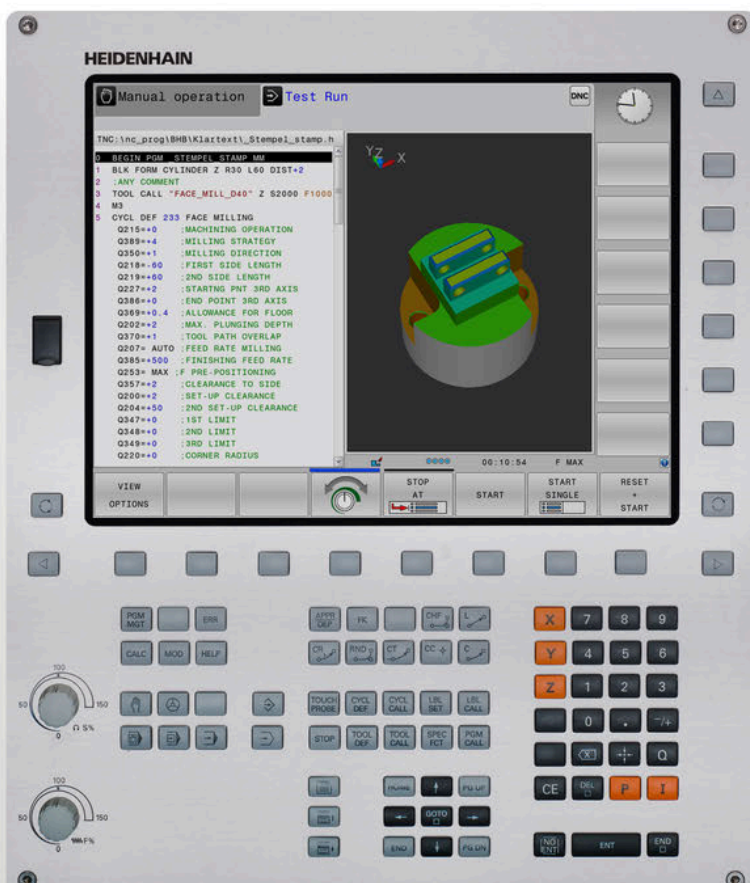




HEIDENHAIN



TNC 320

User's Manual
Setup, Testing and Running
NC Programs







NC Software
771851-06
771855-06

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
	Currently not assigned










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 touch probe cycles
 	Define and call cycles
 	Enter and call subprograms and program section repeats
	Enter program stop in an NC program

Program path contours

Key	Function
	Contour approach and departure
	FK free contour programming
	Straight line
	Circle center/pole for polar coordinates
	Circular arc with center
	Circular arc with radius
	Circular arc with tangential transition
 	Chamfer/rounding arc

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed
	

Contents

1 Fundamentals.....21

2 First steps.....35

3 Fundamentals.....47

4 Tools..... 111

5 Setup..... 147

6 Testing and running..... 217

7 Special Functions..... 275

8 MOD functions.....279

9 HEROS functions..... 299

10 Tables and Overviews.....377

1	Fundamentals.....	21
1.1	About this manual.....	22
1.2	Control model, software and features.....	24
	Software options.....	25
	New functions 77185x-05.....	27
	New functions 77185x-06.....	31

2	First steps.....	35
2.1	Overview.....	36
2.2	Switching on the machine.....	37
	Acknowledging the power interruption and moving to the reference points.....	37
2.3	Graphically testing the workpiece.....	38
	Selecting the Test Run operating mode.....	38
	Selecting the tool table.....	38
	Selecting the NC program.....	39
	Selecting the screen layout and the view.....	39
	Starting the test run.....	40
2.4	Setting up tools.....	41
	Selecting the Manual operation mode.....	41
	Preparing and measuring tools.....	41
	Editing the TOOL.T tool table.....	42
	Editing the TOOL_PTCH pocket table.....	43
2.5	Workpiece setup.....	44
	Selecting the correct operating mode.....	44
	Clamping the workpiece.....	44
	Presetting with a 3-D touch probe.....	45
2.6	Machining the workpiece.....	46
	Selecting the Program run, single block or Program run, full sequence operating mode.....	46
	Selecting the NC program.....	46
	Starting an NC program.....	46

3	Fundamentals.....	47
3.1	The TNC 320.....	48
	HEIDENHAIN Klartext and DIN/ISO.....	48
	Compatibility.....	48
	Data security and data protection.....	49
3.2	Visual display unit and operating panel.....	51
	Display screen.....	51
	Setting the screen layout.....	52
	Control panel.....	52
	Screen keypad.....	53
3.3	Modes of operation.....	54
	Manual Operation and El. Handwheel.....	54
	Positioning with Manual Data Input.....	54
	Programming.....	55
	Test Run.....	55
	Program Run, Full Sequence and Program Run, Single Block.....	56
3.4	Status displays.....	57
	General status display.....	57
	Additional status displays.....	59
3.5	File management.....	64
	Files.....	64
	Displaying externally generated files on the control.....	66
	Directories.....	66
	Paths.....	66
	Calling the file manager.....	67
	Additional functions.....	68
	Selecting drives, directories and files.....	69
	Choosing one of the last files selected.....	71
	USB devices on the control.....	72
	Data transfer to or from an external data carrier.....	73
	The control in a network.....	74
	Data backup.....	76
	Importing a file from an iTNC 530.....	76
	Additional tools for management of external file types.....	77
3.6	Error messages and help system.....	86
	Error messages.....	86
	TNCguide context-sensitive help system.....	91
3.7	NC fundamentals.....	97
	Position encoders and reference marks.....	97

Programmable axes.....	98
Reference systems.....	99

3.8 Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels.....109

3-D touch probes.....	109
HR electronic handwheels.....	110

4	Tools.....	111
4.1	Tool data.....	112
	Tool number, tool name.....	112
	Tool length L.....	112
	Tool radius R.....	112
	Tool table fundamentals.....	113
	Entering tool data into the table.....	117
	Importing tool tables.....	122
	Overwriting tool data from an external PC.....	124
	Pocket table for tool changer.....	125
	Tool change.....	128
	Tool usage test.....	129
4.2	Tool management.....	132
	Basics.....	132
	Calling tool management.....	133
	Editing tool management.....	134
	Available tool types.....	138
	Importing and exporting tool data.....	139
4.3	Tool carrier management.....	142
	Fundamentals.....	142
	Saving tool carrier templates.....	142
	Assigning input parameters to tool carriers.....	143
	Allocating parameterized tool carriers.....	146

5	Setup.....	147
5.1	Switch-on, switch-off.....	148
	Switch-on.....	148
	Traversing reference points.....	150
	Switch-off.....	152
5.2	Moving the machine axes.....	153
	Note.....	153
	Moving the axis with the axis direction keys.....	153
	Incremental jog positioning.....	154
	Traverse with electronic handwheels.....	155
5.3	Spindle speed S, feed rate F and miscellaneous function M.....	165
	Application.....	165
	Entering values.....	165
	Changing the spindle speed and feed rate.....	166
	Feed rate limit F MAX.....	166
5.4	Preset management.....	168
	Note.....	168
	Saving presets in the table.....	169
	Protecting presets from being overwritten.....	173
	Activating a preset.....	175
5.5	Presetting without a 3-D touch probe.....	176
	Note.....	176
	Preparation.....	176
	Presetting with an end mill.....	177
	Using touch probe functions with mechanical probes or measuring dials.....	178
5.6	Using a 3-D touch probe.....	179
	Introduction.....	179
	Overview.....	180
	Suppressing touch probe monitoring.....	182
	Functions in touch probe cycles.....	182
	Selecting the probing cycle.....	185
	Recording measured values from the touch probe cycles.....	185
	Writing measured values from the touch probe cycles to a datum table.....	186
	Writing measured values from the touch probe cycles to the preset table.....	187
5.7	Calibrating 3-D touch probes.....	188
	Introduction.....	188
	Calibrating the effective length.....	189
	Calibrating the effective radius and compensating center misalignment.....	190
	Displaying calibration values.....	193

5.8	Compensating workpiece misalignment with 3-D touch probe.....	194
	Introduction.....	194
	Measuring a basic rotation.....	195
	Saving the basic rotation in the preset table.....	195
	Compensation of workpiece misalignment by rotating the table.....	196
	Showing basic rotation and offset.....	197
	Rescinding basic rotation or offset.....	197
	Measuring 3-D basic rotation.....	198
5.9	Presetting with a 3-D touch probe.....	201
	Overview.....	201
	Presetting on any axis.....	202
	Corner as preset.....	203
	Circle center as preset.....	204
	Setting a center line as preset.....	207
	Measuring workpieces with a 3-D touch probe.....	208
5.10	Tilting the working plane (option 8).....	211
	Application, function.....	211
	Position display in a tilted system.....	212
	Limitations on working with the tilting function.....	212
	Activating manual tilting:.....	213
	Setting the tool-axis direction as the active machining direction.....	216
	Setting a preset in a tilted coordinate system.....	216

6	Testing and running.....	217
6.1	Graphics.....	218
	Application.....	218
	VIEW OPTIONS.....	219
	Tool.....	220
	View.....	221
	Rotating, zooming and moving a graphic.....	223
	Speed of the setting test runs.....	224
	Repeating graphic simulation.....	225
	Shifting the sectional plane.....	225
6.2	Measurement of machining time.....	226
6.3	Showing the workpiece blank in the working space.....	227
	Application.....	227
6.4	Measuring.....	229
	Application.....	229
6.5	Optional program-run interruption.....	230
	Application.....	230
6.6	Skipping NC blocks.....	231
	Test Run and Program Run.....	231
	Positioning w/ Manual Data Input.....	232
6.7	Test run.....	233
	Application.....	233
	Test run execution.....	235
	Executing a Test Run up to a certain NC block.....	236
	GOTO function.....	237
	Display of NC programs.....	238
6.8	Program run.....	239
	Application.....	239
	Running an NC program.....	240
	Structuring NC programs.....	241
	Checking and changing Q parameters.....	242
	Interrupting, stopping or aborting machining.....	244
	Moving the machine axes during an interruption.....	247
	Resuming program run after an interruption.....	249
	Retraction after a power interruption.....	250
	Any entry into the NC program: Block scan.....	253
	Returning to the contour.....	258

6.9	Functions for program display.....	259
	Overview.....	259
6.10	Automatic program start.....	260
	Application.....	260
6.11	Positioning w/ Manual Data Input operating mode.....	261
	Positioning with manual data input (MDI).....	262
	Saving NC programs from \$MDI.....	264
6.12	Entering miscellaneous functions M and STOP.....	265
	Fundamentals.....	265
6.13	Miscellaneous functions for program run inspection, spindle and coolant.....	267
	Overview.....	267
6.14	Miscellaneous functions for coordinate entries.....	268
	Programming machine-referenced coordinates: M91/M92.....	268
	Moving to positions in a non-tilted coordinate system with a tilted working plane: M130.....	270
6.15	Miscellaneous functions for path behavior.....	271
	Superimposing handwheel positioning during program run: M118.....	271
	Deleting basic rotation: M143.....	273
	Automatically retracting the tool from the contour at an NC stop: M148.....	274

7	Special Functions.....	275
7.1	Defining a counter.....	276
	Application.....	276
	Define FUNCTION COUNT.....	277

8	MOD functions.....	279
8.1	MOD function.....	280
	Selecting MOD functions.....	280
	Changing the settings.....	280
	Exiting MOD functions.....	280
	Overview of MOD functions.....	281
8.2	Display software numbers.....	282
	Application.....	282
8.3	Entering the code number.....	282
	Application.....	282
	Functions provided to the machine tool builder by the code number dialog.....	283
8.4	Loading the machine configuration.....	284
	Application.....	284
8.5	Selecting the position display.....	285
	Application.....	285
8.6	Setting the unit of measure.....	287
	Application.....	287
8.7	Graphic settings.....	288
8.8	Setting the counter.....	289
8.9	Changing the machine settings.....	290
	Selecting the kinematics.....	290
	Entering traverse limits.....	291
	Generating a tool usage file.....	292
	Permitting or restricting external access.....	292
8.10	Configuring the HR 550FS radio handwheel.....	295
	Application.....	295
	Assigning the handwheel to a specific handwheel holder.....	295
	Setting the transmission channel.....	296
	Selecting the transmitter power.....	296
	Statistical data.....	297
8.11	Changing the system settings.....	298
	Set the system time.....	298
8.12	Displaying operating times.....	298
	Application.....	298

9	HEROS functions.....	299
9.1	Remote Desktop Manager (option 133).....	300
	Introduction.....	300
	Configuring connections – Windows Terminal Service (RemoteFX).....	301
	Configuring the connection – VNC.....	304
	Shutting down or rebooting an external computer.....	305
	Starting and stopping the connection.....	307
9.2	Additional tools for ITCs.....	308
9.3	Window manager.....	310
	Overview of taskbar.....	311
	Portscan.....	314
	Remote Service.....	315
	Printer.....	317
	SELinux security software.....	319
	State Reporting Interface (option 137).....	320
	VNC.....	323
	Backup and restore.....	326
9.4	Firewall.....	329
	Application.....	329
9.5	Setting up data interfaces.....	332
	Serial interfaces on the TNC 320.....	332
	Application.....	332
	Setting the RS-232 interface.....	332
	Set BAUD RATE (baud rate no. 106701).....	332
	Set protocol (protocol no. 106702).....	333
	Set data bits (dataBits no. 106703).....	333
	Check parity (parity no. 106704).....	333
	Set stop bits (stopBits no. 106705).....	333
	Set handshake (flowControl no. 106706).....	334
	File system for file operation (fileSystem no. 106707).....	334
	Block check character (bccAvoidCtrlChar no. 106708).....	334
	Condition of RTS line (rtsLow no. 106709).....	334
	Define behavior after receipt of ETX (noEotAfterEtx no. 106710).....	335
	Settings for the transmission of data using PC software TNCserver.....	335
	Setting the operating mode of the external device (fileSystem).....	335
	Software for data transfer.....	336
9.6	Ethernet interface.....	338
	Introduction.....	338
	Connection possibilities.....	338
	General network settings.....	338
	Device-specific network settings.....	345

9.7	SELinux security software.....	347
9.8	User administration.....	348
	Configuring the user administration.....	349
	Local LDAP database.....	351
	LDAP on remote computer.....	352
	Connection to a Windows domain.....	353
	Creating further users.....	356
	Access rights.....	359
	HEIDENHAIN function users.....	360
	Definition of roles.....	361
	Rights.....	365
	DNC connection with user authentication.....	366
	Login to user administration.....	370
	Switching or logging out users.....	371
	Screensaver with lockout.....	372
	HOME directory.....	373
	Current User.....	374
	Dialog for requesting additional rights.....	375
9.9	Changing the HEROS conversational language.....	376

10	Tables and Overviews.....	377
10.1	Machine-specific user parameters.....	378
	Application.....	378
	List of user parameter.....	381
10.2	Pin layout and cables for data interfaces.....	393
	RS-232-C/V.24 interface for HEIDENHAIN devices.....	393
	Non-HEIDENHAIN devices.....	395
	Ethernet interface RJ45 socket.....	395
10.3	Specifications.....	396
	User functions.....	398
	Accessories.....	402
10.4	Differences between the TNC 320 and the iTNC 530.....	403
	Comparison: Specifications.....	403
	Comparison: Data interfaces.....	403
	Comparison: PC software.....	404
	Comparison: User functions.....	404
	Comparison: Touch probe cycles in the Manual operation and Electronic handwheel operating modesElectronic handwheel.....	409
	Comparison: Differences in programming.....	410
	Comparison: Differences in Test Run, functionality.....	413
	Comparison: Differences in Test Run, operation.....	414
	Comparison: Differences in Manual Operation, functionality.....	415
	Comparison: Differences in Manual Operation, operation.....	416
	Comparison: Differences in Program Run, operation.....	416
	Comparison: Differences in Program Run, traverse movements.....	417
	Comparison: Differences in MDI operation.....	422
	Comparison: Differences in programming station.....	422

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 320	771851-06
TNC 320 Programming Station	771855-06

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:

- Tool measurement with the TT

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.



Cycle Programming User's Manual:

All of the cycle functions (touch probe cycles and fixed cycles) are described in the **Cycle Programming** User's Manual. If you need this User's Manual, please contact HEIDENHAIN.
ID: 1096959-xx



User's Manuals for Conversational Programming and ISO Programming:

All information regarding NC programming (except touch probe cycles and fixed cycles) is provided in the **Conversational Programming** and **ISO Programming** User's Manuals. If you need these User's Manuals, please contact HEIDENHAIN.
ID for Conversational Programming: 1096950-xx
ID for ISO Programming: 1096983-xx

Software options

The TNC 320 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
------------------------	----------------------------------

Advanced Function Set 1 (option 8)

Expanded functions Group 1	Machining with rotary tables <ul style="list-style-type: none"> ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute Coordinate conversions: Tilting the working plane
-----------------------------------	--

HEIDENHAIN DNC (option 18)

Communication with external PC applications over COM component

CAD Import (option 42)

CAD import	<ul style="list-style-type: none"> ■ Support for DXF, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs
-------------------	--

Extended Tool Management (option 93)

Extended tool management	Python-based
---------------------------------	--------------

Remote Desktop Manager (option 133)

Remote operation of external computer units	<ul style="list-style-type: none"> ■ Windows on a separate computer unit ■ Incorporated in the control's interface
--	--

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 77185x-05

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

- New **FUNCTION COUNT** function for controlling a counter.
- New **FUNCTION LIFTOFF** function for retracting the tool from the contour upon an NC stop.
- It is possible to comment out NC blocks.
- The CAD viewer exports points with **FMAX** to an H file.
- When multiple instances of the CAD viewer are open, they are shown somewhat smaller on the third desktop.
- The CAD viewer now enables you to extract data from STEP, IGES and STEP files.
- 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.
- If a pallet table is selected in a Program Run operating mode, the **Tooling list** and **T usage order** are calculated for the entire pallet table, see "Tool management", Page 132
- You can also open the tool-carrier files in the file management, see "Tool carrier management", Page 142
- With the **ADAPT NC PGM / TABLE** function, you can also import and modify freely definable tables, see "Importing tool tables", Page 122
- 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 122
- A quick search for the tool name is possible in the tool table, see "Entering tool data into the table", Page 117
- The machine tool builder can disable the setting of presets in individual axes, see "Saving presets in the table", Page 169, see "Presetting with a 3-D touch probe ", Page 201
- Line 0 of the preset table can also be edited manually, see "Saving presets in the table", Page 169
- 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 57
- 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 227
- The tool data of touch probes can also be displayed and entered in the tool management (option 93), see "Editing tool management", Page 134
- With the **TCH PROBE MONITOR OFF** soft key you can suppress touch-probe monitoring for 30 seconds, see "Suppressing touch probe monitoring", Page 182
- During manual probing **ROT** and **P**, workpiece misalignment can be compensated by aligning a rotary table, see "Compensation of workpiece misalignment by rotating the table", Page 196, see "Corner as preset", Page 203

- 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 378
- 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 378
- New optional machine parameter **CfgDisplayCoordSys** (no. 127500) for selecting the coordinate system in which a datum shift is to be shown in the status display, see "Machine-specific user parameters", Page 378

Modified functions 77185x-05**Further information:** User's Manual for **Conversational Programming** or **ISO Programming**

- 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 233
- The control provides a positioning logic for returning to the contour, see "Returning to the contour", Page 258
- The positioning logic for returning to the contour with a replacement tool has changed, see "Tool change", Page 128
- 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 253
- Axes that are not active in the current kinematic model can also be referenced in a tilted working plane, see "Crossing the reference point in a tilted working plane", Page 151
- 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 220
- 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 225
- 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 165
- 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 86
- To connect a USB stick you no longer have to press a soft key, see "Connecting and removing USB storage devices", Page 68
- The speed of setting the jog increment, spindle speed and feed rate was adjusted for electronic handwheels.
- The icons of basic rotation, 3-D basic rotation and tilted working plane were modified to make them easier to distinguish, see "General status display", Page 57

- The control automatically recognizes whether a table is to be imported or the table format is to be adapted, see "Importing tool tables", Page 122
- When you place the cursor in an input field of the tool management, the entire input field is highlighted.
- 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 150
- The control issues a warning if the handwheel potentiometers are still active when the handwheel is deactivated, see "Traverse with electronic handwheels", Page 155
- 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"
- 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 119
- The machine tool builder can change the simulated tool change position, see "Test run", Page 233
- 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 378

New and modified cycle functions 77185x-05

Further information: Cycle Programming User's Manual

- New Cycle 441 **FAST PROBING**. With this cycle you can set various touch probe parameters (e.g. positioning feed rate) that are globally effective for all subsequently used touch probe cycles.
- The parameters Q215, Q385, Q369 and Q386 were added to Cycles 256 **RECTANGULAR STUD** and 257 **CIRCULAR STUD**.
- The feed rate behavior in Cycles 205 and 241 was changed.
- 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.
- Cycle 225 Engraving can engrave the current counter reading by using a new syntax.
- New SERIAL column in the touch probe table.
- Enhancement of the contour train: Cycle 25 with Residual Material Machining, Cycle 276 3-D Contour Train.

New functions 77185x-06

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

- It is now possible to work with cutting data tables.
- New **PLANE XY ZX YZ** soft key for selecting the working plane during FK programming.
- 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.
- In the CAD viewer, you can define the preset or the datum by directly entering the values in the list view window.
- 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 348
- With the new HOST COMPUTER MODE function, you can turn command over to an external host computer, see "Permitting or restricting external access", Page 292
- 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 320
- The basic rotation is taken into account in the **Manual Operation** mode, see "Activating manual tilting:", Page 213
- The screen layout soft keys were adapted, see "Modes of operation", Page 54
- The additional status display shows the path and angle tolerances without Cycle 32 being active, see "Additional status displays", Page 59
- 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 73.
- In the **Positioning w/ Manual Data Input** operating mode, you can now skip NC blocks, see "Skipping NC blocks", Page 231
- Two new tool types have been added to the tool table: **Ball-nose cutter** and **Toroid cutter**, see "Available tool types", Page 138
- During probing in a plane (Probing PL) you can select the solution when aligning the rotary axes, see "Measuring 3-D basic rotation", Page 198
- The appearance of the **Optional program run stop** has changed, see "Optional program-run interruption", Page 230

- 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 72
- 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 165
- 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 76
- New machine parameter **CfgProgramCheck** (no. 129800) for defining settings for the tool usage files, see "List of user parameter", Page 381

Modified functions 77185x-06

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

- The **PLANE** functions provide the alternative selection option **SYM** in addition to **SEQ**.
- The cutting data calculator has been improved.
- The **CAD-Viewer** now outputs **PLANE SPATIAL** instead of **PLANE VECTOR**.
- The **CAD-Viewer** now outputs 2-D contours by default.
- When programming straight-line blocks, the **&Z** option is no longer shown by default.
- 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.
- The control issues an error message if you combine an FK block with M89.
- 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 237
- If axis angle not equal to tilt angle, the control no longer issues an error message during presetting with manual probing functions, but opens the **Working plane is inconsistent** menu, see "Using a 3-D touch probe ", Page 179
- The **ACTIVATE PRESET** soft key also updates the values of a line activated in the preset management, see "Activating a preset", Page 175
- 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 59

- The control allows updating of the web browser, see "Additional tools for management of external file types", Page 77
- The Remote Desktop Manager allows you to enter an additional waiting time for the shutdown connection, see "Shutting down or rebooting an external computer", Page 305
- The obsolete tool types were removed from the tool table. The **Undefined** type is assigned to any existing tools of this tool type see "Available tool types", Page 138
- In the expanded tool management, you can now go to the context-sensitive on-line help even while editing the tool form.
- The screensaver glideshow was removed.
- The machine tool builder can specify which M functions to allow in the **Manual Operation** mode, see "Application", Page 165
- 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 117

New and modified cycle functions 77185x-06

Further information: Cycle Programming User's Manual

- New Cycle 1410 PROBING ON EDGE.
- New Cycle 1411 PROBING TWO CIRCLES.
- New Cycle 1420 PROBING IN PLANE.
- Automatic Touch Probe Cycles 408 to 419 take `chkTiltingAxes` (no. 204600) into account during presetting.
- Touch Probe Cycles 41x, automatic preset measurement: New behavior of cycle parameters Q303 MEAS. VALUE TRANSFER and Q305 NUMBER IN TABLE.
- In Cycle 420 MEASURE ANGLE, the data from the cycle and the touch probe table is taken into account during prepositioning.
- The REACTION column was added to the touch probe table.
- In Cycle 24 SIDE FINISHING, a tangential helix is used for approaching and departing in the last infeed.
- Parameter Q367 SURFACE POSITION was added to Cycle 233 FACE MILLING.
- Cycle 257 CIRCULAR STUD also uses Q207 FEED RATE FOR MILLING for roughing.
- Machine parameter `CfgThreadSpindle` (no. 113600) is available for use.

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 User's Manuals for Conversational Programming and ISO Programming cover the following topics:

- Switching on the machine
- Programming the workpiece

2.2 Switching on the machine

Acknowledging the power interruption and moving to the reference points

DANGER

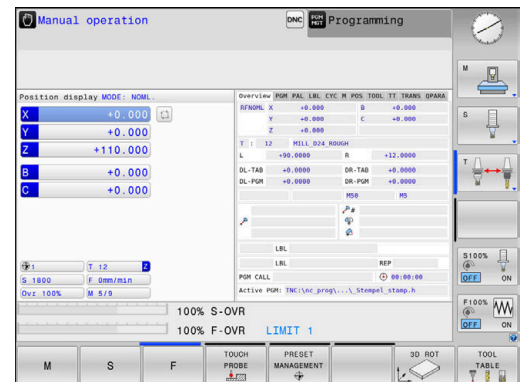
Caution: Danger for the operator!

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.

I

- ▶ Switch on the machine control voltage
- ▶ The control checks operation of the emergency stop circuit and goes into Reference Run mode.



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

Further information on this topic

- Approaching reference points
Further information: "Switch-on", Page 148
- Operating modes
Further information: "Programming", Page 55

2.3 Graphically testing the workpiece

Selecting the Test Run operating mode

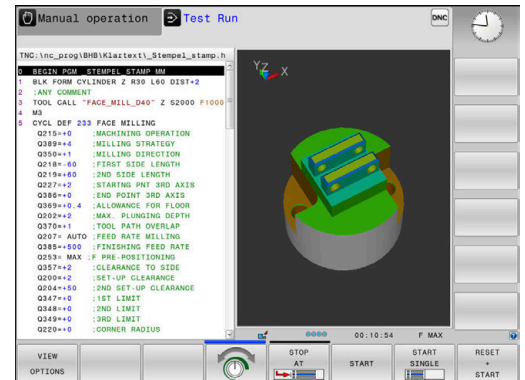
You can test NC programs in the **Test Run** operating mode:



- ▶ 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 54
- Testing NC programs
Further information: "Test run", Page 233



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 117
- Testing NC programs
Further information: "Test run", Page 233

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 218
- Performing a test run
Further information: "Test run", Page 233

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 233
- Graphic functions
Further information: "Graphics ", Page 218
- Adjusting the simulation speed
Further information: "Speed of the setting test runs", Page 224

2.4 Setting up tools

Selecting the Manual operation mode

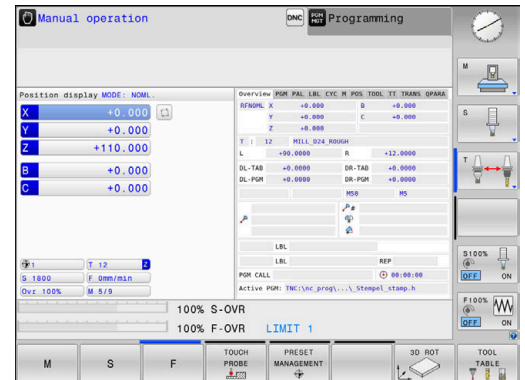
Tools are set up in the **Manual operation** mode:



- ▶ 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 54



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: store the tools in the tool changer

Further information: "Editing the TOOL_PTCH pocket table", Page 43

Editing the TOOL.T tool table



Refer to your machine manual.

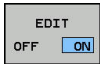
The procedure for calling the tool management may differ from that described below.

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	NULLWKRZUG	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 54
- Working with the tool table
Further information: "Entering tool data into the table", Page 117
- Using the tool management (option 93)
Further information: "Calling tool management", Page 133

Editing the TOOL_P.TCH pocket table



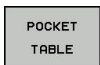
Refer to your machine manual.

The function of the pocket table depends on the machine.

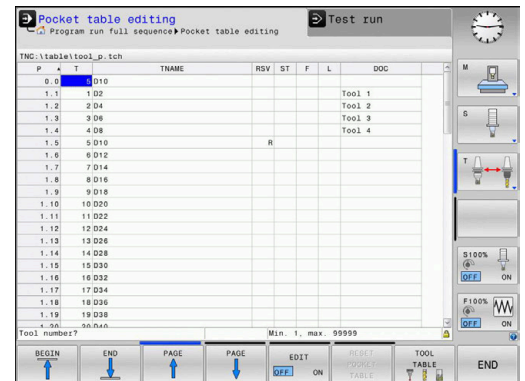
In the pocket table TOOL_P.TCH (permanently saved under **TNC:\table**) you specify which tools your tool magazine contains. To enter data in the pocket table TOOL_P.TCH, proceed as follows:



- ▶ 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 54
- Working with the pocket table
Further information: "Pocket table for tool changer", Page 125

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 153

Clamping the workpiece

Mount the workpiece with a fixture on the machine table. If you have a 3-D touch probe on your machine, then you do not need to clamp the workpiece parallel to the axes.

If you do not have a 3-D touch probe available, you have to align the workpiece so that it is fixed with its edges parallel to the machine axes.

Further information on this topic

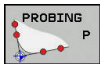
- Presetting with a 3-D touch probe
Further information: "Presetting with a 3-D touch probe ", Page 201
- Presetting without 3-D touch probe
Further information: "Presetting without a 3-D touch probe", Page 176

Presetting with a 3-D touch probe

- ▶ 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.



- ▶ Set the preset at a workpiece corner, for example
- ▶ 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.
- ▶ Use the axis direction keys to pre-position the touch probe near the second touch point on the first workpiece edge
- ▶ 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.
- ▶ Use the axis direction keys to pre-position the touch probe near the first touch point on the second 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.
- ▶ Use the axis direction keys to pre-position the touch probe near the second touch point on the second workpiece edge
- ▶ 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 corner point.



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

Further information on this topic

- Presetting
Further information: "Presetting with a 3-D touch probe ",
 Page 201

2.6 Machining the workpiece

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

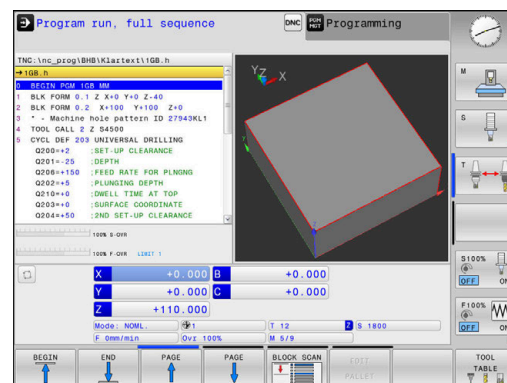
You can run NC programs either in the **Program run, single block** or the **Program run, full sequence** operating mode:



- ▶ 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 54
- Running NC programs
Further information: "Program run", Page 239

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 239

3

Fundamentals

3.1 The TNC 320

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional milling and drilling operations right at the machine in easy-to-use Klartext conversational language. They are designed for milling, drilling and boring machines, as well as machining centers, with up to 6 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 and DIN/ISO

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. If no NC-dimensioned drawing is available, then the FK free contour programming will help. Workpiece machining can be graphically simulated either during a test run or during a program run.

It is also possible to program in ISO format or DNC mode.

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

Further information: User's Manual for Conversational Programming or ISO Programming

Compatibility

NC programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 320. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.



Please also note the detailed description of the differences between the iTNC 530 and the TNC 320.

Further information: "Differences between the TNC 320 and the iTNC 530", Page 403

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 319
- Firewall
Further information: "Firewall", Page 329
- Sandbox
Further information: "Sandbox tab", Page 344
- Integrated browser
Further information: "Displaying Internet files", Page 80
- Management of external access
Further information: "Permitting or restricting external access", Page 292
- Monitoring of TCP and UDP ports
Further information: "Portscan", Page 314
- Remote diagnosis
Further information: "Remote Service", Page 315
- User administration
Further information: "User administration", Page 348

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 available either as a compact version or with a separate display unit and operating panel. Both variants of the control come with a 15-inch TFT color flat-panel display.

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 (exception: if the control only displays graphics).

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

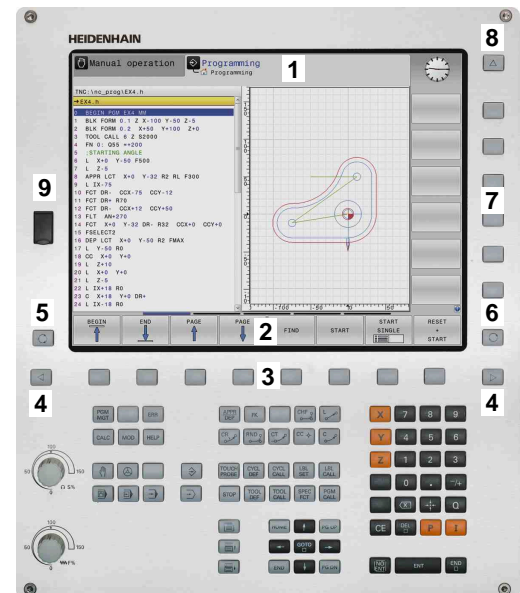
5 Setting the screen layout

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 54



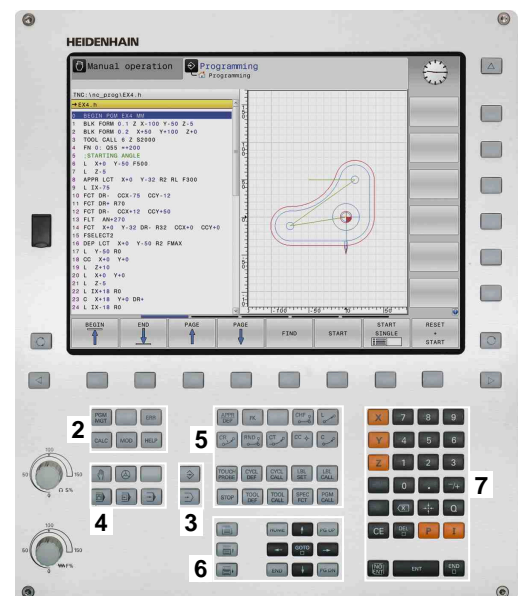
- Select the desired screen layout with a soft key

Control panel

The TNC 320 is delivered with an integrated operating panel. As an alternative, the TNC 320 is also available with a separate display unit and an operating panel with an alphabetic keyboard.

- 1 Alphabetic keyboard for entering texts and file names, as well as for ISO programming
- 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 and axis selection
- 10 Machine operating panel

More information Machine manual



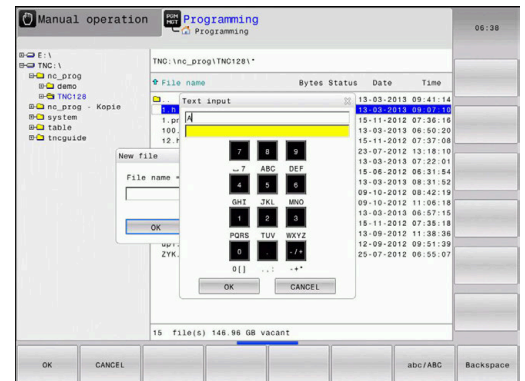
Refer to your machine manual.

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

If you are using the compact version of the control (without alphabetic keyboard), you can enter letters and special characters with the screen keypad or 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, set the presets and tilt the working plane.

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)

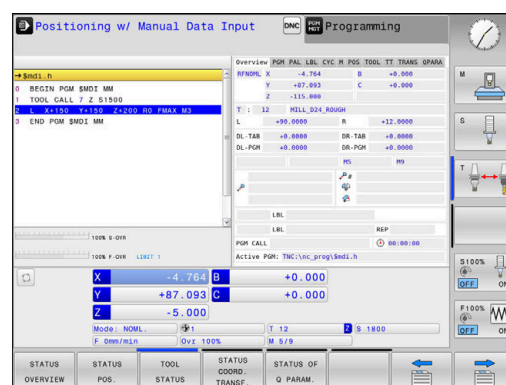
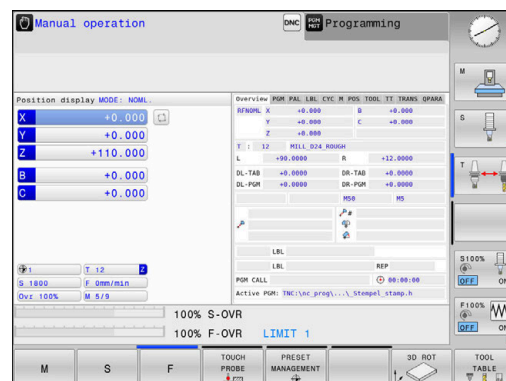
Soft key	Window
POSITION	Positions
POSITION + STATUS	Left: positions, right: status display
POSITION + WORKPIECE	Left: positions, right: workpiece
POSITION + MACHINE	Left: positions, right: collision objects and workpiece

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

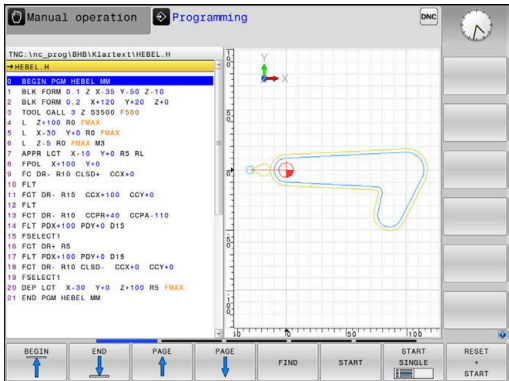


Programming

In this mode of operation you create NC programs. The FK free programming feature, 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

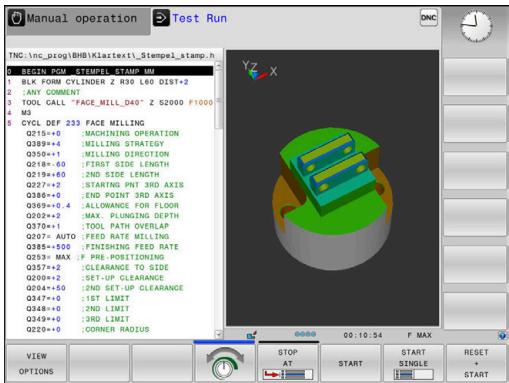


Test Run

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



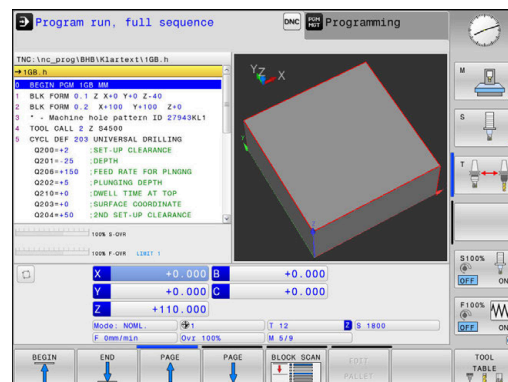
Program Run, Full Sequence and Program Run, Single Block

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

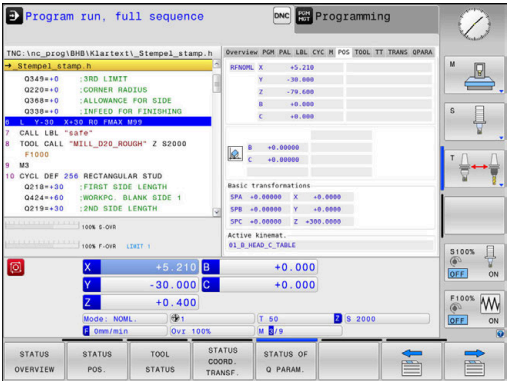











If the **GRAPHICS** screen layout is selected the status display is not shown.

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: Actual, nominal or distance-to-go coordinates mode
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
	Number of the active preset from the preset table. If the preset was set manually, the control displays the text MAN behind the symbol
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
	A basic rotation is active in the active preset
	Axes are moving, taking the basic rotation into account
	A 3-D basic rotation is active in the active preset
	Axes are moving, taking the 3-D basic rotation into account



Icon	Meaning
	Axes are moving in a tilted working plane
	Axes are mirrored and moved
	The function for traversing in the tool-axis direction is active
	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
	NC program stopped, e.g. in Program run, full sequence operating mode after pressing the NC stop key For safety reasons, the control permits no handling in this condition
	NC program interrupted, e.g. in Positioning w/ Manual Data Input operating mode, following the error-free execution of an NC block 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! Further information: "Positioning w/ Manual Data Input operating mode", Page 261 and "Program-controlled interruptions", Page 245
	NC program aborted or terminated
	Pulsing spindle speed function is active



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.

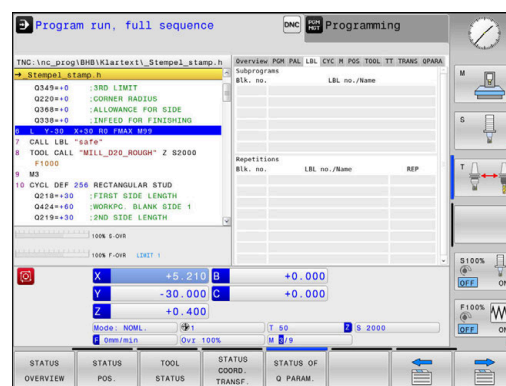
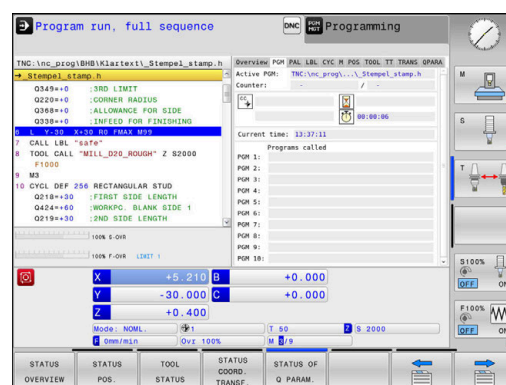
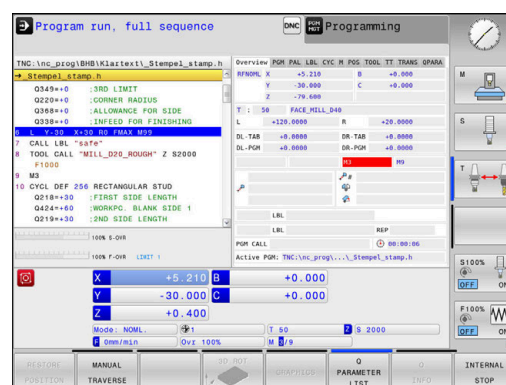
Soft key	Meaning
	Position display
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	NC program called with PGM CALL
	Current machining time
	Name and path of the active main program

General program information (PGM tab)

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

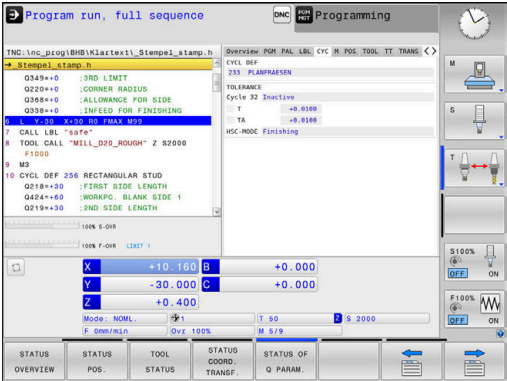
Program section repeats and subprograms (LBL tab)

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



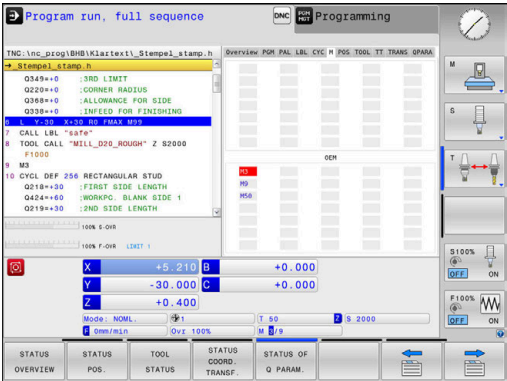
Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active fixed cycle
	Active path and angle tolerances
	Depending on which path and angle tolerances are active, the following values are displayed:
	<div><div>■ Values of Cycle 32, Tolerance</div><div>■ Values defined by the machine tool builder</div></div>



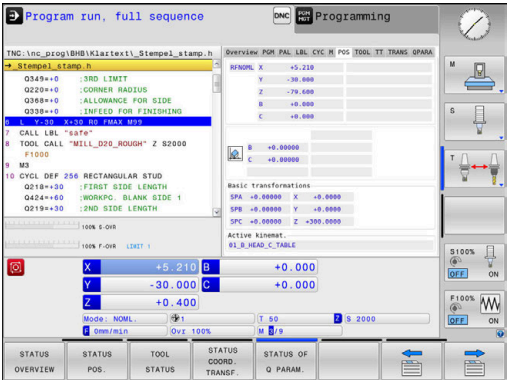
Active miscellaneous functions M (M tab)

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
	Tilt angle of the working plane
	Angle of basic transformations
	Active kinematics



Information on tools (TOOL tab)

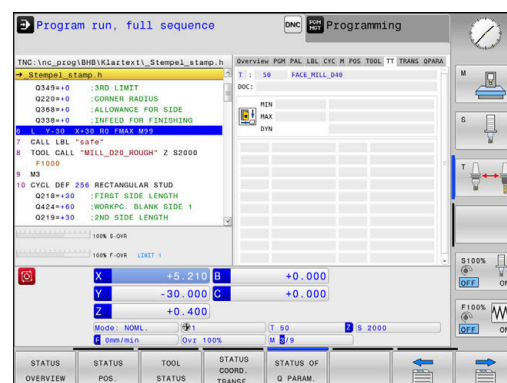
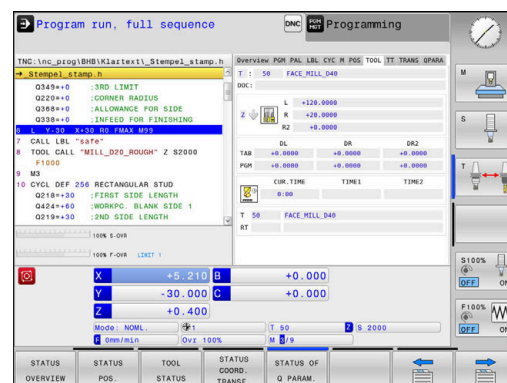
Soft key	Meaning
	Display of active tool: <ul style="list-style-type: none"> ■ T: Tool number and tool name ■ RT: Number and name of a replacement tool
	Tool axis
	Tool length and tool radii
	Oversizes (delta values) from the tool table (TAB) and the TOOL CALL (PGM)
	Tool life, maximum tool life (TIME 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
STATUS COORD. TRANSF.	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 8 axes
	Mirrored axes (Cycle 8)
	Active rotation angle (Cycle 10)
	Active scaling factor/factors (Cycle 11 / 26); the control displays an active scaling factor in up to 6 axes
	Scaling datum



In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

Further information: Cycle Programming User's Manual

Further information: User's Manual for Conversational Programming or ISO Programming

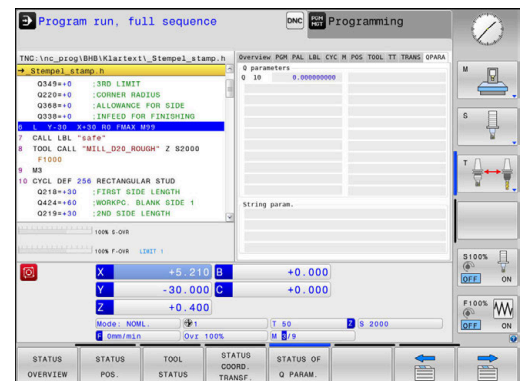
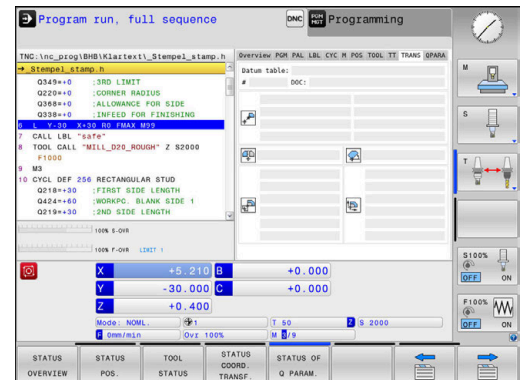
Displaying Q parameters (QPARA tab)

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



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
in DIN/ISO format	.I
Compatible NC programs	
HEIDENHAIN unit programs	.HU
HEIDENHAIN contour programs	.HC
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
CAD files as	
ASCII files	.DXF .IGES .STEP

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 66

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 77

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 \.



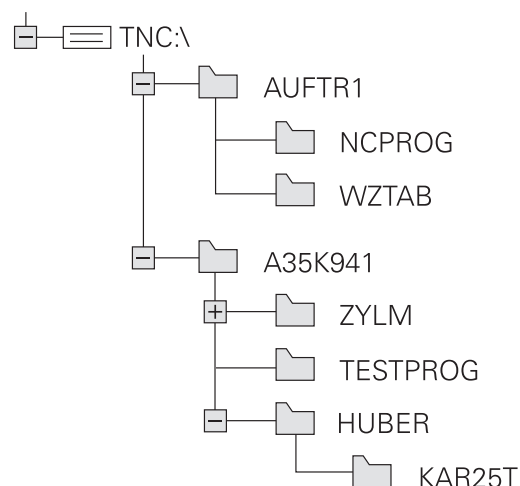
The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

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

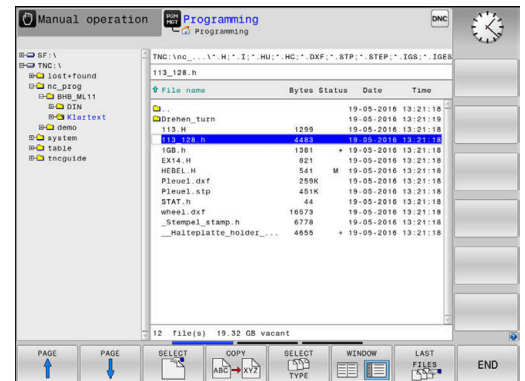


- Press the **PGM MGT** key
- The control displays the file management window (see figure for default setting. If the control displays a different screen layout, press the **WINDOW** soft key).

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



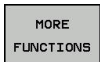
- The file is tagged with the "protected" symbol.



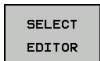
- Cancel file protection:
Press the **UNPROTECT** soft key

Selecting the editor

- Place the cursor on the file you want to open



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

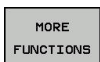


- Select the editor:
Press the **SELECT EDITOR** soft key
- Mark the desired editor
 - **TEXT EDITOR** for text files, e.g. **.A** or **.TXT**
 - **PROGRAM EDITOR** for NC programs **.H** and **.I**
 - **TABLE EDITOR** for tables, e.g. **.TAB** or **.T**
 - **BPM EDITOR** for pallet tables **.P**
- Press 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 72

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



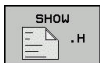
- Press the **ENT** key

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

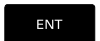


- Use wildcards, e.g. **4*.h**: Show all files of type .h starting with a 4

- 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.

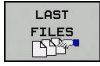


If you enter the first letter of the file you are looking for in the file manager, the cursor automatically jumps to the first NC program with the same letter.

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:



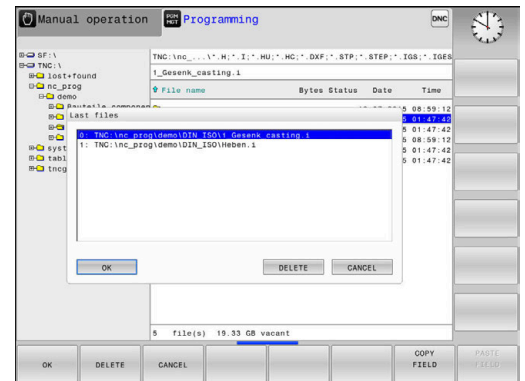
- Moves the cursor up and down within a window



- 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 319

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.

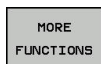
Your machine tool builder can assign permanent names for USB devices.

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 332



- ▶ 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.

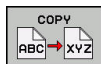
If you wish to copy from the external data medium to the control, move the cursor in the right window to the file to be transferred.



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



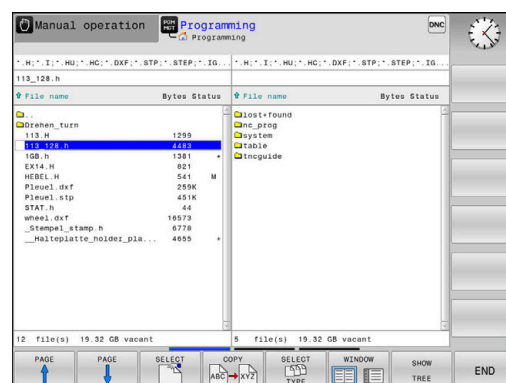
- ▶ Press the **SHOW FILES** soft key
- ▶ Use the arrow keys to select the desired file
- ▶ Press the **COPY** soft key



- ▶ 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.

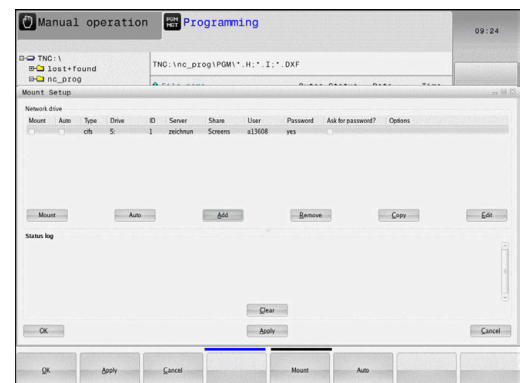
Further information: "Ethernet interface ", Page 338

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

 A small rectangular button with the text "PGM MGT" inside.

- ▶ Press the **PGM MGT** key


 A small rectangular button with the text "NET" inside.

- ▶ 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 326

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 320, 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 320 and saves the changes to the selected file.

Further information: "Importing tool tables", Page 122

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 78
Excel spreadsheets (xls, csv)	Page 79
Internet files (htm, html)	Page 80
ZIP archives (zip)	Page 82
Text files (ASCII files, e.g. txt, ini)	Page 83
Video files (ogg, oga, ogv, ogx)	Page 84
Graphic files (bmp, gif, jpg, png)	Page 84





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**).

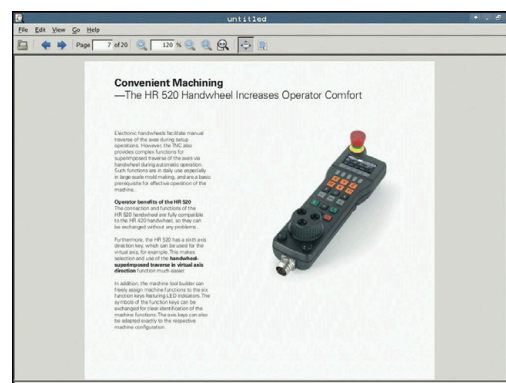
Displaying PDF files

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



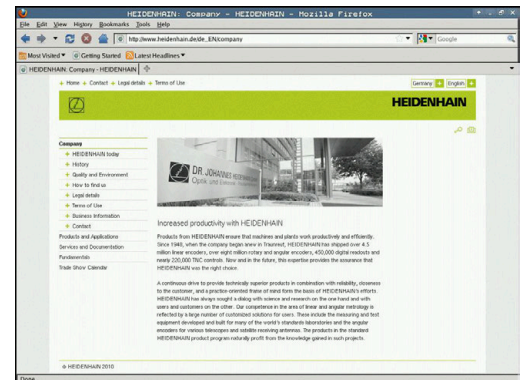
The network must ensure protection against viruses and malicious software. The same applies to access to the Internet or other networks.

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 344



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.

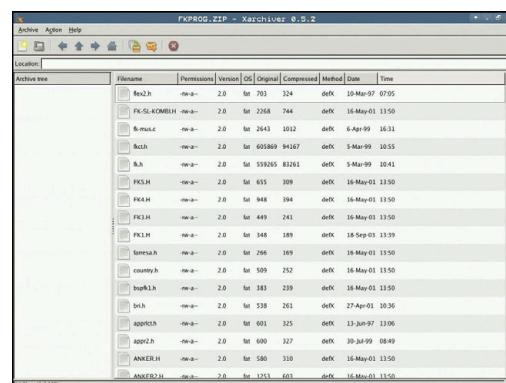
ENT



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.

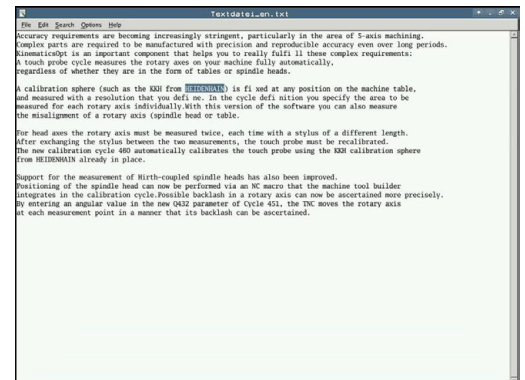
ENT



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



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

Closing the error window



- ▶ Press the **END** soft key, or



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

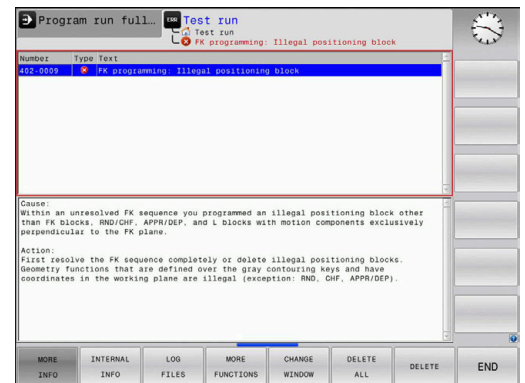
Detailed error messages

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

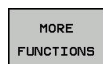


- Detailed information about the error message: Position the cursor on the error message and press the **INTERNAL INFO** soft key
- The control opens a window with internal information about the error.
- Exit the details: Press the **INTERNAL INFO** soft key again

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

CE

- ▶ 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

DELETE

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

**DELETE
ALL**

- ▶ 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.

**LOG
FILES**

- ▶ Press the **LOG FILES** soft key

**ERROR
LOG**

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

**PREVIOUS
FILE**

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





**CURRENT
FILE**

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

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









Keystroke log

The control stores each key pressed and important events (e.g. system start) in a keystroke log. The capacity of the keystroke log is limited. 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

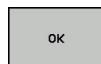
- Open the error window



- 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

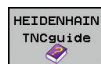
Calling the TNCguide help system

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.



- Call the help for HEIDENHAIN error messages



- Call the help for HEIDENHAIN machine-specific error messages, if available

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 96

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.

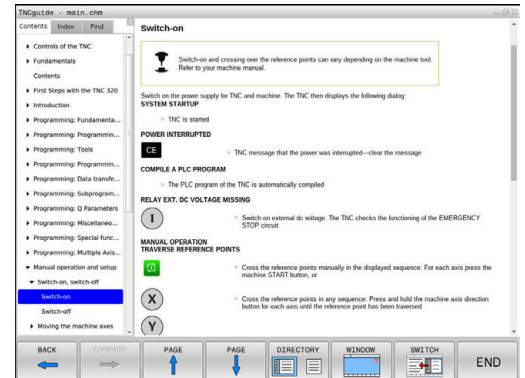
The following user documentation is available in TNCguide:

- Conversational Programming User's Manual (**BHBKlartext.chm**)
- ISO User's Manual (**BHBIso.chm**)
- User's Manual for Setup, Testing and Running NC Programs (**BHBoperate.chm**)
- User's Manual for Cycle Programming (**BHBtchprobe.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



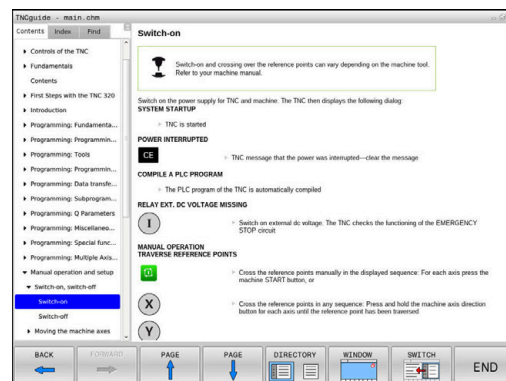
On the Windows programming station, the TNCguide is opened in the internally defined standard browser.

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.

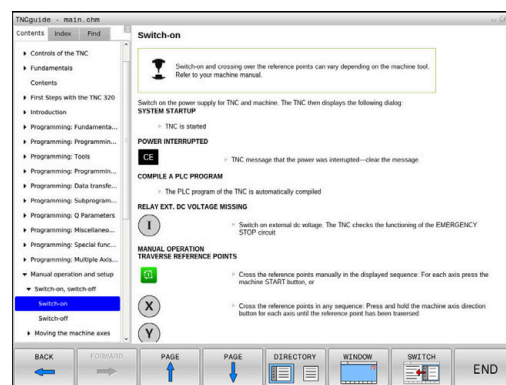
The left side is active.



- ▶ 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



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.

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 300
- ▶ Desired NC software number, e.g. TNC 320 (77185x-06)
- ▶ 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

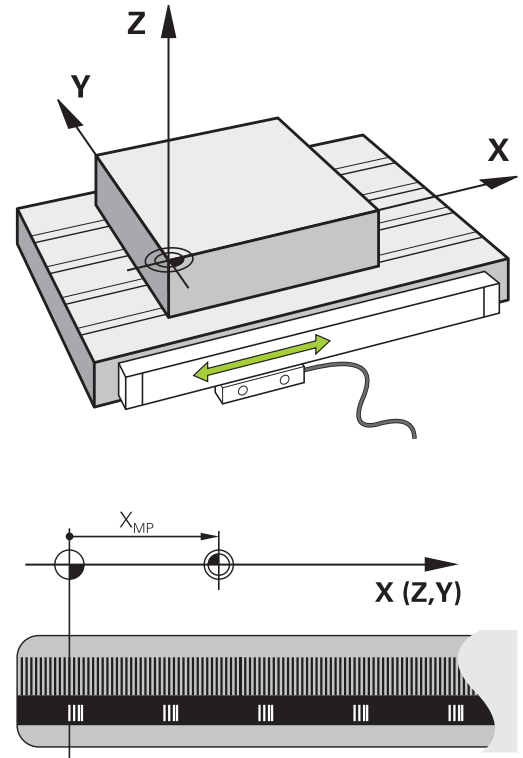
Position encoders and reference marks

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, rotary tables and tilting axes with angle 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, for angle encoders by no more than 20°.

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.



Programmable axes

In the default setting, the programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

The designations of the programmable axes are given in the table below.

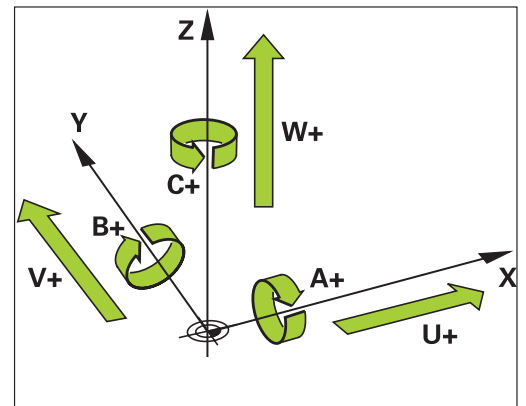
Principal axis	Parallel axis	Rotary axis
X	U	A
Y	V	B
Z	W	C



Refer to your machine manual.

The number, designation and assignment of the programmable axes depend on the machine.

Your machine tool builder can define further axes, such as PLC axes.



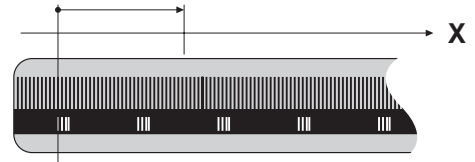
Reference systems

For the control to traverse an axis according to a defined path it requires a **reference system**.

A paraxially mounted linear encoder on a machine tool serves as a simple reference system for linear axes. The linear encoder represents a **number ray**, a unidimensional coordinate system.

To approach a point on the **plane**, the control requires two axes and therefore a reference system with two dimensions.

To approach a point in the **space**, the control requires three axes and therefore a reference system with three dimensions. If these three axes are configured perpendicular to each other this creates a so-called **three-dimensional Cartesian coordinate system**.



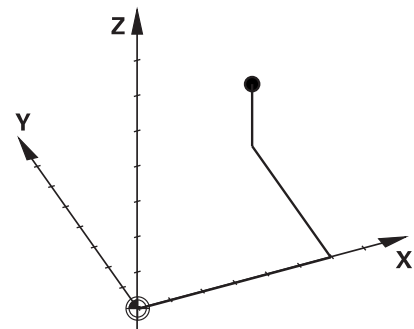
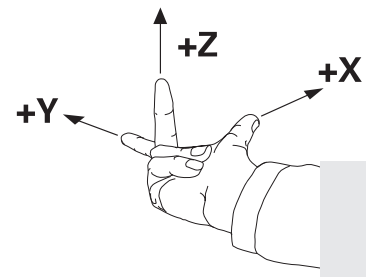
According to the right-hand rule the fingertips point in the positive directions of the three main axes.

For a point to be uniquely determined in space, a **coordinate origin** is needed in addition to the configuration of the three dimensions. The common intersection serves as the coordinate origin in a 3-D coordinate system. This intersection has the coordinates **X+0, Y+0** and **Z+0**.

The control must differentiate between various reference systems for it to always perform a tool change at the same position for example, or carry out a machining operation always related to the current workpiece position.

The control differentiates between the following reference systems:

- Machine coordinate system M-CS:
Machine **C**oordinate **S**ystem
- Basic coordinate system B-CS:
Basic **C**oordinate **S**ystem
- Workpiece coordinate system W-CS:
Workpiece **C**oordinate **S**ystem
- Working plane coordinate system WPLCS:
Working **P**lane **C**oordinate **S**ystem
- Input coordinate system I-CS:
Intput **C**oordinate **S**ystem
- Tool coordinate system T-CS:
Tool **C**oordinate **S**ystem



All reference systems build upon each other. They are subject to the kinematic chain of the specific machine tool.

The machine coordinate system is the reference system.

Machine coordinate system M-CS

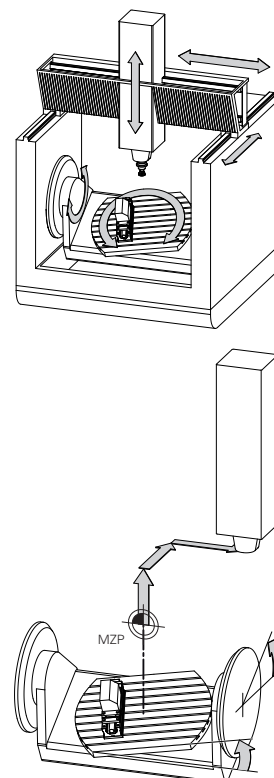
The machine coordinate system corresponds to the description of kinematics and therefore to the actual mechanical design of the machine tool.

Because the mechanics of a machine tool never precisely correspond to a Cartesian coordinate system, the machine coordinate system consists of several one-dimensional coordinate systems. These one-dimensional coordinate systems correspond to the physical machine axes that are not necessarily perpendicular to each other.

The position and orientation of the one-dimensional coordinate systems are defined with the aid of translations and rotations based on the spindle tip in the description of kinematics.

The position of the coordinate origin, the machine datum, is defined by the machine manufacturer during machine configuration. The values in the machine configuration define the zero positions of the encoders and the corresponding machine axes. The machine datum does not necessarily have to be located in the theoretical intersection of the physical axes. It can therefore also be located outside of the traverse range.

Because the machine configuration values cannot be modified by the user, the machine coordinate system is used for determining constant positions, e.g. the tool change point.



Machine datum (MCP)

Soft key	Application
<div>BASE TRANSFORM.</div> <div>OFFSET</div>	The user can define shifts in the machine coordinate system according to the specific axis with use of the OFFSET values of the preset table.



The machine tool builder configures the **OFFSET** columns of the preset management in accordance with the machine.

Further information: "Preset management", Page 168



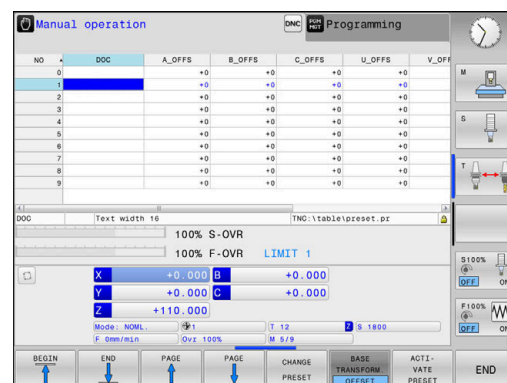
Another feature is **OEM-OFFSET**, which is available only to the machine tool builder. **OEM-OFFSET** can be used to define additive axis shifts for rotary and parallel axes.

The sum of all **OFFSET** values (from all the above **OFFSET** input possibilities) result in the difference between the **ACTL.** position and the **RFACTL** position of an axis.

The control converts all movements in the machine coordinate system, independent of the reference system used for value input.

Example of a 3-axis machine tool with a Y axis as oblique axis, not arranged perpendicularly to the ZX plane:

- ▶ In the **Positioning w/ Manual Data Input** operating mode, run an NC block with **L IY+10**
- > The control determines the required axis nominal values from the defined values.
- > During positioning the control moves the **Y and Z** machine axes.



- > The **RFACTL** and **RFNOML** displays show movements of the Y axis and Z axis in the machine coordinate system.
- > The **ACTL.** and **NOML.** displays only show one movement of the Y axis in the input coordinate system.
- ▶ In the **Positioning w/ Manual Data Input** operating mode, run an NC block with **L IY-10 M91**
- > The control determines the required axis nominal values from the defined values.
- > During positioning the control only moves the **Y** machine axis.
- > The **RFACTL** and **RFNOML** displays only show one movement of the Y axis in the machine coordinate system.
- > The **ACTL.** and **NOML.** displays show movements of the Y axis and Z axis in the input coordinate system.

The user can program positions related to the machine datum, e.g. by using the miscellaneous function **M91**.

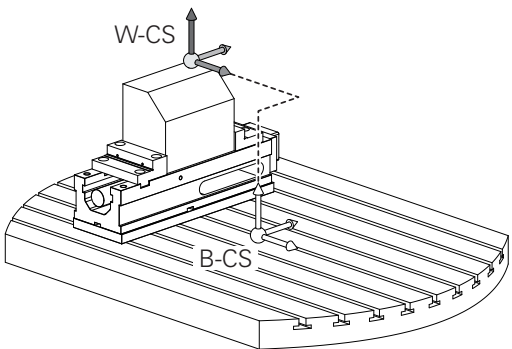
Basic coordinate system B-CS

The basic coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the end of the kinematics model.

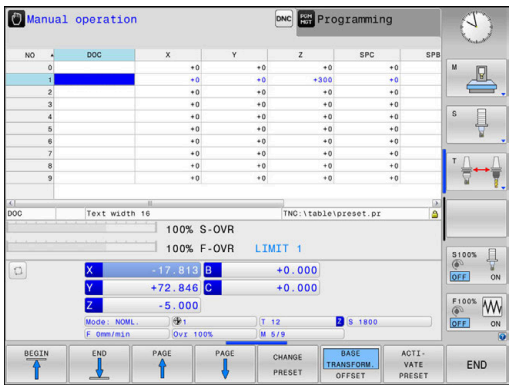
The orientation of the basic coordinate system in most cases corresponds to that of the machine coordinate system. There may be exceptions to this if a machine manufacturer uses additional kinematic transformations.

The kinematic model and thus the position of the coordinate origin for the basic coordinate system is defined by the machine manufacturer in the machine configuration. The user cannot modify the machine configuration values.

The basic coordinate system serves to determine the position and orientation of the workpiece coordinate system.



Soft key	Application
<div> <div>BASE</div> <div>TRANSFORM.</div> <div>OFFSET</div> </div>	<p>The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe for example. The control saves the values determined with respect to the basic coordinate system as BASE TRANSFORM. values in the preset management.</p>
<div> <div>⚙️</div> </div>	<p>The machine tool builder configures the BASE TRANSFORM. columns of the preset management in accordance with the machine.</p>




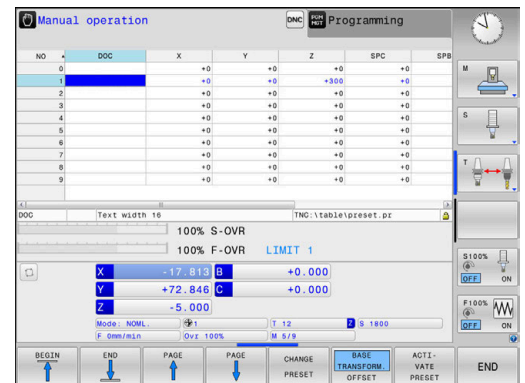
Further information: "Preset management", Page 168

Workpiece coordinate system W-CS

The workpiece coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the active reference point.

The position and orientation of the workpiece coordinate system depend on the **BASE TRANSFORM.** values of the active line in the preset table.

Soft key	Application
	<p>The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe for example. The control saves the values determined with respect to the basic coordinate system as BASE TRANSFORM. values in the preset management.</p>



Further information: "Preset management", Page 168

In the workpiece coordinate system the user defines the position and orientation of the working plane coordinate system with use of transformations.

Transformations in the workpiece coordinate system:

- **3D ROT** functions
 - **PLANE** functions
 - Cycle 19 **WORKING PLANE**
- Cycle 7 **DATUM SHIFT**
(shifting **before** tilting the working plane)
- Cycle 8 **MIRROR IMAGE**
(mirroring **before** tilting the working plane)

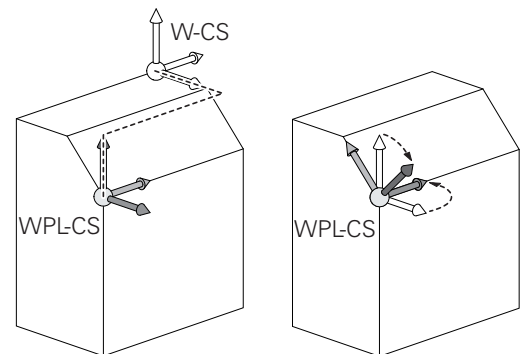
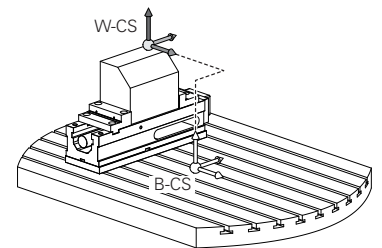


The result of transformations built up on each other depends on the programming sequence.

In every coordinate system, program only the specified (recommended) transformations. This applies to both setting and resetting the transformations. Any other use may lead to unexpected or undesired results. Please observe the following programming notes.

Programming notes:

- Transformations (mirroring and shifting) that are programmed before the **PLANE** functions (except for **PLANE AXIAL**) will change the position of the tilt datum (origin of the working plane coordinate system WPL-CS) and the orientation of the rotary axes
 - If you just program a shift, then only the position of the tilt datum will change
 - If you just program mirroring, then only the orientation of the rotary axes will change
- When used in conjunction with **PLANE AXIAL** and Cycle 19, the programmed transformations (mirroring, rotation and scaling) do not affect the position of the tilt datum or the orientation of the rotary axes





Without active transformations in the workpiece coordinate system, the position and orientation of the working plane coordinate system and workpiece coordinate system are identical.

There are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the working plane coordinate system with this assumption.

Other transformations are of course possible in the working plane coordinate system

Further information: "Working plane coordinate system WPLCS", Page 105

Working plane coordinate system WPL-CS

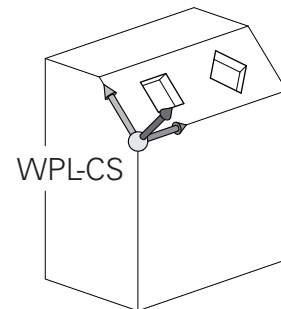
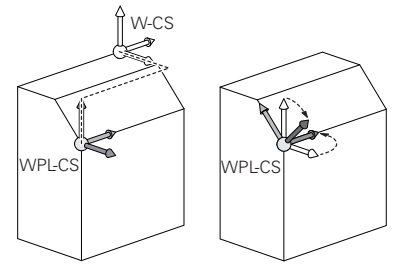
The working plane coordinate system is a 3-D Cartesian coordinate system.

The position and orientation of the working plane coordinate system depend on the active transformations in the workpiece coordinate system.



Without active transformations in the workpiece coordinate system, the position and orientation of the working plane coordinate system and workpiece coordinate system are identical.

There are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the working plane coordinate system with this assumption.



In the working plane coordinate system the user defines the position and orientation of the input coordinate system with use of transformations.

Transformations in the working plane coordinate system:

- Cycle 7 **DATUM SHIFT**
- Cycle 8 **MIRROR IMAGE**
- Cycle 10 **ROTATION**
- Cycle 11 **SCALING**
- Cycle 26 **AXIS-SPECIFIC SCALING**
- **PLANE RELATIVE**



As a **PLANE** function, the **PLANE RELATIVE** is effective in the workpiece coordinate system and aligns the working plane coordinate system.

The values of additive tilting always relate to the current working plane coordinate system.

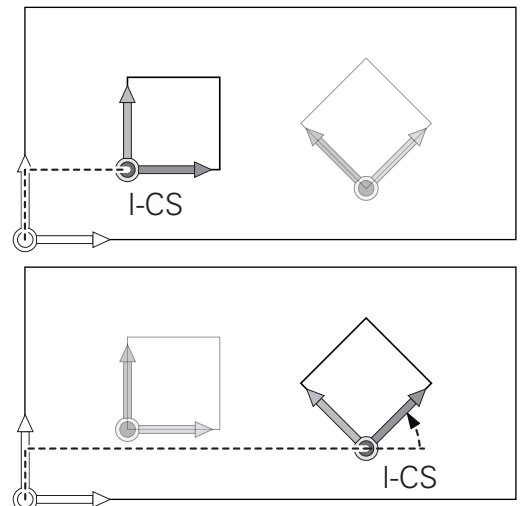


The result of transformations built up on each other depends on the programming sequence.



Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.

There are also no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.



Input coordinate system I-CS

The input coordinate system is a 3-D Cartesian coordinate system. The position and orientation of the input coordinate system depend on the active transformations in the working plane coordinate system.



Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.

There are also no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.

With the aid of positioning blocks in the input coordinate system, the user defines the position of the tool and therefore the position of the tool coordinate system.



The **NOML.**, **ACTL.**, **LAG** and **ACTDST** displays are also based on the input coordinate system.

Positioning blocks in input coordinate system:

- Paraxial positioning blocks
- Positioning blocks with Cartesian or polar coordinates
- Positioning blocks with Cartesian coordinates and surface normal vectors

Example

7 X+48 R+

7 L X+48 Y+102 Z-1.5 R0

7 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007
NZ0.8848844 R0



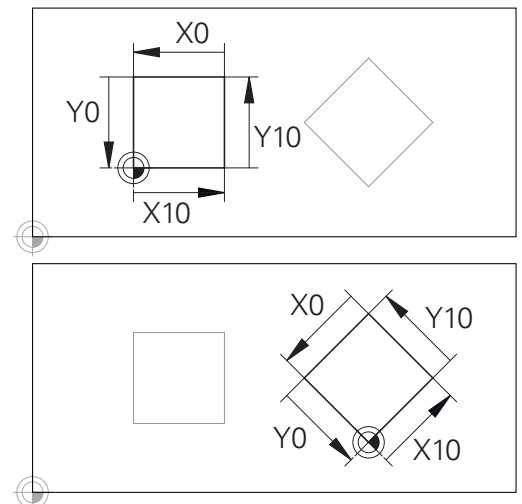
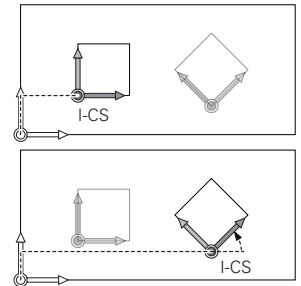
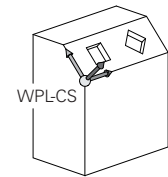
The position of the tool coordinate system is determined by the Cartesian coordinates X, Y and Z also for positioning blocks with surface normal vectors.

In conjunction with 3-D tool compensation, the position of the tool coordinate system can be shifted along the surface normal vectors.



Orientation of the tool coordinate system can be performed in various reference systems.

Further information: "Tool coordinate system T-CS", Page 107



A contour referencing the input coordinate system origin can easily be transformed any way you need.

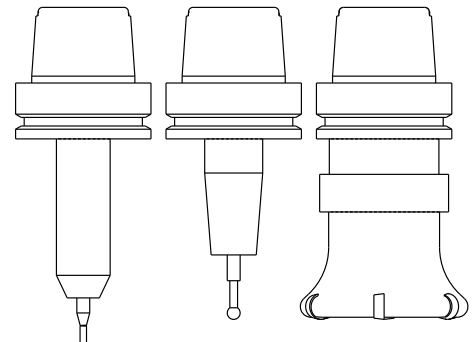
Tool coordinate system T-CS

The tool coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the tool reference point. The values of the tool table, **L** and **R** with milling tools and **ZL**, **XL** and **YL** with turning tools, reference this point.

Further information: "Entering tool data into the table", Page 117

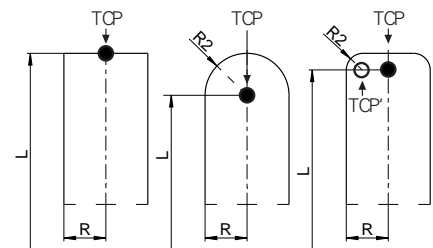
In accordance with the values from the tool table, the coordinate origin of the tool coordinate system is shifted to the tool center point TCP. TCP stands for **T**ool **C**enter **P**oint.

If the NC program does not reference the tool tip, the tool center point must be shifted. The required shift is implemented in the NC program using the delta values during a tool call.



i The position of the TCP as shown in the diagram is obligatory in conjunction with the 3-D tool compensation.

i With the aid of positioning blocks in the input coordinate system, the user defines the position of the tool and therefore the position of the tool coordinate system.



If the **TCPM** function or miscellaneous function **M128** is active, the orientation of the tool coordinate system depends on the tool's current angle of inclination.

The user defines the tool's angle of inclination either in the machine coordinate system or in the working plane coordinate system.

Tool angle of inclination in the machine coordinate system:

Example

```
7 L X+10 Y+45 A+10 C+5 R0 M128
```

Tool angle of inclination in the working plane coordinate system:

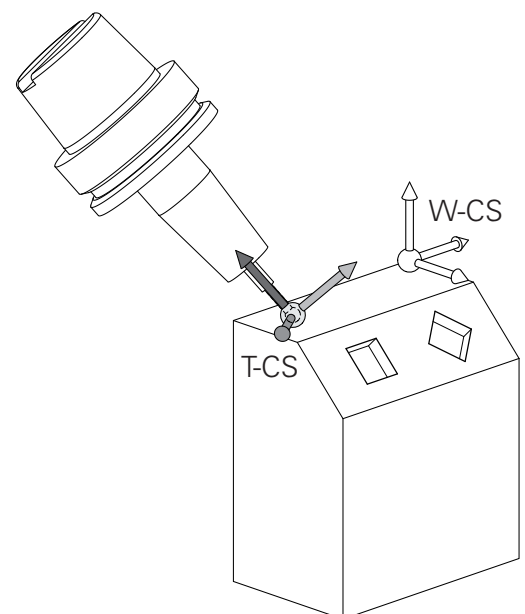
Example

```
6 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS
```

```
7 L A+0 B+45 C+0 R0 F2500
```

```
7 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007
  NZ0.8848844 TX-0.08076201 TY-0.34090025 TZ0.93600126 R0
  M128
```

```
7 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007
  NZ0.8848844 R0 M128
```





With the shown positioning blocks with vectors, 3-D tool compensation is possible with compensation values **DL**, **DR** and **DR2** from the **TOOL CALL** block.

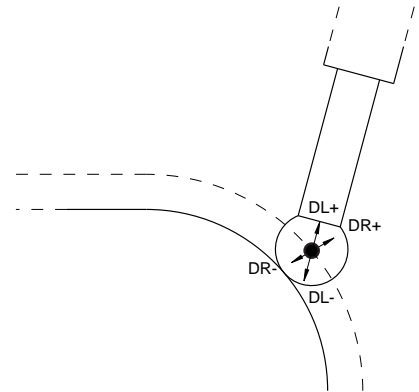
The methods of function of the compensation values depend on the type of tool.

The control detects the various tool types with the columns **L**, **R** and **R2** of the tool table:

- $R2_{TAB} + DR2_{TAB} + DR2_{PROG} = 0$
→ end mill
- $R2_{TAB} + DR2_{TAB} + DR2_{PROG} = R_{TAB} + DR_{TAB} + DR_{PROG}$
→ radius cutter or ball cutter
- $0 < R2_{TAB} + DR2_{TAB} + DR2_{PROG} < R_{TAB} + DR_{TAB} + DR_{PROG}$
→ toroid cutter or toroidal cutter



Without the **TCPM** function or miscellaneous function **M128**, orientation of the tool coordinate system and input coordinate system is identical.



3.8 Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels

3-D touch probes

Applications for HEIDENHAIN 3-D touch probes:

- Automatically align workpieces
- Quickly and precisely set presets
- Measure the workpiece during program run
- Measure and inspect tools



All of the cycle functions (touch probe cycles and fixed cycles) are described in the **Cycle Programming** User's Manual. If you need this User's Manual, please contact HEIDENHAIN.
ID: 1096959-xx

Touch trigger probes TS 260, TS 444, TS 460, TS 642 and TS 740

The TS 248 and TS 260 touch probes are particularly cost-effective and transmit the trigger signals via a cable.

The wireless TS 740 and TS 642 touch probes as well as the smaller TS 460 and TS 444 touch probes are suitable for use on machines with tool changers. All of the above touch probes feature infrared signal transmission. TS 460 also supports wireless transmission and offers optional collision protection. Thanks to an integrated air turbine generator, the TS 444 touch probe is battery-free.

HEIDENHAIN touch trigger probes feature either a wear-resistant optical switch or several high-precision pressure sensors (TS 740) that detect 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 probes TT 160 and TT 460

The TT 160 and TT 460 touch probes are 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. The TT 460 supports infrared and radio transmission.



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: "Traverse with electronic handwheels",
Page 155



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

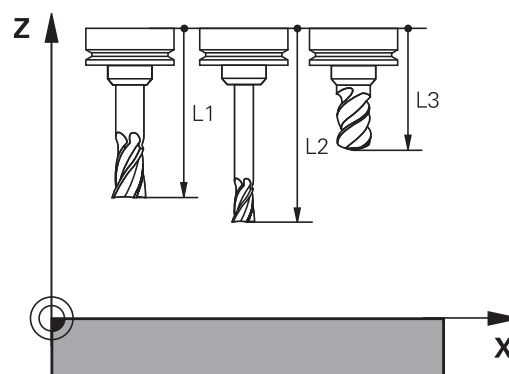
The control automatically replaces lowercase letters with corresponding uppercase letters during saving.

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. The entire tool length is essential for the control in order to perform numerous functions involving multi-axis machining.



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 114

- If your machine has an automatic tool changer
- If you want to apply fine roughing with the fixed cycle 22

Further information: Cycle Programming User's Manual

- If you want to work with fixed cycles 251 to 254

Further information: Cycle Programming User's Manual

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.

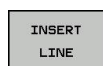
Other settings, such as **SORT/ HIDE COLUMNS**, can be made after the file has been opened.

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 3-D radius compensation or 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 for Cycles 22, 233, 256, 257	Tooth length in the tool axis?
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	Maximum plunge angle?
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?

Abbr.	Inputs	Dialog
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]
LIFTOFF	Definition of whether the control should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour. If Y is defined, the control retracts the tool from the contour, provided M148 has been activated. Further information: "Automatically retracting the tool from the contour at an NC stop: M148", Page 274	Retract allowed? Yes=ENT/ No=NOENT
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. Is used by the Centering cycle (Cycle 240) in order to calculate the centering depth from the diameter entry	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
PTYP	Tool type for evaluation in the pocket table Function is defined by the machine manufacturer. Refer to your machine manual.	Tool type for pocket table?
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 146	Tool-carrier kinematics
OVRTIME	Time for exceeding the tool life in minutes Further information: "Overtime for tool life", Page 129 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?



For a description of the cycles governing automatic tool measurement,

Further information: Cycle Programming User's Manual

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:

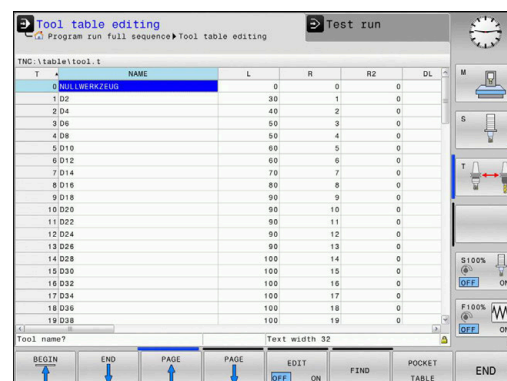
- Select any machine operating mode



- 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 320, you have to adapt its format and content before you can use the tool table. On the TNC 320, 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 320 and saves the changes to the selected file.

Proceed as follows:

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



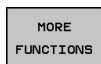
- ▶ Select the **Programming** operating mode



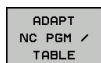
- ▶ Press the **PGM MGT** key



- ▶ 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!

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

Further information: User's Manual for Conversational Programming or ISO Programming

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 336

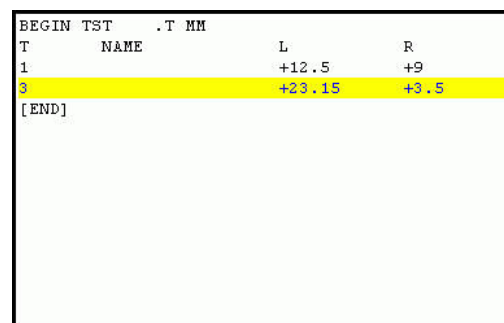
This application case occurs if you wish to determine tool data on an external tool presetter and then transfer this to the control.

Requirements

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

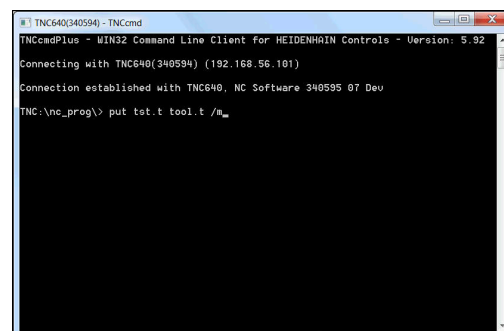
Procedure

- ▶ 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: User's Manual for Conversational Programming or ISO Programming

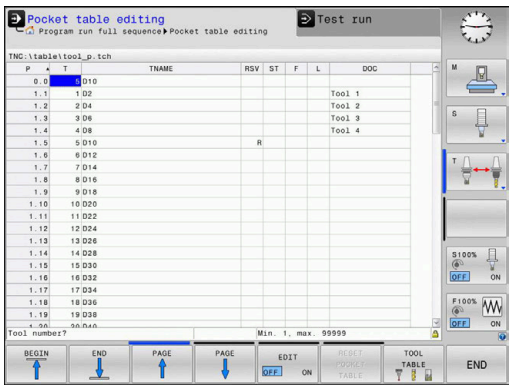
Pocket table for tool changer




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.

Automatic tool change if the tool life expires: M101



Refer to your machine manual.

The function of **M101** can vary depending on the individual machine tool.

When the specified tool life has expired, the control can automatically insert a replacement tool and continue machining with it. Activate the miscellaneous function **M101** for this. **M101** is reset with **M102**.

Enter the respective tool life after which machining is to be continued with a replacement tool in the **TIME2** column of the tool table. In the **CUR_TIME** column the control enters the current tool life.

If the current tool life is higher than the value entered in the **TIME2** column, a replacement tool will be inserted at the next possible point in the program no later than one minute after expiration of the tool life. The change is made only after the NC block has been completed.

NOTICE

Danger of collision!

During an automatic tool change with **M101**, the control always retracts the tool in the tool axis first. There is danger of collision when retracting tools for machining undercuts, such as side milling cutters or T-slot milling cutters!

- Deactivate the tool change with **M102**

After the tool change the control positions the tool according to the following logic, unless otherwise specified by the machine tool builder:

- If the target position in the tool axis is below the current position, the tool axis is positioned last
- If the target position in the tool axis is above the current position, the tool axis is positioned first

Prerequisites for a tool change with M101



As replacement tools, use only tools with the same radius. The control does not automatically check the radius of the tool.

If you want the control to check the radius of the replacement tool, enter **M108** in the NC program.

The control performs the automatic tool change at a suitable point in the program. The automatic tool change is not performed:

- During execution of fixed cycles
- While radius compensation (**RR/RL**) is active
- Directly after an approach function **APPR**
- Directly before a departure function **DEP**
- Directly before and after **CHF** and **RND**
- During execution of macros
- During execution of a tool change
- Directly after a **TOOL CALL** or **TOOL DEF**
- During execution of SL cycles

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 292

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

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

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**

There are two ways to run a tool usage test for a pallet file:

- If the cursor in the pallet file is on a pallet entry, the control runs the tool usage test for the entire pallet.
- If the cursor in the pallet file is on a program entry, the control runs the tool usage test only for the selected NC program.

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.

TOOL
USAGE

▶ Press the **TOOL USAGE** soft key

TOOL
USAGE
TEST

▶ Press the **TOOL USAGE TEST** soft key

> The control opens the **Tool usage test** pop-up window indicating the result of the usage test.

OK

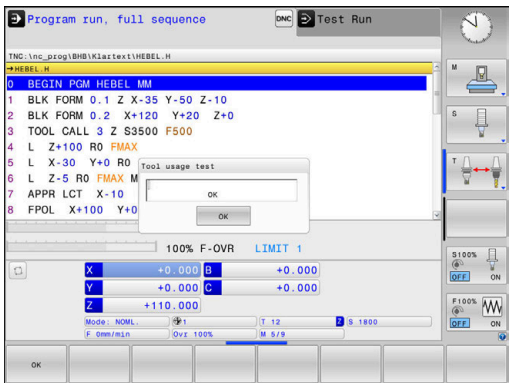
▶ Press the **OK** soft key

> The control closes the pop-up window.

ENT

▶ Alternative: Press the **ENT** key

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



4.2 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:

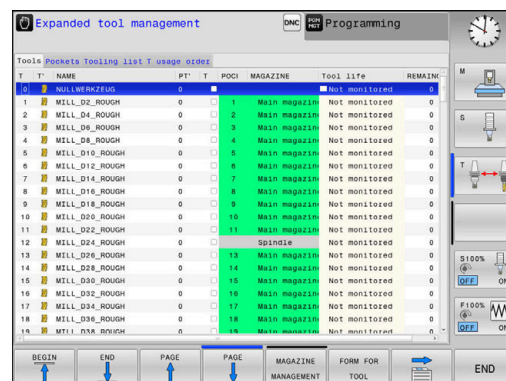
- Display and editing of all tool data from the tool table and the touch probe table
- Easily readable and adaptable representation of the tool data in fillable forms
- Any description of the individual tool data in the new table view
- Mixed representation of data from the tool table and the pocket 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

The Expanded Tool Management (option 93) additionally provides:


- Program-specific or pallet-specific usage sequence of all tools
- Program-specific or pallet-specific list of all available tools



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.**






Calling tool management

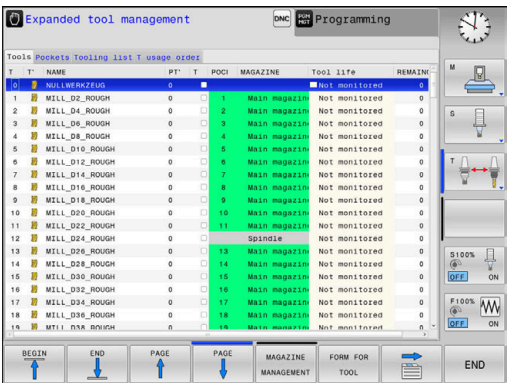


Refer to your machine manual.

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

- 
- ▶ Select the tool table: Press the **TOOL TABLE** soft key
- 
- ▶ Scroll through the soft-key row
- 
- ▶ Press the **TOOL MANAGEMENT** soft key

▶ The control switches to the new table view.




Tool management view

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

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

The Expanded Tool Management (option 93) additionally provides:
















- **Tooling list:** List of all tools in the NC program that is selected in the Program Run mode (only if you have already created a tool usage file)
Further information: "Tool usage test", Page 129
- **T usage order:** List of the sequence of all tools that are inserted in the NC program selected in the Program Run mode (only if you have already created a tool usage file)
Further information: "Tool usage test", Page 129

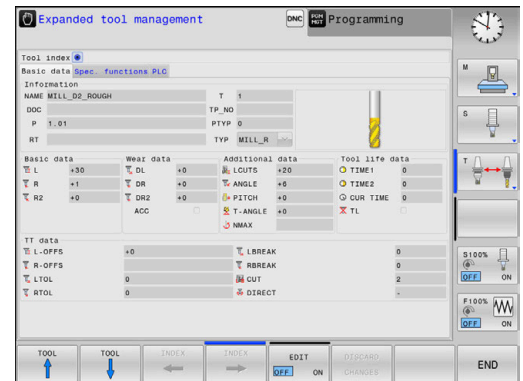


If a pallet table is selected in the Program Run operating mode, the **Tooling list** and **T usage order** are calculated for the entire pallet table.

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 Option 93 additionally provides: Tooling list and T usage order
	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.

Available tool types

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

Icon	Tool type	Tool type number
	Undefined,****	99
	Milling cutter,MILL	0
	Roughing cutter,MILL_R	9
	Finishing cutter,MILL_F	10
	Ball-nose cutter, BALL	22
	Toroid cutter, TORUS	23
	Drill,DRILL	1
	Tap,TAP	2
	Center drill,CENT	4
	Touch probe,TCHP	21
	Ream,REAM	3
	Countersink,CSINK	5
	Piloted counterbore,TSINK	6
	Boring tool,BOR	7
	Back boring tool,BCKBOR	8
	Thread mill,GF	15
	Thread mill w/ countersink,GSF	16
	Thread mill w/ single thread,EP	17
	Thread mill w/ indxbl insert,WSP	18
	Thread milling drill,BGF	19
	Circular thread mill,ZBGF	20

Importing and exporting tool data

Importing tool data



Refer to your machine manual.

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

Using this function you can simply import tool data that you have measured externally on a presetting device, for example. The file to be imported must have the CSV format (comma separated value). The **CSV** file format describes the structure of a text file for exchanging simply structured data. Accordingly, the import file must have the following structure:

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

Follow the steps outlined below for importing:

- ▶ Copy the tool table to be imported to the **TNC:\system\tooltab** directory on the hard disk of the control
- ▶ Start expanded tool management
- ▶ Press the **TOOL IMPORT** soft key in the tool management
- ▶ The control shows a pop-up window with the CSV files that are saved in the **TNC:\system\tooltab** directory
- ▶ Use the arrow keys or mouse to select the file to be imported and confirm with the **ENT** key
- ▶ The control shows a pop-up window with the content of the CSV file
- ▶ Start the import procedure with the **EXECUTE** soft key.



- The CSV file to be imported must be stored in the **TNC:\system\tooltab** directory.
- If you import the tool data of existing tools (whose numbers are in the pocket table) the control issues an error message. You can then decide whether to skip this data record or insert a new tool. The control inserts a new tool into the first empty line of the tool table.
- If the imported CSV file contains unknown table columns, the control displays a message during import. An additional note informs you that the data will not be transferred.
- Make sure that the column designations have been specified correctly.
Further information: "Entering tool data into the table", Page 117
- You can import any tool data, the associated data record does not have to contain all the columns (or data) of the tool table.
- The column names can be in any order, the data must be defined in the corresponding order.

Example

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

Exporting tool data

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

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

Follow the steps outlined below for exporting:

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



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

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.

Along with software option number 8, **Advanced Function Set 1**, you can tilt the working plane to the angle of the removable angled heads and thus keep working with the **Z** tool axis.

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!

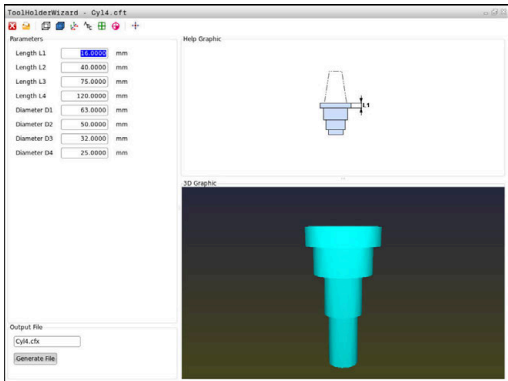
Assigning input parameters to tool carriers

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

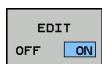
Proceed as follows to parameterize tool carrier templates and save these parameters:



- ▶ 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!

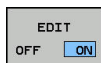
Proceed as follows to allocate a parameterized tool carrier to a tool:



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



- ▶ Press the **TOOL TABLE** soft key



- ▶ Press the **EDIT** soft key



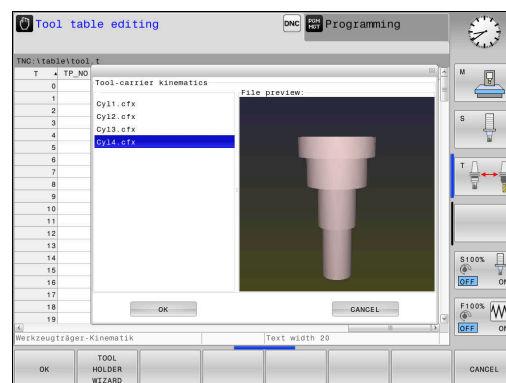
- ▶ Move the cursor to the **KINEMATIC** column of the required tool



- ▶ 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.

NOTICE

Danger of collision!

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

- ▶ If possible, reset the tilted condition before switching the machine off
- ▶ Check the tilted condition when switching the machine back on

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.

Crossing the reference point in a tilted working plane

If the **Tilt working plane** function was active before the control was switched off, then the control automatically activates the function after restarting. This means that movements via the axis keys take place in the tilted working plane.

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

Further information: "Activating manual tilting:", Page 213

NOTICE**Danger of collision!**

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

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



If the machine does not have any absolute encoders, the position of the rotary axes must be confirmed. The position shown in the pop-up window is the last position before the control was switched off.

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 move several axes at a time with these two methods. The control then shows the feed rate. 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 165

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.



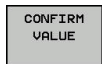
- ▶ Operating mode: Press the **Manual operation** or **Electronic handwheel** key



- ▶ Shift the soft-key row



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



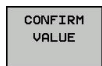
- ▶ Enter the infeed of the **linear axes** and confirm with the **CONFIRM VALUE** soft key



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



- ▶ Use the arrow keys to position the cursor on the **rotary axis**



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



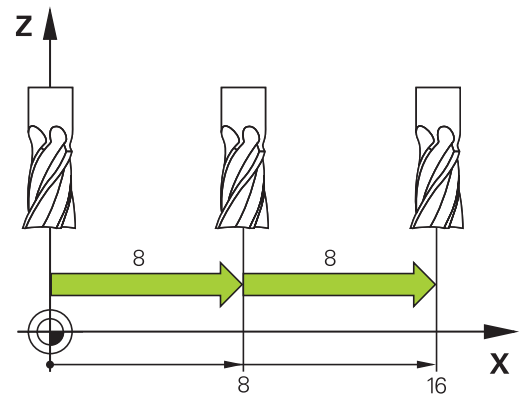
- ▶ Alternatively, confirm with the **ENT** key



- ▶ 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 electronic 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 510: Simple handwheel without display, data transfer via cable
- HR 520: Handwheel with display, data transfer via cable
- HR 550FS: Handwheel with display, data transmission via radio

In addition to this, the control continues to support the cable handwheels HR 410 (without display) and HR 420 (with display).



Refer to your machine manual.

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



If you wish to use the function **Handwheel superimp.** in virtual tool axis **VT**, an HR 5xx handwheel is recommended.

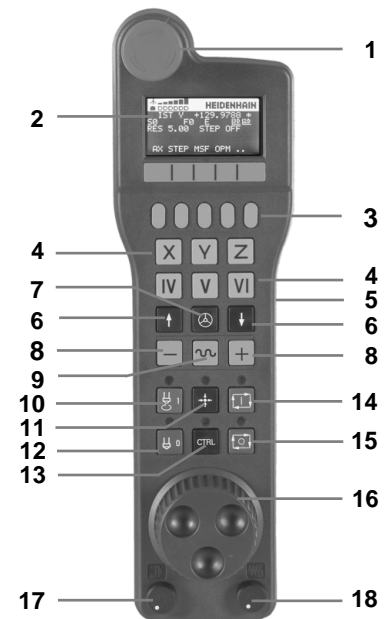
Further information: "Virtual tool axis VT", Page 272

The HR 520 and HR 550FS portable handwheels are equipped with a display that is used by the control to show different types of 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 key
- 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-specific 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, omitted with HR 550FS radio handwheel



Handwheel display

- 1 Only with HR 550FS radio handwheel:** Shows whether the handwheel is in the docking station or whether radio operation is active
- 2 Only with HR 550FS handwheel:** Display of field strength, six bars = maximum field strength
- 3 Only with HR 550FS radio handwheel:** Charging state of battery, six bars = maximum charging state. 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 3D:** Tilted-working-plane function is active
- 11 2D:** Basic rotation function is active
- 12 RES 5.0:** Active handwheel resolution. Path traversed by the selected axis with a handwheel revolution
- 13 STEP ON or OFF:** Incremental jog active or inactive. If the function is active, the control additionally displays the current traversing step
- 14 Soft-key row:** Selection of various functions, described in the following sections



Special features of the HR 550FS radio handwheel

⚠ 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)



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

The HR 550FS can be operated by battery for up to eight hours, before it needs recharging. When the handwheel is completely discharged, it takes about 3 hours until it is fully recharged in the handwheel holder. 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.

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.



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

NOTICE**Caution: Danger to the tool and workpiece!**

The wireless handwheel triggers an emergency stop reaction if the radio transmission is interrupted, the battery is fully empty, or if there is a defect. Emergency stop reactions during machining can cause damage to the tool or workpiece.

- ▶ Place the handwheel in the handwheel holder when it is not in use
- ▶ Keep the distance between the handwheel and the handwheel holder small (pay attention to the vibration alarm)
- ▶ Test the handwheel before machining

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

- ▶ Press the **MOD** key to select the MOD function
- ▶ Select **Machine settings**



- ▶ 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 295

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 handwheel soft keys **F1 (->)** or **F2 (<-)** 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 predefined 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 with the + key, or



- ▶ Move the active axis in the negative direction with the - key



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

Entering the spindle speed S

- ▶ Press the **F3 (MSF)** handwheel soft key
- ▶ Press the **F2 (S)** handwheel soft key
- ▶ 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 the **F3 (MSF)** handwheel soft key
- ▶ Press the **F3 (F)** handwheel soft key
- ▶ Select the desired feed rate by pressing the **F1** or **F2** key
- ▶ Confirm 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**.

Presetting



Refer to your machine manual.

The machine tool builder can disable presetting in individual axes.

- ▶ Press the **F3 (MSF)** handwheel soft key
- ▶ Press the **F4 (PRS)** handwheel soft key
- ▶ 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 modes of operation

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

- ▶ Press the **F4 (OPM)** handwheel soft key
- ▶ 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



Refer to your machine manual.

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 control's keyboard to select the NC block after which the new traversing 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.

Features in the program run modes of operation

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**)
- Returning to the contour after the axes were moved manually during a program interruption (**MOP** and then **REPO** handwheel soft keys). The handwheel soft keys, which function similarly to the screen soft keys, are used for operating.
Further information: "Returning to the contour", Page 258
- On/off switch for the Tilt working plane function (handwheel soft keys **MOP** and then **3D**)

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: "Entering miscellaneous functions M and STOP ", Page 265



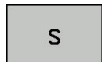
Refer to your machine manual.

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.
 - When **3D ROT** is active the machining feed rate is shown if several axes are moved
 - If **3-D ROT** is inactive, the feed rate display remains empty when more than one axis is moved simultaneously

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

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 and basic rotations in the preset table:

- Manual input
- Using the touch probe cycles in the **Manual operation** and **Electronic handwheel** operating modes
- Using touch probe cycles 400 to 402 and 410 to 419 in automatic mode

Further information: Cycle Programming User's Manual



Operating notes:

- In the 3-D ROT menu, you can specify that the basic rotation is also effective in the **Manual Operation** mode.
Further information: "Activating manual tilting:", Page 213
- When presetting, the positions of the tilting axes must match the tilted situation.
- The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601).
Further information: "Introduction", Page 179
- **PLANE RESET** does not reset the active 3-D ROT.
- 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). Only use this function if your machine has a rotary table, and you want to set the preset to the center of the rotary table by entering 0. 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
	Select the BASE TRANSFORM./OFFSET view. The standard BASE TRANSFORM. view shows the X, Y and Z columns. Depending on the machine, the columns SPA, SPB, and SPC are displayed in addition. The control saves the basic rotation here (with the Z tool axis the control uses the SPC column). The OFFSET view shows the offset values for the preset.
	Write the currently active preset to a selectable line in the table: This function saves the preset in all axes, and then activates the appropriate line in the table automatically. If inch display is active: Enter the value in inches, and the control will internally convert the entered values to mm

Editing the preset table

Soft key	Editing function in table mode
	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
	Choose between showing the Basic Transformation or the Axis Offset
	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

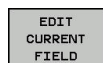
Proceed as follows to protect a preset from being overwritten:



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

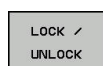


- ▶ Select the **LOCKED** column



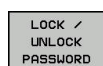
- ▶ Press the **EDIT CURRENT FIELD** soft key

Protection for a preset without using a password:

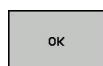


- ▶ Press the **LOCK / UNLOCK** soft key
- ▶ The control writes an **L** in the **LOCKED** column.

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

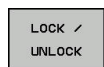


- ▶ Select the **LOCKED** column



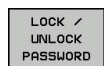
- ▶ Press the **EDIT CURRENT FIELD** soft key

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, rotation, or scaling factor.
- On the other hand, the **Tilt working plane** function (Cycle **19** or **PLANE**) remains active.



- Select the **Manual operation** mode



- Press the **PRESET MANAGEMENT** soft key



- Select the preset number that you want to activate



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



- Confirm with the **ENT** key



- Press the **ACTIVATE PRESET** soft key



- Confirm activation of the preset
- The control sets the display and the basic rotation.



- 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: Cycle 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", Page 201



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:



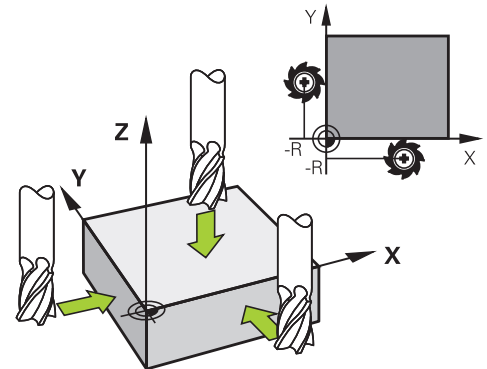
- Select the axis
- The control opens the **PRESETTING Z=** dialog window



- 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.
- The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601).

Further information: "Introduction", Page 179

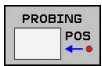
Using touch probe functions with mechanical probes or measuring dials

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 ", Page 179

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 186
- Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 187
- ▶ 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

Introduction

The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601):

- **chkTiltingAxes: NoCheck** The control does not check whether the current coordinates of the rotary axes (actual positions) agree with the tilt angles that you defined.
- **chkTiltingAxes: CheckIfTilted** With an active tilted working plane, the control checks during presetting in the X, Y, and Z axes whether the current coordinates of the rotary axes agree with the tilt angles that you defined (3-D ROT menu). If the positions do not match, the control opens the **Working plane is inconsistent** menu.
- **chkTiltingAxes: CheckAlways** With an active tilted working plane, the control checks during presetting in the X, Y, and Z axes whether the current coordinates of the rotary axes match. If the positions do not match, the control opens the **Working plane is inconsistent** menu.



Operating notes:

- If the checking function is off, the **PL** and **ROT** probing functions use a rotary axis position that is equal to 0 in their calculations.
- Always set a preset in all three principal axes. This clearly and correctly defines the preset. That way you also taken into account possible deviations resulting from the tilting of the axes.
- The control issues an error message if you do not use a 3-D touch probe for presetting and the positions do not match.

If the machine parameter has not been set, the control checks as if **chkTiltingAxes: CheckAlways** were set

Behavior with tilted axes

If the positions do not match, the control opens the **Working plane is inconsistent** menu.

Soft key	Function
<div>ADOPT 3-D ROTAT. STATUS</div>	<p>In the 3-D ROT menu, the control sets Manual Operation 3-D ROT to Active. The axes move in a tilted working plane.</p> <p>Manual Operation 3-D ROT remains active until you set it to Inactive.</p>
<div>IGNORE 3-D ROTAT. STATUS</div>	<p>The control ignores the tilted working plane.</p> <p>The preset defined applies only to this tilted condition.</p>

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.



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	188
	Measuring a 3-D basic rotation by probing a plane	198
	Measuring a basic rotation using a line	195
	Setting the preset on any axis	202
	Setting a corner as preset	203
	Setting a circle center as preset	204
	Setting the centerline as preset	207
	Touch probe system data management	See Cycle Programming User's Manual



Further information: Cycle Programming User's Manual

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

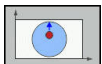
If you use a probing routine for automatic probing of a hole, stud, or a pattern circle, the control opens a form with the required entry fields.

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)
Starting angle?	Angle for the first probing operation (0° = positive direction of principal axis, i.e. in X+ for spindle axis Z). All other probe angles result from the number of touch points.
Number of touch points?	Number of probing operations (3 to 8)
Angular length?	Probing a full circle (360°) or a circle segment (angular length<360°)

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:

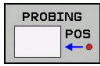
- Before starting an automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance opposite to the probing direction. The safety clearance is derived from the sum of the values in the touch-probe table and in 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 187

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

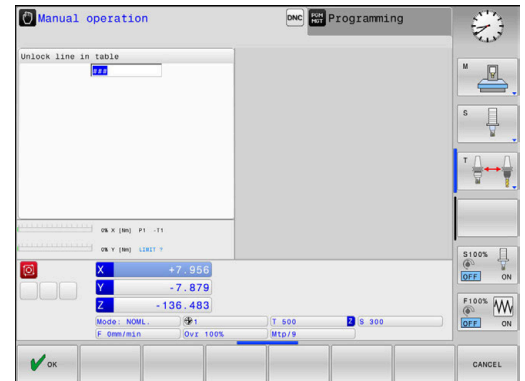
 If you want to save measured values in the basic coordinate system, use the **ENTRY IN PRESET TABLE** function. If you want to save measured values in the workpiece coordinate system, use the **ENTER IN DATUM TABLE** function.

Further information: "Writing measured values from the touch probe cycles to a datum table", Page 186

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

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:

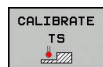
- Always calibrate the touch probe again in the following cases:
 - Initial configuration
 - Broken stylus
 - Stylus exchange
 - Change in the probe feed rate
 - Irregularities caused, for example, when the machine heats up
 - Change of active tool axis
- When you press the **OK** soft key after calibration, 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	189
	Measure the radius and the center offset using a calibration ring	190
	Measure the radius and the center offset using a stud or a calibration pin	190
	Measure the radius and the center offset using a calibration sphere	190

Calibrating the effective length

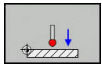


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

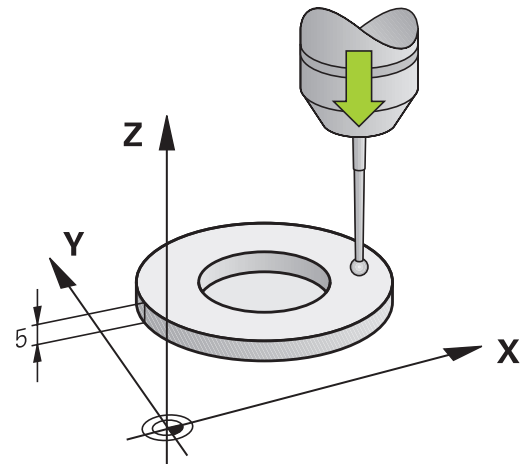


The effective length of the touch probe is always referenced to the tool reference point. The tool reference point is often on the spindle nose (and face of the spindle). The machine manufacturer may also place the tool reference point at a different point.

- 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

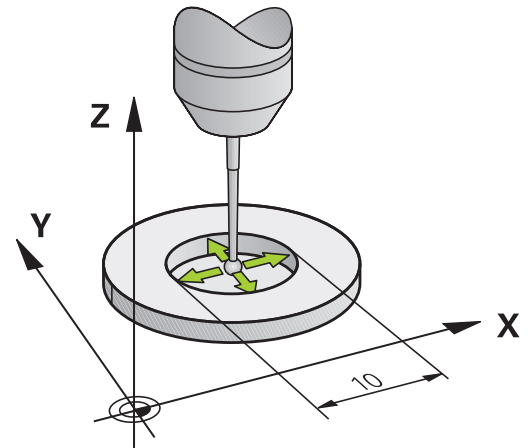


HEIDENHAIN only gives warranty for the function of the probing cycles if HEIDENHAIN touch probes are used.

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.

After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The calibration function can determine the offset between touch probe axis and spindle axis by probing from opposite orientations (rotation by 180°) and can calculate and implement the necessary compensation.



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
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ 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.

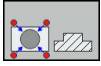


Refer to your machine manual.

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
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ 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.

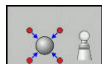


Refer to your machine manual.

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
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ 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.

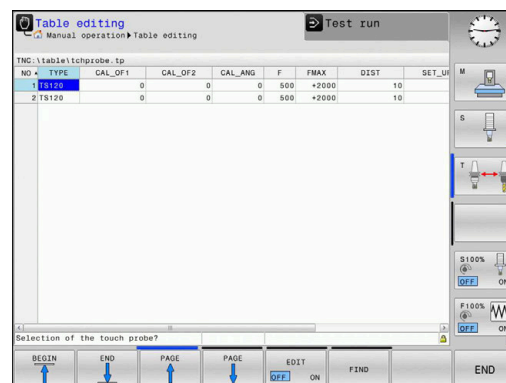
During calibration, the control automatically creates the TCHPRMAN.html log file to which the calibration values are saved.



Ensure that the tool number of the tool table and the touch-probe number of the touch-probe table are correct. This is regardless of whether you want to use a touch-probe cycle in automatic mode or **Manual operation** mode.



Further information: Cycle Programming User's Manual



5.8 Compensating workpiece misalignment with 3-D touch probe

Introduction

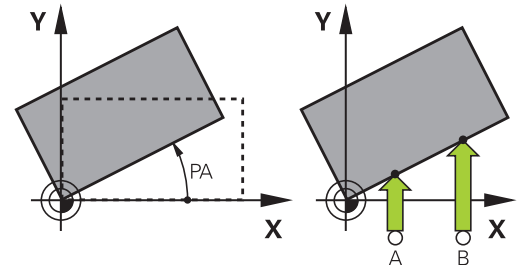


Refer to your machine manual.

It depends on the machine whether you can compensate workpiece misalignment with an offset (angle for table rotation).



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



The control compensates workpiece misalignment either mathematically by computing a basic rotation (angle of basic rotation) or by an offset (angle for table rotation)

For this purpose, the control sets the rotation angle to the desired angle with respect to the reference axis in the working plane.

Basic rotation: The control interprets the measured angle as rotation around the tool direction, and saves the values in the columns SPA, SPB, or SPC of the preset table.

Offset: The control interprets the measured angle as a shift in each axis in the machine coordinate system, and saves the values in the columns A_OFFS, B_OFFS, or C_OFFS of the preset table.

In order to identify the basic rotation or offset, probe two points on the side of the workpiece. The sequence in which you probe the points influences the calculated angle. The measured angle goes from the first to the second probing point. You can also determine the basic rotation or offset using holes or studs.

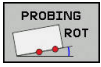


Operating and programming notes:

- Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.
- To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.
- You can also use a basic rotation in conjunction with the **PLANE** function (except for **PLANE AXIAL**). In this case first activate the basic rotation and then the **PLANE** function.
- You can also activate a basic rotation or offset without probing a workpiece. To do so, enter a value in the corresponding input field and press the **SET BASIC ROTATION** or **SET TABLE ROTATION** soft key.
- The behavior of the control during presetting depends on the setting in the machine parameter **chkTiltingAxes** (no. 204601).

Further information: "Introduction", Page 179

Measuring a basic rotation



- ▶ Press the **Probe rotation** soft key
- > The control opens the **Probing of rotation** menu.
- ▶ The following input fields are displayed:
 - **Angle of basic rotation**
 - **Offset of rotary table**
 - **Number in table?**
- > The control displays any current basic rotation or offset in the input field.
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probe direction or probing routine by soft key
- ▶ Press the **NC Start** key
- ▶ Position the touch probe at a position near the second touch point
- ▶ Press the **NC Start** key
- > The control determines the basic rotation and offset and displays them.
- ▶ Press the **SET BASIC ROTATION** soft key
- ▶ Press the **END** soft key

The control logs the probing process in TCHPRMAN.html.

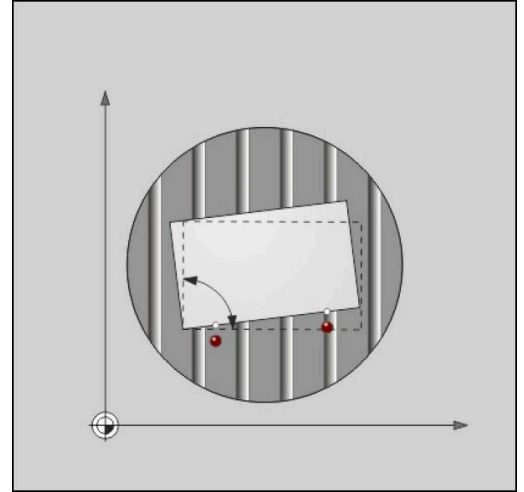
Saving the basic rotation in the preset table

- ▶ After the probing process, enter the preset number in which the control is to save the active basic rotation in the **Number in table?** input field
- ▶ Press the **BASIC ROT. IN PRESET TABLE** soft key
- > If appropriate, the control opens the **Overwrite active preset?** menu.
- ▶ Press the **OVERWRITE PRESET** soft key
- > The control saves the basic rotation in the preset table.

Compensation of workpiece misalignment by rotating the table

There are three possibilities for compensating workpiece misalignment by rotating the table:

- Align rotary table
- Set table rotation
- Save table rotation in the preset table



Align rotary table

You can compensate the ascertained misalignment by positioning the rotary table.



Pre-position all axes before rotating the table, in order to preclude collisions resulting from compensating movements. The control additionally outputs a warning before table rotation.

- ▶ Press the **ALIGN ROT. TABLE** soft key after the probing procedure
- > The control opens the warning.
- ▶ Clear with the **OK** soft key if needed
- ▶ Press the **NC Start** key
- > The control aligns the rotary table.

Set table rotation

You can set a manual preset in the axis of the rotary table.

- ▶ Press the **SET TABLE ROTATION** soft key after the probing procedure
- > If a basic rotation is already set, the control opens the **Reset basic rotation?** menu.
- ▶ Press the **DELETE BASIC ROT.** soft key
- > The control deletes the basic rotation from the preset table, and inserts the offset.
- ▶ Or press **KEEP BASIC ROT.**
- > The control inserts the offset in the preset table, and the basic rotation also remains.

Saving table rotation in the preset table

You can save the misalignment of the rotary table in any row of the preset table. The control stores the angle in the offset column of the rotary table, e.g. in the C_OFFS column for a C axis.

- ▶ Press the **TABLE ROT. IN PRESET TABLE** soft key after the probing procedure
- > If appropriate, the control opens the **Overwrite active preset?** menu.
- ▶ Press the **OVERWRITE PRESET** soft key
- > The control saves the offset in the preset table.

You may have to change the view in the preset table with the **BASE TRANSFORM./OFFSET** soft key for this column to be displayed.

Showing basic rotation and offset

If you select the **PROBING ROT** function, the control displays the active angle of the basic rotation in the **Angle of basic rotation** input field and the active offset in the **Offset of rotary table** input field.

In addition, the basic rotation and the offset are shown in the **PROGRAM + STATUS** screen layout on the **STATUS POS.** tab.

When the control moves the machine axis in accordance with the basic rotation, a symbol for the basic rotation is shown in the status display.

Rescinding basic rotation or offset

- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Enter the **Angle of basic rotation: 0**
- ▶ Alternative: Enter the **Offset of rotary table: 0**
- ▶ Apply with the **SET BASIC ROTATION** soft key
- ▶ Alternative: Apply with the **SET TABLE ROTATION** soft key
- ▶ Terminate the probing function: Press the **END** soft key

Measuring 3-D basic rotation

You can measure the inclined position of any tilted surface by probing three positions. The **Probe in plane** function enables you to measure this inclined position and save it as a 3-D basic rotation in the preset table.



Operating and programming notes:

- The sequence and position of the touch points determines how the control will calculate the orientation of the plane.
- With the first two points, you specify the direction of the reference axis. Define the second point in the positive direction of the desired reference axis. The position of the third point determines the direction of the minor axis and of the tool axis. Define the third point in the positive Y axis of the desired workpiece coordinate system.
 - 1st point: On the reference axis
 - 2nd point: On the reference axis, in a positive direction from the first point
 - 3rd point: On the minor axis, in a positive direction of the desired workpiece coordinate system

Optionally inputting a datum angle enables you to define the nominal direction of the probed plane.

Procedure



- ▶ Select the probing function: Press the **PROBING PL** soft key
- ▶ The control displays the current 3-D basic rotation.
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probe direction or probing routine 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
- ▶ Position the touch probe near the third touch point
- ▶ Probe: Press the **NC Start** key.
- ▶ The control measures the 3-D basic rotation and displays the values for SPA, SPB, and SPC in relation to the active coordinate system.
- ▶ If required, enter the reference angle

Activate 3-D basic rotation:



- ▶ Press the **SET BASIC ROTATION** soft key

Save the 3-D basic rotation in the preset table:




- ▶ Press the **BASIC ROT. IN PRESET TABLE** soft key



- ▶ Terminate the probing function: Press the **END** soft key

The control saves the 3-D basic rotation in the SPA, SPB, and SPC columns of the preset table.

Displaying 3-D basic rotation

If a 3-D basic rotation is saved in the active preset, the control shows the  icon for the 3-D basic rotation in the status display. The control moves the machine axes according to the 3-D basic rotation.

Aligning 3-D basic rotation

If the machine has two rotary axes and the probed 3-D basic rotation is activated, you can use the rotary axes to align the 3-D basic rotation.

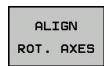
NOTICE

Danger of collision!

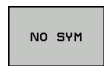
The control does not run a collision test before aligning the rotary axes. Danger of collision if there is no pre-positioning movement.

- ▶ Move to a safe position before the alignment

Proceed as follows:



- ▶ Press the **ALIGN ROT. AXES** soft key
- > The control displays the calculated axis angles.
- ▶ Enter a feed rate



- ▶ Select a solution where necessary
- > The control activates the 3-D rotation and updates the axis-angle display.



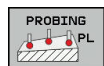
- ▶ Select the positioning behavior



- ▶ Press the **NC start** key
- > The control aligns the axes. The Tilt working plane function becomes active for this.

After aligning the plane, you can align the reference axis with the **Probing rot** function.


Canceling a 3-D basic rotation



- ▶ Select the probe function by pressing the **PROBING PL** soft key
- ▶ Enter 0 for all angles
- ▶ Press the **SET BASIC ROTATION** soft key
- ▶ Terminate the probing function: Press the **END** soft key

5.9 Presetting with a 3-D touch probe

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.

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	202
	Setting a corner as preset	203
	Setting a circle center as preset	204
	Setting the center line as preset	207

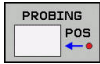


With an active datum shift the determined value is with respect to the current preset (possibly a manual preset from the **Manual operation** mode). The datum shift is taken into account in the position display.

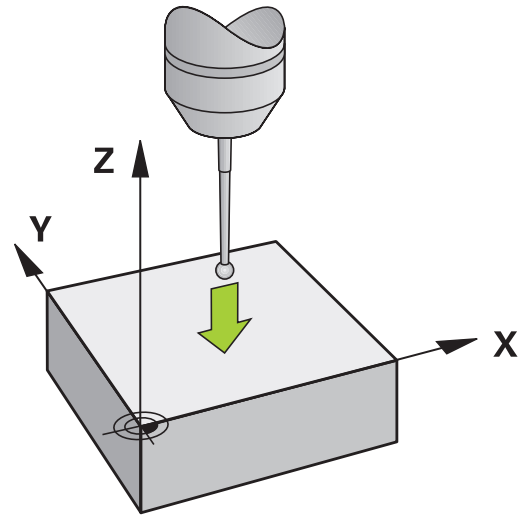
Presetting on any axis



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



- ▶ 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 186
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 187
- ▶ Terminate the probing function: Press the **END** soft key



Corner as preset

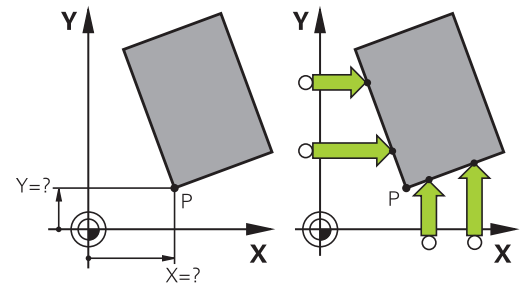


Refer to your machine manual.

It depends on the machine whether you can compensate workpiece misalignment with an offset (angle for table rotation).



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



The "Corner as preset" probing cycle identifies the angle and intersection of two straight lines.



- ▶ Select the probing function: Press the **PROBING P** soft key
- ▶ Position the touch probe near the first touch point on the first workpiece edge
- ▶ Select the probe direction by soft key
- ▶ Probe: Press the **NC Start** key
- ▶ Position the touch probe near the second touch point on the same workpiece edge
- ▶ Probe: Press the **NC Start** key
- ▶ Position the touch probe near the first touch point on the second workpiece edge
- ▶ Select the probe direction by soft key
- ▶ Probe: Press the **NC Start** key
- ▶ Position the touch probe near the second touch point on the same workpiece edge
- ▶ Probe: Press the **NC Start** key
- ▶ **Preset:** Enter both coordinates of the preset 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 186
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 187
- ▶ Terminate the probing function: Press the **END** soft key



You can identify the intersection of two straight lines by holes or studs and set this as the preset.

The **ROT 1** soft key activates the angle of the first straight line as basic rotation or as offset and the **ROT 2** soft key activates the angle or offset of the second straight line.

If you activate the basic rotation, the control automatically writes the positions and the basic rotation to the preset table.

If you activate the offset, the control automatically writes the positions and the offset or only the positions to the preset table.

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.

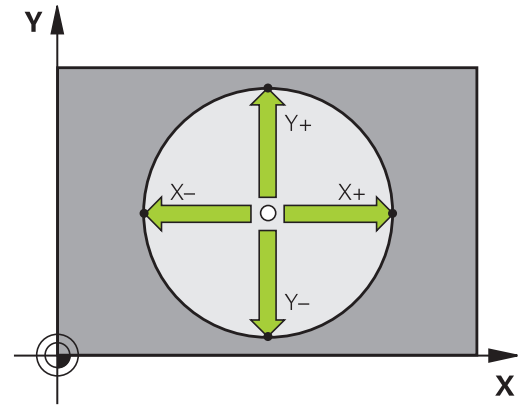
Inside circle:

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.



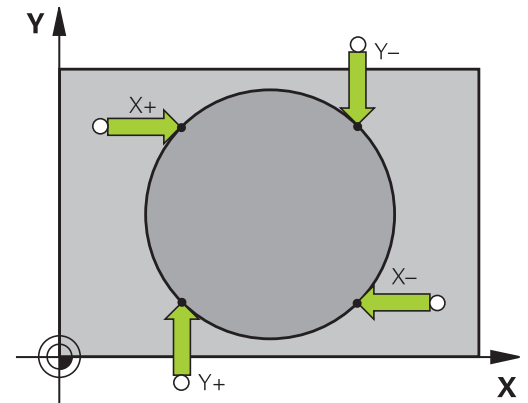
- ▶ Position the touch probe approximately in the center of the circle
- ▶ Select the probing function: Press the **PROBING CC** soft key
- ▶ Select the soft key for the desired probing direction
- ▶ Probe: Press the **NC Start** key. The touch probe probes the inside wall of the circle in the selected 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 186
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 187
- ▶ 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 186
- ▶ **Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 187
- ▶ 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.

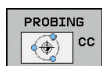
Setting the preset in the intersection of multiple holes or circular studs:

- Pre-position touch probe

Select the **Probing CC** probing function

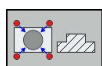


- Select the probing function: Press the **PROBING CC** soft key

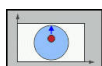


- Press the **PROBING CC (probing of circular pattern)** soft key

Probe a circular stud



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



- Enter starting angle or select using soft key

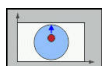


- Start probing function: Press the **NC start** key

Probe the hole.



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



- Enter starting angle 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

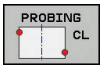
- Load with the **SET PRESET** soft key

Further information: "Writing measured values from the touch probe cycles to a datum table", Page 186

Further information: "Writing measured values from the touch probe cycles to the preset table", Page 187

- Terminate the probing function: Press the **END** soft key

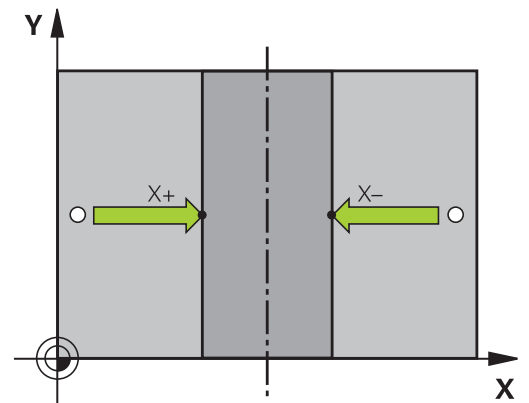
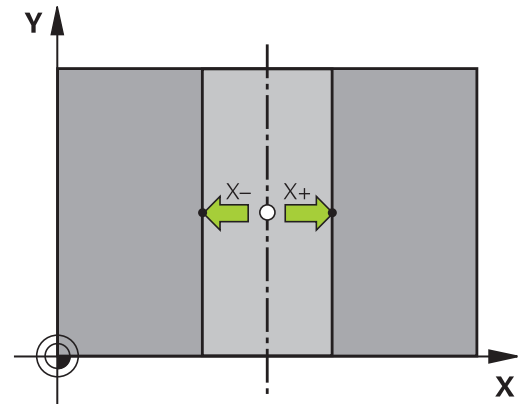
Setting a center line as preset



- ▶ 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 186
- Further information:** "Writing measured values from the touch probe cycles to the preset table", Page 187
- ▶ 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

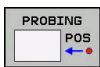
You can also use the touch probe in the **Manual operation** and **Electronic handwheel** operating modes to perform simple measurements on the workpiece. Numerous programmable touch probe cycles are available for more complex measuring tasks.

Further information: Cycle Programming User's Manual

With a 3-D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions and angles 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.

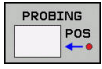
Finding the coordinates of a corner point on the working plane

Find the coordinates of the corner point.

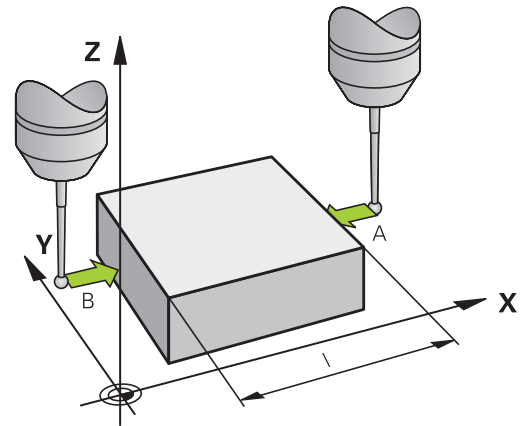
Further information: "Corner as preset", Page 203

The control displays the coordinates of the probed corner 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

Measuring angles

You can use the 3-D touch probe to measure angles in the working plane. You can measure

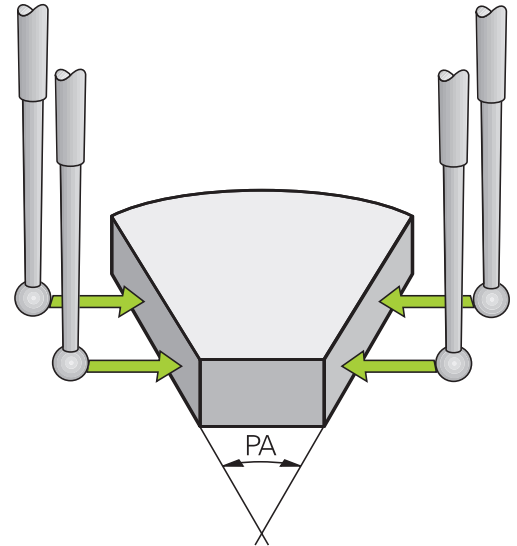
- The angle between the angle reference axis and a workpiece edge; or
- the angle between two sides

The measured angle is displayed as a value of max. 90°.

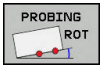
Finding the angle between the angle reference axis and a workpiece edge



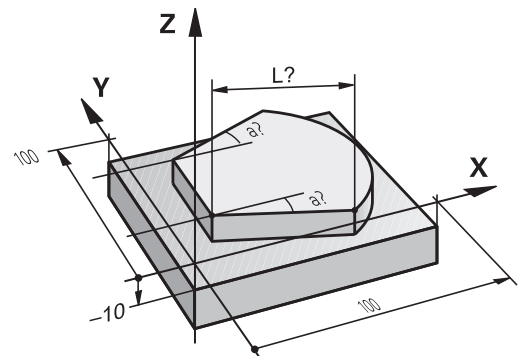
- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Rotation angle: If you wish to restore the current basic rotation later, note the value that appears under Rotation Angle
- ▶ Perform a basic rotation with the workpiece edge to be compared
Further information: "Compensating workpiece misalignment with 3-D touch probe", Page 194
- ▶ Press the **PROBING ROT** soft key to display the angle between the angle reference axis and the workpiece edge as the rotation angle
- ▶ Cancel the basic rotation, or restore the previous basic rotation
- ▶ Set the rotation angle to the value that you previously wrote down



Measuring the angle between two workpiece edges



- ▶ Select the probe function by pressing the **PROBING ROT** soft key
- ▶ Rotation angle: If you wish to restore the current basic rotation later, note the value that appears under Rotation Angle
- ▶ Perform a basic rotation with the workpiece edge to be compared
Further information: "Compensating workpiece misalignment with 3-D touch probe", Page 194
- ▶ Probe the second edge in the same way as for a basic rotation, but do not set the rotation angle to 0
- ▶ Press the **ROTATION PROBING** soft key to display the angle PA between the workpiece edges as the rotation angle
- ▶ Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously



5.10 Tilting the working plane (option 8)

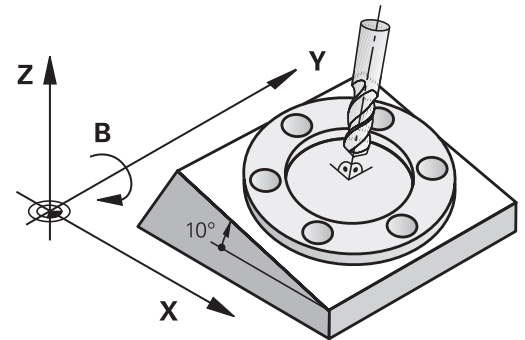
Application, function



Refer to your machine manual.

The **Tilt working plane** functions are interfaced to the control and the machine tool by the machine tool builder.

The machine tool builder also specifies whether the programmed angles are interpreted as coordinates of the rotary axes (axis angles) or as angular components of a tilted plane (spatial angles).



The control supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are three functions available for tilting the working plane:

- Manual tilting with the **3-D ROT** soft key in the **Manual operation** and **Electronic handwheel** operating modes
Further information: "Activating manual tilting:", Page 213
- Controlled tilting, Cycle **19 WORKING PLANE** in the NC program
Further information: Cycle Programming User's Manual
- Controlled tilting, **PLANE** function in the NC program
Further information: User's Manuals for Conversational Programming or ISO Programming

The control functions for tilting the working plane are coordinate transformations. The working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the control differentiates between two machine types:

■ **Machine with tilting table**

- You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with an L block.
- The position of the transformed tool axis **does not change** in relation to the machine coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction key in the **Manual operation** mode, the tool moves in Z+ direction.
- In calculating the transformed coordinate system, the control considers only the mechanically influenced offsets of the particular tilting table (the translational components).

■ **Machine with swivel head**

- You must tilt the workpiece into the desired position for machining by positioning the swivel head, for example with an L block
- The position of the tilted (transformed) tool axis changes in relation to the machine coordinate system. Thus if you rotate the swivel head of your machine—and therefore the tool—in the B axis by +90° for example, the coordinate system also rotates. If you press the Z+ axis direction key in the **Manual operation** mode, the tool moves in the X+ direction of the machine coordinate system.
- In calculating the active coordinate system, the control considers both the mechanically influenced offsets of the particular swivel head (the translational components) as well as offsets caused by tilting of the tool (3-D tool length compensation).



The control only supports the **Tilt working plane** function in combination with the spindle axis Z.

Position display in a tilted system

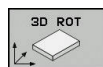
The positions displayed in the status window (**ACTL.** and **NOML.**) are referenced to the tilted coordinate system.

In the optional machine parameter **CfgDisplayCoordSys** (no. 127501) you can specify the coordinate system in which the status display shows an active datum shift.

Limitations on working with the tilting function

- The **Actual-position capture** function is not allowed if the Tilt working plane function is active
- PLC positioning (determined by the machine tool builder) is not possible.

Activating manual tilting:



- ▶ Press the **3-D ROT** soft key
- ▶ The control opens the **Tilt the working plane** pop-up window.



- ▶ Use the arrow keys to position the cursor on the desired function

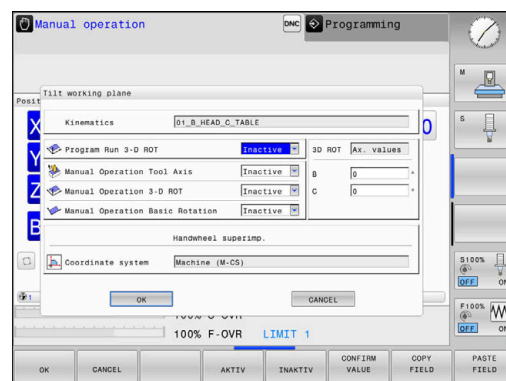
- **Manual Operation Tool Axis**
- **Manual Operation 3-D ROT**
- **Manual Operation Basic Rotation**



- ▶ Press the **ACTIVE** soft key
- ▶ If required, use the arrow keys to position the cursor on the desired rotary axis



- ▶ If required, enter the tilting angle
- ▶ Press the **END** key
- ▶ The input is complete.




If you set the **Manual Operation 3-D ROT** function to **Active**, the values defined in **SPA**, **SPB**, and **SPC** become effective. These values are ignored by the other functions.

Manual Operation Tool Axis



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


If the Traverse in Tool Axis function is active, the control displays the  icon in the status display.

You can only move in the direction of the tool axis. The control locks all other axes.

The traverse is effective in the T-CS tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 107

Manual Operation 3-D ROT

If the 3-D ROT function is active, the control displays the  icon in the status display.


All axes move in a tilted working plane.

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

The axis movements are effective in the WPL-CS working plane coordinate system.

Further information: "Working plane coordinate system WPL-CS", Page 105

Manual Operation Basic Rotation

If the basic rotation function is active, the control displays the  icon in the status display.

If a basic rotation or a 3-D basic rotation has already been saved to the preset table, the control additionally displays the corresponding icon.



If the **Manual Operation Basic Rotation** function is active, an active basic rotation or 3-D basic rotation is taken into account during manual movement of the axes. The control displays two icons in the status display.

The axis movements are effective in the W-CS workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 103

Program Run 3-D ROT

If you set the **Tilt working plane** function to **Active** for the **Program run** mode, the tilt angle entered in the menu will be effective starting from the first NC block of the NC program to be run.

If you use Cycle **19 WORKING PLANE** or the **PLANE** function in the NC program, the angle values defined there are in effect. Angle values entered in the menu will be set to 0.



The control uses the following **transformation types** for tilting:

- **COORD ROT**
 - if a **PLANE** function was previously executed with **COORD ROT**
 - after **PLANE RESET**
 - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine tool builder
- **TABLE ROT**
 - if a **PLANE** function was previously executed with **TABLE ROT**
 - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine tool builder



If tilting was active when the control was turned off, then the control also moves in the tilted working plane when it is turned on again.

Further information: "Crossing the reference point in a tilted working plane", Page 151

Deactivating manual tilting

To deactivate, set the desired function to **Inactive** in the **Tilt working plane** menu.

Even if the **3D-ROT** dialog in the **Manual operation** mode is set to **Active**, resetting the tilting (**PLANE RESET**) with an active basic transformation still functions correctly.

Setting the tool-axis direction as the active machining direction



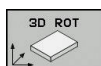
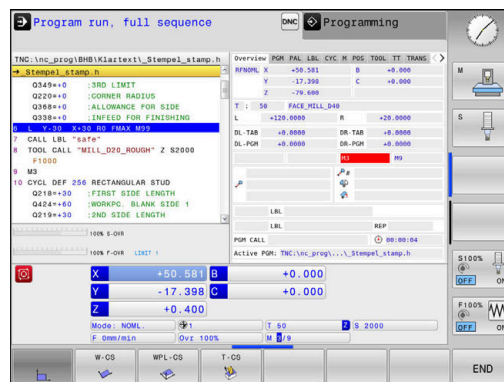
Refer to your machine manual.

Your machine manufacturer enables this function.

Using this function in the **Manual operation** and **Electronic handwheel** operating modes, you can move the tool in the direction in which the tool axis is currently pointed using the axis direction keys or the handwheel.

Use this function if

- You want to retract the tool in the direction of the tool axis during suspension of a 5-axis machining program
- You want to machine with an inclined tool using the handwheel or the axis direction keys in Manual Operation mode



- ▶ To select manual tilting, press the **3-D ROT** soft key.




- ▶ Use the arrow keys to move the cursor to the **Manual Operation Tool Axis** menu item



- ▶ Press the **ACTIVE** soft key
- ▶ Press the **END** key

To deactivate, set the **Manual Operation Tool Axis** menu item to **Inactive** in the Tilt working plane menu.

The  icon appears in the status display when the Move in tool axis direction function is active.

Setting a preset in a tilted coordinate system

After you have positioned the rotary axes, set the preset in the same manner as for a non-tilted system. The behavior of the control during presetting depends on the setting in the optional machine parameter **chkTiltingAxes** (no. 204601):

Further information: "Introduction", Page 179

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

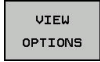


The simulation of NC programs with 5-axis machining or tilted machining might run at reduced speed. With the MOD menu **Graphic settings**, you can decrease the **Model quality** and in that way increase the speed of simulation.

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 220
	Show the tool paths Further information: "Tool", Page 220
	Select the view Further information: "View", Page 221
	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 223

Soft keys	Function
	Shift the sectional plane in the projection in three planes Further information: "Shifting the sectional plane", Page 225



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 117

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

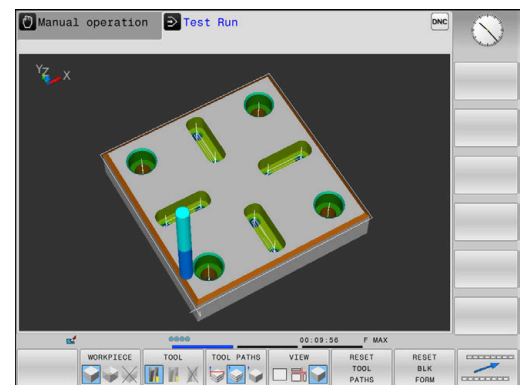
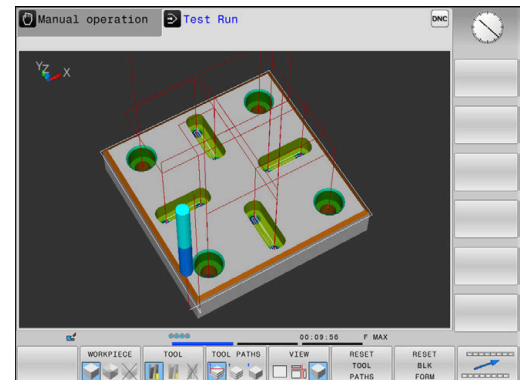
Showing the tool paths

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






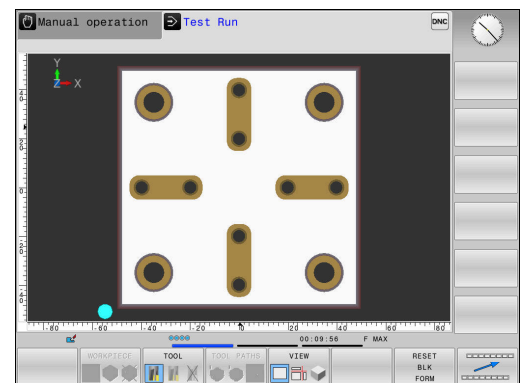
If you move at rapid traverse inside a workpiece, both the movement and the workpiece is shown in red at the respective location.



View

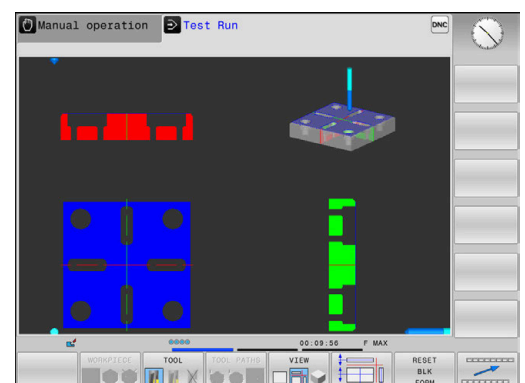
The control provides the following views:

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



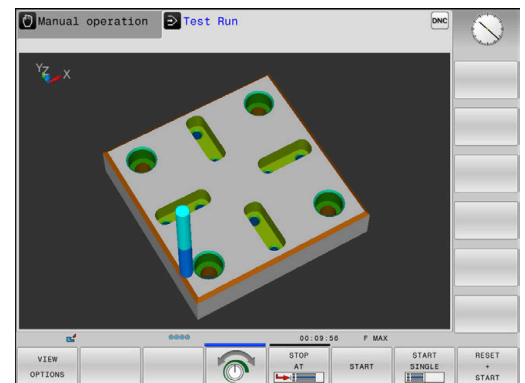
Projection in three planes

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 Measurement of machining time

Machining time in the Test Run operating mode

The control calculates the duration of the tool movements and displays this as machining time in the test run. The control takes feed movements and dwell times into account.

The time determined by the control is only of limited value for calculating the machining time because it does not take any machine-dependent time intervals (e.g., for tool changes) into consideration.


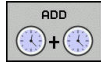

Proceed as follows to select the stopwatch function:



- Select the stopwatch function



- Select the desired function by soft key, e.g. saving the displayed time

Soft key	Stopwatch functions
	Save the displayed time
	Display the sum of saved time and displayed time
	Clear the displayed time

Machining time in the machine operating modes

Time display from program start to program end. The timer stops whenever machining is interrupted.

6.3 Showing the workpiece blank in the working space

Application

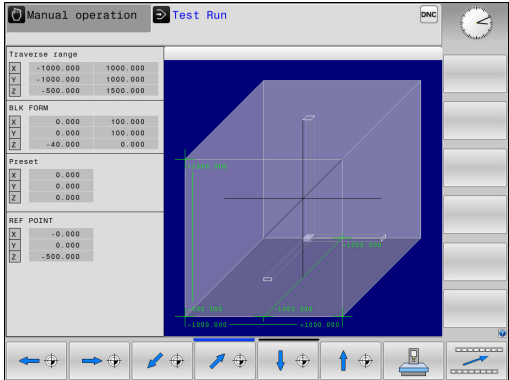
In **Test Run** operating mode, you can graphically check the position of the workpiece blank and the preset in the working space of the machine. The graphic shows the preset that has been set in the NC program using Cycle 247. If you have not defined a preset in the NC program, then the graphic shows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



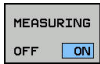
For blank in work space, the control depicts the **BLK FORM** only schematically.

- With **BLK FORM CYLINDER**, a cuboid is depicted as the workpiece blank
- With **BLK FORM ROTATION**, no workpiece blank is depicted

6.4 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 221

6.5 Optional program-run interruption

Application



Refer to your machine manual.

The behavior of this function varies depending on the respective machine.

The control optionally interrupts program run at NC blocks in which an M1 has been programmed. If you use M1 in the **Program run** operating mode, the control does not switch off the spindle or the coolant.



- ▶ Set the **M01** soft key to **OFF**
- > The control does not interrupt the **Program run** or **Test Run** at NC blocks that contain an M1.



- ▶ Set the **M01** soft key to **ON**
- > The control interrupts the **Program run** or **Test Run** at NC blocks with M1.

6.6 Skipping NC blocks

You can have NC blocks skipped in the following operating modes:

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



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 (**/**).

6.7 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**

With rotationally symmetrical workpiece blanks, the control starts a test run after a tool call at the following position:

- In the machining plane at the position $X=0, Y=0$
- In the tool axis 1 mm above the defined workpiece blank

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 prepositioning
- ▶ 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 227

- 
- ▶ 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

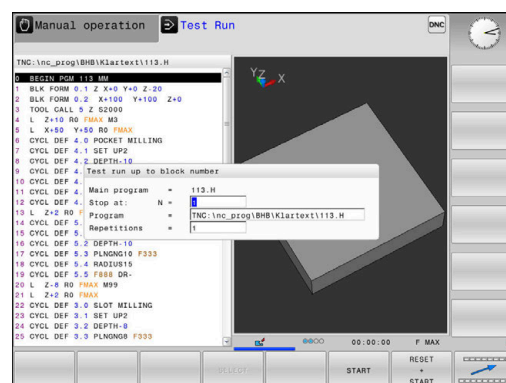
Executing a Test Run up to a certain NC block

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 253

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: Cycle Programming User's Manual

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

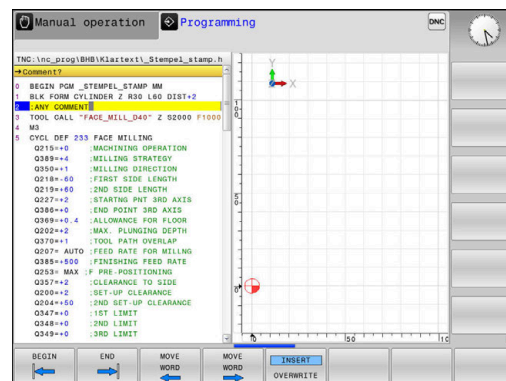
The control displays syntax elements with various colors according to their meaning. Color-highlighting makes the NC programs easier to read and clearer.

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.



6.8 Program run

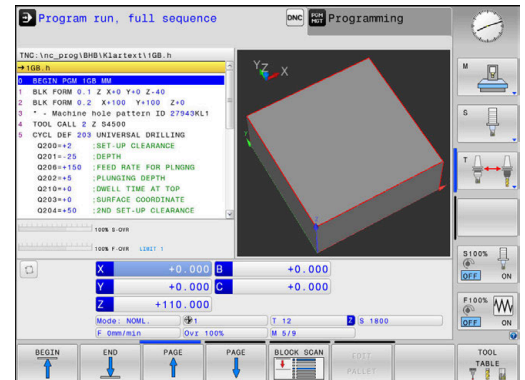
Application

In the **Program run, full sequence** operating mode, the control executes an NC program continuously to its end or up to a program stop.

In the **Program run, single block** operating mode, the control executes each NC block individually after pressing the **NC Start** key. With point pattern cycles and **CYCL CALL PAT** the control stops after each point.

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 necessary tables and 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



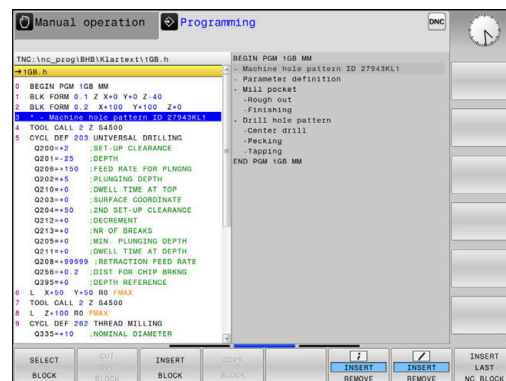
- ▶ Display structure window: For this screen layout press the **PROGRAM + SECTS** soft key



- ▶ 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.



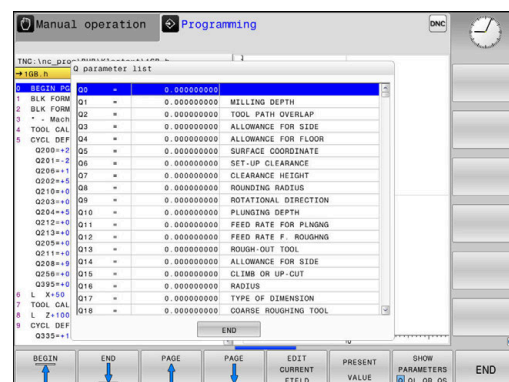
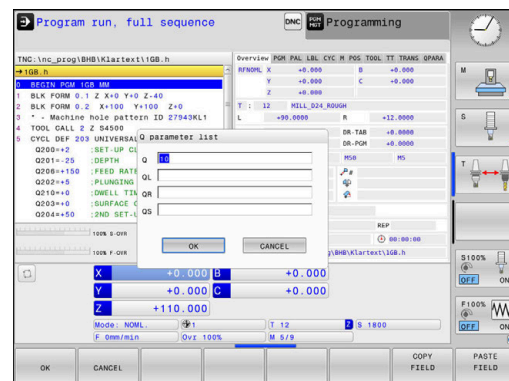
Checking and changing Q parameters

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
 - ▶ 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

i 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).

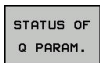
- ▶ If you are in a program run, interrupt it if required (e.g. by pressing the **NC stop** key and the **INTERNAL STOP** soft key), or stop the test run



- ▶ 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 57

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 **STOP** (with and without miscellaneous function)
- 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.

247

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, rotation, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour 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
Tilted system	Movement of all axes in the active coordinate system Effective parameters: Position of the tilting axes
Tool axis	Movements of the tool axis in the active 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



If the **Tilt the working plane** function (option 8) is enabled on your control, then the **Tilted system** traverse mode is also available.

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!**

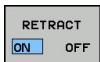
A power failure during the machining operation can cause uncontrolled “coasting” or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last 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 in the tilted working plane was being performed. You have to retract the tap:

- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control 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: If required, reset the tilted working plane

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:

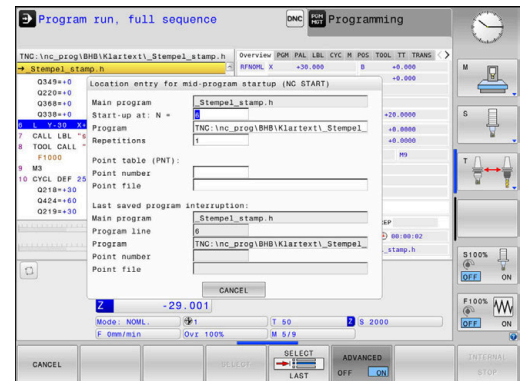
- **INTERNAL STOP** soft key
- Emergency stop
- Power failure

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 255



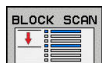
The **BLOCK SCAN** function must not be used in conjunction with the following functions:

- Touch probe cycles 0, 1, 3, and 4 during the search phase of mid-program startup

Procedure for simple mid-program startup

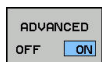


The control only displays the dialogs required by the process in the pop-up window.

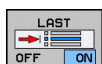


- ▶ Press the **BLOCK SCAN** soft key
- ▶ The control shows a pop-up window with the active main program.
- ▶ **Start-up at: N =:** Enter the number of the NC block where you wish to enter the NC program
- ▶ **Program:** Check the name and path of the NC program containing the NC block, or enter with 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 258



- ▶ 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

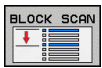
If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this purpose, jump in the main program to the desired subprogram call. With the **CONTINUE BLOCK SCAN** function, you can jump further from this position.



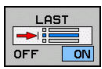
Operating notes:

- 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

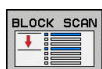


- ▶ Press the **NC Start** key
- ▶ The control resumes execution of the NC program.

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.



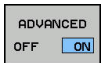
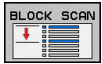
- ▶ Press the **CONTINUE BLOCK SCAN** soft key
- ▶ In the pop-up window enter the following data:
 - **Start-up at: N =8**
 - **Repetitions 1**



- ▶ 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.

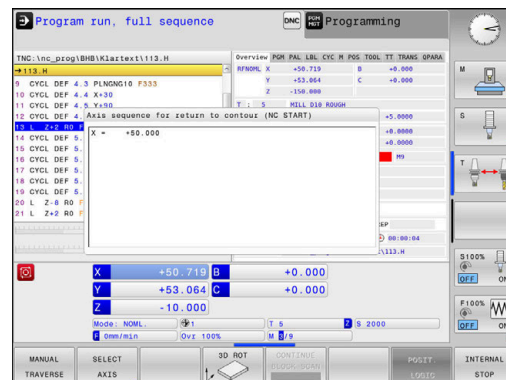


- ▶ Press the **BLOCK SCAN** soft key
- > The control shows a pop-up window.
- ▶ Press the **ADVANCED** soft key
- > The control expands the pop-up window.
- ▶ **Point number:** Enter the line number of the point table you start with
- ▶ **Point file:** Enter the name and path of the point table
- ▶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

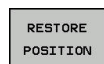
If you would like to use the block scan function to start in a point pattern, then proceed just as you would for starting in the point table. Enter the desired point number in the **Point number** input field. The first point in the point pattern has the point number **0**.

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



Proceed as follows to approach the contour:



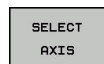
- ▶ Press the **RESTORE POSITION** soft key
- ▶ Restore the machine status, if required

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.

6.9 Functions for program display

Overview

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control displays the following soft keys for displaying the NC program in pages:

Soft key	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

6.10 Automatic program start

Application



Refer to your machine manual.

The control must be specially prepared by the machine tool builder for use of the automatic program start function.

⚠ DANGER

Caution: Danger for the operator!

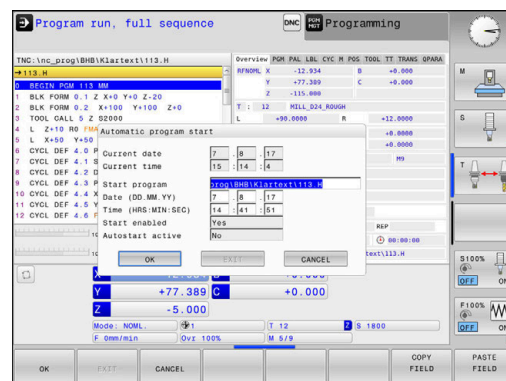
The **AUTOSTART** function automatically starts the machining operation. Open machines with unsecured work envelopes pose a huge danger for the machine operator.

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

In a Program Run operating mode, you can use the **AUTOSTART** soft key to define a specific time at which the NC program that is currently active in this operating mode is to be started:



- Display window for setting the starting time
- **Time (h:min:sec):** Time of day at which the NC program is to be started
- **Date (TT.MM.JJJJ):** Date at which the NC program is to be started
- To activate the start, press the **OK**



6.11 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, depending on machine parameter **programInputMode** (no. 101201), in Klartext conversational programming or in ISO format 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.

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

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 261



Operating and programming notes:

- The following functions are not available in the **Positioning w/ Manual Data Input** operating mode:
 - FK free contour programming
 - 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.

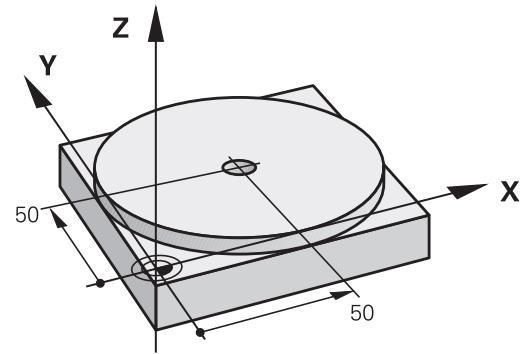
Further information: User's Manual for Conversational Programming or ISO Programming
- You can control and modify Q parameters with the soft keys **Q PARAMETER LIST** and **Q INFO**.

Further information: "Checking and changing Q parameters", Page 242

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 L Z+200 R0 FMAX	Retract the tool (F MAX = rapid traverse)
3 L X+50 Y+50 R0 FMAX M3	Move the tool at F MAX to a position above the hole, spindle on
4 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	Hole depth (algebraic sign=working 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
5 CYCL CALL	Call the DRILLING cycle
6 L Z+200 R0 FMAX M2	Retract the tool
7 END PGM \$MDI MM	End of program

Example : Remove workpiece misalignment on a machine with a rotary table

- ▶ Use a 3-D touch probe to carry out a basic rotation
Further information: "Compensating workpiece misalignment with 3-D touch probe ", Page 194
- ▶ Write down the rotation angle and cancel the basic rotation



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



- ▶ Select the rotary table axis, enter the rotation angle and feed rate you wrote down, e.g. **L C +2.561 F50**



- ▶ Conclude entry



- ▶ Press the **NC Start** button: The rotation of the table corrects the misalignment

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:



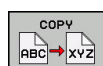
- ▶ Operating mode: Press the **Programming** key



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



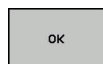
- ▶ Move the highlight to the **\$MDI** file



- ▶ 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

6.12 Entering miscellaneous functions M and STOP

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

Entering a miscellaneous function in a STOP block

If you program a **STOP** block, the program run or test run is interrupted at the block, e.g. for a tool inspection. You can also enter an M (miscellaneous) function in a **STOP** block:

STOP

- ▶ To program an interruption of program run, press the **STOP** key
- ▶ Enter a miscellaneous function **M**

Example

87 STOP M6

6.13 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.

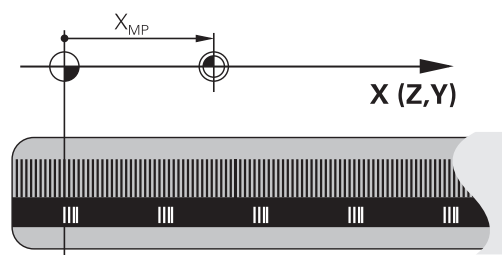
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine tool builder)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 1 Clear status display Functional scope 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			■

6.14 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 176

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 57

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.

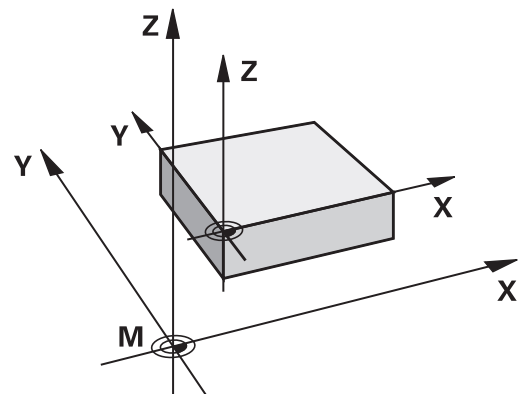
M91 and M92 take effect at the start of block.

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 227

Moving to positions in a non-tilted coordinate system with a tilted working plane: M130

Standard behavior with a tilted working plane

The control references the coordinates in the positioning blocks to the tilted working plane coordinate system.

Behavior with M130

Despite an active tilted working plane, the control references the coordinates in straight line blocks to the non-tilted workpiece coordinate system.

The control then positions the tilted tool at the programmed coordinates of the non-tilted workpiece coordinate system.

NOTICE

Danger of collision!

The **M130** function is only active blockwise. The control executes the subsequent machining operations in the tilted working plane coordinate system again. Danger of collision during machining!

- Check the sequence and positions using a graphic simulation



Programming notes:

- The **M130** function is only allowed if the **Tilt the working plane** function is active.
- If the **M130** function is combined with a cycle call, the control will interrupt the execution with an error message.

Effect

M130 functions blockwise in straight-line blocks without tool radius compensation.

6.15 Miscellaneous functions for path behavior

Superimposing handwheel positioning during program run: M118

Standard behavior

In the Program Run operating modes, the control moves the tool as defined in the NC program.

Behavior with M118

M118 permits manual corrections by handwheel during the program run. For this purpose, you program **M118** and enter an axis-specific value (linear or rotary axis).

NOTICE

Danger of collision!

If you use the **M118** function to modify the position of a rotary axis with the handwheel and then execute the **M140** function, the control ignores the superimposed values with the retraction movement. This results in unwanted and unpredictable movements, especially when using machines with head rotation axes. There is a danger of collision during these compensating movements!

- Do not combine **M118** with **M140** when using machines with head rotation axes.

Input

If you enter **M118** in a positioning block, the control continues the dialog for this block by prompting you for the axis-specific values. Use the orange axis keys or the alphabetic keyboard for entering the coordinates.

Effect

To cancel handwheel positioning, program **M118** once again without coordinate input.

M118 becomes effective at the start of the block.

Example

You want to be able to use the handwheel during program run to move the tool in the working plane X/Y by ± 1 mm and in the rotary axis B by $\pm 5^\circ$ from the programmed value:

```
L X+0 Y+38.5 RL F125 M118 X1 Y1 B5
```



M118 is always effective in the machine coordinate system.

Further information: "Handwheel superimp.", Page **M118** is also effective in the **Positioning w/ Manual Data Input** operating mode!

Virtual tool axis VT

Refer to your machine manual.

Your machine tool builder must have prepared the control for this function.

With the virtual tool axis, you can also traverse with the handwheel in the direction of a sloping tool on a machine with swivel heads.

To traverse in a virtual tool axis direction, select the **VT** axis on the display of your handwheel.

Further information: "Traverse with electronic handwheels", Page 155

When using a HR 5xx handwheel, you can select the virtual axis directly with the orange **VI** axis key, if necessary.

In conjunction with the **M118** function, it is also possible to carry out handwheel superimpositioning in the currently active tool axis direction. For this purpose, program at least the spindle axis with its permitted range of traverse in the **M118** function (e.g. **M118 Z5**) and select the **VT** axis on the handwheel.

Deleting basic rotation: M143

Standard behavior

The basic rotation remains in effect until it is reset or is overwritten with a new value.

Behavior with M143

The control deletes a basic rotation from the NC program.



The function **M143** is not permitted with mid-program startup.

Effect

M143 is effective only from the NC block in which it is programmed.

M143 becomes effective at the start of the block.



M143 clears the entries from the **SPA**, **SPB** and **SPC** columns in the preset table. When the corresponding line is reactivated, the basic rotation is **0** in all columns.

Automatically retracting the tool from the contour at an NC stop: M148

Standard behavior

In case of an NC stop, the control stops all traverse movements. The tool stops moving at the point of interruption.

Behavior with M148



Refer to your machine manual.

This function must be configured and enabled by your machine tool builder.

In the **CfgLiftOff** (no. 201400) machine parameter, the machine tool builder defines the path the control is to traverse for a **LIFTOFF** command. You can also use the **CfgLiftOff** machine parameter to deactivate the function.

Set the **Y** parameter in the **LIFTOFF** column of the tool table for the active tool. The control then retracts the tool from the contour by 2 mm max. in the direction of the tool axis.

Further information: "Entering tool data into the table", Page 117

LIFTOFF takes effect in the following situations:

- An NC stop triggered by you
- An NC stop triggered by the software, e.g. if an error occurred in the drive system
- When a power interruption occurs

Effect

M148 remains in effect until deactivated with **M149**.

M148 becomes effective at the start of the block, **M149** at the end of the block.

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.



You can use Cycle 225 to engrave the current counter value into the workpiece.

Further information: Cycle Programming User's Manual

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 L ...	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

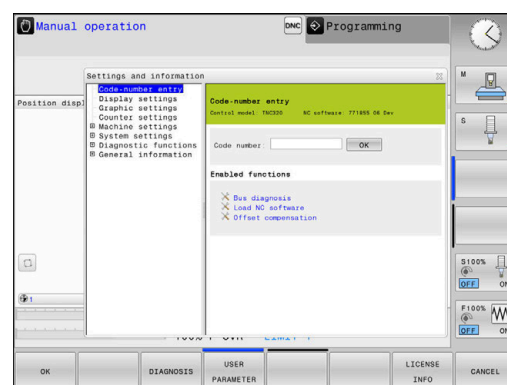
8.1 MOD function

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

As well as with the mouse, navigation with the alphabetic keyboard is also possible in the MOD functions:

- ▶ Switch from the input area in the right window to the MOD function selections in the left window with the tab key
- ▶ Select MOD function
- ▶ Switch to the input field with the tab key or ENT key
- ▶ Enter value according to function and confirm with **OK** or make selection and confirm with **Apply**



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 **END** 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

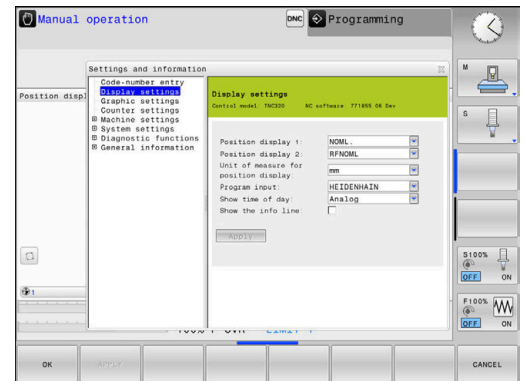
- Set the system time
- Define the network connection
- Network: IP configuration

Diagnostic functions

- Bus diagnosis
- Diagnosis of drives
- 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

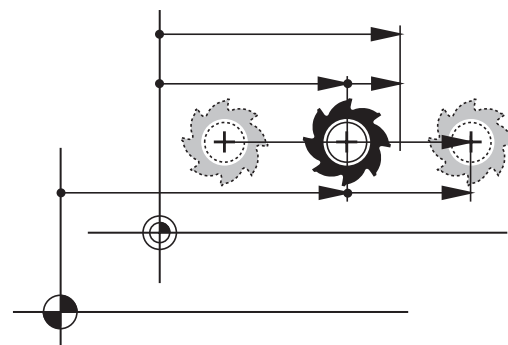
Application



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 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 276

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.

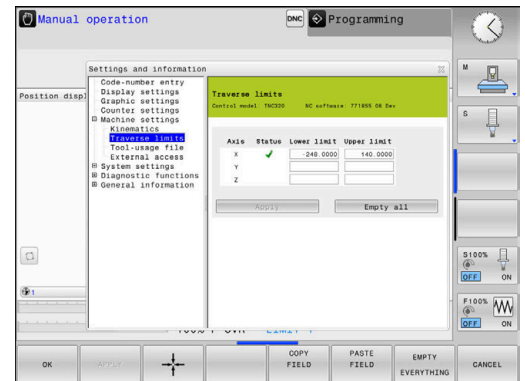
To enter traverse limits:

- ▶ 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.

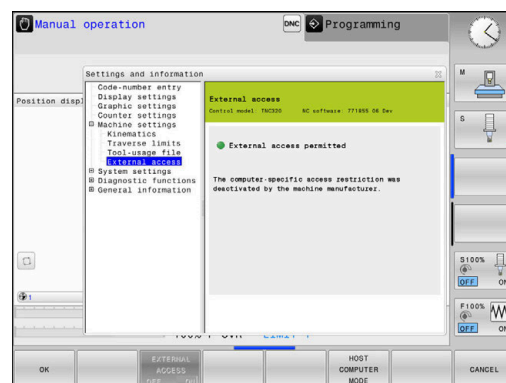
The machine tool builder can configure the external access options.

Depending on the machine, you can grant or restrict access for an external diagnostics or commissioning software application using the **TNCOPT** soft key.

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:

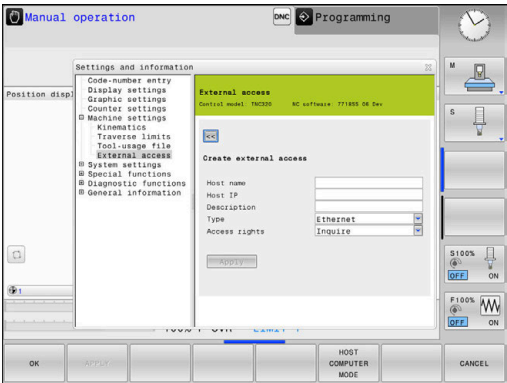
Inquire	For external access, the control opens a query dialog
Deny	Do not permit network access
Permit	Permit network access without query

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.
Connections without access rights are shown gray in the overview list.



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.

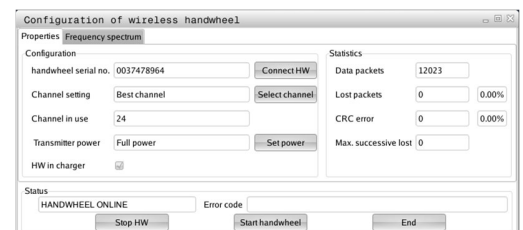
This device complies with Part 15 of the FCC Rules and the RSS standard(s) of Industry Canada for license-exempt equipment.

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

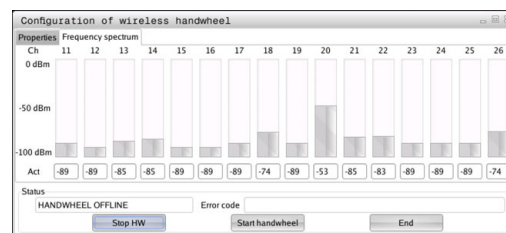
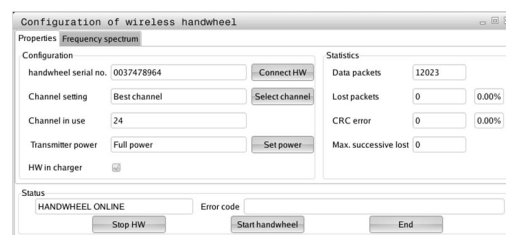
- ▶ Make sure that the handwheel holder is connected to the control hardware.
- ▶ Place the wireless handwheel you want to assign to the handwheel holder in the handwheel holder
- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration 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

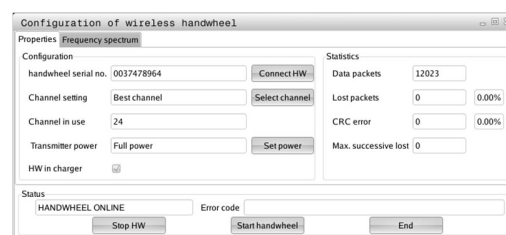


Selecting the transmitter power



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

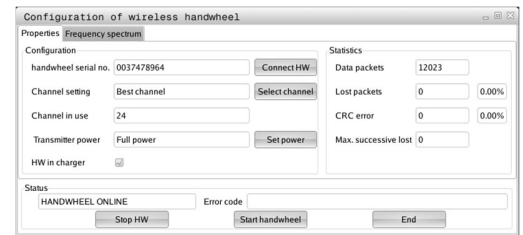
- ▶ Press the **MOD** key to select the MOD function
- ▶ Select the **Machine settings** menu
- ▶ Select the configuration menu for the radio handwheel: Press the **SET UP WIRELESS HANDWHEEL** soft key
- ▶ Click the **Set power** button
- ▶ The control displays the three available power settings. Click 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 296

Further information: "Selecting the transmitter power", Page 296

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 NTP server
- ▶ Press the **Add** soft key
- ▶ Press the **OK** soft key

8.12 Displaying operating times

Application

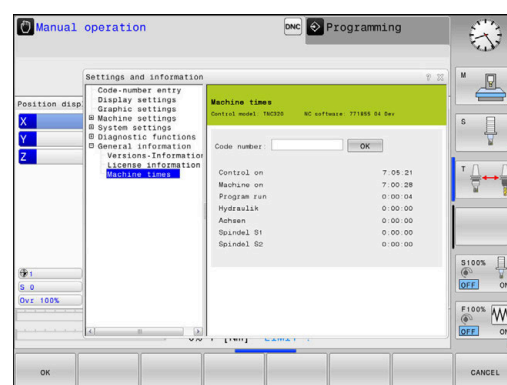
The **MACHINE TIME** MOD function enables you to see various types of operating times:

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 Remote Desktop Manager (option 133)

Introduction

The Remote Desktop Manager enables you to display external computer units on the control screen that are connected via Ethernet, and to operate them via the control. You can also start programs specifically under HEROS or display web pages of an external server.

HEIDENHAIN offers you the IPC 6641 as a Windows computer. With the IPC 6641 Windows computer you can start and operate Windows-based applications directly from the control.

The following connection options are available:

- **Windows Terminal Server (RemoteFX):** Displays the desktop of a remote Windows computer on the control
- **VNC:** Connection to an external computer. Displays the desktop of a remote Windows, Apple or Unix computer on the control
- **Switch-off/restart of a computer:** Configure automatic shutdown of a Windows computer
- **Web browser:** Available only to authorized specialists
- **SSH:** Available only to authorized specialists
- **XDMCP:** Available only to authorized specialists
- **User-defined connection:** Available only to authorized specialists



HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.



If you are using a TNC 320 with touch control, you can replace some keystrokes with hand-to-screen contact.
Further information: "Operating the Touchscreen", Page

Configuring connections – Windows Terminal Service (RemoteFX)

Configuring an external computer



You do not need additional software for your external computer for connecting to the Windows Terminal Service.



Proceed as follows to configure the external computer, e.g. in the Windows 7 operating system:





- ▶ Press the Windows Start button and select **Control Panel** on the Start menu
- ▶ Select **System and Security**
- ▶ Select **System**
- ▶ Select **Remote settings**
- ▶ Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
- ▶ Under **Remote Desktop**, select **Allow connections from computers running any version of Remote Desktop**
- ▶ Press **OK** to confirm your settings

Configuring the control

Proceed as follows to configure the control:

- ▶ Press the **DIADUR** key to open the HeROS menu
- ▶ Select **Remote Desktop Manager**
- > The control opens the **Remote Desktop Manager**.
- ▶ Press **New connection**
- ▶ Press **Windows Terminal Service (RemoteFX)**
- > The control opens the **Select server operating system** pop-up window.
- ▶ Select the desired operating system
 - Win XP
 - Win 7
 - Win 8.X
 - Win 10
 - Another Windows
- ▶ Press **OK**
- > The control opens the **Edit the connection** pop-up window.
- ▶ Edit the connection

Setting	Meaning	Input
Connection name	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection	Behavior with terminated connection: <ul style="list-style-type: none"> ■ Always restart ■ Never restart ■ Always after an error ■ Ask after an error 	Required
Automatic starting upon login	Connection automatically established during control power-up	Required
Add to favorites	Connection icon in the task bar: <ul style="list-style-type: none"> ► Single click with the left mouse button > The control switches to the desktop of the connection. ► Single click with the right mouse button > The control displays the connection menu. 	Required
Move to the following workspace	Number of desktop for the connection, whereby desktops 0 and 1 are reserved for the NC software Default setting: Third desktop	Required
Release USB mass memory	Enable access to connected USB mass memory	Required
Computer	Host name or IP address of the external computer HEIDENHAIN recommends the following setting for the IPC (6641): IPC6641.machine.net The host name IPC6641 must be assigned to the IPC in the Windows operating system for this setting. <div data-bbox="507 1261 1326 1440">  The .machine.net code is very important in this context. When .machine.net is entered, the control automatically searches the Ethernet interface X116, and not the interface X26—this reduces the time needed for access. </div>	Required
User name	Name of the user	Required
Password	User password	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or User-defined window size	Size of the connection window	Required
Multimedia enhancements	Enables hardware acceleration when playing videos Certain formats, e.g. MP4 files, require the paid-for Fluendo Codec Pack <div data-bbox="507 1787 1326 1888">  Any additional software is installed by your machine tool builder. </div>	Optional
Touchscreen input	Enables operation of multitouch systems and applications	Optional

Setting	Meaning	Input
Encryption	<p>Sets the encryption appropriate for the selected Windows system</p> <div>  <p>If you activate the Encryption function, you need to remove the entries -sec-tls -sec-nla from the Additional options input field.</p> <p>If problems occur, you should try to set up a connection with the function deactivated. The Windows log files are required for an analysis.</p> </div>	Required
Color depth	Setting for the display of the external system on the control	Required
Locally effective keys	<p>Shortcuts for automatically switching to the next active connection and workspace or desktop</p> <p>Default setting:</p> <ul style="list-style-type: none"> ■ Super_R equals the right DIADUR key and switches between the active connections ■ F12 switches between the workspaces or desktops <div>  <p>Touchscreens no longer provide an F12 key. Here, the vacant key between PGM MGT and ERR serves to switch the workspaces or desktops.</p> </div> <p>It is possible to adjust the default settings or to make additional entries</p>	Required
Max. connection time (seconds)	<p>Waiting time for connection</p> <p>Exceeding this time corresponds to a broken connection</p>	Required
Additional options	<p>Available only to authorized specialists</p> <p>Additional command lines with transfer parameters</p> <div>  <p>If you activate the Encryption function, you need to remove the entries -sec-tls -sec-nla from the Additional options input field.</p> </div>	Required
Pass USB devices through	<p>Passing the USB devices (e.g. 3-D mouse for operating CAD programs) connected to the control through to the Windows computer</p> <p>This requires that the Eltima EveUSB software be installed on the Windows computer.</p> <div>  <p>All USB devices that have been passed through are not available on the control while the connection to the Windows computer is active.</p> </div>	Optional

For integrating the IPC 6641, HEIDENHAIN recommends using a RemoteFX connection.

For RemoteFX, the screen of the external computer is not mirrored, as for VNC. Instead, a separate desktop is opened. The desktop that is active on the external computer when the connection is established is then locked and the user is logged off. This prevents that two users access the control simultaneously.

Configuring the connection – VNC

Configuring an external computer



You do not need an additional VNC server for your external computer for connecting to VNC.
Install and configure the VNC server, e.g. the TightVNC server, before configuring the control.

Configuring the control

Proceed as follows to configure the control:

- ▶ Press the **DIADUR** key to open the HeROS menu
- ▶ Select **Remote Desktop Manager**
- > The control opens the **Remote Desktop Manager**.
- ▶ Press **New connection**
- ▶ Press **VNC**
- > The control opens the **Edit the connection** pop-up window.
- ▶ Edit the connection

Setting	Meaning	Input
Connection name:	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection:	Behavior with terminated connection: <ul style="list-style-type: none"> ■ Always restart ■ Never restart ■ Always after an error ■ Ask after an error 	Required
Automatic starting upon login	Connection automatically established during control power-up	Required
Add to favorites	Connection icon in the task bar: <ul style="list-style-type: none"> ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu. 	Required
Move to the following workspace	Number of desktop for the connection, whereby desktops 0 and 1 are reserved for the NC software Default setting: Third desktop	Required
Release USB mass memory	Permit access to connected USB mass memory	Required
Calculator	Host name or IP address of the external computer. In the recommended configuration of the IPC 6641, the IP address 192.168.254.3 is used	Required
User name:	Name of the user to be logged on	Required
Password	Password for connecting to the VNC server	Required

Setting	Meaning	Input
Full-screen mode or User-defined window size:	Size of the connection window	Required
Permit further connections (share)	Enable access to the VNC server also by other VNC connections	Required
View only	The external computer cannot be operated in display mode	Required
Entries in the Advanced options area	Available only to authorized specialists	Optional



If you are using **Extended Workspace Compact**, select the **Extended Workspace, Compact** function to enable the corresponding configuration for your connection.

By selecting the **Extended Workspace, Compact** function, you automatically adapt the connections to the additional workspace.

Further information: "Extended Workspace Compact", Page

With VNC, the screen of the external computer is mirrored directly. The active desktop on the external computer is not locked automatically.

With a VNC connection, it is also possible to completely shut down the external computer via the Windows menu. As the computer cannot be rebooted over a connection, it must actually be switched off and on again.

Shutting down or rebooting an external computer

NOTICE

Caution: Data may be lost!

If you do not shut down the external computer properly, data may be irreversibly damaged or deleted.

- Configure the automatic shutdown of the Window computer

Proceed as follows to configure the control:

- Press the **DIADUR** key to open the HeROS menu
- Select **Remote Desktop Manager**
- > The control opens the **Remote Desktop Manager**.
- Press **New connection**
- Press **Switch-off/restart of a computer**
- > The control opens the **Edit the connection** pop-up window.
- Edit the connection

Setting	Meaning	Input
Connection name:	Name of the connection in the Remote Desktop Manager	Required
Restarting after end of connection:	Not necessary with this connection	–

Setting	Meaning	Input
Automatic starting upon login	Not necessary with this connection	–
Add to favorites	Connection icon in the task bar: <ul style="list-style-type: none"> ▶ Single click with the left mouse button > The control switches to the desktop of the connection. ▶ Single click with the right mouse button > The control displays the connection menu. 	Required
Move to the following workspace	Not active with this connection	–
Release USB mass memory	Not advisable with this connection	–
Calculator	Host name or IP address of the external computer. In the recommended configuration of the IPC 6641, the IP address 192.168.254.3 is used	Required
User name	User name with which the connection should be logged on	Required
Password	Password for connecting to the VNC server	Required
Windows domain:	Domain of the target computer, if required	Optional
Max. waiting time (seconds):	When the control shuts down, it commands the Windows computer to shut down. Before the control displays the message Now you can switch off. , it waits for <Timeout> seconds. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the <Timeout> seconds have expired, the control stops waiting.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after closing port 445.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If Force is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Reboot the Windows computer	Required
Run during restart	Reboot the Windows computer when the control reboots. Effective only if the control is rebooted using the shutdown icon at the bottom right in the taskbar or if it is rebooted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	The Windows computer is switched off when the control is shut down (no reboot). This is the standard scenario. The END key will then no longer trigger a reboot, either.	Required
Entries in the Advanced options area	Available only to authorized specialists	Optional

Starting and stopping the connection

Once a connection has been configured, this connection is shown as an icon in the Remote Desktop Manager window. Click the connection icon with the right mouse key to open a menu in which the display can be started and stopped.

If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there.

When the HEROS 5 operating system is shut down, the control automatically terminates all connections. Please note, however, that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Further information: "Shutting down or rebooting an external computer", Page 305

Proceed as follows to switch between the third desktop and the control interface:

- Use the right DIADUR key on the alphabetic keyboard
- Use the taskbar
- Use the operating mode key

9.2 Additional tools for ITCs

The following additional tools allow you to apply various settings for the touch screens on connected ITCs.

ITCs are industrial PCs without their own memory media, and therefore they do not have their own operating system. This feature is what makes ITCs different from IPCs.

ITCs are frequently used with large machinery, e.g. as a clone of the actual control system.



Refer to your machine manual.

The machine manufacturer defines and configures the display and function of the connected ITCs and IPCs.

Additional tool	Application
ITC Calibration	4-point calibration
ITC Gestures	Configuration of gesture control
ITC touchscreen configuration	Selection of touch sensitivity



The additional tools for the ITCs are only provided by the control in the taskbar with connected ITCs.

ITC Calibration

Using the additional tool **ITC Calibration**, you align the position for the mouse cursor displayed with the actual movement position of your finger.

Calibration using the additional **ITC Calibration** tool is recommended in the following cases:

- After replacing the touchscreen
- When changing the touch screen position (parallel axis error due to amended viewing angle)

Calibration involves the following steps:

- ▶ Start the tool in control using the task bar
- > The ITC opens the calibration screen with four touch points in the corners of the screen
- ▶ Touch the four touch points shown one after the other
- > The ITC closes the calibration screen once calibration has been successfully completed

ITC Gestures

Using the additional **ITC Gestures** tool, the machine manufacturer configures the gesture control on the touch screen.



Refer to your machine manual.

This function may only be used with the permission of your machine manufacturer.

ITC touchscreen configuration

Using the additional **ITC Touchscreen Configuration** tool, you can select the touch sensitivity of the touch screen.

The ITC gives you the following options:

- **Normal Sensitivity (Cfg 0)**
- **High Sensitivity (Cfg 1)**
- **Low Sensitivity (Cfg 2)**

Use the **Normal Sensitivity (Cfg 0)** setting as standard. If you find it difficult to operate the equipment while wearing gloves in this setting, select the **High Sensitivity (Cfg 1)** setting.



If the ITC touch screen is not splash-proof, select the **Low Sensitivity (Cfg 2)** setting. This stops the ITC interpreting drops of water as touches.

Configuration involves the following steps:

- ▶ Start the tool in control using the task bar
- > The ITC opens a pop-up window with three options
- ▶ Select Touch Sensitivity
- ▶ Press the **OK** button
- > The ITC closes the pop-up window

9.3 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.

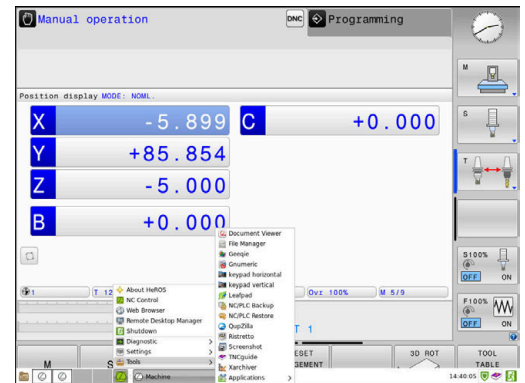
Overview of taskbar

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: CAD-Viewer 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 314
 - **Portscan OEM:** Available only to authorized specialists
 - **RemoteService:** Start and stop remote maintenance
Further information: "Remote Service", Page 315
 - **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 317
- **Screensaver:** Setting the screensaver
Further information: "Screensaver with lockout", Page 372
- **Current User:** Show the current user
Further information: "Current User", Page 374
- **UserAdmin:** Configure the user administration
Further information: "Configuring the user administration", Page 349
- **OEM Function Users:** Edit the OEM function users
Further information: "HEIDENHAIN function users", Page 360
- **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 320
- **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 323
- **WindowManagerConfig:** Available only to authorized specialists
- **Firewall:** Configure the firewall
Further information: "Firewall", Page 329
- **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 326
 - **NC/PLC Restore:** Restore backup file
Further information: "Backup and restore", Page 326
 - **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 371



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 77

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 310
- ▶ 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 310
- ▶ 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.

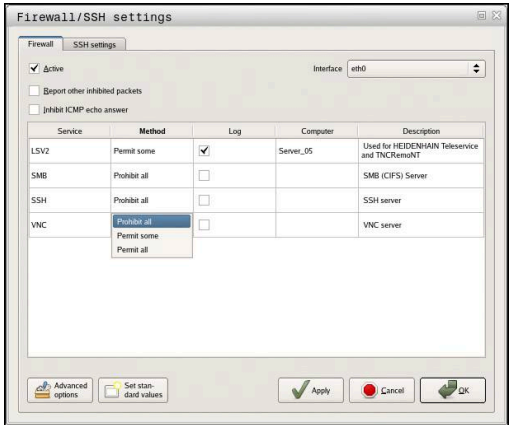
Further information: "General network settings", Page 338


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 310
- ▶ 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.

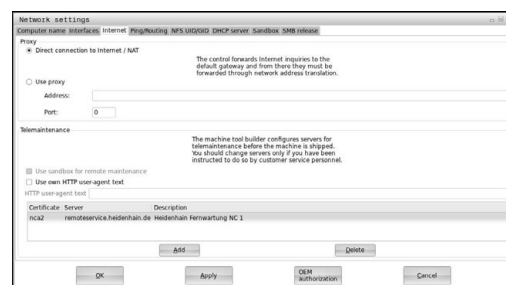
Proceed as follows to install the certificate on the control:

- ▶ Taskbar at the bottom edge of the screen
- ▶ **Further information:** "Window manager", Page 310
- ▶ 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 310
- ▶ 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 329

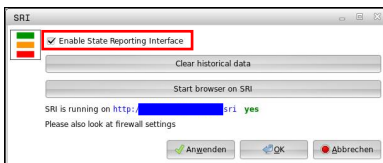


If the **SRI** is accessed locally via an IPC connected to the machine network (X116), eth0 (X26) can also remain locked.

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 311



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:

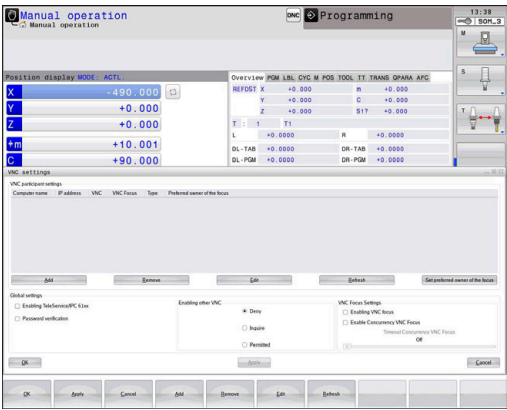
- List of permitted clients (IP address or name)
- Password for the connection
- Additional server options
- Additional settings for assigning the focus



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 310
- ▶ 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 329.</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 310
- ▶ 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.


9.4 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 310
- ▶ 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 344</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 366 ■ 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

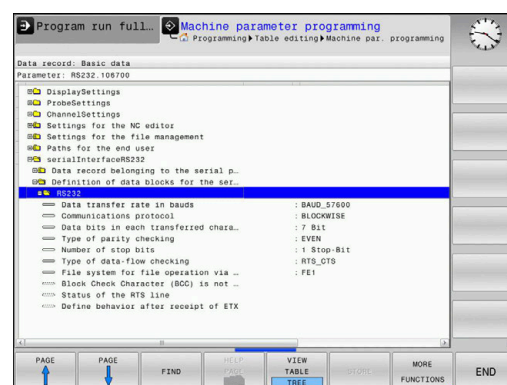
9.5 Setting up data interfaces

Serial interfaces on the TNC 320

The TNC 320 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.

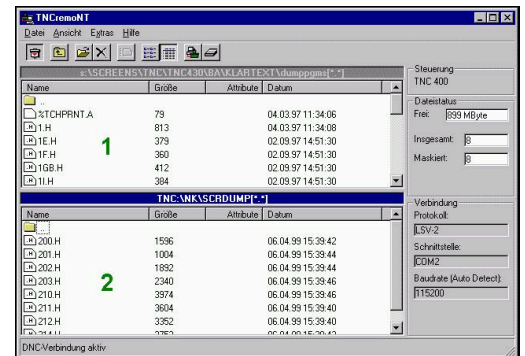
If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- ▶ 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 73



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

Further information: "Available tool types", Page 138

End TNCremo

Select <File>, <Exit>



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

9.6 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.

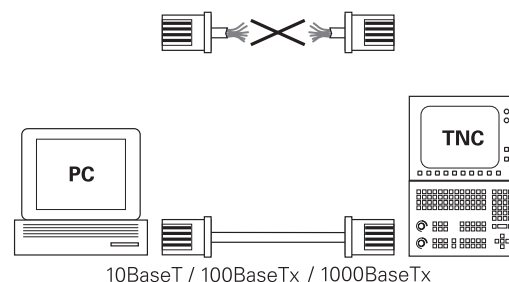
Connection possibilities

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:



- ▶ Press the **MOD** key



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




- ▶ Press the **NET** soft key



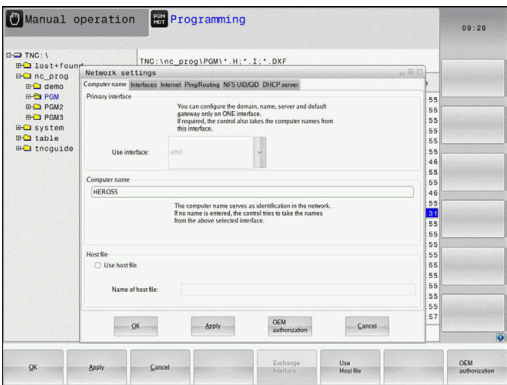
- ▶ Press the **CONFIGURE NETWORK** soft key

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>
----------------------------	---

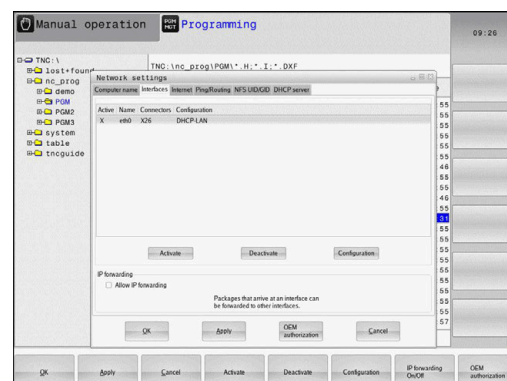
Proceed as follows to access the configuration menu:

- 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 option: The control is to procure the IP address from the DHCP server ■ Manually set IP address option: 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
-------------------	--

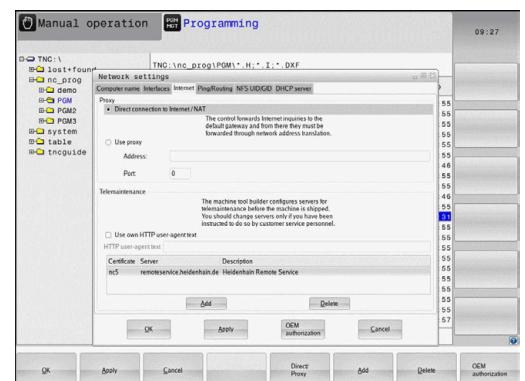


Setting	Meaning
Domain Name Server (DNS)	<ul style="list-style-type: none"> ■ Automatically procure DNS option: The control is to automatically procure the IP address of the domain name server ■ Manually configure the DNS option: Manually enter the IP addresses of the servers and the domain name
Default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

Setting	Meaning
Proxy	<ul style="list-style-type: none"> ■ Direct connection to Internet / NAT: The control forwards Internet inquiries to the default gateway and from there they must be forwarded through network address translation (e.g. if a direct connection to a modem is available) ■ Use proxy: Define the Address and Port of the Internet router in your network, ask your network administrator for the correct address and port
Telemaintenance	The machine manufacturer configures the server for telemaintenance here. Changes must always be made in agreement with your machine tool builder



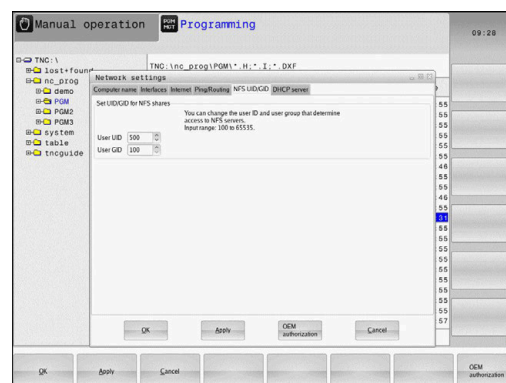
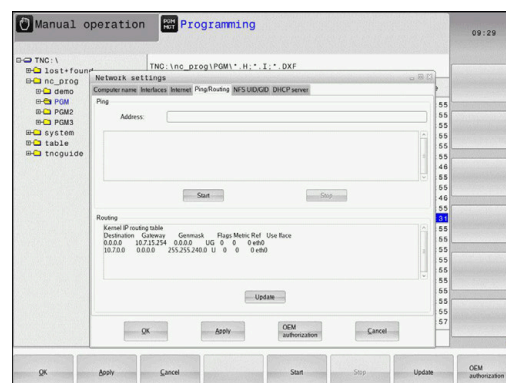
Ping/Routing tab

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

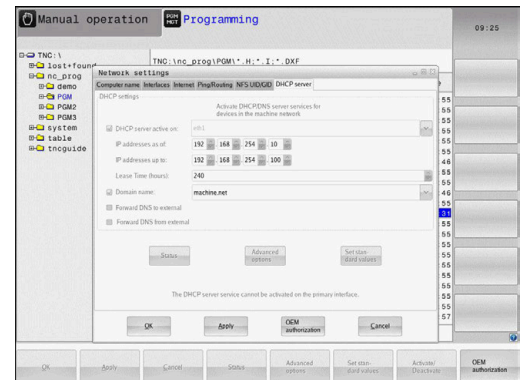
On the **NFS UID/GID** tab, you enter the user and group identifications.

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 329

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:



- ▶ Press the **MOD** key

Alternative:



- ▶ Press the **PGM MGT** key



- ▶ Press the **NET** soft key



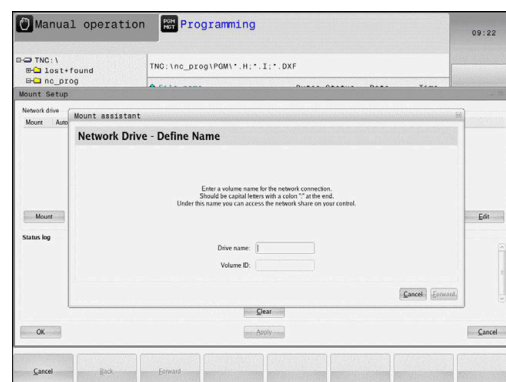
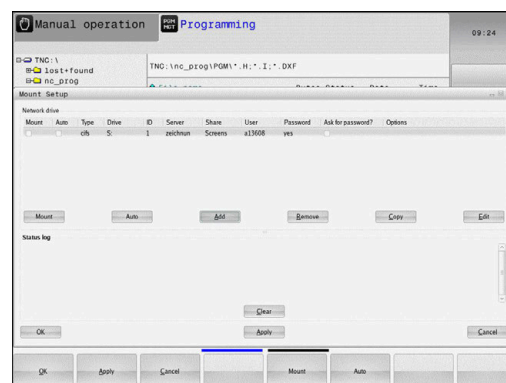
- ▶ Press the **DEFINE NETWORK CONNECTN.** soft key

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>



9.7 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.

9.8 User administration



Refer to your machine manual.

Some user administration areas are configured by the machine tool builder.

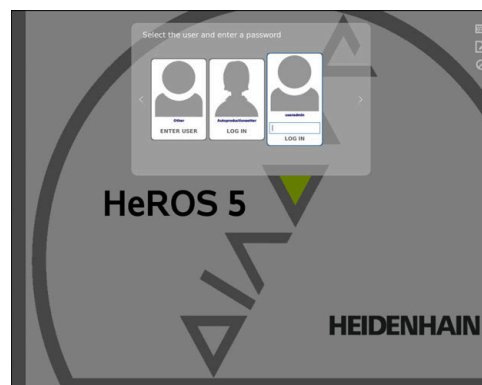
If you want to use the user administration feature on a control without HEIDENHAIN keyboard, you need to connect an external alphabetic keyboard to the control.

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 351

- **LDAP database on remote computer**

- Importing an LDAP server configuration file

Further information: "LDAP on remote computer", Page 352

- **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 353



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 348

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

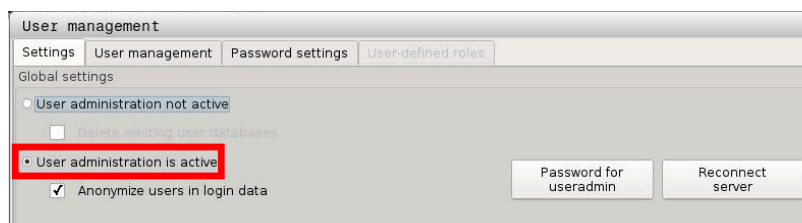
- **Further information:** "Local LDAP database", Page 351
- **Further information:** "LDAP on remote computer", Page 352
- **Further information:** "Connection to a Windows domain", Page 353

3. Create further users

Further information: "Creating further users", Page 356

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 361
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.

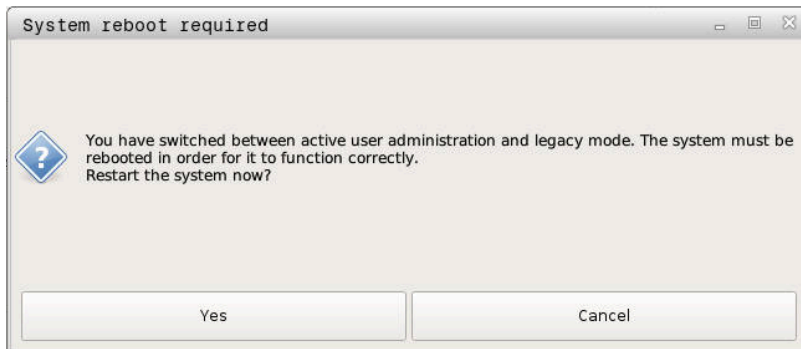
2. Proceed as follows to set up the database:

- ▶ Select the database for saving the user data
 - Local LDAP database

Further information: "Local LDAP database", Page 351
 - LDAP on remote computer

Further information: "LDAP on remote computer", Page 352
 - Connection to a Windows domain

Further information: "Connection to a Windows domain", Page 353
- ▶ 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.



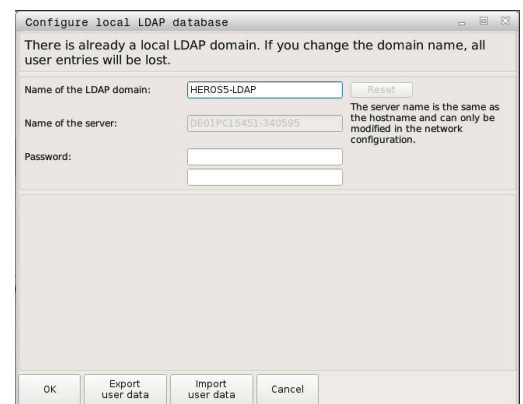
Local LDAP database

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 356

LDAP on remote computer

Before you can use the **LDAP on remote computer** function, the following requirements must be met:

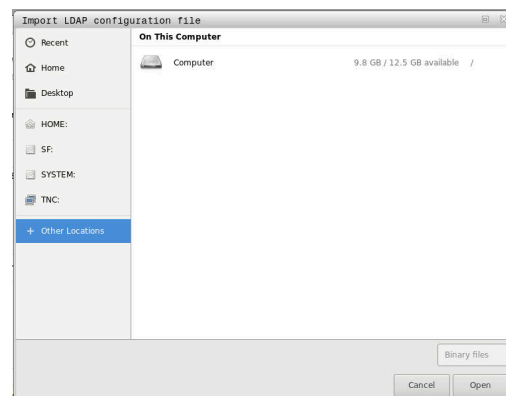
- The user administration is active
- The **useradmin** user has been configured
- An LDAP database has been set up in the company network
- A server configuration file of an existing LDAP database must be stored on the control or on a PC in the network
- The PC with the existing configuration file is 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

Configuration

Find domain

Add role definition

HEROS role base:

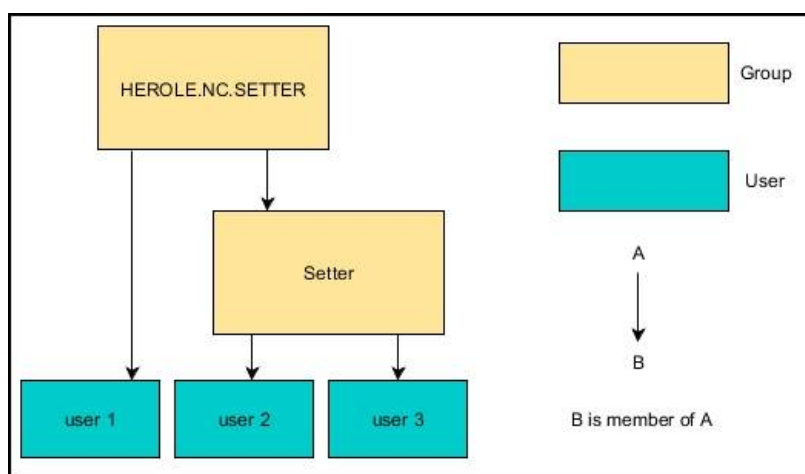
There are the following ways to create groups corresponding to the different roles:

- 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

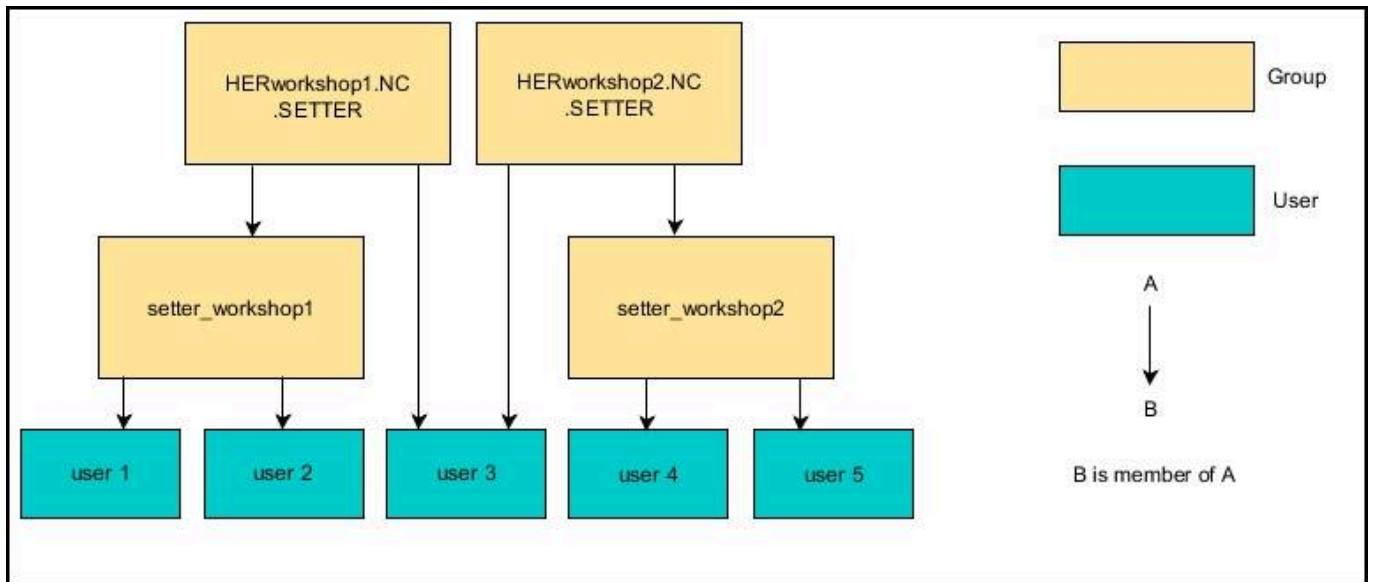
The users need to be added manually by the Windows administrator to the roles (security groups) on the domain controller.

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 353

- ▶ 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:

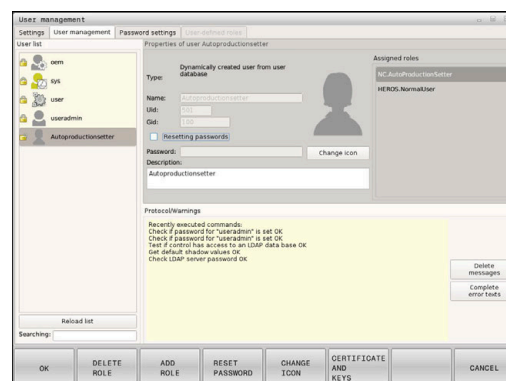
- ▶ Press the **CREATE NEW USER** soft key
- ▶ The control opens a window for creating a user.
- ▶ Enter the user name
- ▶ Enter a password for the user



The user has to change this password the first time he logs on.

Further information: "Login to user administration", Page 370

- ▶ 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 361
- ▶ 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 361

- ▶ 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 349

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 370

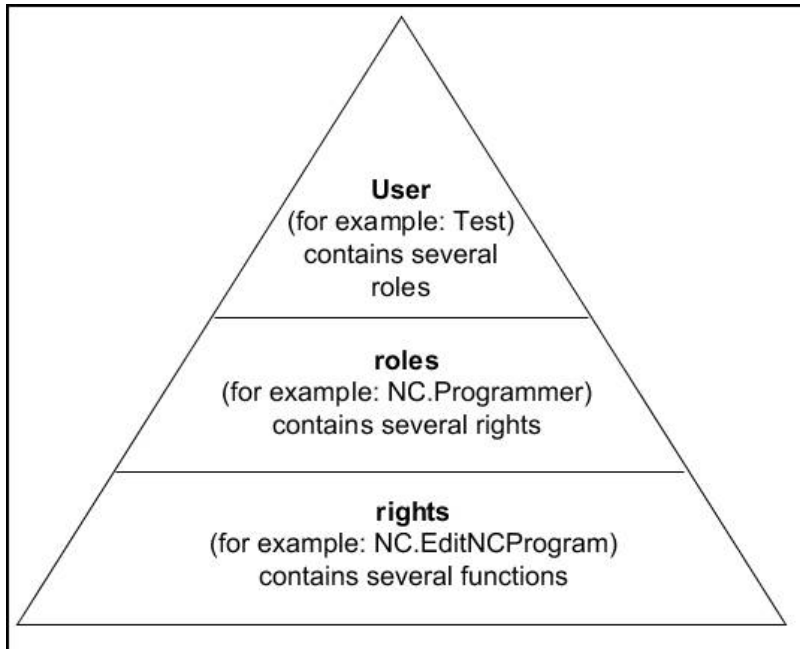
- ▶ Call the user administration
- ▶ Select the **User management** tab
- ▶ Press the **EDIT USER** soft key
- ▶ Press the **CHANGE ICON** soft key
- ▶ Select image
- ▶ Use the **CHOOSE ICON** soft key to select an image
- ▶ Press the **OK** soft key
- ▶ Press the **APPLY** soft key.
- > The changes 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 360

- **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 356

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**



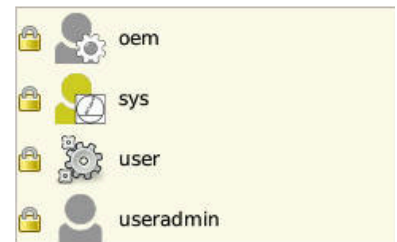
Refer to your machine manual.

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 374



■ **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 356

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

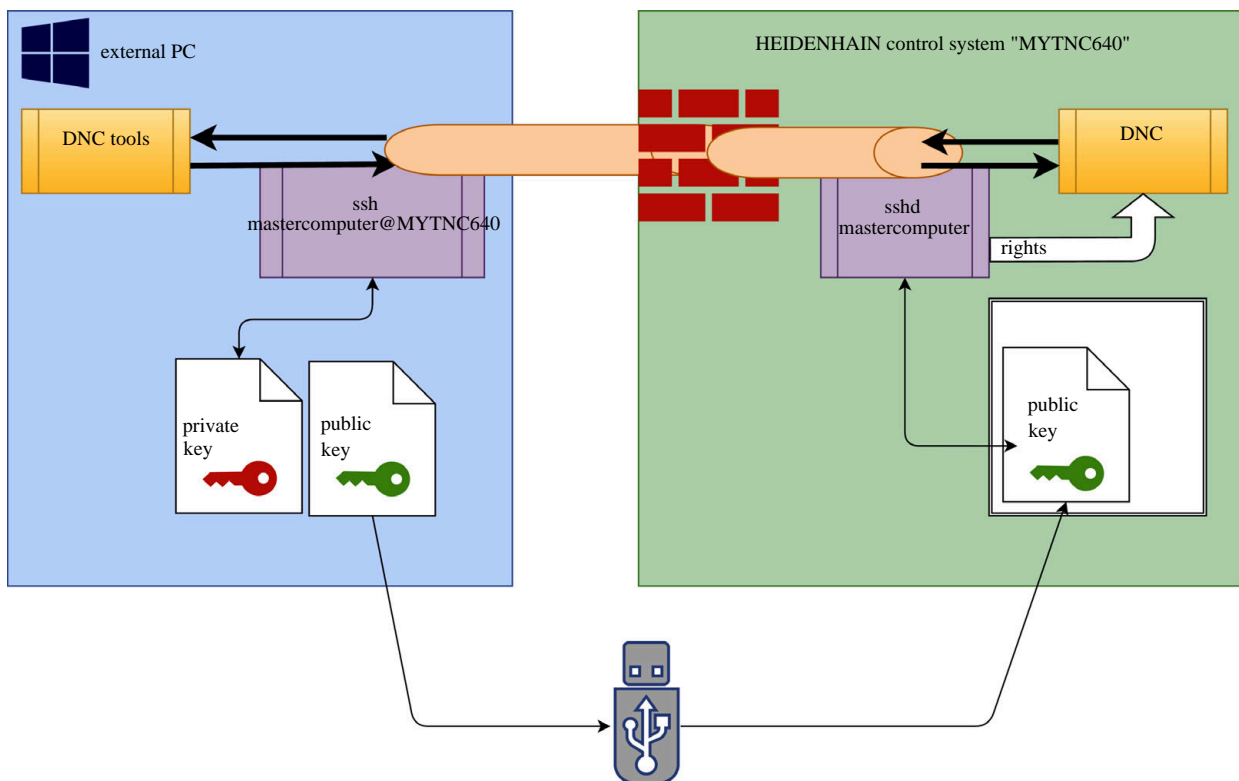
Prerequisites:

- 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

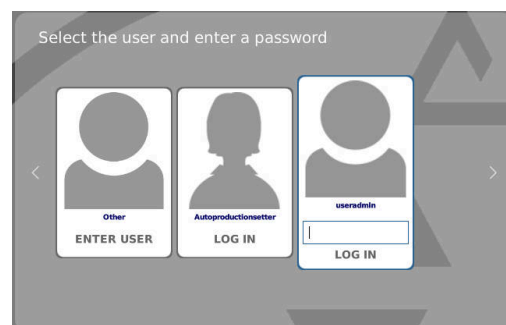
Proceed as follows to log in a user who is already shown in the login dialog:

- ▶ 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

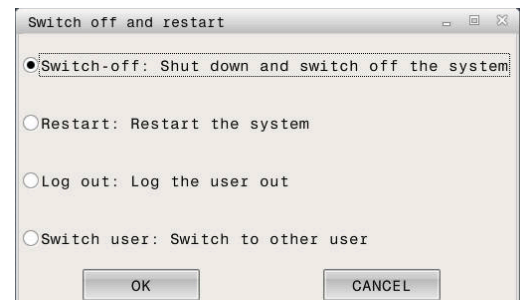


To continue, you need to log in a new user and enter his password.
The NC program continues running under the previously logged-on user.

- **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.



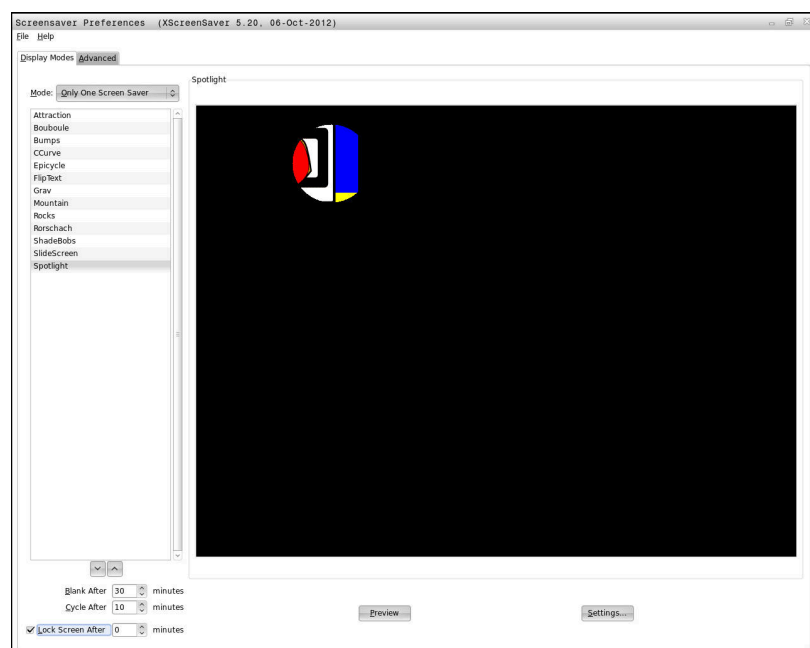
Screensaver with lockout

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 370



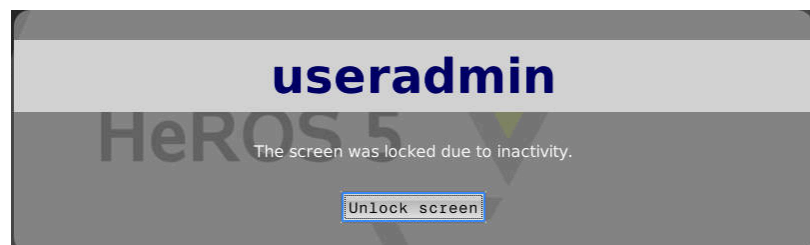
Proceed as follows to access the screensaver settings in the HEROS menu:

- ▶ 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.



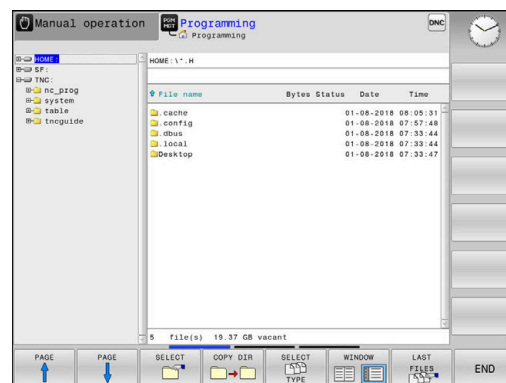
Use **Unlock screen** or Enter to call the login window again.

Further information: "Login to user administration", Page 370

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 360

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 360

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

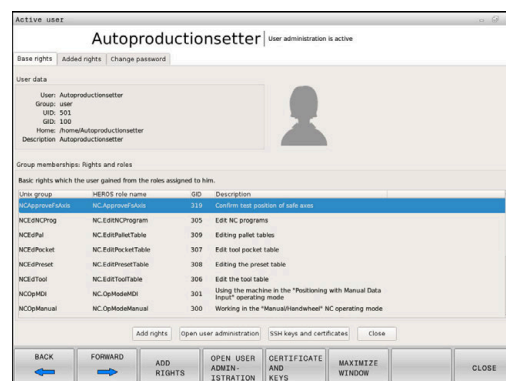
Proceed as follows to select the **REMOVE ADDED RIGHTS** soft key:

- ▶ 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 370

- > 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:

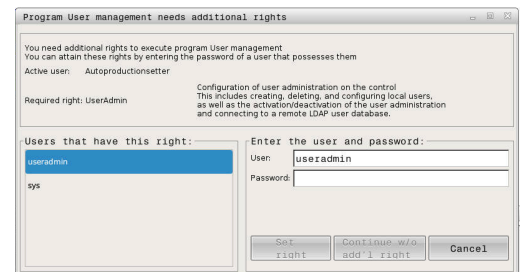
In this window, the control offers you to temporarily increase your rights by adding another user's rights.

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 374

9.9 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 319

The following link provides information on how to change the NC conversational language:

Further information: "List of user parameter", Page 381

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.

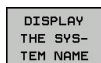
Changing the display of the parameters

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

DisplaySettings

Setting the coordinate systems for the display

Coordinate system for the datum shift

WorkplaneSystem: Datum is displayed in the system of the tilted plane, WPL-CS

WorkpieceSystem: Datum is displayed in the workpiece coordinate system, W-CS

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

Geometry tolerances

Permissible deviation of the circle radius

0.0001 to 0.016 [mm]: Permissible deviation of the radius at the circle end-point compared with the circle start-point

Permissible error in successive threads

0.0001 bis 999.9999 [mm]: Permissible deviation of the dynamically smoothed contour from the programmed thread contour

Configuration of the fixed cycles

Overlap factor for pocket milling

0.001 to 1.414: Overlap factor for Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET

Traverse after machining a contour pocket

PosBeforeMachining: Tool moves to the same position as before the cycle was machined**ToolAxClearanceHeight: Position the tool axis to clearance height**Display the **Spindle ?** error message if M3/M4 is not active**On: Issue error message****Off: No error message**Display the **Enter depth as negative****On: Issue error message****Off: No error message**

Behavior when moving to wall of slot in the cylinder surface

LineNormal. Approach on a straight line**CircleTangential: Approach on a circular path**

Parameter settings

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

Do not show the **Residual material** warning

On: Warning is not displayed

Off: Warning is displayed

Geometry filter for culling linear elements

Type of stretch filter

- Off: No filter active

- ShortCut: Omit individual points on a polygon

- Average: The geometry filter smooths corners

Maximum distance of the filtered to the unfiltered contour

0 to 10 [mm]: The filtered points lie within this tolerance to the resulting new path

Maximum length of the distance resulting from filtering

0 to 1000 [mm]: Length over which geometry filtering is active

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

Parameter settings

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

Behavior of paraxial positioning blocks

TRUE: Paraxial positioning blocks enabled

FALSE: Paraxial positioning blocks disabled

Line number to which identical syntax elements are searched for

500 to 50000: Search for selected elements with up or down arrow keys

Parameter settings

Behavior of PARAXMODE function with UVW axes

FALSE: PARAXMODE function enabled

TRUE: PARAXMODE function disabled

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 332

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	Assignment	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°
Interpolation	<ul style="list-style-type: none"> ■ Linear in 4 axes ■ Circular in 2 axes ■ Helical: superimposition of circular and straight paths
Block processing time 3-D straight line without radius compensation	<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, reversal peaks during circular movements, 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 +40 °C ■ Storage: -20 °C to +60 °C

Input formats and units of control functions

Positions, coordinates, circle radii, 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) [°]
Angle for polar coordinates, rotation, tilting the working plane	–360.0000 to 360.0000 (3, 4) [°]
Polar coordination for screw line interpolation (CP)	–5 400.0000 to 5 400.0000 (4, 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 ■ Fourth NC axis plus auxiliary axis ■ or □ Additional axis for 4 axes plus closed-loop spindle □ Additional axis for 5 axes plus closed-loop spindle
Program entry	In HEIDENHAIN conversational format and DIN/ISO
Position entry	<ul style="list-style-type: none"> ■ Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates ■ Incremental or absolute dimensions ■ Display and entry in mm or inches
Tool compensation	<ul style="list-style-type: none"> ■ Tool radius in the working plane and tool length ■ Radius-compensated contour look-ahead for up to 99 NC blocks (M120)
Tool tables	Multiple tool tables with any number of tools
Constant contour speed	<ul style="list-style-type: none"> ■ With respect to the path of the tool center ■ With respect to the cutting edge
Parallel operation	Creating an NC program with graphical support while another NC program is being run
Rotary table machining (Advanced Function Set 1)	1 Programming of cylindrical contours as if in two axes 1 Feed rate in distance per minute

User functions

Contour elements	<ul style="list-style-type: none"> ■ Straight line ■ Chamfer ■ Circular path ■ Circle center ■ Circle radius ■ Tangentially connected arc ■ Rounded corners
Approaching and departing the contour	<ul style="list-style-type: none"> ■ Via straight line: tangential or perpendicular ■ Via circular arc
FK free contour programming	<ul style="list-style-type: none"> ■ FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	<ul style="list-style-type: none"> ■ Subprograms ■ Program section repeat ■ Any desired NC program as subprogram
Machining cycles	<ul style="list-style-type: none"> ■ Cycles for drilling, and conventional and rigid tapping ■ Roughing of rectangular and circular pockets ■ Cycles for pecking, reaming, boring, and counterboring ■ Cycles for milling internal and external threads ■ Finishing of rectangular and circular pockets ■ Cycles for clearing level and inclined surfaces ■ Cycles for milling linear and circular slots ■ Cartesian and polar point patterns ■ Contour-parallel contour pocket ■ Contour train ■ OEM cycles (special cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	<ul style="list-style-type: none"> ■ Datum shift, rotation, mirroring ■ Scaling factor (axis-specific) 1 Tilting the working plane (Advanced Function Set 1)

User functions

Q parameters

Programming with variables

- Mathematical functions: =, +, −, *, sin α , cos α , root
 - Logical operations (=, ≠, <, >)
 - Calculating with parentheses
 - tan α , arc sin, arc cos, arc tan, a^n , e^n , ln, log, absolute value of a number, constant π , negation, truncation of digits before or after the decimal point
 - Functions for calculation of circles
 - String parameters
-

Programming aids

- Calculator
 - Color highlighting of syntax elements
 - Complete list of all current error messages
 - Context-sensitive help function for error messages
 - Graphic support for the programming of cycles
 - Comment blocks in NC program
-

Teach-In

- Actual positions can be transferred directly to the NC program
-

User functions

Test graphics Display modes	■	Graphic simulation before a program run, even while another NC program is being run
	■	Plan view / projection in 3 planes / 3-D view / 3-D line graphic
	■	Detail enlargement
Programming graphics	■	In the Programming 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	■	Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■	Calculation of machining time in the Test Run operating mode Test Run
	■	Display of the current machining time in the Program Run operating modes
Contour, returning to	■	Block scan in any NC block in the NC program, returning the tool to the calculated nominal position to continue machining
	■	NC program interruption, contour departure and return
Datum tables	■	Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	■	Calibrating the touch probe
	■	Compensation of workpiece misalignment, manual or automatic
	■	Presetting, manual or automatic
	■	Automatically measuring workpieces
	■	Cycles for automatic tool measurement

Accessories

Accessories

Electronic handwheels	<ul style="list-style-type: none"> ■ 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
Touch probes	<ul style="list-style-type: none"> ■ TS 248: 3-D touch trigger probe with cable connection ■ TS 260: 3-D touch trigger probe with cable connection ■ TS 444: Battery-free 3-D touch trigger probe with infrared transmission ■ TS 460: 3-D touch trigger probe with infrared and radio transmission ■ TS 642: 3-D touch trigger probe with infrared transmission ■ TS 740: High-precision 3-D touch trigger probe with infrared transmission ■ TT 160: 3-D touch trigger probe for tool measurement ■ TT 460: 3-D touch trigger probe for tool measurement with infrared transmission

10.4 Differences between the TNC 320 and the iTNC 530

Comparison: Specifications

Function	TNC 320	iTNC 530
Control loops	Maximum 6 control loops (including up to 2 spindles)	18 maximum
Input resolution and display step:		
■ Linear axes	■ 0.1 µm	■ 0.1 µm
■ Rotary axes	■ 0.001°	■ 0.0001°
Display	15.1-inch screen with soft keys	19-inch screen or 15.1-inch screen with soft keys
Memory media for NC, PLC programs, and system files	CompactFlash memory card	Hard disk or SSDR solid state disk
Program memory for NC programs	2 GB	> 21 GB
Block processing time	3 ms	0.5 ms
Interpolation:		
■ Straight line	■ 5 axes	■ 5 axes
■ Circle	■ 3 axes	■ 3 axes
■ Helix	■ yes	■ Yes
■ Spline	■ No	■ Yes with option 9
Hardware	Compact in operating panel	Modular in electrical cabinet

Comparison: Data interfaces

Function	TNC 320	iTNC 530
RS-422/V.11 serial interface	-	X

Further information: "Setting up data interfaces", Page 332

Comparison: PC software

Function	TNC 320	iTNC 530
ConfigDesign for the configuration of machine parameters	Available	Not available
TNCAnalyzer for the analysis and evaluation of service files	Available	Not available

Comparison: User functions

Function	TNC 320	iTNC 530
Program entry		
■ smarT.NC	■ –	■ X
■ ASCII editor	■ X, directly editable	■ X, editable after conversion
Position entry		
■ Set the last tool position as pole (empty CC block)	■ X (error message if pole transfer is ambiguous)	■ X
■ Spline sets (SPL)	■ –	■ X, with option 9
Tool compensation		
■ Three-Dimensional Tool Radius Compensation	■ –	■ X, with option 9
Tool table		
■ Flexible management of tool types	■ X	■ –
■ Filtered display of selectable tools	■ X	■ –
■ Sorting function	■ X	■ –
■ Column names	■ Sometimes with _	■ Sometimes with -
■ Form view	■ Switchover with Screen Layout key	■ Switchover by soft key
■ Exchange of tool table between TNC 320 and iTNC 530	■ X	■ Not possible
Touch probe table for managing different 3-D touch probes	X	–
Cutting data calculator: Automatic calculation of spindle speed and feed rate	<ul style="list-style-type: none"> ■ Simple cutting data calculator without stored table ■ Cutting data calculator with stored technology tables 	Using stored technology tables

Function	TNC 320	iTNC 530
Define any tables	<ul style="list-style-type: none"> ■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions ■ Definable via config. data ■ The names of tables and table columns must start with a letter, and no arithmetic operators are permitted ■ Reading and writing with SQL functions 	<ul style="list-style-type: none"> ■ Freely definable tables (.TAB files) ■ Reading and writing with FN functions
Traverse in tool-axis direction		
■ Manual operation (3-D ROT menu)	■ X	■ X, FCL2 function
■ With handwheel superimpositioning	■ X	■ X, option 44
Entry of feed rates:		
■ FT (time in seconds for path)	■ –	■ X
■ FMAXT (only for active rapid traverse potentiometer: time in seconds for path)	■ –	■ X
FK free contour programming		
■ Conversion of FK program to Klartext conversational language	■ –	■ X
■ FK blocks in combination with M89	■ –	■ X
Program jumps:		
■ Maximum number of labels	■ 65535	■ 1000
■ Subprograms	■ X	■ X
■ Nesting depth for subprograms	■ 20	■ 6

Function	TNC 320	iTNC 530
Q parameter programming:		
■ FN 15: PRINT	■ –	■ X
■ FN 25: PRESET	■ –	■ X
■ FN 29: PLC LIST	■ X	■ –
■ FN31: RANGE SELECT	■ –	■ X
■ FN32: PLC PRESET	■ –	■ X
■ FN37: EXPORT	■ X	■ –
■ Write to LOG file with FN16	■ X	■ –
■ Displaying parameter contents in the additional status display	■ X	■ –
■ SQL functions for writing and reading tables	■ X	■ –
Graphic support		
■ 2-D programming graphics	■ X	■ X
■ REDRAW function (REDRAW)	■ –	■ X
■ Show grid lines as the background	■ X	■ –
■ Test graphics (plan view, projection on 3 planes, 3-D view)	■ X	■ X
■ Coordinates of line intersection for projection in 3 planes	■ –	■ X
■ Factor in tool change macro	■ X (differing to actual execution)	■ X
Preset table		
■ Line 0 of the preset table can be edited manually	■ X	■ –
Pallet management		
■ Support of pallet files	■ –	■ X
■ Tool-oriented machining	■ –	■ X
■ Management of presets for a pallet in a table	■ –	■ X

Function	TNC 320	iTNC 530
Programming aids:		
■ Color highlighting of syntax elements	■ X	■ –
■ Calculator	■ X (scientific)	■ X (standard)
■ Convert NC blocks to comments	■ X	■ –
■ Structure blocks in NC program	■ X	■ X
■ Structure view in test run	■ –	■ X
Dynamic Collision Monitoring (DCM):		
■ Collision monitoring in Automatic operation	■ –	■ X, option 40
■ Collision monitoring in Manual operation	■ –	■ X, option 40
■ Graphic depiction of the defined collision objects	■ –	■ X, option 40
■ Collision checking in test run	■ –	■ X, option 40
■ Fixture monitoring	■ –	■ X, option 40
■ Tool carrier management	■ X	■ X, option 40
CAM support:		
■ Load contours from Step data and Iges data	■ X, option 42	■ –
■ Load machining positions from Step data and Iges data	■ X, option 42	■ –
■ Offline filter for CAM files	■ –	■ X
■ Stretch filter	■ X	■ –
MOD functions:		
■ User parameters	■ Config data	■ Numerical structure
■ OEM help files with service functions	■ –	■ X
■ Data medium inspection	■ –	■ X
■ Load service packs	■ –	■ X
■ Specify the axes for actual position capture	■ –	■ X
■ Configure counter	■ X	■ –

Function	TNC 320	iTNC 530
Special functions:		
■ Create reverse program	■ –	■ X
■ Adaptive Feed Control AFC	■ –	■ X, option 45
■ Define the counter with FUNCTION COUNT	■ X	■ –
■ Define the dwell time with FUNCTION FEED	■ X	■ –
Functions for large molds and dies:		
■ Global program settings (GS)	■ –	■ X, option 44
■ Expanded M128: FUNCTION TCPM	■ –	■ X
Status displays:		
■ Dynamic display of Q-parameter contents, definable number ranges	■ X	■ –
■ Graphic display of residual run time	■ –	■ X
Individual color settings of user interface	–	X

Comparison: Touch probe cycles in the Manual operation and Electronic handwheel operating modes

Cycle	TNC 320	iTNC 530
Touch-probe table for managing 3-D touch probes	X	–
Calibrating the effective length	X	X
Calibrating the effective radius	X	X
Measuring a basic rotation using a line	X	X
Setting the preset on any axis	X	X
Setting a corner as preset	X	X
Setting a circle center as preset	X	X
Setting a center line as preset	X	X
Measuring a basic rotation using two holes/cylindrical studs	X	X
Setting the preset using four holes/cylindrical studs	X	X
Setting the circle center using three holes/cylindrical studs	X	X
Determine and offset misalignment of a plane	X	–
Support of mechanical touch probes by manually capturing the current position	By soft key or hard key	By hard key
Write measurement values to the preset table	X	X
Write measurement values to the datum table	X	X

Comparison: Differences in programming

Function	TNC 320	iTNC 530
File management:		
■ Entry of name	■ Opens Select file pop-up window	■ Synchronizes the cursor
■ Support of key combinations	■ Not available	■ Available
■ Favorites Management	■ Not available	■ Available
■ Configuration of column structure	■ Not available	■ Available
Selecting a tool from the table	Selection via split-screen menu	Selection in a pop-up window
Programming special functions with the SPEC FCT key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the SPEC FCT key again; then the control shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the SPEC FCT key again; then the control shows the last active soft-key row
Programming approach and departure motions with the APPR DEP key	Pressing the key opens a soft-key row as a submenu. To exit the submenu, press the APPR DEP key again; then the control shows the last active soft-key row	Pressing the key adds the soft-key row as the last row. To exit the menu, press the APPR DEP key again; then the control shows the last active soft-key row
Pressing the hard key END with active CYCLE DEF and TOUCH PROBE menus	Terminates the editing process and calls the file manager	Exits the respective menu
Calling the file manager while the CYCLE DEF and TOUCH PROBE menus are active	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	Key non-functional error message
Calling the file manager while CYCL CALL , SPEC FCT , PGM CALL and APPR/DEP menus are active	Terminates the editing process and calls the file manager. The respective soft-key row remains selected when the file manager is exited	Terminates the editing process and calls the file manager. The basic soft-key row is selected when the file manager is exited

Function	TNC 320	iTNC 530
Datum table:		
■ Sorting function by values within an axis	■ Available	■ Not available
■ Resetting the table	■ Available	■ Not available
■ Switching the list/form view	■ Switch via the screen layout key	■ Switchover by toggle soft key
■ Inserting individual line	■ Allowed everywhere, renumbering possible after request. Empty line is inserted, must be filled with zeros manually	■ Only allowed at the end of the table. Line with value 0 in all columns is inserted
■ Transfer of actual position values on individual axis to the datum table using the keys	■ Not available	■ Available
■ Transfer of actual position values on all active axes to the datum table using the keys	■ Not available	■ Available
■ Capturing the last positions measured by TS using the keys	■ Not available	■ Available
FK free contour programming:		
■ Programming of parallel axes	■ With X/Y coordinates, independent of machine type; switchover with FUNCTION PARAXMODE	■ Machine-dependent with the existing parallel axes
■ Automatic correction of relative references	■ Relative references in contour subprograms are not corrected automatically	■ All relative references are corrected automatically
■ Specify the working plane during programming	■ BLK form ■ Plane XY ZX YZ soft key if the working plane differs	■ BLK form
Q-parameter programming:		
■ Q-parameter formula with SGN	Q12 = SGN Q50 ■ if Q 50 = 0 then Q12 = 0 ■ if Q50 > 0 then Q12 = 1 ■ if Q50 < 0 then Q12 = -1	Q12 = SGN Q50 ■ if Q50 >= 0 then Q12 = 1 ■ if Q50 < 0 then Q12 = -1

Function	TNC 320	iTNC 530
Handling of error messages:		
■ Help with error messages	■ Call via ERR key	■ Call via HELP key
■ Switching the operating mode while help menu is active	■ Help menu is closed when the operating mode is switched	■ Operating mode switchover is not allowed (key is non-functional)
■ Selecting the background operating mode while help menu is active	■ Help menu is closed when F12 is used for switching	■ Help menu remains open when F12 is used for switching
■ Identical error messages	■ Are collected in a list	■ Are displayed only once
■ Acknowledgment of error messages	■ Every error message (even if it is displayed more than once) must be acknowledged, the DELETE ALL function is available	■ Error message to be acknowledged only once
■ Access to protocol functions	■ Log and powerful filter functions (errors, keystrokes) are available	■ Complete log without filter functions available
■ Saving service files	■ Available. No service file is created when the system crashes	■ Available. A service file is automatically created when the system crashes
Find function:		
■ List of words recently searched for	■ Not available	■ Available
■ Show elements of active block	■ Not available	■ Available
■ Show list of all available NC blocks	■ Not available	■ Available
Starting the search function with the up/down arrow keys when highlighted	Works up to max. 50000 NC blocks, can be set via configuration datum	No limitation regarding program length
Programming graphics:		
■ True-to-scale display of grid	■ Available	■ Not available
■ Editing contour subprograms in SLII cycles with AUTO DRAW ON	■ With error messages, in the main program the cursor is positioned on the CYCL CALL NC block	■ With error messages, the cursor is positioned on the NC block in the contour subprogram that caused the error
■ Moving the zoom window	■ Repeat function not available	■ Repeat function available

Function	TNC 320	iTNC 530
Programming minor axes:		
■ Syntax FUNCTION PARAXCOMP : Define the behavior of the display and the paths of traverse	■ Available	■ Not available
■ Syntax FUNCTION PARAXMODE : Define the assignment of the parallel axes to be traversed	■ Available	■ Not available
Programming OEM cycles		
■ Access to table data	■ Via SQL commands and via FN 17/FN 18 or TABREAD-TABWRITE functions	■ Via FN 17/FN 18 or TABREAD-TABWRITE functions
■ Access to machine parameters	■ With the CFGREAD function	■ Via FN 18 functions
■ Creating interactive cycles with CYCLE QUERY , e.g. touch probe cycles in Manual Operation	■ Available	■ Not available

Comparison: Differences in Test Run, functionality

Function	TNC 320	iTNC 530
Entering a program with the GOTO key	Function only possible if the START SINGLE soft key was not pressed	Function also possible after START SINGLE
Calculation of machining time	Each time the simulation is repeated by pressing the START soft key, the machining time is totaled	Each time the simulation is repeated by pressing the START soft key, time calculation starts at 0
Single block	With point pattern cycles and CYCL CALL PAT , the control stops after each point	Point pattern cycles and CYCL CALL PAT are handled by the control as a single NC block

Comparison: Differences in Test Run, operation

Function	TNC 320	iTNC 530
Zoom function	Each sectional plane can be selected by individual soft keys	Sectional plane can be selected via three toggle soft keys
Machine-specific miscellaneous functions M	Lead to error messages if they are not integrated in the PLC	Are ignored during Test Run
Displaying/editing the tool table	Function available via soft key	Function not available
Tool depiction	<ul style="list-style-type: none"> ■ Turquoise: Tool length ■ Red: Length of cutting edge and tool is engaged ■ Blue: Length of cutting edge and tool is not engaged 	<ul style="list-style-type: none"> ■ - ■ Red: Tool is engaged ■ Green: Tool is not engaged
View options of 3-D view	Available	Function not available
Adjustable model quality	Available	Function not available

Comparison: Differences in Manual Operation, functionality

Function	TNC 320	iTNC 530
Jog increment function	The jog increment can be defined separately for linear and rotary axes	The jog increment applies for both linear and rotary axes
Preset table	<p>Basic transformation (translation and rotation) of machine table system to workpiece system via the X, Y and Z columns, as well as spatial angles SPA, SPB and SPC. In addition, the X_OFFSETS to W_OFFSETS columns can be used to define the axis offset of each individual axis. The function of the axis offsets can be configured. Line 0 can also be edited manually.</p>	<p>Basic transformation (translation) of machine table system to workpiece system via the columns X, Y and Z, as well as a ROT basic rotation in the working plane (rotation). In addition, columns A to W can be used to define presets on the rotary and parallel axes. Line 0 can only be edited by manual probing cycles.</p>
Behavior when presetting	<p>Presetting in a rotary axis has the same effect as an axis offset. This offset is also effective for kinematics calculations and during tilting of the working plane.</p> <p>In machine parameter preset-ToAlignAxis (no. 300203) your machine tool builder specifies for each axis what effect an offset of a rotational axis has on the preset.</p> <ul style="list-style-type: none"> ■ True (default): The offset is subtracted from the axis value before the calculation of the kinematics ■ False: The offset only affects the position display 	<p>Rotary axis offsets defined by machine parameters do not influence the axis positions that were defined in the Tilt working plane function.</p> <p>MP7500, bit 3 permits you to define whether the current rotary axis position relative to the machine datum is taken into account or whether the first rotary axis (usually the C axis) is assumed to be in 0° position.</p>
Presetting	Only after a reference run is it possible to set a preset or to modify a preset via the preset table.	A preset can be set or modified via the preset table before a reference run.
Handling of the preset table:		
Defining the feed rates	<p>Feed rates can be defined separately for linear and rotary axes</p> <p>Press the F soft key in the Manual operation mode to define a separate feed rate for linear and rotary axes. These feed rates apply only in the Manual operation mode.</p>	Only one feed-rate can be defined for linear and rotary axes

Comparison: Differences in Manual Operation, operation

Function	TNC 320	iTNC 530
Capturing the position values from mechanical probes	Confirm actual position with a soft key or hard key	Actual-position capture by hard key

Comparison: Differences in Program Run, operation

Function	TNC 320	iTNC 530
Operating mode switchover after program run has been suspended by switching to the Program run, single block operating mode and canceled with INTERNAL STOP	When you return to the Program run, full sequence mode: error message Current block not selected . Use block scan to select the point of interruption	Switching the operating mode is allowed, modal information is saved, program run can be continued by pressing NC start
GOTO is used to go to FK sequences after program run was interrupted there before switching the operating mode	Error message FK programming: Undefined starting position Entering with mid-program startup is permitted	GOTO allowed
Mid-program startup:		
Switching the screen layout for mid-program startup	Only possible, if startup position has already been approached	Possible in all operating states
Error messages	Error messages are still active after the error has been corrected and must be acknowledged separately	Error messages are sometimes acknowledged automatically after the error has been corrected
Point patterns in single block	With point pattern cycles and CYCL CALL PAT , the control stops after each point.	Point pattern cycles and CYCL CALL PAT are handled by the control as a single NC block

Comparison: Differences in Program Run, traverse movements

NOTICE

Danger of collision!

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

- ▶ Check the NC program or program section using the graphic simulation
- ▶ Carefully test the NC program or program section in the **Program run, single block** operating mode
- ▶ Pay attention to the following known differences (the list below might not be complete!)

Function	TNC 320	iTNC 530
Handwheel-superimposed traversing with M118	Effective in the machine coordinate system	Effective in the machine coordinate system
Deleting basic rotation with M143	M143 clears the entries from the SPA , SPB and SPC columns in the preset table	M143 does not delete the entry in the ROT column in the preset table; but only in the NC program; reactivating the corresponding preset table line activates the deleted basic rotation
Scaling approach/departure movements (APPR/DEP/RND)	Axis-specific scaling factor is allowed, radius is not scaled	Error message
Approach/departure with APPR/DEP	Error message if R0 is programmed for APPR/DEP LN or APPR/DEP CT	Tool radius 0 and compensation direction RR are assumed
Approach/departure with APPR/DEP if contour elements with length 0 are defined	Contour elements with length 0 are ignored. The approach/departure movements are calculated for the first and last valid contour element	An error message is issued if a contour element with length 0 is programmed after the APPR block (relative to the first contour point programmed in the APPR block) For a contour element with length 0 before a DEP block, the iTNC 530 does not issue an error message, but uses the last valid contour element to calculate the departure movement

Function	TNC 320	iTNC 530
Effect of Q parameters	Q60 to Q99 (QS60 to QS99) are always local.	Q60 to Q99 (QS60 to QS99) are local or global, depending on MP7251 in converted cycle programs (.cyc). Nested calls may cause problems
Automatic cancelation of tool radius compensation	<ul style="list-style-type: none"> ■ NC block with R0 ■ DEP block ■ Program selection ■ END PGM 	<ul style="list-style-type: none"> ■ NC block with R0 ■ DEP block ■ Program selection ■ Programming of Cycle 10 ROTATION ■ PGM CALL
NC blocks with M91	No consideration of tool radius compensation	Consideration of tool radius compensation
Behavior with M120 LA1	No effect on processing, as the control interprets the input internally as an LA0	Possible undesired effect on processing, as the control interprets the entry internally as an LA2
Block scan in a point table	The tool is positioned above the next position to be machined	The tool is positioned above the last position that has been completely machined
Empty CC block (pole adoption from last tool position) in the NC program	Last positioning block in the working plane must contain both coordinates of the working plane	Last positioning block in the working plane does not necessarily need to contain both coordinates of the working plane. Can cause problems with RND or CHF blocks
Axis-specific scaling of RND block	RND block is scaled, the result is an ellipse	Error message is issued
Reaction if a contour element with length 0 is defined before or after a RND or CHF block	Error message is issued	<p>Error message is issued if a contour element with length 0 is located before the RND or CHF block</p> <p>Contour element with length 0 is ignored if the contour element with length 0 is located after the RND or CHF block</p>

Function	TNC 320	iTNC 530
Circle programming with polar coordinates	The incremental rotation angle IPA and the direction of rotation DR must have the same sign. Otherwise, an error message will be issued	The algebraic sign of the direction of rotation is used if the sign defined for DR differs from the one defined for IPA
Tool radius compensation on circular arc or helix with angular length = 0	The transition between the adjacent elements of the arc/helix is generated. Also, the tool axis motion is executed right before this transition. If the element is the first or last element to be corrected, the next or previous element is dealt with in the same way as the first or last element to be corrected	The equidistant line of the arc/helix is used for generating the tool path
SLII Cycles 20 to 24:		
<ul style="list-style-type: none"> ■ Number of definable contour elements 	<ul style="list-style-type: none"> ■ Max. 16384 blocks in up to 12 subcontours 	<ul style="list-style-type: none"> ■ Max. 8192 contour elements in up to 12 subcontours, no restrictions for subcontour
<ul style="list-style-type: none"> ■ Define the working plane 	<ul style="list-style-type: none"> ■ Tool axis in TOOL CALL block defines the working plane 	<ul style="list-style-type: none"> ■ The axes of the first positioning block in the first subcontour define the working plane
<ul style="list-style-type: none"> ■ Position at end of SL cycle 	<ul style="list-style-type: none"> ■ With the posAfterContPocket (no. 201007) parameter, you can define whether the end position is above the last programmed position, or whether the tool moves to clearance height in the tool axis ■ If the tool moves to clearance height in the tool axis, both coordinates must be programmed with the first traverse movement 	<ul style="list-style-type: none"> ■ With MP7420, you can define whether the end position is above the last programmed position, or whether the tool moves only to clearance height in the tool axis ■ If the tool moves to clearance height in the tool axis, one coordinate must be programmed with the first traverse movement

Function	TNC 320	iTNC 530
SLII Cycles 20 to 24:		
<ul style="list-style-type: none"> ■ Behavior with islands not contained in pockets ■ Set operations for SL cycles with complex contour formulas ■ Radius compensation is active during CYCL CALL ■ Paraxial positioning blocks in contour subprogram ■ Miscellaneous functions M in contour subprogram 	<ul style="list-style-type: none"> ■ Cannot be defined with complex contour formula ■ Real set operation possible ■ Error message is issued ■ Error message is issued ■ Error message is issued 	<ul style="list-style-type: none"> ■ Restricted definition in complex contour formula is possible ■ Only restricted performance of real set operation possible ■ Radius compensation is canceled, NC program is being run ■ NC program is being run ■ M functions are ignored
Cylinder surface machining in general:		
<ul style="list-style-type: none"> ■ Contour definition ■ Offset definition on cylinder surface ■ Offset definition for basic rotation ■ Circle programming with C/CC ■ APPR/DEP blocks in contour definition 	<ul style="list-style-type: none"> ■ With X/Y coordinates, independent of machine type ■ With datum shift in X/Y, regardless of machine type ■ Function available ■ Function available ■ Function not available 	<ul style="list-style-type: none"> ■ Machine-dependent, with existing rotary axes ■ Machine-specific datum shift in rotary axes ■ Function not available ■ Function not available ■ Function available
Cylinder surface machining with Cycle 28:		
Complete roughing-out of slot	Function available	Function not available
Cylinder surface machining with Cycle 29		
	Direct plunging to contour of ridge	Circular approach to contour of ridge
Cycles 25x for pockets, studs and slots:		
<ul style="list-style-type: none"> ■ Plunging movements 	In limit ranges (geometrical conditions of tool/contour) error messages are triggered if plunging movements lead to unreasonable/critical behavior	In limit ranges (geometrical conditions of tool/contour), vertical plunging is used if required

Function	TNC 320	iTNC 530
PLANE function:		
<ul style="list-style-type: none"> ■ TABLE ROT/COORD ROT 	<p>Effect:</p> <ul style="list-style-type: none"> ■ The transformation types are effective on all free rotary axes ■ The control does not always position the free rotary axis with TABLE ROT, but depending on the current position, the programmed spatial angle and the machine kinematics <p>Default with missing selection:</p> <ul style="list-style-type: none"> ■ COORD ROT is used 	<p>Effect</p> <ul style="list-style-type: none"> ■ The transformation types are only effective with a C rotary axis ■ With TABLE ROT the control always positions the rotary axis <p>Default with missing selection:</p> <ul style="list-style-type: none"> ■ COORD ROT is used
<ul style="list-style-type: none"> ■ Positioning behavior 	<ul style="list-style-type: none"> ■ SYM ■ SEQ 	<ul style="list-style-type: none"> ■ SEQ
<ul style="list-style-type: none"> ■ Machine is configured for axis angle 	<ul style="list-style-type: none"> ■ All PLANE functions can be used 	<ul style="list-style-type: none"> ■ Only PLANE AXIAL is executed
<ul style="list-style-type: none"> ■ Programming an incremental spatial angle according to PLANE AXIAL 	<ul style="list-style-type: none"> ■ Error message is issued 	<ul style="list-style-type: none"> ■ Incremental spatial angle is interpreted as an absolute value
<ul style="list-style-type: none"> ■ Programming an incremental axis angle according to PLANE SPATIAL if the machine is configured for spatial angles 	<ul style="list-style-type: none"> ■ Error message is issued 	<ul style="list-style-type: none"> ■ Incremental axis angle is interpreted as an absolute value
<ul style="list-style-type: none"> ■ Programming of PLANE functions with active Cycle 8 MIRROR IMAGE 	<ul style="list-style-type: none"> ■ Mirroring has no influence on tilting using AXIAL PLANE and Cycle 19 	<ul style="list-style-type: none"> ■ Function is available with all PLANE functions
<ul style="list-style-type: none"> ■ Axis positioning on machines with two rotary axes e.g. L A+0 B+0 C+0 or L A+Q120 B+Q121 C+Q122 	<ul style="list-style-type: none"> ■ Only possible after a tilting function (error message if without a tilting function) ■ Parameters that are not defined are given the status UNDEFINED; they are not given the value 0 	<ul style="list-style-type: none"> ■ Possible at any time if spatial angles are used (machine parameter setting) ■ The control assigns the value 0 to parameters that are not defined
Special functions for cycle programming:		
<ul style="list-style-type: none"> ■ FN 17 	<ul style="list-style-type: none"> ■ Values are always output in metric form 	<ul style="list-style-type: none"> ■ Values are output in the units of the active NC program
<ul style="list-style-type: none"> ■ FN 18 	<ul style="list-style-type: none"> ■ Values are always output in metric form 	<ul style="list-style-type: none"> ■ Values are output in the unit of the active NC program
Compensation of tool length in the position display	The tool length entries L and DL from the tool table are taken into account in the position display, from TOOL CALL block depending on the machine parameter progToolCallIDL (no. 124501)	The tool length entries L and DL from the tool table are taken into account in the position display

Comparison: Differences in MDI operation

Function	TNC 320	iTNC 530
Miscellaneous functions	<ul style="list-style-type: none"> ■ Status display for Q parameters ■ Block functions, e.g. COPY BLOCK ■ Miscellaneous program functions, e.g. FUNCTION DWELL 	
Skip NC blocks	Separate soft key for MDI mode	The soft key from the Program Run Full Sequence operating mode is effective

Comparison: Differences in programming station

Function	TNC 320	iTNC 530
Demo version	NC programs with more than 100 NC blocks cannot be selected; an error message is issued	NC programs can be selected, max. 100 NC blocks are displayed, further NC blocks are truncated in the display
Demo version	If nesting with PGM CALL results in more than 100 NC blocks, there is no test graphic display; an error message is not issued	Nested NC programs can be simulated
Demo version	You can transfer up to 10 elements from the CAD viewer to an NC program.	You can transfer up to 31 lines from the DXF converter to an NC program.
Copying NC programs	Copying to and from the directory TNC:\ is possible with Windows Explorer	TNCremo or file manager of programming station must be used for copying
Shifting the horizontal soft-key row	Clicking on the soft-key bar shifts one soft-key row to the right or left	Clicking any soft-key bar activates the respective soft-key row

Index

3

3-D basic rotation.....	198
3-D touch probe	
Calibrating.....	188
using.....	179

A

About this manual.....	22
Accessories.....	109
Adding comments.....	238
Automatic program start.....	260
Automatic tool measurement...	119
Axis position, checking.....	149

B

Backup.....	326
Basic rotation.....	195
measuring in Manual operation	
mode.....	195
Behavior after receipt of ETX....	335
Block check character.....	334
Block scan.....	253
after a power failure.....	253
In a point table.....	257
Browser.....	80

C

Calling tool management.....	133
Code number entry.....	282
Comparison of functions.....	403
Compensating workpiece	
misalignment	
By measuring two points on a	
straight surface.....	194
Condition of RTS line.....	334
Configuration data.....	378
Context-sensitive help.....	91
Control panel.....	52
Counter.....	276
Counter settings.....	289

D

Data Backup.....	76, 326
Data interface.....	332
pin layout.....	393
setting up.....	332
Data transfer	
File system.....	334
Software.....	336
Data transmission	
Behavior after receipt of	
ETX.....	335
Block check character.....	334
Condition of RTS line.....	334
Data bits.....	333
Handshake.....	334

Parity.....	333
Protocol.....	333
Software TNC server.....	335
Stop bits.....	333

Datum table	
transferring probed values....	186

Directory.....	66
Displaying an HTML file.....	80
Displaying an Internet file.....	80
Display of the NC program.....	238
Display screen.....	51
DNC.....	330
Downloading help files.....	96

E

EnDat encoder.....	149
Error message.....	86
help with.....	86
Ethernet interface.....	338
configuration.....	338
configuring.....	345
connecting and disconnecting a	
network drive.....	74
connection possibility.....	338
introduction.....	338
External access.....	292
External data transfer.....	73

F

FCL.....	282
FCL function.....	26
Feature Content Level.....	26
Feed rate.....	165
changing.....	166
File	
importing.....	76
protecting.....	68
File management.....	64
external data transfer.....	73
External file types.....	66
File manager	
Calling.....	67
Directory.....	66
File type.....	64
Selecting files.....	69
File status.....	67
Firewall.....	329
FUNCTION COUNT.....	276
Fundamentals.....	97

G

GOTO.....	237
Graphics.....	218
view options.....	219
Graphic settings.....	288
Graphic simulation.....	225
tool.....	220

H

Handwheel.....	155
Hard disk.....	64
Help system.....	91
Help with error message.....	86

I

Import	
file from iTNC 530.....	76
table from iTNC 530.....	122
Indexed tool.....	114
Interrupt machining.....	244
iTNC 530.....	48

J

Jumping	
with GOTO.....	237

K

Kinematics.....	290
-----------------	-----

L

Loading the machine configuration.	
284	

M

M91, M92.....	268
Machine parameters.....	378
changing.....	378
changing the display.....	380
list.....	381
Machine settings.....	290
Manual datum setting	
circle center as preset.....	204
Manual datum setting.....	201
Manual presetting	
Corner as datum.....	203
On any axis.....	202
Setting a center line as	
preset.....	207
without a 3-D touch probe...	176
MDI.....	261
Measurement of machining	
time.....	226
Measuring workpieces.....	208
Miscellaneous functions.....	265
enter.....	265
For path behavior.....	271
For program run inspection.	267
For spindle and coolant.....	267
Miscellaneous functions for	
coordinate entries.....	268
Modes of Operation.....	54
MOD function.....	280
exiting.....	280
overview.....	281
selecting.....	280
Move machine axes	

Jog positioning.....	154
Moving the machine axes.....	153
With axis direction keys.....	153
with the handwheel.....	155

N

NC error message.....	86
NC program structuring.....	241
Network connection.....	74
Network settings	
control-specific.....	345
general.....	338

O

Opening a BMP file.....	84
Opening a GIF file.....	84
Opening a graphics file.....	84
Opening a JPG file.....	84
Opening an INI file.....	83
Opening a PNG file.....	84
Opening a TXT file.....	83
Opening a video file.....	84
Opening Excel files.....	79
Operating times.....	298

P

Path.....	66
PDF viewer.....	78
Pin layout	
data interface.....	393
Pocket table.....	125
Positioning.....	261
with manual data input.....	261
With tilted working plane.....	270
Preset	
managing.....	168
Preset table.....	168, 168
transferring probed values....	187
Probing	
with a 3-D touch probe.....	179
With end mill.....	177
Probing a plane.....	198
Program	
structuring.....	241
Program run.....	239
block scan.....	253
Interrupt.....	244
measuring.....	229
Overview.....	239
Resuming after interruption.	249
Retraction.....	250
running.....	240
skipping NC blocks.....	231
Program Test	
Overview.....	233
Protection zone.....	291

Q

Q parameters	
checking.....	242

R

Radio handwheel.....	158
configuring.....	295
Reference system.....	99
Basic.....	102
Input.....	106
Machine.....	100
Tool.....	107
Working plane.....	105
Workpiece.....	103
Restore.....	326
Retraction.....	250
After a power interruption....	250
Returning to the contour.....	258
Rotating, zooming and moving a graphic.....	223

S

Save service files.....	90
Screen keypad.....	53, 53
Screen layout.....	52
Set BAUD rate.....	332
Set data transmission speed....	332
Shifting the sectional plane.....	225
Software number.....	282
Spindle speed	
changing.....	166
Status display.....	57
additional.....	59
General.....	57
Stop at.....	236
Structuring NC programs.....	241
Superimposing handwheel positioning M118.....	271
Switch-off.....	152
Switch-on.....	148
System settings.....	298

T

Taskbar.....	311
test run.....	259
executing up to a certain NC block.....	236
Execution.....	235
Setting speed.....	224
Text file	
opening.....	83
Tilting the working plane.....	211
Manual.....	211
TNCguide.....	91
TNCremo.....	336
Tool carrier management.....	142
Tool change.....	128
Tool data.....	112

entering into the table.....	117
Export.....	139
Import.....	139
indexing.....	121
Tool length.....	112
Tool management.....	132
Editing.....	134
Tool types.....	138
Tool measurement.....	119
Tool name.....	112
Tool number.....	112
Tool radius.....	112
Tool table.....	113
Edit, exit.....	120
Editing functions.....	120
filter function.....	115
fundamentals.....	113
importing.....	122
input options.....	117
Tool usage file.....	129, 292
Tool usage test.....	129
Touch probe cycles.....	180
manual.....	180
Manual operation mode.....	180
Traverse limits.....	291
Traversing reference marks.....	148

U

USB device	
Connecting.....	72
Removing.....	73
User administration.....	348
User parameters.....	378, 381
Using touch probe functions with mechanical probes or measuring dials.....	178

V

Version number.....	282
Version numbers.....	284
Virtual tool axis.....	272

W

Window Manager.....	310
Wireless handwheel	
Assign handwheel holder.....	295
Selecting transmitter power.	296
Setting channel.....	296
Statistical data.....	297
Working space monitoring.....	227
Workspace monitoring.....	235
Writing probing values	
log.....	185
to the datum table.....	186
to the preset table.....	187

Z

ZIP archive.....	82
------------------	----

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.

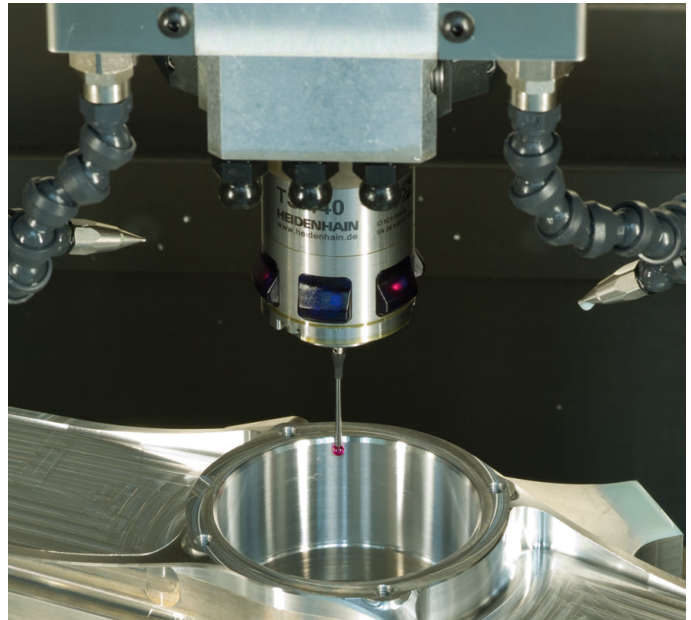
Workpiece touch probes

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

