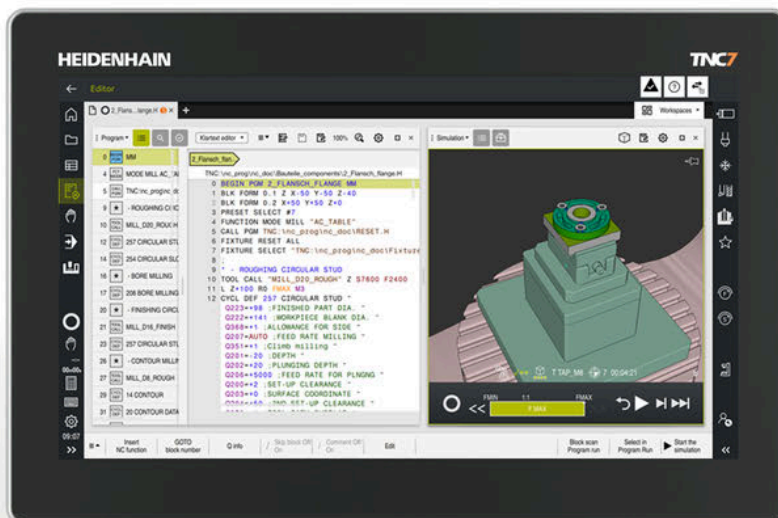




HEIDENHAIN



TNC7 basic

User's Manual
Setup and Program Run

NC Software
81762x-20

English (en)
10/2025

Table of contents

1	New and modified functions.....	33
2	About the User's Manual.....	47
3	About the product.....	61
4	First steps.....	105
5	Status displays.....	119
6	Powering on and off.....	151
7	Manual operation.....	159
8	NC fundamentals.....	165
9	Tools.....	169
10	Coordinate transformation.....	195
11	Collision monitoring.....	223
12	Control functions.....	255
13	CAD Viewer.....	267
14	User aids.....	293
15	The MDI application.....	301
16	Touch probe functions in the Manual operating mode (#17 / #1-05-1).....	305
17	Program run.....	339
18	Tables.....	369
19	Electronic handwheel.....	429
20	Override controller.....	447
21	Embedded Workspace and Extended Workspace.....	457
22	Integrated functional safety (FS).....	461
23	The Settings application.....	469
24	User administration.....	549
25	HEROS operating system.....	577
26	Overviews.....	601

1	New and modified functions.....	33
1.1	New functions.....	35
1.1.1	Process Tracking Interface (#3-04-1*).....	35
1.1.2	Dynamic Collision Monitoring DCM (#140 / #5-03-2).....	35
1.1.3	About the product.....	36
1.1.4	Status displays.....	36
1.1.5	Tools.....	37
1.1.6	CAD Viewer	37
1.1.7	Program Run.....	37
1.1.8	Tables.....	37
1.1.9	The Settings application.....	38
1.1.10	Machine parameters.....	38
1.1.11	HEROS operating system.....	39
1.2	Modified or extended functions.....	40
1.2.1	About the product.....	40
1.2.2	Status displays.....	40
1.2.3	Powering on and off.....	40
1.2.4	Tools.....	40
1.2.5	Collision monitoring (#140 / #5-03-2).....	41
1.2.6	User aids.....	41
1.2.7	Touch probe functions in the Manual operating mode (#17 / #1-05-1).....	42
1.2.8	Program run.....	42
1.2.9	Tables.....	42
1.2.10	Electronic handwheel.....	44
1.2.11	The Settings application.....	44
1.2.12	Machine parameters.....	45
1.2.13	User administration.....	45
1.2.14	HEROS operating system.....	45

2	About the User's Manual.....	47
2.1	Target group: Users.....	48
2.2	Available user documentation.....	49
2.3	Types of notes used.....	51
2.4	Notes on using NC programs.....	53
2.5	User's Manual as integrated product aid: TNCguide.....	54
2.5.1	Searching in TNCguide.....	57
2.5.2	Copying NC examples to clipboard.....	58
2.6	Contacting the editorial staff.....	59

3	About the product.....	61
3.1	The TNC7 basic.....	62
3.1.1	Proper and intended use.....	62
3.1.2	Data Architecture and Access Options.....	63
3.1.3	Intended place of operation.....	64
3.2	Safety precautions.....	65
3.3	Software.....	68
3.3.1	Software options.....	69
3.3.2	Information on licensing and use.....	75
3.4	Hardware.....	77
3.4.1	Touchscreen and keyboard unit.....	77
3.4.2	Hardware enhancements.....	81
3.5	Areas of the control's user interface.....	83
3.6	Overview of the operating modes.....	85
3.7	Workspaces.....	87
3.7.1	Operating elements within the workspaces.....	87
3.7.2	Icons within the workspaces.....	88
3.7.3	Overview of workspaces.....	88
3.8	Operating elements.....	91
3.8.1	Common gestures for the touchscreen.....	91
3.8.2	Operating elements of the keyboard unit.....	91
3.8.3	Keyboard shortcuts for operating the control.....	98
3.8.4	Icons on the control's user interface.....	99
3.8.5	The Desktop menu workspace.....	102

4	First steps.....	105
4.1	Chapter overview.....	106
4.2	Switching the machine and the control on.....	107
4.3	Setting up a tool.....	109
4.3.1	Example tool.....	109
4.3.2	Preparing the tools.....	109
4.3.3	Opening the Tool management application and the Form workspace.....	110
4.3.4	Inserting a tool in the tool management.....	111
4.3.5	Defining the tool.....	111
4.3.6	Entering the tool in the pocket table.....	112
4.4	Setting up a workpiece.....	113
4.4.1	Selecting an operating mode.....	113
4.4.2	Clamping the workpiece.....	113
4.4.3	Workpiece presetting with a touch probe.....	113
4.5	Machining a workpiece.....	116
4.5.1	Selecting an operating mode.....	116
4.5.2	Opening an NC program.....	116
4.5.3	Starting an NC program.....	116
4.5.4	Interrupting an NC program.....	116
4.5.5	Aborting program run.....	117
4.5.6	Safe restart with block scan.....	117
4.6	Switching the machine off.....	118

5	Status displays.....	119
5.1	Overview.....	120
5.2	The Positions workspace.....	121
5.3	Status overview of the TNC bar.....	127
5.4	The Status workspace.....	129
5.5	The Simulation status workspace.....	145
5.6	Display of the program run time.....	146
5.7	Position displays.....	148
5.7.1	Switching the position display mode.....	149
5.8	Defining the contents of the QPARA tab.....	150

6	Powering on and off.....	151
6.1	Switch-on.....	152
6.1.1	Powering the machine and the control on.....	153
6.2	The Referencing workspace.....	155
6.2.1	Axis reference run.....	155
6.3	Powering off.....	157
6.3.1	Shutting down the control and powering-off the machine.....	157

7	Manual operation.....	159
7.1	The Manual operation application.....	160
7.2	Moving the machine axes.....	162
7.2.1	Using axis keys to move the axes.....	162
7.2.2	Incremental jog positioning of axes.....	163

8	NC fundamentals.....	165
8.1	NC fundamentals.....	166
8.1.1	Programmable axes.....	166
8.1.2	Designation of the axes of milling machines.....	166
8.1.3	Position encoders and reference marks.....	167
8.1.4	Presets in the machine.....	167

9	Tools.....	169
9.1	Fundamentals.....	170
9.2	Presets on the tool.....	171
9.2.1	Tool carrier reference point.....	171
9.2.2	Tool tip TIP	172
9.2.3	Tool center point (TCP, tool center point).....	172
9.2.4	Tool location point (TLP, tool location point).....	173
9.2.5	Tool rotation point (TRP, tool rotation point).....	173
9.2.6	Tool radius 2 center (CR2, center R2).....	174
9.3	Tool parameters.....	175
9.3.1	Tool ID number.....	176
9.3.2	Tool name.....	176
9.3.3	Database ID.....	177
9.3.4	Indexed tool.....	178
9.3.5	Tool types.....	182
9.4	Tool management.....	184
9.5	Tool carrier management.....	186
9.5.1	Assigning a tool carrier.....	187
9.6	Customizing tool carrier templates with ToolHolderWizard.....	188
9.6.1	Parameterizing tool carrier templates.....	189
9.7	Tool model (#140 / #5-03-2).....	190
9.7.1	Assigning a tool model.....	192
9.8	Tool usage test.....	193
9.8.1	Performing the tool usage test.....	194

10	Coordinate transformation.....	195
10.1	Reference systems.....	196
10.1.1	Overview.....	196
10.1.2	Basics of coordinate systems.....	197
10.1.3	Machine coordinate system M-CS	198
10.1.4	Basic coordinate system B-CS	200
10.1.5	Workpiece coordinate system W-CS	202
10.1.6	Working plane coordinate system WPL-CS	204
10.1.7	Input coordinate system I-CS	206
10.1.8	Tool coordinate system T-CS	207
10.2	Preset management.....	209
10.2.1	Setting a preset manually.....	212
10.2.2	Activating a preset manually.....	213
10.3	Tilting the working plane (#8 / #1-01-1).....	214
10.3.1	Fundamentals.....	214
10.3.2	The 3-D rotation window (#8 / #1-01-1).....	218

11 Collision monitoring.....	223
11.1 Dynamic Collision Monitoring DCM (#140 / #5-03-2).....	224
11.2 Fixture management.....	231
11.2.1 Fundamentals.....	231
11.2.2 Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2).....	234
11.2.3 Editing CFG files with KinematicsDesign	244
11.2.4 Combining fixtures in the New Fixture window.....	252

12 Control functions.....	255
12.1 Adaptive Feed Control (AFC) (#45 / #2-31-1).....	256
12.1.1 Fundamentals.....	256
12.1.2 Activating and deactivating AFC.....	259
12.1.3 AFC teach-in cut.....	262
12.1.4 Monitoring tool wear and tool load.....	264
12.2 Active Chatter Control (ACC) (#145 / #2-30-1).....	266

13 CAD Viewer.....	267
13.1 Fundamentals.....	268
13.2 Workpiece preset in the CAD file.....	275
13.2.1 Setting the workpiece preset or workpiece datum and orienting the coordinate system.....	276
13.3 Workpiece datum in the CAD file.....	278
13.4 Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1).....	280
13.4.1 Selecting and saving a contour.....	284
13.4.2 Select positions.....	285
13.5 Generating STL files with 3D mesh (#152 / #1-04-1).....	288
13.5.1 Positioning the 3D model for rear-face machining.....	291

14	User aids.....	293
14.1	Virtual keyboard of the control bar.....	294
14.1.1	Opening and closing the virtual keyboard.....	296
14.2	Message menu on the information bar.....	297
14.2.1	Creating a service file manually.....	299
14.2.2	Creating a service file automatically.....	300

15 The MDI application.....	301

16 Touch probe functions in the Manual operating mode (#17 / #1-05-1).....	305
16.1 Fundamentals.....	306
16.1.1 Setting a preset in a linear axis.....	315
16.1.2 Setting the circle center point of a stud by means of the automatic probing method.....	316
16.1.3 Determining and compensating for the basic rotation of a workpiece.....	318
16.1.4 Using touch probe functions with mechanical probes or dial gages.....	319
16.2 Calibrating the workpiece touch probe.....	321
16.2.1 Calibrating the length of the workpiece touch probe.....	323
16.2.2 Calibrating the radius of the workpiece touch probe.....	324
16.3 Setting up the workpiece with graphical support (#159 / #1-07-1).....	326
16.3.1 Setting up a workpiece.....	332
16.4 Measuring the tool by scratching.....	334
16.4.1 Tool measurement by scratching.....	335
16.5 Suppressing touch probe monitoring.....	336
16.5.1 Deactivating touch probe monitoring.....	336
16.6 Comparison of offset and 3D basic rotation.....	337

17 Program run.....	339
17.1 The Program Run operating mode.....	340
17.1.1 Fundamentals.....	340
17.1.2 Navigation path in the Program workspace.....	348
17.1.3 Manual traverse during an interruption.....	350
17.1.4 Block scan for mid-program startup.....	352
17.1.5 Returning to the contour.....	360
17.1.6 Automatic program start.....	361
17.2 Compensation during program run.....	364
17.2.1 Opening tables from within the Program Run operating mode.....	364
17.3 The Retract application.....	366

18 Tables.....	369
18.1 The Tables operating mode.....	370
18.1.1 Editing the contents of tables.....	372
18.2 The Create new table window.....	374
18.3 The Table workspace.....	376
18.4 The Form workspace for tables.....	384
18.4.1 Adding a column in the workspace.....	386
18.5 Importing and exporting of table contents.....	387
18.5.1 Exporting table contents.....	389
18.5.2 Importing table contents.....	390
18.6 Tool tables.....	392
18.6.1 Overview.....	392
18.6.2 Tool table tool.t	392
18.6.3 Touch probe table tchprobe.tp (#17 / #1-05-1).....	401
18.6.4 Creating a tool table in inches.....	405
18.7 Pocket table tool_p.tch.....	406
18.8 Tool usage file.....	409
18.9 T usage order (#93 / #2-03-1).....	411
18.10 Tooling list (#93 / #2-03-1).....	413
18.11 Preset table *.pr.....	415
18.11.1 Activating write protection.....	419
18.11.2 Removing write protection.....	420
18.11.3 Creating a preset table in inches.....	421
18.12 Tables for AFC (#45 / #2-31-1).....	423
18.12.1 Basic AFC settings in AFC.tab	423
18.12.2 AFC.DEP settings file for teach-in cuts.....	425
18.12.3 Log file AFC2.DEP	427
18.12.4 Editing the tables for AFC.....	428

19 Electronic handwheel.....	429
19.1 Fundamentals.....	430
19.2 Handwheel without display.....	432
19.2.1 Activating and deactivating a handwheel.....	433
19.2.2 Creating an NC block with the current position.....	434
19.2.3 Incremental jog positioning using handwheels without display.....	434
19.3 Handwheel with display.....	435
19.3.1 Activating and deactivating a handwheel with display.....	440
19.3.2 Creating an NC block with the current position.....	441
19.3.3 Defining the spindle speed S	441
19.3.4 Defining the handwheel feed rate F	442
19.3.5 Incremental jog positioning.....	442
19.4 Setting up a wireless handwheel.....	444
19.4.1 Setting up a new wireless handwheel.....	445

20 Override controller..... 447

21	Embedded Workspace and Extended Workspace.....	457
21.1	Embedded Workspace (#133 / #3-01-1).....	458
21.2	Extended Workspace.....	460

22 Integrated functional safety (FS).....	461
22.1 Checking axis positions manually.....	467

23 The Settings application.....	469
23.1 Overview.....	470
23.2 Code numbers.....	474
23.3 The Machine Settings menu item.....	475
23.4 The General Information menu item.....	478
23.5 The SIK menu item.....	480
23.5.1 Viewing of software options.....	481
23.5.2 Enabling software options.....	482
23.6 The Machine Times menu item.....	483
23.7 Overview of touch probes menu item.....	484
23.7.1 Connecting a new touch probe with radio transmission.....	486
23.7.2 Changing the radio channel.....	486
23.8 The Adjustment of analog voltage offset menu item.....	487
23.9 The Adjust system time window.....	488
23.10 Conversational language of the control.....	489
23.10.1 Changing the language.....	490
23.11 SELinux security software.....	491
23.12 Network drives on the control.....	492
23.13 Ethernet interface.....	496
23.13.1 The Network settings window.....	497
23.14 PKI Admin.....	503
23.15 OPC UA NC Server (#56-61 / #3-02-1*).....	505
23.15.1 Fundamentals.....	505
23.15.2 The OPC UA (#56-61 / #3-02-1*) menu item.....	510
23.15.3 The OPC UA connection assistant function (#56-61 / #3-02-1*).....	511
23.15.4 The OPC UA license settings function (#56-61 / #3-02-1*).....	512
23.16 The DNC menu item.....	513
23.17 Printers.....	516
23.17.1 Creating a printer.....	518
23.17.2 Copying a printer.....	518

23.18 The VNC menu item.....	519
23.19 The Remote Desktop Manager window (#133 / #3-01-1).....	522
23.19.1 Configuring an external computer for Windows Terminal Service (RemoteFX)	526
23.19.2 Establishing and starting a connection.....	526
23.19.3 Exporting and importing connections.....	527
23.20 Firewall.....	528
23.21 Portscan.....	533
23.22 Backup and restore.....	534
23.22.1 Backing up data.....	535
23.22.2 Restoring data.....	536
23.22.3 Selecting or deselecting multiple files.....	536
23.23 TNCdiag.....	537
23.24 Update the documentation.....	538
23.24.1 Transferring TNCguide	539
23.25 Machine parameters.....	540
23.26 Configuring the control's user interface.....	547
23.26.1 Exporting and importing configurations.....	548

24 User administration.....	549
24.1 Fundamentals.....	550
24.1.1 Configuring user administration.....	554
24.1.2 Deactivating user administration.....	557
24.2 The User administration window.....	558
24.3 The Active user window.....	559
24.4 Saving user data.....	560
24.4.1 Overview.....	560
24.4.2 Local LDAP database	560
24.4.3 LDAP database on a remote computer.....	561
24.4.4 Connection to Windows domain	562
24.5 Autologin with user administration.....	567
24.6 Logging on with user administration.....	568
24.6.1 Logging on a user with password.....	569
24.6.2 Assigning a smartcard to a user.....	569
24.7 Window for requesting additional rights.....	571
24.8 SSH-secured DNC connection.....	572
24.8.1 Setting up SSH-secured DNC connections.....	574
24.8.2 Removing a secure connection.....	574

25 HEROS operating system.....	577
25.1 Fundamentals.....	578
25.2 HEROS menu.....	579
25.3 Serial data transfer.....	584
25.4 PC software for data transfer.....	586
25.5 File transfer with SFTP (SSH File Transfer Protocol).....	588
25.5.1 Setting up an SFTP connection with CreateConnections.....	589
25.6 Secure Remote Access.....	590
25.7 Data backup.....	592
25.7.1 Example: Transmission duration of different transmission types.....	593
25.8 Opening files with additional software.....	594
25.8.1 Opening tools.....	595
25.9 Network configuration with Advanced Network Configuration.....	596
25.9.1 The Editing network connection window.....	597

26	Overviews.....	601
26.1	Pin layout and cables for data interfaces.....	602
26.1.1	V.24/RS-232-C interface for HEIDENHAIN devices.....	602
26.1.2	Ethernet interface RJ45 socket.....	602
26.2	User administration roles and rights.....	603
26.2.1	List of roles.....	603
26.2.2	List of rights.....	606
26.3	Overview of signals for Process Tracking Interface (PTI) (#3-04-1*).....	608

1

**New and modified
functions**

Available documentation



TNC7 basic complete edition

The split editions of the User's Manual contain only the new and modified functions that are relevant to the corresponding User's Manual. The **complete edition** contains all new and modified functions of this software version that are relevant to the user.

ID: 1411730-xx

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

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



Overview of new and modified software functions

The additional documentation **Overview of New and Modified Software Functions** contains all new and modified functions of the current and previous software versions that are relevant to the user.

ID: 1443541-xx

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

TNCguide

1.1 New functions

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

Topic	Description
Software option PTI 8 channels (#3-04-1*)	This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with SIK2. You can order this software option up to two times, each time selecting up to eight channels for recording. Further information: "Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)", Page 608

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

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

1.1.3 About the product

Topic	Description
Workspaces	When you select the title of a workspace, the control will open a selection menu. This selection menu allows you to switch workspaces without changing their size or position. You can only switch between workspaces that can be closed. Further information: "Operating elements within the workspaces", Page 87
Customizing the user interface	The machine manufacturer can use the machine parameter state (no. 143601) to dim or hide the following user interface elements, for example: ■ Operating modes ■ Applications ■ Workspaces ■ Buttons Thus, it is possible to remove certain areas from the control's user interface for training purposes. Refer to your machine manual. If the machine is equipped with the required sensor technology, the machine manufacturer can set up a Resource Usage Monitor (RUM). In RUM, usage of various supplies can be displayed, such as electricity or compressed air. Refer to your machine manual.

1.1.4 Status displays

Topic	Description
Position display mode	The Actual distance to go (ACTDST) and Nominal distance to go (REFDST) modes have been added to the position display: ■ Actual distance to go (ACTDST) displays the distance-to-go of the programmed axes in the input coordinate system I-CS. ■ Nominal distance to go (REFDST) displays the distance-to-go of the physical axes in the machine coordinate system M-CS that are required for the programmed traverse. Further information: "Position displays", Page 148
The Status workspace	On the COMPMON tab of the Status workspace, the Enforced reactions of component monitoring (#155 / #5-02-1) area was added. The area contains a table with all reactions that have been triggered since the control was switched on. You can filter the contents of this table. Further information: "The COMPMON (#155 / #5-02-1) tab", Page 133

1.1.5 Tools

Topic	Description
Tool type	The Calibration mandrel (CAL_PIN) tool type was added. You can use the Calibration mandrel to calibrate the tool touch probe with Cycles 480 CALIBRATE TT (ISO: G480) and 484 CALIBRATE IR TT (ISO: G484). Further information: "Tool types", Page 182

1.1.6 CAD Viewer

Topic	Description
The Hover mode	The Hover icon was added to replace the Rotate mode. If Hover is active and you swipe across the CAD model during touch operation, the control will display selectable options in orange. The function is comparable to hovering with a mouse. As soon as you stop the movement, the control will mark the current selection option. If Hover is active, you can no longer rotate the CAD model. Further information: "Screen layout", Page 269
Loading machining positions (#42 / #1-03-1)	Using the H file with FMAX M99 at each position (for older NC controls) icon, you can determine whether the control adds a cycle call for each position in the NC program or not. Further information: "Applying positions", Page 282

1.1.7 Program Run

Topic	Description
The Manual traverse function	Using the optional machine parameter userControl (no. 144101), the machine manufacturer defines if the control will display the Handwheel toggle switch or not. Further information: "Manual traverse during an interruption", Page 350

1.1.8 Tables

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

1.1.9 The Settings application

Topic	Description
Code numbers	The Home operating mode icon shown in the TNC bar changes if the code number 123 is enabled. The icon additionally shows an unlocked padlock to illustrate that machine parameters can be accessed. Further information: "Code numbers", Page 474
The Machine Settings menu item	If the machine manufacturer has enabled the function, you can define the operating station active by default for movements during a program-run interruption. Further information: "The Machine and Simulation areas", Page 475
The Expanded logging menu item	The Expanded logging menu item has been added to replace the optional machine parameter extendedDiagnosis (no. 124204). If this function is active, the control will save graphics journal data. These data may be needed by the HEIDENHAIN Customer Service for diagnostic purposes. If you activate this function, you need to restart the control. Deactivate this function after the fault has been fixed to reduce the amount of memory used.

1.1.10 Machine parameters

Topic	Description
Open context menu	For all objects in the configuration editor, the control provides an icon that opens the context menu. Objects are shown by a folder icon.
The Highlight change button	The Highlight change button has been added to the function bar of the configuration editor. The control marks the selected machine parameter as changed, even though the value remains the same. The machine manufacturer can use this function for testing, for example. Further information: "Icons, buttons and shortcuts", Page 542
Read-only NC programs	Using the optional machine parameter editWhileProtected (no. 105416), you can define whether it is possible to edit a read-only file. By default, you can edit such a file, but only save the changes in a new file with Save as. If you set this machine parameter to FALSE, the control will dim any editing functions.
Tool management	With the optional machine parameter resetOnTypeChange (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to TRUE and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters. Further information: "Tool management ", Page 184
Expanded logging	The optional machine parameter extendedDiagnosis (no. 134204) has been removed. Instead, you can activate the saving of graphics journal data in the Expanded logging menu item of the Settings application. Further information: "Overview", Page 470

1.1.11 HEROS operating system

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

1.2 Modified or extended functions

1.2.1 About the product

Topic	Description
Workspaces	As an alternative to using the Move icon in the title bar, you can move workspaces by dragging the title area.
Scaling windows	If a window is displayed in the right half of the screen, the control displays the scaling icon in the left corner of the window instead of the right corner. Further information: "Icons on the control's user interface", Page 99
The Start workspace	The Start/Login workspace was renamed Start . Once startup has been completed, the control will close the Start workspace. Further information: "Powering on and off", Page 151

1.2.2 Status displays

Topic	Description
The Status workspace	On the Tool tab, the control shows the tool life in minutes instead of in hours and minutes. This format is also used in tool management. Further information: "The Tool tab", Page 143 On the Tool tab, the information about the tool axis and tool type has been integrated in the Tool information area. There is no longer a separate Tool type area.
The Simulation status workspace	The following contents were removed from the POS tab of the Simulation status workspace: ■ Active Feed in mm/min ■ Active Feed-rate override in % ■ Active Spindle override in % Further information: "The POS tab", Page 138

1.2.3 Powering on and off

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

1.2.4 Tools

Topic	Description
Tool management	If a tool has been saved to the pocket table, you cannot reset the corresponding row in tool management. The tool must be removed from the magazine first.

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

Topic	Description
Set up fixtures	For the selection of the 3D model, the Last files area has been added. The control displays the most recently selected 3D models of the file types *.stl and *.cfg. Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234

1.2.6 User aids

Topic	Description
Context menu	The context menu of the Document workspace was removed.
Cutting data calculator	The representation of the cutting data calculator was improved. The cutting data calculator now includes only those input fields that are relevant to the currently active NC function.
Message menu	The message menu was revised. Using the hand pointer, you can enlarge or reduce the size of both areas. Further information: "Message menu on the information bar", Page 297
FUNCTION REPORT	The FUNCTION REPORT NC function now includes the ARGS syntax element for texts in PO files. You can enter additional text that will be displayed in the message details.
The GOTO function in the Program workspace	In the GOTO jump instruction window, the GOTO 0 and Select last buttons were added. With GOTO 0, the control jumps to the beginning of the NC program and then closes the window. If the user clicks the Select last button in the Program Run operating mode, the control will copy the number of the last executed NC block to the Block number row.

1.2.7 Touch probe functions in the Manual operating mode (#17 / #1-05-1)

Topic	Description
The Change the preset window	The control displays a symbol in front of the row of the current preset, like in the preset table. If you apply the changes made in the Change the preset window while the active preset is locked, the control will open a window with a confirmation prompt. Further information: "The Change the preset window", Page 313
Applying the result	If you use the Compensate the active preset or Correct the pallet reference point function to write to a locked table row, the control will open a window with a confirmation prompt.
Set up the workpiece (#159 / #1-07-1)	For the selection of the 3D model, the Last files area has been added. The control displays the most recently selected 3D models of the file types *.stl and *.cfg. Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 326
Log file	The maximum file size of the automatic log file TCHPRMAN.html was increased to 512 kB. Further information: "Log file of touch-probe functions", Page 314

1.2.8 Program run

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

1.2.9 Tables

Topic	Description
The Table workspace	In the dialog bar of the Table workspace, the control displays the unit of the selected table cell. Further information: "The Table workspace", Page 376 If you insert the contents of the clipboard while editing is inactive, the control displays the Editing disabled. Enable? prompt. If you select Yes, the control inserts the copied contents. The sequence of columns defined in the Tables window applies to all filters for the table. If user administration is active, the control stores the column sequence for each user. After updating the software to version 20, you must redefine the column sequence if you changed it.

Topic	Description
The Form workspace	You can scale the content of the Form workspace to make it look larger or smaller. Further information: "Icons", Page 385 In the tool manager, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the Configure the layout icon, the control will also display the hidden form areas. Further information: "The Form workspace for tables", Page 384 For the following tool types, the CUT parameter was added for the number of teeth in the Additional geometry data area: ■ Thread mill (GF) ■ Thread miller with chamfer (GSF) ■ Thread mill with single thread (EP) ■ Thread mill w/ indexable insert (WSP) ■ Circular thread mill (ZBGF)
Tool management	When you open the selection window for the KINEMATIC parameter, you can directly enter characters to search for a tool carrier kinematic model.
Pocket table tool_p.tch	If you define a tool in the column T of the pocket table, you can enter the tool number directly. The control opens the Tool selection window and selects the defined tool. You can load the tool to the column T .
Freely definable tables	When you edit cells of the type PATHNAME , the control automatically opens the Open File selection window.
Column names	Column names are not case-sensitive. If the only difference in column names is their case, the control will not open the table, but display an error message.
The Incomplete table layout window	If you modify a table format in the Incomplete table layout window, the control applies the same logic as is used for the Update TAB / PGM button.
Exporting table contents	When you export the contents of the Tooling list table, the CSV file contains all contents of the tool, and not just the columns of the Tooling list. This allows you to import this CSV file to the tool management. Further information: "Importing and exporting of table contents", Page 387 The pocket table no longer permits the export of table contents. The table T usage order (#93 / #2-03-1) no longer permits the export of table contents.
Importing table contents	The Confirm import window has been revised: ■ The control opens the window for every import. ■ The window shows the entire content to be imported. ■ You can select separately for each row whether the control will append, overwrite, or ignore the content. ■ For example, if you append tool data, you can optionally enter a new tool number. Further information: "The Confirm import window", Page 388

Topic	Description
Machine manufacturer settings	The optional machine parameter choice (no. 105704) for machine manufacturers is now available for additional parameters and tables. The machine manufacturer can thus modify the appearance of toggle switches in the Form workspace.

1.2.10 Electronic handwheel

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

1.2.11 The Settings application

Topic	Description
Enabling software options with SIK2	To enable a software option for a control with SIK2 , you need a key file (from the machine manufacturer, for example). If a key file is present in the TNC:\SIK2 folder, the control will read that file automatically during the next startup. Further information: "Code numbers", Page 474
The General Information menu item	In the Version Information area, the control displays the Operating System subsection and perhaps also the OEM cycles subsection. In the Machine information area, the control displays the Hardware subsection with the machine's RAM. Further information: "The General Information menu item", Page 478
The Adjustment of analog voltage offset menu item	The table for analog axes was revised. Instead of the Offset in mV column, the table now contains the columns Current offset , New offset , and Deviation . Further information: "The Adjustment of analog voltage offset menu item", Page 487
Network drives	You can use the Single Sign On setting not only for private networks, but also for all networks. The control uses the respective user's logon data for the public network drive. Further information: "Network drives on the control", Page 492
OPC UA connection assistant (#56-61 / #3-02-1*)	The OPC UA connection assistant also exports self-generated server certificates. Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 511
The DNC menu item	If the DNC access permitted toggle switch is inactive, the TNCopt full access allowed toggle switch is dimmed instead of hidden. Further information: "The DNC menu item", Page 513
Configurations	With the Enable autosave toggle switch, you can define whether the control automatically saves changes to the current configuration. If you reset or save a configuration, the control displays a confirmation prompt. If you reset to a saved configuration, the control will display the date of saving. Further information: "Configuring the control's user interface", Page 547

1.2.12 Machine parameters

Topic	Description
Configuration editor	The search function takes all machine parameters into account, independent of the current cursor position. Further information: "Machine parameters", Page 540 In the Configuration window, the Do not show empty folders toggle switch was added. This toggle switch can be used to hide empty attributes or folders.

1.2.13 User administration

Topic	Description
The Active user window	The control displays the assigned roles in the Active user window. Further information: "The Active user window", Page 559

1.2.14 HEROS operating system

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

2

**About the
User's Manual**

2.1 Target group: Users

A user is anyone who uses the control to perform at least one of the following tasks:

- Operating the machine
 - Setting up tools
 - Setting up workpieces
 - Machining workpieces
 - Eliminating possible errors during program run
- Creating and testing NC programs
 - Creating NC programs at the control or externally using a CAM system
 - Using the Simulation mode to test the NC programs
 - Eliminating possible errors during program test

The depth of information in the User's Manual results in the following qualification requirements on the user:

- Basic technical understanding (e.g., spatial imagination and the ability to read technical drawings)
- Basic knowledge in the field of metal cutting (e.g., understanding the meaning of material-specific parameters)
- Safety instructions (e.g., understanding possible dangers and how to avoid them)
- Training on the machine (e.g., comprehending axis directions and the machine configuration)



HEIDENHAIN offers separate information products for other target groups:

- Leaflets and overview of the product portfolio for potential buyers
- Service Manual for service technicians
- Technical Manual for machine manufacturers

Additionally, HEIDENHAIN provides users and lateral entrants with a wide range of training opportunities in the field of NC programming.

HEIDENHAIN training portal

In line with the target group, this User's Manual only contains information on the operation and use of the control. The information products for other target groups contain information on further product life phases.

2.2 Available user documentation

User's Manual

HEIDENHAIN refers to this information product as a User's Manual, regardless of the output or transport medium. Well-known designations with the same meaning include operator's manual and operating instructions.

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

- As a PDF file containing a **complete edition** of the User's Manual with all contents
ID: 1411730-xx
https://content.heidenhain.de/doku/tnc_guide/html/en/
- As an HTML file with all contents for use as the integrated product aid **TNCguide**: directly on the control
https://content.heidenhain.de/doku/tnc_guide/html/en/
- As a printed version, sub-divided into the modules below:
 - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for executing NC programs.
ID: 1410286-xx
 - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe cycles and machining cycles are not included.
ID: 1409856-xx
 - The **Machining Cycles** User's Manual contains all machining cycles functions.
ID: 1410289-xx
 - The **Measuring Cycles for Workpieces and Tools** User's Manual contains all functions related to the touch-probe cycles.
ID: 1410290-xx

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

Further information: "Proper and intended use", Page 62



Complete edition and TNCguide for the TNC7 basic

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

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

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

Further information products for users

The following information products are available:

- The **Overview of New and Modified Software Functions** informs you about the innovations of specific software versions.

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

- **Overview of the machine parameters, error numbers and system data**, providing the following functions:
 - Machine parameters of the **MPs for setters** application
 - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
 - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions

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

- **HEIDENHAIN brochures** inform you about products and services from HEIDENHAIN (e.g., software options of the control).

HEIDENHAIN brochures

- The **NC Solutions** database offers solutions for frequently occurring tasks.
HEIDENHAIN NC solutions

2.3 Types of notes used

Safety precautions

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

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

⚠ DANGER
Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard will result in death or severe injury .
⚠ WARNING
Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard could result in death or serious injury .
⚠ CAUTION
Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard could result in minor or moderate injury .
NOTICE
Caution indicates danger to material or data. If you do not follow the avoidance instructions, the hazard could result in things other than personal injury, such as property damage .

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

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



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



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



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

2.4 Notes on using NC programs

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

Change the following contents as needed:

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

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

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



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

2.5 User's Manual as integrated product aid: TNCguide

Application

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

Further information: "Available user documentation", Page 49

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

Further information: "Proper and intended use", Page 62

Related topics

- The **Help** workspace

Further information: Programming and Testing User's Manual

Requirement

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

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

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



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

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

Description of function

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

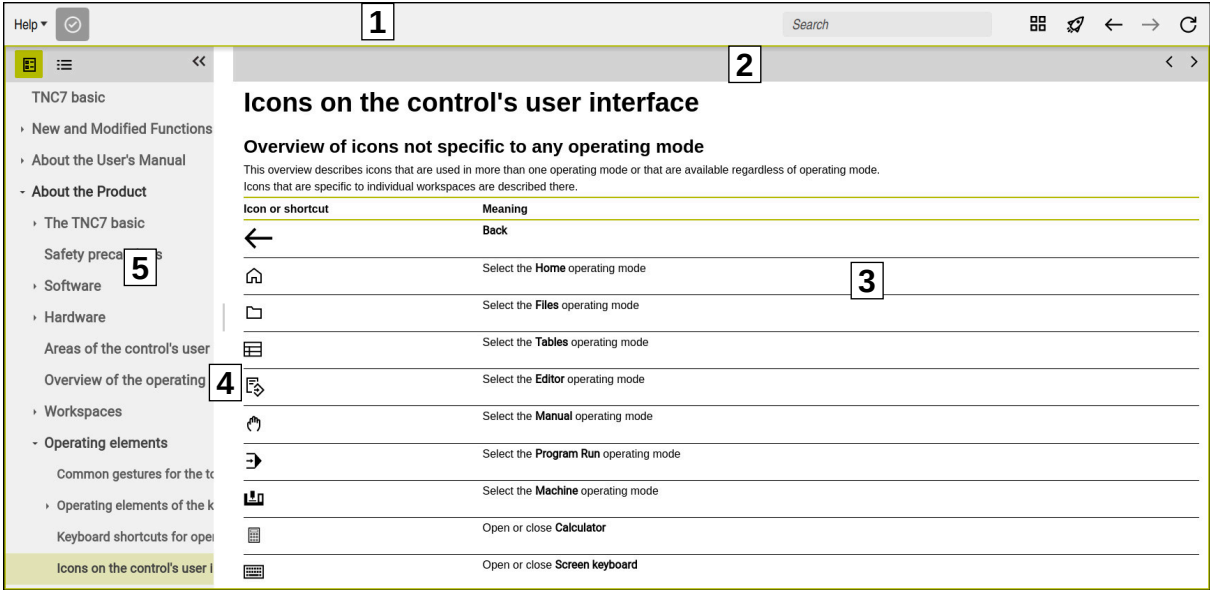
Further information: "The Help application", Page 55

Further information: Programming and Testing User's Manual

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

Further information: "Icons", Page 56

The Help application



Open **TNCguide** in the **Help** workspace

TNCguide includes the following areas:

- 1 Title bar of the **Help** workspace
Further information: "The Help workspace", Page 56
- 2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 56
- 3 Content column of the **TNCguide**
- 4 Separator between the columns of the **TNCguide**
Adjust the column width by means of the separator.
- 5 Navigation column of the **TNCguide**

Icons

The Help workspace

The **Help** workspace within the **Help** application includes the following icons:

Icon	Meaning
	Open or close the Search results column Further information: "Searching in TNCguide", Page 57
	Open Home page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., TNCguide). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function.
	Open Tutorials
	Navigate Navigate between the contents opened recently
	Refresh

TNCguide

The integrated **TNCguide** product aid includes the following icons:

Icon	Meaning
	Open Structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Open Index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
	Navigate Display previous or next page within the documentation
	Open or close Display or hide the navigation
	Copy Copy NC examples to the clipboard Further information: "Copying NC examples to clipboard", Page 58

Context-sensitive help

You can open **TNCguide** for the current context. Context-sensitive help means that the relevant information is displayed directly (e.g., for the selected item or the current NC function).

To call context-sensitive help, the following elements are available:

Icon or key	Meaning
	Help icon If you select the icon and then one of the items in the user interface, the control will open the associated information in TNCguide .
	HELP key If you press the HELP key while editing an NC block, the control will display the associated information in TNCguide .

If you call TNCguide in a certain context, the control opens the contents in a pop-up window. If you select the **Show more** button, the control will open **TNCguide** in the **Help** application.

Further information: "The Help application", Page 55

If the **Help** workspace is already open, the control displays **TNCguide** there and will not open a pop-up window.

Further information: Programming and Testing User's Manual

2.5.1 Searching in TNCguide

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

Use the search function as follows:

- ▶ Enter a string in **Search**



The search starts automatically after you enter a character.
If you wish to delete the entry, use the X symbol within the entry field.

- > The control opens the column containing the search results.
- > The control marks references also within open content pages.
- ▶ Select the reference
- > The control opens the selected content.
- > The control continues displaying the results of the last search.
- ▶ Select an alternative reference if necessary
- ▶ Enter a new character string if required

2.5.2 Copying NC examples to clipboard

Use the copy function to copy NC examples from the documentation to the NC editor.

To use the copy function:

- ▶ Navigate to the desired NC example
- ▶ Expand **Notes on using NC programs**
- ▶ Read and follow **Notes on using NC programs**

Further information: "Notes on using NC programs", Page 53



- ▶ Copy NC example to clipboard



- > The button switches colors while copying.
 - > The clipboard contains the entire content of the copied NC example.
 - ▶ Insert the NC example into the NC program
 - ▶ Adapt the inserted content according to the **Notes on using NC programs**
 - ▶ Use the Simulation mode to test the NC program
- Further information:** Programming and Testing User's Manual

2.6 Contacting the editorial staff

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

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

tnc-userdoc@heidenhain.com

3

About the product

3.1 The TNC7 basic

Every HEIDENHAIN control supports you with dialog-guided programming and finely detailed simulation. The TNC7 basic additionally offers you form-based or graphical programming to reach the desired result quickly and easily.

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

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

Further information: "Hardware enhancements", Page 81

Definitions

Abbreviation	Definition
TNC	TNC is derived from the acronym CNC (computerized numerical control). The T (tip or touch) stands for the capability of entering NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.
basic	The addition basic indicates that the control provides all required basic functions for universal milling or drilling.

3.1.1 Proper and intended use

The information about proper and intended use supports you in safely handling a product such as a machine tool.

The control is a machine component but not a complete machine. This User's Manual describes the use of the control. Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.



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

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



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

Further information: "Contacting the editorial staff", Page 59

3.1.2 Data Architecture and Access Options

Directories

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

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

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

Data acquisition

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

Data acquisition examples:

- Backup
You can manually create a backup copy of the TNC partition in order to avoid potential loss of data.
- Service file
In the event of a fault, you can manually create a service file that supports troubleshooting by a service technician.
- User administration
If user administration is active, the control assigns defined roles and permissions to the logged-on users. When a user logs on, the control will acquire personal or anonymous data in connection with its use.
- Tool-usage file
Using tool-usage files, the control can, for example, check whether the tool's remaining life is sufficient for the current NC program.
- Log functions or log files (e.g., from touch-probe cycles)
Using various log files, you can, for example, check and compare the results of touch-probe cycles.
The control needs other function-dependent log files to, for example, detect process variations.



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

Data transfer and data access

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

Examples of data transfer:

- **HEIDENHAIN DNC** (#18 / #3-03-1)
This software option enables external Windows applications to access data of the control via the TCP/IP protocol.
- **OPC UA NC Server** (#56-61 / #3-02-1*)
These software options include OPC UA, a standardized interface that allows external access to the data and functions of your control.
- **USB devices**
USB devices can be used to transfer data or make external backups.

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

Data access via file management:

- **Drives, folders, and files**
You can access external data or make external backups by using connected drives or USB devices. For example, you can manage, create, edit, or delete folders and files.
- **Functions and settings**
In the file manager you can search for certain contents or display hidden or dependent files.

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

3.1.3 Intended place of operation

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

Definitions

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

3.2 Safety precautions

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

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



Refer to your machine manual.

Before using the machine including the control, take the OEM documentation to inform yourself about the safety-related aspects, the necessary safety equipment as well as the requirements on the qualified personnel.

The following overview contains exclusively the generally valid safety precautions. Pay attention to additional safety precautions that may vary with the configuration and are given in the following chapters.



For ensuring maximum safety, all safety precautions are repeated at the relevant places within the chapters.

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- ▶ Devices should be connected or removed only by authorized service technicians
- ▶ Only switch on the machine via a connected handwheel or a secured connection

DANGER

Caution: hazard to the user!

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

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

WARNING

Caution: hazard to the user!

Manipulated data records or software can lead to an unexpected behavior of the machine. Malicious software (viruses, trojans, malware, or worms) can cause changes to data records and software.

- ▶ Check any removable memory media for malicious software before using them
- ▶ Start the internal web browser only from within the sandbox

NOTICE**Danger of collision!**

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

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

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

A power failure during the machining operation can cause uncontrolled "coasting" or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

- ▶ Use a low feed rate
- ▶ Please keep in mind that the traverse range monitoring is not available for non-referenced axes

NOTICE**Danger of collision!**

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

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

NOTICE**Danger of collision!**

The control uses the defined tool length from the tool table for compensating for the tool length. Incorrect tool lengths will result in an incorrect tool length compensation. The control does not perform tool length compensation or a collision check for tools with a length of **0** and after a **TOOL CALL 0**. There is a risk of collision during subsequent tool positioning movements!

- ▶ Always define the actual tool length of a tool (not just the difference)
- ▶ Use **TOOL CALL 0** only to empty the spindle

NOTICE**Caution: Significant property damage!**

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

NOTICE**Danger of collision!**

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

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

NOTICE**Danger of collision!**

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

- ▶ Check the NC program or program section using the graphic simulation
- ▶ Carefully test the NC program or program section in **Program Run Single Block mode**

NOTICE**Caution: Data may be lost!**

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

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

NOTICE**Caution: Data may be lost!**

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

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

3.3 Software

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



The actual range of functions depends, among other things, on the enabled software options.

Further information: "Software options", Page 69

The table shows the NC software numbers described in this User's Manual.



HEIDENHAIN has simplified the version schema, starting with NC software version 16:

- The publication period determines the version number.
- All control models of a publication period have the same version number.
- The version number of the programming stations corresponds to the version number of the NC software.

NC software number	Product
817620-20	TNC7 basic
817625-20	TNC7 basic Programming Station



Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

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

If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

3.3.1 Software options

Software options define the range of functions of the control. The optional functions are either machine- or application-specific. The software options give you the possibility of adapting the control to your individual needs.

You can check which software options are enabled on your machine.

Further information: "Viewing of software options", Page 481

The TNC7 basic features various software options that the machine manufacturer may enable separately, even at a later point in time. The following overview includes only those software options that are relevant for you.

The software options are saved on the **SIK** (System Identification Key) plug-in board. The TNC7 basic can be equipped with a **SIK** or **SIK2** plug-in board. Depending on which one is used, the numbers of the software options differ.



The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions.

The parentheses enclose the **SIK** and **SIK2** option numbers, separated by a slash, for example: (#18 / #3-03-1).

The Technical Manual contains information about additional software options that are relevant to the machine manufacturer.

SIK2 definitions

SIK2 option numbers are structured by <class>-<option>-<version>:

Class	The function is effective for the following areas: ■ 1: Programming, simulation, and process setup ■ 2: Part quality and productivity ■ 3: Interfaces ■ 4: Technology functions and quality assessment ■ 5: Process stability and monitoring ■ 6: Machine configuration ■ 7: Developer tools
Option	Sequential number within each class
Version	New versions of software options are released if, for example, its features have been changed.

You can order some software options with **SIK2** more than once in order to obtain multiple variants of the same function (e.g., if you need to enable multiple control loops for the axes). In the User's Manual, these software option numbers are identified by an asterisk (*).

The control indicates in the **SIK** menu item of the **Settings** application whether a software option has been enabled, and if so, how often. The control also shows whether is equipped with the **SIK** or **SIK2**.

Further information: "The SIK menu item", Page 480

Overview



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

Further information: "Hardware", Page 77

Software option	Definition and application
Control Loop Qty. (#0-3 / #6-01-1*)	Additional control loop A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required, for example, for detachable and motor-driven tilting tables. If your control features a SIK2 , you can order this software option multiple times and enable up to 8 control loops.
Adv. Function Set 1 (#8 / #1-01-1)	Advanced functions (set 1) On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: ■ Tilting the working plane (e.g., with PLANE SPATIAL) Further information: Programming and Testing User's Manual ■ Programming of contours on an unrolled cylinder surface (e.g., with Cycle 27 CYLINDER SURFACE) Further information: User's Manual for Machining Cycles ■ Programming the rotary axis feed rate in mm/min with M116 Further information: Programming and Testing User's Manual ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.
Adv. Function Set 2 (#9 / #4-01-1)	Advanced functions (set 2) On machines with rotary axes, this software option enables simultaneous 4-axis machining of workpieces. The software option includes the following functions: ■ TCPM (tool center point management): Automatic tracking of linear axes during rotary axis positioning Further information: Programming and Testing User's Manual ■ Running of NC programs with vectors, including optional 3D tool compensation Further information: Programming and Testing User's Manual ■ Manual moving of axes in the active tool coordinate system T-CS

Software option	Definition and application
Touch Probe Functions (#17 / #1-05-1)	Touch-probe functions This software option is used to program and execute automatic probing processes. If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled. The software option includes the following functions: ■ Automatic compensation of workpiece misalignment ■ Automatic setting of workpiece presets ■ Automatic measurement of workpieces ■ Automatic measurement of tools The touch-probe functions reduce the setup effort and increase accuracy when machining workpieces. Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
HEIDENHAIN DNC (#18 / #3-03-1)	HEIDENHAIN DNC This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
Adv. Function Set 3 (#21 / #4-02-1)	Advanced functions (set 3) This software option offers additional ease of use with two powerful miscellaneous functions. The software option includes the following miscellaneous functions: ■ M120 for machining small contour steps without error message and contour damage Further information: Programming and Testing User's Manual ■ M118 for superimposed handwheel positioning during program run Further information: Programming and Testing User's Manual The advanced functions (set 3) reduce the setup effort and increase flexibility during program run.
CAD Import (#42 / #1-03-1)	CAD Import This software option is used to select positions and contours from CAD files and to transfer them into an NC program. With the CAD Import option you reduce the programming effort and prevent typical errors such as the incorrect entry of values. In addition, CAD Import contributes to paperless manufacturing. Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280

Software option	Definition and application
Adaptive Feed Contr. (#45 / #2-31-1)	Adaptive Feed Control (AFC) This software option enables an automatic feed control that depends on the current spindle load. The control increases the feed rate as the load decreases and reduces the feed rate as the load increases. With AFC you can shorten machining times without adapting the NC program, while at the same time preventing machine damage from overload. Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256
KinematicsOpt (#48 / #2-01-1)	KinematicsOpt This software option uses automatic probing processes to check and optimize the active kinematics. With KinematicsOpt the control can correct position errors on rotary axes and thus increase the accuracy of machining operations in the tilted working plane and of simultaneous machining operations. In part, the control can compensate for temperature-induced deviations through repeated measurements and corrections. Further information: Measuring Cycles for Workpieces and Tools User's Manual
OPC UA NC Server (#56-61 / #3-02-1*)	OPC UA NC Server These software options include OPC UA, a standardized interface for remote access to the control's data and functions. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection. If more than one parallel connection is required, you need to enable multiple of these software options. If your control features the SIK2, you can order this software option multiple times and enable up to ten connections. Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505
4 Additional Axes (#77 / #6-01-1*)	Four additional control loops Further information: "Control Loop Qty. (#0-3 / #6-01-1*)", Page 70
Ext. Tool Management (#93 / #2-03-1)	Extended tool management This software option extends tool management by the two tables Tooling list and T usage order. The tables show the following contents: ■ The Tooling list shows the tool requirements of the NC program or pallet to be run Further information: "Tooling list (#93 / #2-03-1)", Page 413 ■ The T usage order shows the tool order of the NC program or pallet to be run Further information: "T usage order (#93 / #2-03-1)", Page 411 Extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.

Software option	Definition and application
Remote Desk. Manager (#133 / #3-01-1)	Remote Desktop Manager This software option is used to display and operate externally linked computer units. With Remote Desktop Manager you don't need to move between different workstations as much, and so increase your efficiency. Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522
Collision Monitoring v2 (#140 / #5-03-2)	Dynamic Collision Monitoring (DCM) The machine manufacturer can use this software option to define machine components as collision objects. The control monitors the defined collision objects during all machine movements. The software option includes the following functions: ■ Automatic interruption of program run whenever a collision is imminent ■ Warnings in case of manual axis movements ■ Collision monitoring in Test Run mode ■ Collision monitoring of fixtures Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234 ■ Defining a reduced minimum distance between fixture and tool With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or machine downtime.
Cross Talk Comp. (#141 / #2-20-1)	Compensation of axis couplings (CTC) Using this software option, the machine manufacturer can, for example, compensate for acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.
Position Adapt. Contr. (#142 / #2-21-1)	Position Adaptive Control (PAC) Using this software option, the machine manufacturer can, for example, compensate for position-induced deviations at the tool and thus increase accuracy and dynamic performance.
Load Adapt. Contr. (#143 / #2-22-1)	Load Adaptive Control (LAC) Using this software option, the machine manufacturer can, for example, compensate for load-induced deviations at the tool and thus increase accuracy and dynamic performance.
Motion Adapt. Contr. (#144 / #2-23-1)	Motion Adaptive Control (MAC) Using this software option, the machine manufacturer can, for example, change speed-dependent machine settings and thus increase the dynamic performance.
Active Chatter Contr. (#145 / #2-30-1)	Active Chatter Control (ACC) With this software option the chatter tendency of a machine used for heavy machining can be reduced. The control can use ACC to improve the surface quality of the workpiece, increase the tool life and reduce the machine load. Depending on the type of machine, the metal-removal rate can be increased by more than 25%. Further information: "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 266

Software option	Definition and application
Machine Vibr. Contr. (#146 / #2-24-1)	Vibration damping for machines (MVC) Damping of machine oscillations for improving the workpiece surface quality through the following functions: ■ AVD Active Vibration Damping ■ FSC Frequency Shaping Control
CAD Model Optimizer (#152 / #1-04-1)	Optimization of CAD models This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288
Batch Process Mngr. (#154 / #2-05-1)	Batch Process Manager (BPM) This software option makes it easy to plan and execute multiple production jobs. By extending and combining the pallet management and extended tool management functions (#93 / #2-03-1), BPM offers the following additional data, for example: ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs Further information: Programming and Testing User's Manual
Component Monitoring (#155 / #5-02-1)	Component monitoring This software option enables the automatic monitoring of machine components configured by the machine manufacturer. Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.
Model Aided Setup (#159 / #1-07-1)	Graphically supported setup This software option is used to determine the position and misalignment of a workpiece with only one touch-probe function. You can probe complex workpieces with, for example, free-form surfaces or undercuts, which is not possible with all of the other touch-probe functions. The control supports you additionally by showing the setup situation and possible touch points in the Simulation workspace by means of a 3D model. Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 326

Software option	Definition and application
Opt. Contour Milling (#167 / #1-02-1)	Optimized contour machining (OCM) This software option enables trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions. The software option includes the following cycles: ■ Cycle 271 OCM CONTOUR DATA ■ Cycle 272 OCM ROUGHING ■ Cycle 273 OCM FINISHING FLOOR and Cycle 274 OCM FINISHING SIDE ■ Cycle 277 OCM CHAMFERING ■ In addition, the control provides OCM STANDARD FIGURES for frequently needed contours With OCM you can shorten machining times while at the same time reducing tool wear. Further information: User's Manual for Machining Cycles
PTI 8 channels (#3-04-1*)	Process Tracking Interface PTI This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with SIK2. You can order this software option up to two times, each time selecting up to eight channels for recording. Further information: "Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)", Page 608  For more information, refer to the integrated help system of RemoTools SDK. ID 340442-xx

3.3.2 Information on licensing and use

Open-source software

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

To get to the licensing terms on the control:



- ▶ Select the **Home** operating mode



- ▶ Select the **Settings** application



- ▶ Select the **Operating system** tab
- ▶ Double-tap or double-click **About HeROS**
- The control opens the **HEROS License Viewer** window.

OPC UA

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

The control's behavior can be influenced by means of the OPC UA NC Server (#56-61 / #3-02-1*) and HEIDENHAIN DNC (#18 / #3-03-1). Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505

3.4 Hardware

This User's Manual describes functions for setting up and operating the machine. These functions primarily depend on the installed software.

Further information: "Software", Page 68

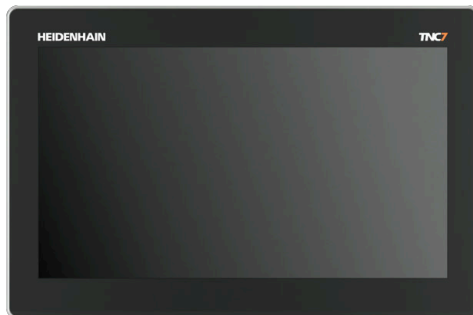
The actual range of functions also depends on hardware enhancements and the enabled software options.

Further information: "Hardware enhancements", Page 81

Further information: "Software options", Page 69

The control must have at least 16 GB of RAM, as the control will otherwise display a warning.

3.4.1 Touchscreen and keyboard unit



16" MC 345 with TE 340 (FS)

The TNC7 basic is delivered with a 16-inch screen.

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

Further information: "Common gestures for the touchscreen", Page 91

Further information: "Operating elements of the keyboard unit", Page 91

The machine operating panel is machine-dependent.



MB 340 (FS)

Operating and cleaning the touchscreen

Touchscreens can even be operated with dirty hands, as long as the touch sensors are able to detect the skin resistance. Small amounts of liquid do not affect the function of the touchscreen, but large amounts may cause incorrect input.

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

To navigate to this function:

Settings application ► Diagnostics/Maintenance group ► Touchscreen Cleaning

Further information: "The Settings application", Page 469

Do not apply the cleaning agent directly to the screen, but slightly dampen a clean, lint-free cleaning cloth with it.

The following cleaning agents are permitted for the screen:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

The following cleaning agents are prohibited for the screen:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners



- Touchscreens are sensitive to electrostatic charges from the user. Dissipate the static charge by touching metallic, grounded objects or wear ESD clothing.
- Wear operating gloves to prevent the screen from getting dirty.
- You can operate the touchscreen with special touchscreen operating gloves.

Cleaning the keyboard unit

Switch the control off before cleaning the keyboard unit.

NOTICE

Caution: risk of property damage

Incorrect cleaning agents and incorrect cleaning procedures can damage the keyboard unit or parts of it.

- ▶ Use permitted cleaning agents only
- ▶ Use a clean, lint-free cleaning cloth to apply the cleaning agent

The following cleaners are permitted for the keyboard unit:

- Cleaning agents containing anionic surfactants
- Cleaning agents containing nonionic surfactants

The following cleaning agents are prohibited for the keyboard unit:

- Cleaning agents for machines
- Acetone
- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners



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

If a trackball is embedded in the keyboard, you need to clean it only if it no longer works properly.

To clean a trackball (if needed):

- ▶ Shut down the control
- ▶ Turn the pull-off ring by 100° in counterclockwise direction
- > Turning the removable pull-off ring moves it upwards out of the keyboard unit.
- ▶ Remove the pull-off ring
- ▶ Take out the ball
- ▶ Carefully remove sand, chips, or dust from the shell area



Scratches in the shell area may impair the functionality or prevent proper functioning.

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

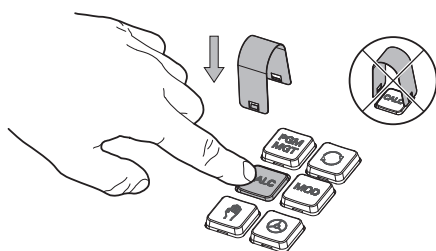
Exchanging keycaps

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



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

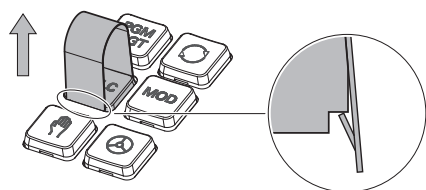
To exchange the keycaps:



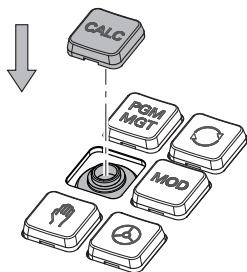
- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage



Pressing the key will make it easier to apply the keycap puller.



- Pull off the keycap



- Place the keycap onto the seal and push it down



The seal must not be damaged; otherwise IP54 protection cannot be guaranteed.

- Verify proper seating and correct functionality

3.4.2 Hardware enhancements

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

The TNC7 basic features various hardware extensions that the machine manufacturer may add separately, even at a later point in time. The following overview includes only those extensions that are relevant to you.



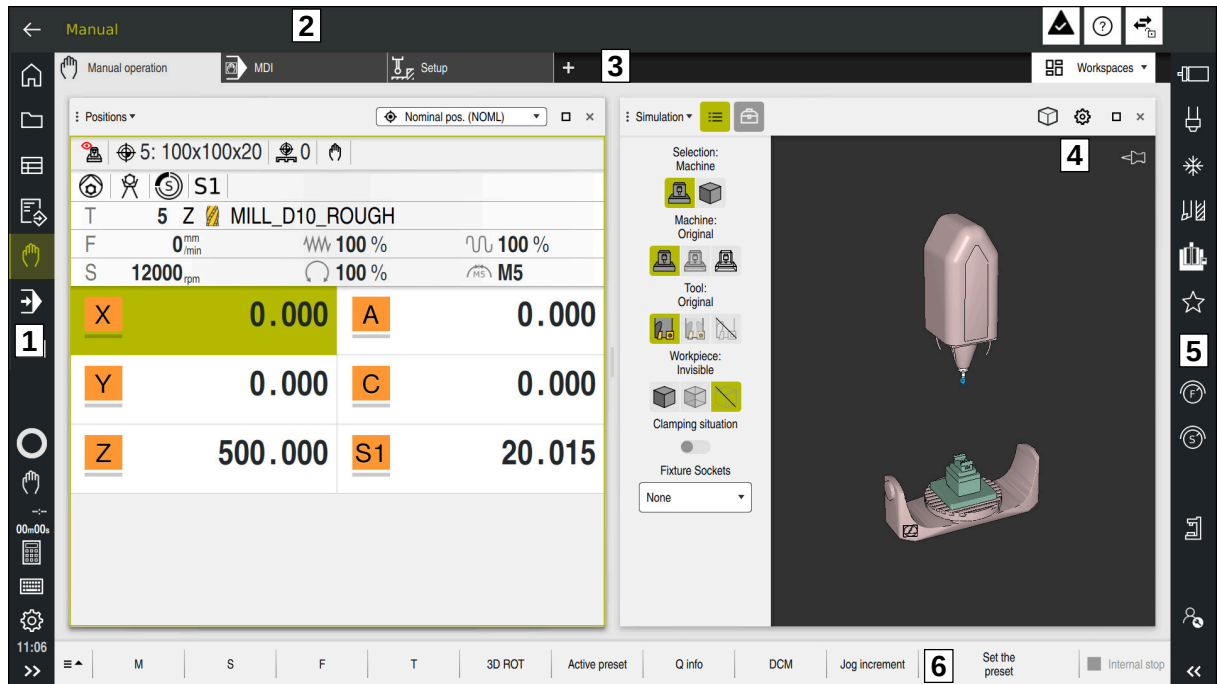
Keep in mind that particular hardware enhancements require additional software options.

Further information: "Software options", Page 69

Hardware enhancements	Definition and application
Electronic handwheels	You use this enhancement for exact manual positioning of machine axes. The wireless portable variants improve ergonomics and increase versatility. The handwheels have the following differing features: ■ Portable or installed in the machine operating panel ■ With or without display ■ With or without functional safety Electronic handwheels, for example, greatly simplify workpiece setup. Further information: "Electronic handwheel", Page 429
Workpiece touch probes	The control uses this enhancement for automatic and precise detection of workpiece positions and misalignments (#17 / #1-05-1). The workpiece touch probes have the following differing features: ■ With radio or infrared transmission ■ With or without cable Workpiece touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions during program run. Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
Tool touch probes	With this extension, the control can measure tools automatically and precisely, directly in the machine (#17 / #1-05-1). Tool touch probes have the following differing features: ■ Contact-free or tactile measurement ■ With radio or infrared transmission ■ With or without cable Tool touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions and breakage control during program run. Further information: Measuring Cycles for Workpieces and Tools User's Manual

Hardware enhancements	Definition and application
Additional operating stations	This enhancement adds a second screen, to facilitate operation of the control. The additional ITC (industrial thin client) operating stations are differentiated by their intended use: ■ The ITC 755 is a compact, additional operating station that mirrors the control's main screen, making it possible to operate the control. ■ The ITC 860 is an auxiliary screen that increases the area of the main screen. This allows multiple applications to be viewed simultaneously.  By adding a keyboard unit, the ITC 860 can be used as a full-fledged additional operating station. The additional operating stations increase operator comfort, especially on large machining centers.
Industrial PC	You use this enhancement to install and run Windows-based applications. With Remote Desktop Manager (#133 / #3-01-1), you can display applications on the control screen. Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522 The industrial PC is a secure and powerful alternative to external PCs.
Override controller	This extension allows you to define breakpoints at which the control stops during program run (e.g., before a tilting function). The override controller enables the feed rate or rapid traverse value to be changed as well as starting or continuing the NC program. Further information: "Override controller", Page 447

3.5 Areas of the control's user interface



User interface of the control in the **Manual operation** application

The control's user interface shows the following areas:

- 1 TNC bar
 - **Back**
Use this function to go backwards in the application history since booting the control.
 - **Operating modes**
Further information: "Overview of the operating modes", Page 85
 - **Status overview**
Further information: "Status overview of the TNC bar", Page 127
 - **Calculator**
Further information: Programming and Testing User's Manual
 - **Screen keyboard**
Further information: "Virtual keyboard of the control bar", Page 294
 - **Settings**
The Settings menu enables you to change the control interface:
 - **Left-hand mode**
The control swaps the positions of the TNC bar and the machine manufacturer bar.
 - **Dark Mode**
In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether the **Dark Mode** function is available for selection.

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

- Date and time
- 2 Information bar
 - Active operating mode
 - Message menu

Further information: "Message menu on the information bar", Page 297
 - Symbols

Further information: "Icons on the control's user interface", Page 99
- 3 Application bar
 - Tabs of opened applications

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

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

Further information: "Workspaces", Page 87
- 5 Machine manufacturer bar

The machine manufacturer configures the machine manufacturer bar.
- 6 Function bar
 - Selection menu for buttons

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

With the buttons you activate individual functions of the control.



Refer to your machine manual.

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

- Operating modes
- Applications
- Workspaces
- Buttons

3.6 Overview of the operating modes

The control provides the following operating modes:

Icon	Operating modes	Further information
	The Home operating mode contains the following applications: ■ The Start/Login application During the startup process, the control is in the Start/Login application. ■ The Settings application ■ The Help application ■ Applications for machine parameters	Page 469 See the User's Manual for Programming and Testing Page 540
	In the Files operating mode the control displays drives, folders and files. You can, for example, create or delete folders or files and can also connect drives.	See the User's Manual for Programming and Testing
	In the Tables operating mode you can open various tables and edit them as necessary.	Page 370
	In the Editor operating mode you can do the following: ■ Create, edit and simulate NC programs ■ Create and edit contours ■ Create and edit pallet tables	See the User's Manual for Programming and Testing
	The Manual operating mode contains the following applications: ■ The Manual operation application ■ The MDI Application ■ The Setup application ■ The Move to ref. point application ■ The Retract application You can move the tool away from the workpiece, for example after a power failure.	Page 160 Page 301 Page 305 Page 155 Page 366
	In the Program Run operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence. You also execute pallet tables in this operating mode.	Page 340
	If the machine manufacturer has defined an embedded workspace, then you can open full-screen mode with this operating mode. The machine manufacturer defines the name of the operating mode. Refer to your machine manual.	Page 457
	In the Machine operating mode, the machine manufacturers define their own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	



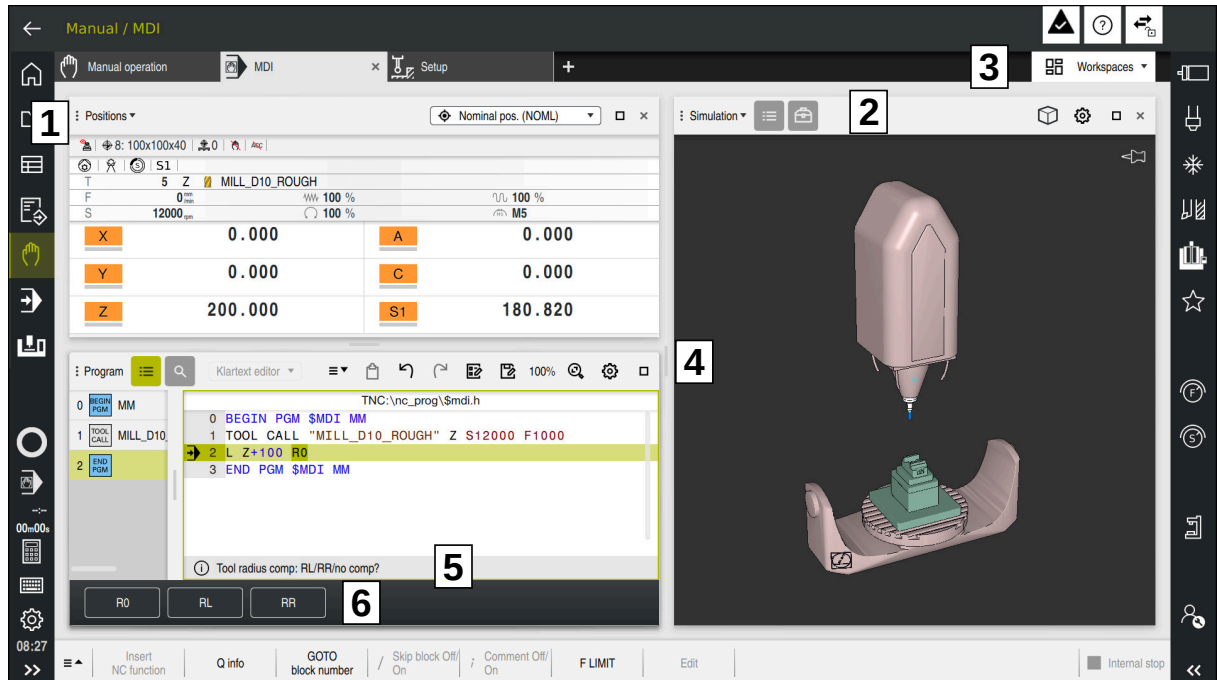
Refer to your machine manual.

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

- Operating modes
- Applications
- Buttons

3.7 Workspaces

3.7.1 Operating elements within the workspaces



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

The control displays the following operating elements:

- 1 The **Move** icon
Use the **Move** icon in the title bar to change the position of the workspaces. You can also align two workspaces vertically above each other. Alternatively, you can drag the title to move a workspace.
- 2 Title bar
In the title bar the control shows the title of the workspace, and different icons or settings, depending on the workspace.
When you select the title of a workspace, the control will open a selection menu. This selection menu allows you to switch workspaces without changing their size or position.
You can only switch between workspaces that can be closed.
- 3 Selection menu for workspaces
Use the selection menu for workspaces in the application bar to open individual workspaces. The available workspaces depend on the active application.
- 4 Separator
You use the separator between two workspaces to change the scaling of the workspaces.
- 5 Dialog bar
In the dialog bar, the control shows information on the current input, such as a description or unit of measure.
- 6 Action bar
In the action bar the control shows selection options for the current dialog; for example, an NC function.

3.7.2 Icons within the workspaces

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

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

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

Icons that are specific to individual workspaces are described there.

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

Further information: "Overview of icons not specific to any operating mode", Page 99

3.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function (#17 / #1-05-1) In the Probing function workspace you set presets on the workpiece and determine and compensate for workpiece misalignment and rotations. You can also calibrate the touch probe, measure tools or set up fixtures.	Page 305
Job list In the Job list workspace, you edit and execute pallet tables.	See the User's Manual for Programming and Testing
Open File In the Open File workspace you select or create files, for example.	See the User's Manual for Programming and Testing
Files In the file manager, the control displays drives, folders, and files. You can, for example, create or delete folders or files and can also connect drives. The Files workspace is part of the Files operating mode.	See the User's Manual for Programming and Testing
Details In the Details workspace, the control displays information on the selected machine parameter or the last change you made.	Page 545
Document You can open files for viewing in the Document workspace, for example a technical drawing.	See the User's Manual for Programming and Testing

Workspace	Further information
Settings In the Settings workspace, you can display and edit, if required, various settings of the control (e.g., set up the traverse limits). The Settings workspace is part of the Settings application.	Page 469
The Form for tables In the Form workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.	Page 384
The Form for pallets In the Form workspace the control shows the contents of the pallet table for the selected row.	See the User's Manual for Programming and Testing
Retract In the Retract workspace, you can disengage the tool after a power interruption.	Page 366
Desktop menu In the Desktop menu workspace, the control displays selected control and HEROS functions.	Page 102
Help In the Help workspace, the control displays a help graphic for the current syntax element of an NC function or the integrated product aid TNCguide .	See the User's Manual for Programming and Testing
Contour graphics In the Contour graphics workspace, the control can draw contours directly during programming. You can also use graphical programming by drawing contours and exporting them as NC blocks. In addition, you can import contours from existing NC programs and edit them graphically.	See the User's Manual for Programming and Testing
List In the List workspace, the control shows the machine parameter structure; you might be able to edit some of the parameters.	Page 541
Positions In the Positions workspace, the control displays information about the status of various functions of the control and about current axis positions.	Page 121
Program The control displays the NC program in the Program workspace.	See the User's Manual for Programming and Testing
Referencing On machines with incremental linear and angle encoders, the control shows in the Referencing workspace which axes need to be referenced.	Page 155
Remote Desktop Manager (#133 / #3-01-1) If the machine manufacturer has defined an embedded workspace, you can see and operate the screen of an external computer on the control. The machine manufacturer can change the name of the workspace. Refer to your machine manual.	Page 457

Workspace	Further information
Quick selection In the Quick selection new table and Quick selection new file workspaces, you can create files or open existing files, depending on the active operating mode.	See the User's Manual for Programming and Testing
Simulation In the Simulation workspace, the control shows the simulated or current movements, depending on the operating mode.	See the User's Manual for Programming and Testing
Simulation status In the Simulation status workspace the control shows data based on the simulation of the NC program.	Page 145
Start In the Start workspace, the control shows the steps that are performed during startup.	Page 107
Status In the Status workspace, the control shows the status and values of individual functions.	Page 129
Table In the Table workspace, the control shows the contents of a table. You can search in all tables and filter the table content.	Page 376
The Table for machine parameters In the Table workspace the control shows the machine parameters; you might be able to edit some of them.	Page 541
Keyboard In the Keyboard workspace, you can enter NC functions, letters and numbers, and also navigate.	Page 294
Text editor Use the Text editor workspace to create and edit text files.	See the User's Manual for Programming and Testing
Overview In the Overview workspace, the control displays information on the status of individual functional safety (FS) safety functions.	Page 464



Refer to your machine manual.

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

- Operating modes
- Applications
- Workspaces
- Buttons

3.8 Operating elements

3.8.1 Common gestures for the touchscreen

The screen of the control is multi-touch capable. That means the control can distinguish various gestures, even with two or more fingers at once.

You can use the following gestures:

Icon	Gesture	Meaning
	Tap	Select element
	Double tap	■ Open an element (e.g., window in the Settings application) ■ Edit an NC block ■ Reset the graphic or 3D model to its original size
	Long press	Open context menu i ■ If you are working with a mouse, click with the right mouse key. ■ If you do not stop holding, the control will automatically cancel the holding gesture after approximately ten seconds.
	Swipe	■ Scroll ■ Rotate the graphic or 3D model
	Drag	■ Change the selected area ■ Shift elements
	Two-finger drag	■ Move a graphic or 3D model ■ Shift drawing view in the Contour graphics workspace
	Spread	■ Change font size ■ Enlarge a graphic or 3D model
	Pinch	■ Reduce font size ■ Reduce a graphic or 3D model

3.8.2 Operating elements of the keyboard unit

Application

You operate the TNC7 basic primarily through the touchscreen, meaning with gestures.

Further information: "Common gestures for the touchscreen", Page 91

In addition, the control's keyboard unit offers keys and other elements for alternative operating sequences.

Description of function

The tables below describe the keyboard unit's operating elements.



If there are deviations from the virtual keyboard, the table also indicates the corresponding keys on the virtual keyboard.

Further information: "Virtual keyboard of the control bar", Page 294

Keycaps for alphabetic keyboard

Key	Meaning
  	Enter texts (e.g., file names)
	Q
  	With an open NC program, enter a Q parameter formula in the Editor operating mode, or in the Manual operating mode open the Q parameter list window Further information: Programming and Testing User's Manual By selecting the Q key multiple times, you can switch between Q , QL , and QR .
	Close the window or context menu, discard the current input
	Select the next element; for example, an input field, button, or selection option
[SHIFT] + [TAB]	Select the previous element
	Show the hidden NC block
	Hide or show NC blocks
	Create screenshot
	The DIADUR keys provide the following functions: ■ Left DIADUR key Open the HEROS menu ■ Right DIADUR key Open the Remote Desktop Manager connection in the defined desktop Further information: "Connection settings", Page 524
	Open context menu
  	Perform calculations for numerical values in input fields and table cells
  	

Keycaps for operating aids

Key	Meaning
	Open the Open File workspace in the Editor and Program Run operating modes Further information: Programming and Testing User's Manual
	Currently no function
	Open and close the message menu Further information: "Message menu on the information bar", Page 297
	Open and close the calculator Further information: Programming and Testing User's Manual
	Open the Settings application Further information: "The Settings application", Page 469
	Open the online help Further information: "User's Manual as integrated product aid: TNCguide", Page 54

Operating modes

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

Key	Meaning
	Open the Manual operation application in the Manual operating mode Further information: "The Manual operation application", Page 160
	Activate and deactivate the electronic handwheel in the Manual operating mode Further information: "Electronic handwheel", Page 429
	Open the Tool Management tab in the Tables operating mode Further information: "Tool management ", Page 184
	Open the MDI application in the Manual operating mode Further information: "The MDI application ", Page 301
	Open the Program Run operating mode in Single Block mode Further information: "The Program Run operating mode", Page 340
	Open the Program Run operating mode Further information: "The Program Run operating mode", Page 340
	Open the Editor operating mode Further information: Programming and Testing User's Manual
	While the NC program is running, open the Simulation workspace in the Editor operating mode Further information: Programming and Testing User's Manual

Keycaps for NC dialog



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

Key	Meaning
	In the Insert NC function window, open the Path contour folder in order to select an approach or departure function
	Open the Contour graphics workspace (e.g., to draw a milling contour) Only in the Editor operating mode
	Program a chamfer
	Program a straight line segment
	Program a circular arc with radius entry
	Program a rounding arc
	Program a circular arc with tangential connection to the preceding contour element
	Program a circle center or pole
	Program a circular arc with reference to the circle center
	In the Insert NC function window, open the Setup folder in order to select a touch probe cycle Further information: Measuring Cycles for Workpieces and Tools User's Manual
	In the Insert NC function window, open the Fixed cycles folder in order to select a cycle Further information: User's Manual for Machining Cycles
	In the Insert NC function window, open the Cycle call folder to select a machining cycle. Further information: User's Manual for Machining Cycles
	Program a jump label
	Program a subprogram or a program section repeat
	Program an intentional stop
	Pre-select a tool in the NC program
	Call the tool in the NC program
	In the Insert NC function window, open the Special functions folder (e.g., to program a workpiece blank)
	In the Insert NC function window, open the Selection folder (e.g., to call an external NC program)

Further information: Programming and Testing User's Manual

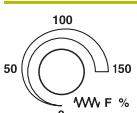
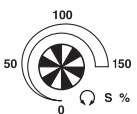
Keycaps for axis input and value input

Key	Meaning
 ... 	Select axes in the Manual operating mode, or enter them in the Editor operating mode
 ... 	Enter numbers (e.g., coordinate values)
	Insert a decimal separator during entry
	Invert algebraic sign of entered value
	Delete values during entry
	Open position display of the status overview to copy axis values Further information: "Status overview of the TNC bar", Page 127 In the Editor operating mode and the MDI application, program a line L with the actual positions of all defined axes.
	In the Editor operating mode, open the FN folder in the Insert NC function window
	
	Clear entries or delete messages
	Delete NC block or cancel a dialog during programming
	Skip or remove optional syntax elements during programming
	Confirm entries and continue dialogs
	Conclude entry (e.g., finish an NC block)
	Switch between entry of polar and Cartesian coordinates
	Switch between entry of incremental and absolute coordinates

Keycaps for navigation

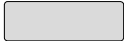
Key	Meaning
 	Position the cursor
 	
	■ Directly position the cursor by using the number of an NC block, table row or machine parameter ■ Open the selection menu while editing
	Jump to first line of an NC program or first column of a table
	Jump to last line of an NC program or last column of a table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

Potentiometers

Potentiometer	Function
	Increase or reduce the feed rate Further information: Programming and Testing User's Manual
	Increase or reduce the spindle speed Further information: Programming and Testing User's Manual

3.8.3 Keyboard shortcuts for operating the control

With a keyboard unit or a USB keyboard, you can use keyboard shortcuts in your control. In the User's Manual, the labels of the keys are used when indicating keyboard shortcuts. Keys without a label are indicated as follows:

Key	Designation
	SHIFT
	SPACE
	RETURN
	TAB
	UP
	DOWN
	RIGHT
	LEFT

3.8.4 Icons on the control's user interface

Overview of icons not specific to any operating mode

This overview describes icons that are used in more than one operating mode or that are available regardless of operating mode.

Icons that are specific to individual workspaces are described there.

Icon or shortcut	Meaning
	Back
	Select the Home operating mode
	Select the Files operating mode
	Select the Tables operating mode
	Select the Editor operating mode
	Select the Manual operating mode
	Select the Program Run operating mode
	Select the Machine operating mode
	Open or close Calculator
	Keyboard for formula input
	Open or close Screen keyboard
	Open or close the Settings selection menu
	Open or close ■ White: expand the TNC bar or machine manufacturer's bar ■ Green: collapse the TNC bar or machine manufacturer's bar ■ Gray: Confirm message
	Add
	Open
	Close
	Move Change the position of workspaces or windows
	Activate or deactivate Remember position The control remembers the position of the window until it is shut down.
	Activate or deactivate Change column width You can change the width of the currently selected column.

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

Icon or shortcut	Meaning
	Show the Message menu
	Call context-sensitive help Further information: "Context-sensitive help", Page 57
	Dynamic Collision Monitoring DCM (#140 / #5-03-2) deactivated Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
	Secure connection configuration External access to the control is active; all connections are using a secure connection configuration.
	Non-secure connection configuration External access to the control is active and at least one connection uses a non-secure connection configuration.
	Automatic program start active Further information: "Automatic program start", Page 361
	Window manager Select active applications in the background (e.g., HEROS functions window)
	ITC is connected and Enabling VNC focus is set Further information: "The VNC Focus Settings area", Page 521
	Both the control and the ITC show the following icons. The focus is on the currently used device You can use the mouse and keyboard for the local and the remote device.
	The focus is on the remote device You cannot use the mouse and keyboard on the local device.
	No focus assigned
	Inputs with the mouse and keyboard are locked until the focus has been assigned to the local or remote device.
Safety self-test	Self-test of the control is running
[CTRL] + [A]	Mark everything

3.8.5 The Desktop menu workspace

Application

In the **Desktop menu** workspace, the control displays selected control and HEROS functions.

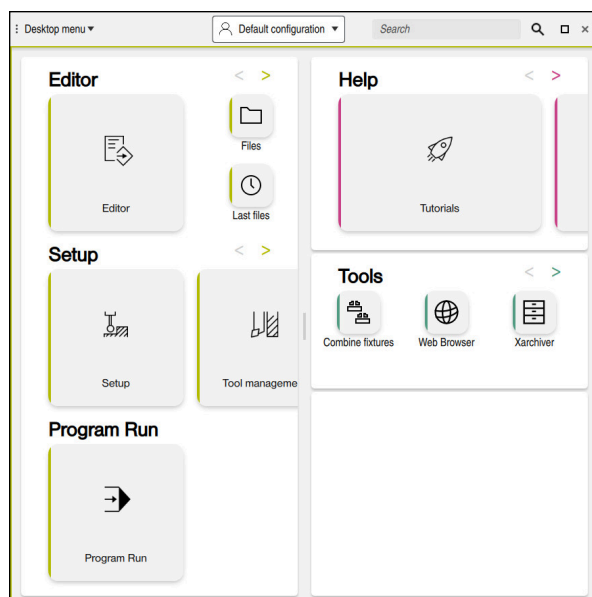
Description of function

The title bar of the **Desktop menu** workspace includes the following functions:

- The **Active Configuration** selection menu
Using the selection menu, you can activate a configuration of the control interface.
Further information: "Configuring the control's user interface", Page 547
- Full-text search
Search for functions in the workspace with the full-text search.
Further information: "Adding and removing favorites", Page 103

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

- **Control**
In this area you can open operating modes or applications.
Further information: "Overview of the operating modes", Page 85
Further information: "Overview of workspaces", Page 88
- **Tools**
In this area you can open some tools from the HEROS operating system.
Further information: "HEROS operating system", Page 577
- **Help**
In this area you can open training videos or **TNCguide**.
Further information: "User's Manual as integrated product aid: TNCguide", Page 54
- **Favorites**
In this area you will find the favorites that you have chosen.
Further information: "Adding and removing favorites", Page 103



The **Desktop menu** workspace

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

Showing or hiding an area

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

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



Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

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

- ▶ Enter the search term in the full-text search (e.g., **dat**)
- ▶ Long-press or right-click the function's icon (e.g., **Date/Time**)
- > The control displays the icon for **adding favorites**.



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

Removing favorites

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

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



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

4

First steps

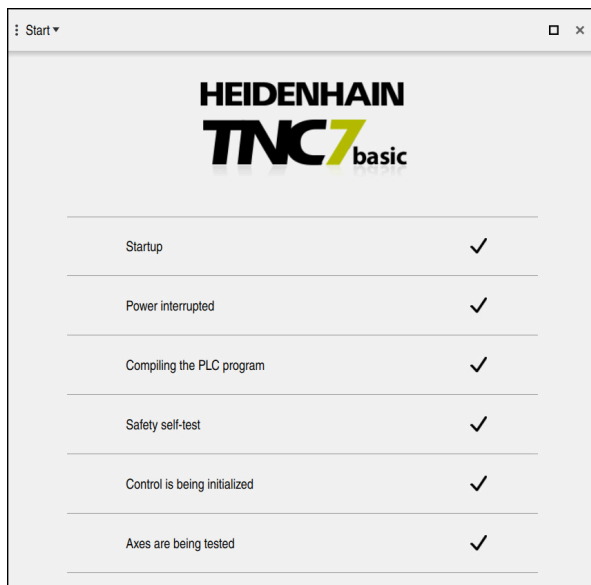
4.1 Chapter overview

This chapter uses an example workpiece to explain how to operate the control: from switching the machine on to the finished workpiece.

The chapter covers the following topics:

- Switching the machine and the control on
- Setting up a tool
- Setting up a workpiece
- Machining a workpiece
- Switching the machine off

4.2 Switching the machine and the control on



The **Start** workspace

⚠ DANGER

Caution: hazard to the user!

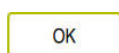
Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

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

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

To switch the machine on:

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



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



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

Further information: "The Referencing workspace",
Page 155



- ▶ Press the **NC Start** key
- The control moves to all necessary reference points.
- The control is ready for operation and the **Manual operation** application is open.

Further information: "The Manual operation application",
Page 160

More detailed information

- Switching on and off
- Position encoders

Further information: "Position encoders and reference marks", Page 167

- Axis reference run

4.3 Setting up a tool

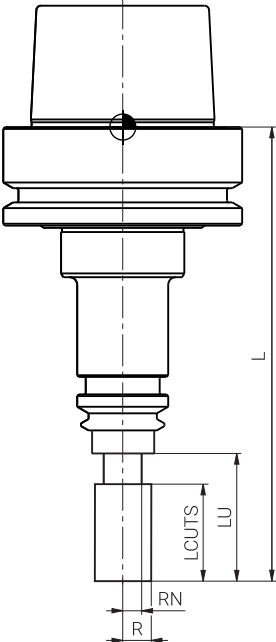
This chapter uses an example to show how to set up a tool.

To set up a tool, the following steps are necessary:

- Preparing the tools
- Opening the **Tool management** application and the **Form** workspace
- Inserting a tool in the tool management
- Defining the tool
- Entering the tool in the pocket table

You have to set up the tools to enable the control to position, simulate and compensate for the tools, for example.

4.3.1 Example tool

Tool	Parameter	Value
	Name	MILL_D12
	L	+120
	R	+6
	R2	+0
	TYP	Roughing mill (MILL_R)
	LCUTS	+20
	LU	28
	RN	4.8
	R-TIP	0
	T-ANGLE	+0
	CUTS	4

4.3.2 Preparing the tools

To prepare tools:

- ▶ Clamp the required tools in the tool holder
- ▶ Measure the tools (e.g., with a presetter)
- ▶ Write down the parameters
- > Now the tool is ready.

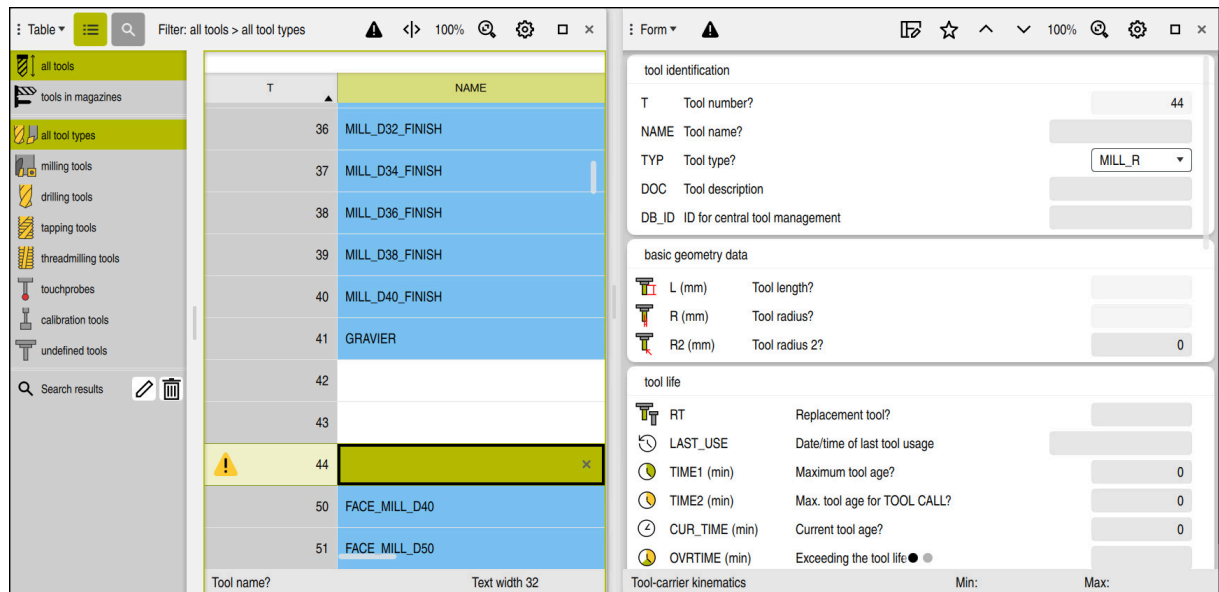


- You can measure tools with the tool touch probe of the control.
- Alternatively, the control makes it possible to measure the tool without a tool touch probe.

Further information: "Measuring the tool by scratching", Page 334

- You cannot and do not have to measure all parameters. Some values can be found in the tool catalog for example.

4.3.3 Opening the Tool management application and the Form workspace



The **Tool management** application with the **Table** and **Form** workspaces

You enter tools in the **Tool management** application of the **Tables** operating mode. You define tool parameters in the **Form** workspace. In the **Form** workspace, the control shows the required parameters of the selected tool.

To open the **Tool management** application and the **Form** workspace:



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



- ▶ Select the **Tool management** application
- The control displays the **Tool management** application.
- ▶ In the application bar, select **Workspaces**
- ▶ Select **Form**
- The control opens the **Form** workspace.

More detailed information

- **Tables** operating mode
Further information: "The Tables operating mode", Page 370
- The **Form** workspace
Further information: "The Form workspace for tables", Page 384
- Tool parameters
Further information: "Tool parameters", Page 175
- Tool management
Further information: "Tool management ", Page 184

4.3.4 Inserting a tool in the tool management

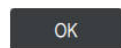
To insert a tool:



- ▶ Activate **Edit**



- ▶ Select **Insert tool**
 - > The control opens the **Insert tool** window.
 - ▶ Select the tool type, for example **Roughing mill (MILL_R)**
 - ▶ Enter the line number, if required



- ▶ Select **OK**
 - > The control inserts the new tool

More detailed information

- Tool types

Further information: "Tool types", Page 182

4.3.5 Defining the tool

To define the tool with the measured parameters:

- ▶ Enter the tool name **NAME** (e.g., **MILL_D12**)
- ▶ Enter the tool length **L** (e.g., **120**)
- ▶ Enter the tool radius **R** (e.g., **6**)
- ▶ Enter the tooth **LCUTS** (e.g., **20**)
- ▶ Enter the usable length **LU** (e.g., **28**)
- ▶ Enter the neck radius **RN** (e.g., **4,8**)
- ▶ Enter the number of teeth **CUT** (e.g., **4**)
- > The control saves the tool with all entered parameters.



Based on the entered parameters, the control shows the tool correctly in the **Simulation** workspace.

Further information: Programming and Testing User's Manual

To use the full performance range, you have to define further parameters, such as **KINEMATIC** (tool-carrier kinematics for collision monitoring). You need different parameters depending on the tool type.

Further information: "Tool parameters", Page 175

4.3.6 Entering the tool in the pocket table



Refer to your machine manual!

Access to the **tool_p.tch** pocket table is machine-dependent.

P	T	RSV	ST	F	L	NAME
1.1	1					MILL_D2_ROUGH
1.2	2					MILL_D4_ROUGH
1.3	3					MILL_D6_ROUGH
1.4	4					MILL_D8_ROUGH
1.5	5	R				MILL_D10_ROUGH
1.6	6					MILL_D12_ROUGH
1.7	7					MILL_D14_ROUGH
1.8	8					MILL_D16_ROUGH
1.9	9					MILL_D18_ROUGH
1.10	10					MILL_D20_ROUGH
1.11	11					MILL_D22_ROUGH
1.12	12					MILL_D24_ROUGH
1.13	13					MILL_D26_ROUGH
1.14	14					MILL_D28_ROUGH
1.15	15					MILL_D30_ROUGH

Tool number? Min: 1 Max: 99999

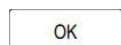
The **Pocket table** application with opened **Table** workspace

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. You must add the tool to the pocket table in order for the control to be able to insert the tool. The tool must be stored in the machine.

To enter a tool in the pocket table:



- ▶ Select the **Pocket table** application
- The control displays the **Pocket table** application.
- ▶ Enter the desired pocket number in the **Table** workspace
- ▶ Open the **Form** workspace
- ▶ Activate **Edit**
- ▶ Select the tool number **T**
- The control opens the **Tool selection** window.
- ▶ Select the tool you want to use
- ▶ Select **OK**
- The control inserts the tool in the pocket table.
- ▶ Define additional parameters if necessary (e.g., to reserve a pocket)



More detailed information

- Pocket table

Further information: "Pocket table tool_p.tch", Page 406

4.4 Setting up a workpiece

4.4.1 Selecting an operating mode

You set up workpieces in the **Manual** operating mode.

To select the **Manual** operating mode:



- ▶ Select the **Manual** operating mode
- The control displays the **Manual** operating mode.

More detailed information

- Operating mode: **Manual**

Further information: "Overview of the operating modes", Page 85

4.4.2 Clamping the workpiece

Mount the workpiece with a fixture on the machine table.

More detailed information

- The **Probing function** workspace

Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305

4.4.3 Workpiece presetting with a touch probe

Inserting a workpiece touch probe

Use a workpiece touch probe to set up the workpiece with the aid of the control and set the workpiece preset.

To insert a workpiece touch probe:

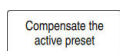


- ▶ Select **T**
- ▶ Enter the tool number of the workpiece touch probe (e.g., **600**)
- ▶ Press the **NC Start** key
- The controls inserts the workpiece touch probe.

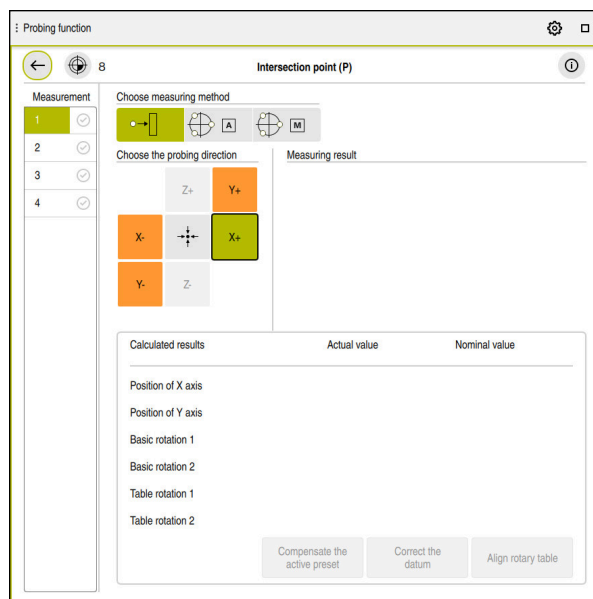


Setting a workpiece preset

To set a workpiece preset at a corner:



- ▶ Select the **Setup** application
- ▶ Select **Intersection point (P)**
 - The control opens the probing cycle.
- ▶ Select **Change preset**, if necessary
 - The control opens the **Change the preset** window.
 - Choose another row in the preset table if necessary.
 - The control highlights the selected line in green.
- ▶ Select **Apply**, if appropriate
 - Manually position the touch probe near the first touch point of the first workpiece edge
- ▶ In the **Choose the probing direction** area, select the direction of probing (e.g., **Y+**)
- ▶ Press the **NC Start** key
 - The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- ▶ Manually position the touch probe near the second touch point of the first workpiece edge
- ▶ Press the **NC Start** key
 - The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- ▶ Manually position the touch probe near the first touch point of the second workpiece edge
- ▶ In the **Choose the probing direction** area, select the direction of probing (e.g., **X+**)
- ▶ Press the **NC Start** key
 - The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
- ▶ Manually position the touch probe near the second touch point of the second workpiece edge
- ▶ Press the **NC Start** key
 - The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.
 - The control then displays the coordinates of the determined corner point in the **Measuring result** area.
- ▶ Select **Compensate the active preset**
 - The control applies the calculated results to the workpiece preset.
- ▶ Select **Exit probing**
 - The control closes the probing cycle.



The **Probing function** workspace with an open manual probing function

More detailed information

- The **Probing function** workspace
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
- Presets in the machine
Further information: "Presets in the machine", Page 167
- Preset management
Further information: "Preset management", Page 209
- Tool change in the **Manual operation** application
Further information: "The Manual operation application", Page 160

4.5 Machining a workpiece

4.5.1 Selecting an operating mode

You can machine workpieces in the **Program Run** operating mode.

To select the **Program Run** operating mode:



- ▶ Select the **Program Run** operating mode
- > The control displays the **Program Run** operating mode and the most recently executed NC program.

More detailed information

- The **Program Run** operating mode

Further information: "The Program Run operating mode", Page 340

4.5.2 Opening an NC program

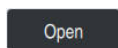
To open an NC program:



- ▶ Select **Open File**
- > The control displays the **Open File** workspace.



- ▶ Select an NC program



- ▶ Select **Open**
- > The control opens the NC program.

More detailed information

- The **Open File** workspace

Further information: Programming and Testing User's Manual

4.5.3 Starting an NC program

To start an NC program:



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

4.5.4 Interrupting an NC program

To interrupt an NC program:



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



While a machining operation is interrupted, it is not possible to change the operating mode and edit the NC program.

To continue program run, press the **NC Start** key.

4.5.5 Aborting program run

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

To abort program run:



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



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



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

4.5.6 Safe restart with block scan

The **Block scan** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations. In a block scan, the control restores the machine status up to the corresponding NC block (i.e., the spindle is switched on or the machine axes are tilted to position). A block scan reduces the risk of collision.

Running a block scan after a program interruption

To perform a block scan:



- ▶ Select the **Program Run** operating mode



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



- ▶ Press the **NC Start** key
- The control starts the block scan and calculates until the entered NC block.



- The control opens the **Restore machine status** window.

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



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



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

4.6 Switching the machine off



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

NOTICE

Caution: Data may be lost!

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

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

To power-off the machine:



- ▶ Select the **Home** operating mode

Shut down

- ▶ Select **Shut down**
- The control opens the **Shut down** window.

Shut down

- ▶ Select **Shut down**
- If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
- ▶ If necessary, save unsaved NC programs with **Save** or **Save as**
- The control shuts down.
- After completion of the shutdown process, the control displays the text **Now you can switch off.**
- ▶ Switch off the main power switch of the machine

5

Status displays

5.1 Overview

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

The control offers the following status displays:

- General status display and position display in the **Positions** workspace
Further information: "The Positions workspace", Page 121
- Status overview on the TNC bar
Further information: "Status overview of the TNC bar", Page 127
- Additional status displays for specific areas in the **Status** workspace
Further information: "The Status workspace", Page 129
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
Further information: "The Simulation status workspace", Page 145

5.2 The Positions workspace

Application

The general status display in the **Positions** workspace provides information about the status of various functions of the control and about current axis positions.

Description of function

Nominal pos. (NOML)	
5: 100x100x20 0	
T 8 Z MILL_D16_ROUGH	
F 0 mm/min	100 %
S 12000 rpm	M5
X 12.000	A 0.000
Y -3.000	C 0.000
Z 40.000	S1 20.000

The **Positions** workspace with general status display

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

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

Further information: "Overview of the operating modes", Page 85

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

- Icons of active and inactive functions (e.g., Dynamic Collision Monitoring (DCM) (#140 / #5-03-2)
- Active tool
- Technology values
- Settings of the spindle and feed-rate potentiometers
- Active miscellaneous functions for the spindle
- Axis values and statuses, such as "Axis not referenced"

Further information: "Test status of the axes", Page 466

Axis display and position display



Refer to your machine manual.

In the machine parameter **axisDisplay** (no. 100810) you define the quantity and sequence of the displayed axes.

Icon	Meaning
IST	Position display mode (e.g., actual or nominal coordinates of the current tool position) You can select the mode in the title bar of the workspace. Further information: "Position displays", Page 148
	Axes The X axis is selected. You can move the selected axis.
	The auxiliary axis m is not selected. The control displays auxiliary axes, such as the tool magazine, as lowercase letters. Further information: "Definition", Page 126
?	The axis is not referenced.
	The axis is not in safe mode. Further information: "Checking axis positions manually", Page 467
Δ	The axis is moving the distance-to-go shown next to the symbol.
	The axis is clamped.
	You can move the axis with the handwheel.
	You cannot move the axis with the handwheel.
 Refer to your machine manual. The machine manufacturer defines which axes you can move with the handwheel.	
	Feed status when stopped Further information: "Functional safety (FS) in the Positions workspace", Page 464
	Spindle status when stopped Further information: "Functional safety (FS) in the Positions workspace", Page 464

Presets and technology values

Icon	Meaning
	Number and comment of the active workpiece preset The number corresponds to the active row number of the preset table. The comment corresponds to the content of the DOC column. Further information: "Preset management", Page 209
	Number of the active pallet preset The number corresponds to the active row number in the pallet preset table. Further information: Programming and Testing User's Manual
T	In the T area, the control shows the following information: ■ Number of the active tool ■ Tool axis of the active tool ■ Symbol of the defined tool type ■ Name of the active tool
F	In the F area, the control shows the following information: ■ Active feed rate in mm/min You can program the feed rate in various units of measurement. The control always converts the programmed feed rate in this display to mm/min. ■ If M136 is active: active feed rate in mm/rev Further information: Programming and Testing User's Manual ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent Further information: "Potentiometers", Page 97 If a feed-rate limitation has been activated with the F LIMIT button, the area is labeled F LIMIT instead of F. The control displays the text F LIMIT and the feed-rate value in orange. Further information: "Feed rate limit F LIMIT", Page 345
S	In the S area, the control shows the following information: ■ Active shaft speed in rpm If you have programmed a cutting speed instead of a rotational speed, the control automatically converts this value to a rotational speed. ■ Setting of the spindle potentiometer in percent ■ Active miscellaneous function for the spindle

Active functions

Icon	Meaning
	The Manual traverse function is active.
	The Manual traverse function is not active. Further information: "The Program Run operating mode", Page 340
	Tool radius compensation RL is active. Further information: Programming and Testing User's Manual
	Tool radius compensation RR is active. Further information: Programming and Testing User's Manual The control displays these symbols as transparent during the Block scan function. Further information: "Block scan for mid-program startup", Page 352
	Tool radius compensation R+ is active. Further information: Programming and Testing User's Manual
	Tool radius compensation R- is active. Further information: Programming and Testing User's Manual These symbols are transparent while the Block scan function of the control is active. Further information: "Block scan for mid-program startup", Page 352
	3D tool compensation is active (#9 / #4-01-1). Further information: Programming and Testing User's Manual This symbol is transparent while the Block scan function of the control is active. Further information: "Block scan for mid-program startup", Page 352
	A basic rotation is defined in the active preset. Further information: "Basic rotation and 3D basic rotation", Page 211
	The basic rotation will be taken into account while moving the axes. Further information: "The Basic rotation selection item", Page 221
	A 3D basic rotation is defined in the active preset. Further information: "Basic rotation and 3D basic rotation", Page 211

Icon	Meaning
	The tilted working plane will be taken into account while moving the axes. Further information: Programming and Testing User's Manual Further information: "The 3D ROT selection item", Page 221
	The Tool axis function is active (#21 / #4-02-1). Further information: "The Tool axis selection item", Page 221
	The TRANS MIRROR function or the Cycle 8 MIRRORING is active. The axes programmed in this function or cycle are moved with mirroring. Further information: User's Manual for Machining Cycles Further information: Programming and Testing User's Manual
	The pulsing spindle speed function S-PULSE is active. Further information: Programming and Testing User's Manual
	The PARAXCOMP DISPLAY function is active.
	The PARAXCOMP MOVE function is active. Further information: Programming and Testing User's Manual
	If one of the following NC functions is active, then the control displays this symbol: ■ CYLINDER SURFACE (#8 / #1-01-1) ■ FUNCTION PARAX MODE ■ POLARKIN (#8 / #1-01-1) This icon might be superimposed on the icons for FUNCTION PARAX COMP DISPLAY and FUNCTION PARAX COMP MOVE. Further information: Programming and Testing User's Manual
TCPM	The function M128 or FUNCTION TCPM is active (#9 / #4-01-1). Further information: Programming and Testing User's Manual
	The Dynamic Collision Monitoring function (DCM (#140 / #5-03-2)) is active.
	The Dynamic Collision Monitoring function (DCM (#140 / #5-03-2)) is not active. Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
	The Dynamic Collision Monitoring function (DCM) is active with a reduced minimum distance (#140 / #5-03-2). Further information: Programming and Testing User's Manual

Icon	Meaning
AFC 	The Adaptive Feed Control function (AFC) is active in teach-in cut mode (#45 / #2-31-1).
AFC	The Adaptive Feed Control function (AFC) is active in closed-loop mode (#45 / #2-31-1). Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256
ACC	The Active Chatter Control function (ACC) is active (#145 / #2-30-1). Further information: "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 266



In the optional machine parameter **iconPrioList** (no. 100813), you can change the sequence in which the control displays these symbols. The symbol for Dynamic Collision Monitoring (DCM (#140 / #5-03-2)) is always visible and cannot be configured.

Definition

Auxiliary axes

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

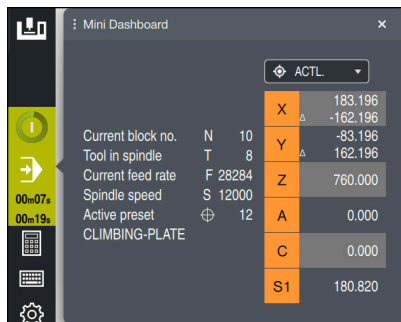
5.3 Status overview of the TNC bar

Application

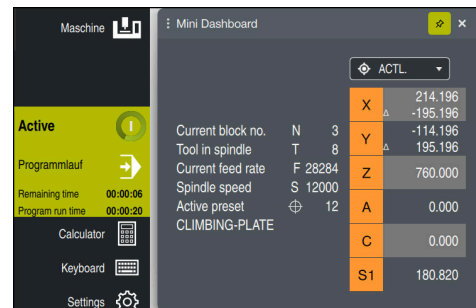
On the TNC bar, the control shows a status overview with the execution status, the current technology values, and the axis positions.

Description of function

General information



Status overview of the collapsed TNC bar with opened position display



Status overview of the expanded TNC bar with opened position display

While an NC program or individual NC blocks are being executed, the control displays the following information in the status overview:

- **Control-in-operation:** Current machining status
Further information: "Definition", Page 128
- Symbol of the application used for machining
- Remaining run time of the NC program
- Program run time
- Active tool
- Active feed rate
- Current spindle speed
- Number and comment of the active workpiece preset
- Position display

If the TNC bar is expanded, then the control displays the run times in the default format hh:mm:ss (e.g., 01:10:30).

If the TNC bar is collapsed, then the control displays the run times in the following formats:

- Minutes and seconds for run times of less than an hour (e.g., 10min30s)
- Hours and minutes for run times of one hour or longer (e.g., 01h04min)

Further information: "Display of the program run time", Page 146

Position display

If you select the status overview area, then the control opens or closes the position display with the current axis positions. The position display mode can be selected independently of the **Positions** workspace (e.g., **Actual pos. (ACT)**).

Further information: "The Positions workspace", Page 121

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

Press the **actual position capture** key to open the position display. The control prompts you to select the value to be copied to the clipboard. If you select the value of an axis while editing an NC block, then the control inserts the value into the NC block.

Further information: Programming and Testing User's Manual

Definition

Control-in-operation:

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

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

Further information: "Interrupting, stopping or canceling program run", Page 346

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

5.4 The Status workspace

Application

In the **Status** workspace the control shows the additional status display. The additional status display shows the current status of various functions on specific tabs. You can use the additional status display to better monitor the running of an NC program by receiving real-time information about active functions and accesses.

Description of function

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

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

Further information: "Overview of the operating modes", Page 85

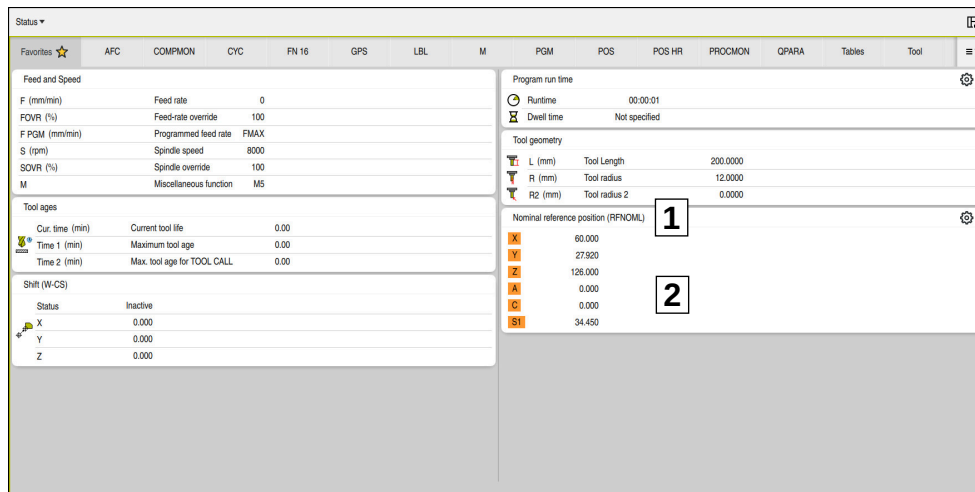
Icons

The following icons are shown in the **Status** workspace:

Icon	Meaning
	Configure the layout You can make the following layout adaptations: ■ Add or remove areas to the Favorites view ■ Rearrange areas using the gripper ■ Add or remove columns
	Settings Some areas have their own settings. Use this icon to customize the contents of the area (e.g., by defining the variable range to be displayed).
	Favorite Further information: "The Favorites tab", Page 130
	Add The control only shows this icon when you are adapting the layout. With this icon you can add the following elements: ■ Column You can divide the workspace into several columns. Further information: "Adding a column in the workspace", Page 386 ■ Area In the Favorites view you can add another area.
	Remove The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

The Favorites tab

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



The **Favorites** tab

- 1 Area
- 2 Contents

Each section of the status display has its own **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

The AFC tab (#45 / #2-31-1)

The control displays information on the Adaptive Feed Control function (AFC) (#45 / #2-31-1) on the **AFC** tab.

Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256



AFC tab

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

Area	Contents
AFC status	■ AFC If AFC is being used to control the feed rate, then Control is displayed in this area. If the control is not controlling the feed rate, then Inactive is displayed in this area. ■ CUT Counts the quantity of cuts that have been performed with FUNCTION AFC CUT BEGIN, starting from zero. ■ FOVR (%) Active factor of the feed-rate potentiometer in percent ■ SACT (%) Current spindle load in percent ■ SREF (%) Reference load of the spindle in percent Define the reference load of the spindle in the syntax element LOAD of the FUNCTION AFC CUT BEGIN function. Further information: "NC functions for AFC (#45 / #2-31-1)", Page 259 ■ S (rpm) Spindle shaft speed in rpm ■ SDEV (%) Current deviation of the speed in percent
AFC graph	The AFC graph visualizes the relationship between the elapsed Time [sec] and the Spindle load/Feed-rate override [%]. The green line in the graph shows the feed-rate override and the blue line shows the spindle load.

The COMPMON (#155 / #5-02-1) tab

On the **COMPMON** tab, the control displays information about monitoring defined machine components using the Component Monitoring function (#155 / #5-02-1).

Further information: Programming and Testing User's Manual



Refer to your machine manual.

The machine manufacturer defines the monitoring functions and the contents of the **COMPMON** tab.



The **COMPMON** tab with configured spindle speed monitoring

The machine manufacturer can define up to four ranges and their content on the **COMPMON** tab.

The machine manufacturer can select the following values for each component:

- Status

The control shows the individual status of up to five components and the combined results of all components as a colored bar.

- Green: component is the range that is defined as safe
- Yellow: component in the warning zone
- Red: component is overloaded

If a component cannot be monitored, then the control displays the status as gray. A component cannot be monitored, for example, if configurations are missing or faulty.

- Graph of current monitoring

The control displays the graph either as a resulting value relative to the defined boundaries or as an absolute display of the signal.

- Histogram

The control shows a graphical evaluation of previous monitoring sessions.



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

In the **Program Run** operating mode, the control monitors the components only during machining. If no machining is active, then the control displays the value of the most recent program run.

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

CYC tab

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

Area	Contents
Active cycle definition	When you use the CYCL DEF function to define a cycle, the control shows the cycle number in this area.
Cycle 32 TOLERANCE	■ Status Shows whether Cycle 32 TOLERANCE is active or inactive ■ Values of Cycle 32 TOLERANCE ■ Values from the machine manufacturer for path and angle tolerance, such as predefined machine-specific roughing or finishing filters ■ Values of Cycle 32 limited by Dynamic Collision Monitoring (DCM) TOLERANCE (#140 / #5-03-2) If a value is limited by DCM, then the control displays the text DCM limited following the value.



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

In the optional machine parameter **maxLinearTolerance** (no. 205305) the machine manufacturer defines the maximum permissible linear tolerance.

In the optional machine parameter **maxAngleTolerance** (no. 205303) the machine manufacturer defines the maximum permissible angle tolerance.

If DCM is active, the control restricts the tolerance defined in Cycle **32 TOLERANCE** to these values.

The FN 16 tab

On the **FN 16** tab, the control displays the contents of a file output to the screen with **FN 16: F-PRINT**.

Further information: Programming and Testing User's Manual

Area	Contents
Output	Contents of an output file that was output with FN 16: F-PRINT, such as measured values or texts. To stop the output: ■ Define the SCLR: output path (Screen Clear) ■ Select the Clear button ■ Select the Reset program button ■ Select a new NC program

The LBL tab

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

Further information: Programming and Testing User's Manual

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

The M tab

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

Further information: Programming and Testing User's Manual

Area	Contents
Active M functions	■ Function Active miscellaneous functions, such as M3 ■ Description Descriptive text about the respective miscellaneous function.  Refer to your machine manual. Only the machine manufacturer can create a descriptive text for machine-specific miscellaneous functions.

The PGM tab

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

Area	Contents
Parts counter	Quantity Actual value and nominal value of the parts counter defined with the FUNCTION COUNT function Further information: Programming and Testing User's Manual The Settings icon lets you open the Counter settings window, in which you can check and, if needed, edit the current count and the target value for the counter. You cannot edit these values while the control is executing an NC program.  With the optional machine parameter userPermission (no. 129101), the machine manufacturer defines which counter settings you can change.
Program run time	■ Runtime Run time of the NC program in the format hh:mm:ss ■ Dwell time Countdown of the waiting time in seconds from the following functions: ■ FUNCTION DWELL ■ Cycle 9 DWELL TIME ■ Parameter Q210 DWELL TIME AT TOP ■ Parameter Q211 DWELL TIME AT DEPTH ■ Parameter Q255 DWELL TIME Further information: "Display of the program run time", Page 146
Programs called	Path of the main program as well as called NC programs including the path
Pole/circle center	Programmed axes and values of the circle center point CC
Radius compensation	Programmed tool radius compensation
Program run options	Active breakpoints in connection with the override controller Further information: "Override controller", Page 447

The POS tab

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

Area	Contents
Position display (e.g., Actual reference position (RFACTL))	In this area the control shows the current position of all axes that are present. You can choose between the following views in the position display: ■ Nominal pos. (NOML) ■ Actual pos. (ACT) ■ Nominal reference position (RFNOML) ■ Actual reference position (RFACTL) ■ Servo lag (LAG) ■ Handwheel superimposed (M118) ■ Actual distance to go (ACTDST) ■ Nominal distance to go (REFDST) Further information: "Position displays", Page 148
Feed and Speed	■ Active Feed in mm/min Only in the Status workspace If a feed rate limit is active, the control displays the line in orange. If the feed rate is limited using the F LIMIT button, the control displays LIMIT in square brackets. Further information: "Feed rate limit F LIMIT", Page 345 If the feed rate is limited by means of functional safety (FS), then the control displays the active safety function in brackets. Further information: "Safety functions", Page 463 ■ Active Feed-rate override in % Only in the Status workspace ■ Active Rapid-traverse override in % ■ Active Programmed feed rate in mm/min Only in the Status workspace If M136 is active: active feed rate in mm/rev Further information: Programming and Testing User's Manual ■ Active Spindle speed in rpm ■ Active Spindle override in % Only in the Status workspace ■ Active Miscellaneous function in reference to the spindle, such as M3

Area	Contents
Orientation of the working plane	Spatial angles or axis angles for the active working plane Further information: Programming and Testing User's Manual If axis angles are active, the control displays in this area only the values of the physically present axes. Defined values in the 3-D rotation window Further information: "The 3D ROT selection item", Page 221
Basic transformations	The control displays the values of the current workpiece preset in this area. Further information: "Preset management", Page 209
Active traverse ranges	Active traverse range, such as Limit 1 for traverse range 1 Traverse ranges are machine-specific. If no traverse range is active, then Traverse range not defined is displayed in this area.
Active kinemat.	Name of the active machine kinematics

The POS HR tab

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

Area	Contents
Coordinate system	■ Machine (M-CS) If you use M118 (#21 / #4-02-1), handwheel superimpositioning is always effective in the machine coordinate system M-CS. Further information: Programming and Testing User's Manual
Handwheel superimp.	■ Max. val. Maximum value of the individual axes, programmed in M118 (#21 / #4-02-1) ■ Actl.val. Current superimpositioning

The QPARA tab

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

Further information: Programming and Testing User's Manual

Using the **Parameter list** and **Variable list** windows, you can define which variables the control shows in the areas. Up to 22 variables can be displayed in each area.

Further information: "Defining the contents of the QPARA tab", Page 150

Area	Contents
Q parameter	Shows the values of the selected Q parameters
QL parameter	Shows the values of the selected QL parameters
QR parameter	Shows the values of the selected QR parameters
QS parameter	Shows the contents of the selected QS parameters
Variables	Shows the contents of the selected named parameters

The Tables tab

On the **Tables** tab, the control shows information about the active tables for program run or the simulation.

Area	Contents
Active tables	In this area the control shows the path for the following active tables: ■ Tool table ■ Preset table ■ Datum table ■ Pocket table ■ Touch-probe table (#17 / #1-05-1)

The TRANS tab

On the **TRANS** tab the control shows information about active transformations in the NC program.

Area	Contents
Active datum	■ Path of the selected datum table ■ Row number of the selected datum table ■ DOC Contents of the DOC column of the datum table
Active datum shift	Datum shift that was defined with the TRANS DATUM function Further information: Programming and Testing User's Manual
Mirrored axes	Axes mirrored with either the TRANS MIRROR function or Cycle 8 MIRRORING Further information: Programming and Testing User's Manual Further information: User's Manual for Machining Cycles
Active angle of rotation	Rotation angle defined with either the TRANS ROTATION function or Cycle 10 ROTATION Further information: Programming and Testing User's Manual Further information: User's Manual for Machining Cycles
Orientation of the working plane	Spatial angles or axis angles for the active working plane Further information: Programming and Testing User's Manual
Center of scaling	Center of scaling that was defined with Cycle 26 AXIS-SPECIFIC SCALING Further information: User's Manual for Machining Cycles
Active scaling factors	Scaling factors that were defined for the individual linear axes with the TRANS SCALE function, Cycle 11 SCALING FACTOR or Cycle 26 AXIS-SPECIFIC SCALING Further information: Programming and Testing User's Manual Further information: User's Manual for Machining Cycles
Shift (WPL-CS)	Active shift in the working plane coordinate system WPL-CS by means of the FUNCTION CORRDATA function Further information: Programming and Testing User's Manual
Compensation value table	■ Path of the selected compensation table *.wco ■ Row number of the selected compensation table *.wco ■ Content of the DOC column of the active row Further information: Programming and Testing User's Manual

The TT tab

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

Further information: "Hardware enhancements", Page 81

Area	Contents
TT: tool measurement	■ T Tool number ■ Name Tool name ■ Measuring method Selected measurement method for tool measurement (e.g., Length) ■ Min (mm) When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge. ■ Max (mm) When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge. ■ DYN Rotation (mm) When measuring milling cutters with a rotating spindle, the control shows values in this area.  In the optional machine parameter tippingTolerance (no. 114206) you define the tipping angle tolerance. The control will determine the tipping angle automatically only if a tolerance is defined.
TT: measurement of individual teeth	Number List of the measurements performed and the measured values of the individual cutting edges

The Tool tab

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

Further information: "Tool types", Page 182

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

Definitions

Tipping angle

If a TT tool touch probe with a cuboid contact cannot be clamped to a machine table so that it is level, the angular offset must be compensated for. This offset is the tipping angle.

Angle of misalignment

In order to exactly measure with TT tool touch probes with a cuboid contact, the misalignment on the machine table relative to the main axis must be compensated for. This offset is the angle of misalignment.

5.5 The Simulation status workspace

Application

You can call additional status displays in the **Editor** operating mode in the **Simulation status** workspace. In the **Simulation status** workspace, the control shows data based on the simulation of the NC program.

Description of function

The following tabs are available in the **Simulation status** workspace:

- **Favorites**
Further information: "The Favorites tab", Page 130
- **CYC**
Further information: "CYC tab", Page 135
- **FN 16**
Further information: "The FN 16 tab", Page 135
- **LBL**
Further information: "The LBL tab", Page 136
- **M**
Further information: "The M tab", Page 136
- **PGM**
Further information: "The PGM tab", Page 137
- **POS**
Further information: "The POS tab", Page 138
- **QPARA**
Further information: "The QPARA tab", Page 139
- **Tables**
Further information: "The Tables tab", Page 140
- **TRANS**
Further information: "The TRANS tab", Page 141
- **TT**
Further information: "The TT tab", Page 142
- **Tool**
Further information: "The Tool tab", Page 143

Notes

- The control displays the current counter reading and the defined target quantity only on the **PGM** tab of the **Status** workspace.
Further information: "The PGM tab", Page 137
Further information: Programming and Testing User's Manual
- The following contents are only displayed on the **POS** tab of the **Status** workspace:
 - Active **Feed** in mm/min
 - Active **Feed-rate override** in %
 - Active **Spindle override** in %
 Further information: "The POS tab", Page 138

5.6 Display of the program run time

Application

The control calculates the duration of all traverse movements and displays them together as the **Program run time**. The control takes traversing movements and dwell times into account.

In addition, the control calculates the remaining run time of the NC program.

Description of function

The control displays the program run time in the following areas:

- **PGM** tab of the **Status** workspace
Further information: "The PGM tab", Page 137
- Status overview on the TNC bar
Further information: "Status overview of the TNC bar", Page 127
- **PGM** tab of the **Simulation status** workspace
- The **Simulation** workspace in the **Editor** operating mode
Further information: Programming and Testing User's Manual

Use the **Settings** button in the **Program run time** area to influence the calculated program run time.

The control opens a selection menu with the following functions:

Function	Meaning
Save	Save the current value under Runtime
Addition	Add the saved time to the value under Runtime
Resetting	Reset the saved time and the contents of the Program run time area to zero

The control counts the time during which the **Control-in-operation** symbol is green. The control adds the time from the **Program Run** operating mode and the **MDI** application.

The following functions reset the program run time:

- Selecting a new NC program for program run
- The **Reset program** button
- The **Resetting** function in the **Program run time** area

Remaining run time of the NC program

If a tool usage file is available, the control calculates for the **Program Run** operating mode the duration of executing the active NC program. During program run, the control updates the remaining run time.

Further information: "Tool usage test", Page 193

The control shows the remaining run time in the status overview on the TNC bar.

The control does not take the feed-rate potentiometer setting into account, but calculates with a feed rate of 100%.

The following functions reset the remaining run time:

- Selecting a new NC program for program run
- **Internal stop** button
- Generate new tool usage file

Notes

- In the machine parameter **operatingTimeReset** (no. 200801) the machine manufacturer defines whether the control resets the program run time when the program is started.
- The control cannot simulate the run time of machine-specific functions such as tool changing. That is why this function is only partially suitable for calculating the production time in the **Simulation** workspace.
- In the **Program Run** operating mode, the control displays the exact time of the NC program while taking all machine-specific actions into account.

Definition

Control-in-operation:

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

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

Further information: "Interrupting, stopping or canceling program run", Page 346

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

5.7 Position displays

Application

The control offers various modes in the position display, for example values from different reference systems. You can choose one of the modes available based on the application.

Description of function

The control has position displays in the following areas:

- The **Positions** workspace
- Status overview on the control bar
- The **POS** tab of the **Status** workspace
- The **POS** tab of the **Simulation status** workspace

On the **POS** tab of the **Simulation status** workspace the control always shows the **Nominal pos. (NOML)** mode. In the **Status** and **Positions** workspaces you can choose the mode of the position display.

The control offers the following modes for the position display:

Mode	Meaning
Nominal pos. (NOML)	This mode shows the value of the currently calculated target position in the input coordinate system I-CS. When the machine moves the axes, the control compares the coordinates of the measured actual position with the calculated nominal position in predefined time intervals. The nominal position is the position at which the axes should be located at the time of comparison, based on the calculation.  The Nominal pos. (NOML) and Actual pos. (ACT) modes differ solely with regard to the servo lag.
Actual pos. (ACT)	This mode shows the currently measured tool position in the input coordinate system I-CS. The actual position is the measured position of the axes, as determined by encoders at the time of comparison.
Nominal reference position (RFNOML)	This mode shows the calculated target position in the machine coordinate system M-CS.  The Nominal reference position (RFNOML) and Actual reference position (RFACTL) modes differ solely with regard to the servo lag.
Actual reference position (RFACTL)	This mode shows the currently measured tool position in the machine coordinate system M-CS.
Servo lag (LAG)	This mode shows the difference between the calculated nominal position and the measured actual position. The control determines the difference in predefined time intervals.
Handwheel superimposed (M118)	This mode shows the values that you move using the M118 miscellaneous function. Further information: Programming and Testing User's Manual

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



Refer to your machine manual.

In the machine parameter **progToolCallDL** (no. 124501), the machine manufacturer defines whether the position display takes the delta value **DL** from the tool call into account. The modes **NOML.** and **ACTL.** as well as **RFNOML** and **RFACTL** then differ from each other by the value **DL**.

5.7.1 Switching the position display mode

To switch the position display mode in the **Status** workspace:

- ▶ Select the **POS** tab



- ▶ Select **Settings** in the position display area
- ▶ Select the desired mode for the position display (e.g., **Actual pos. (ACT)**)
- ▶ The control displays the positions in the selected mode.

Notes

- The machine parameter **CfgPosDisplayPace** (no. 101000) defines the display accuracy by the number of decimal places.
- If the axes are moved, then, in addition to the current positions, the control also displays the remaining distances-to-go with a symbol and the respective value.
Exception: The **Actual distance to go (ACTDST)** and **Nominal distance to go (REFDST)** modes

Further information: "Axis display and position display", Page 122

5.8 Defining the contents of the QPARA tab

On the **QPARA** tab of the **Status** and **Simulation status** workspaces, you can define which variables the control will show.

Further information: "The QPARA tab", Page 139

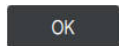
To define the contents of the **QPARA** tab:

- ▶ Select the **QPARA** tab



- ▶ Select the **Settings** in the desired area, such as QL parameters
- > The control opens the **Parameter list** window.

- ▶ Enter the number or name (e.g., **1,3,200-208**)



- ▶ Press **OK**
- > The control displays the values of the defined variables.



- Use commas to separate single variables and connect sequential variables with a hyphen.
- The control always shows eight decimal places on the **QPARA** tab. For example, the control shows the result of **Q1 = COS 89.999** as 0.00001745. Very large and very small values are shown in exponential notation. The control shows the result of **Q1 = COS 89.999 * 0.001** as +1.74532925e-08, with e-08 corresponding to the factor of 10^{-8} .
- If you check the content of a string parameter on the **QPARA** tab of the **Status** workspace, you possibly do not see the complete content.

6

Powering on and off

6.1 Switch-on

Application

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



Refer to your machine manual.

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

Related topics

- Absolute and incremental position encoders

Further information: "Position encoders and reference marks", Page 167

Description of function

DANGER

Caution: hazard to the user!

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

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

Power-on of the control begins with the power supply.

After booting, the control checks the machine status, e.g.:

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

If the control registers an error during or after booting, it issues an error message.

The following step differs depending on position encoders on the machine:

- Absolute position encoders

If the machine has absolute position encoders, the control opens the **Start/Login** application after power-on.

- Incremental position encoders

If the machine has incremental position encoders, you must traverse the reference points in the **Move to ref. point** application. Once all axes have been referenced, the control is in the **Manual operation** application.

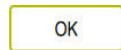
Further information: "The Referencing workspace", Page 155

Further information: "The Manual operation application", Page 160

6.1.1 Powering the machine and the control on

To switch the machine on:

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



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

Further information: "The Referencing workspace",
Page 155



- ▶ Press the **NC Start** key
- > The control moves to all necessary reference points.
- > The control is ready for operation and the **Manual operation** application is open.

Further information: "The Manual operation application",
Page 160



If startup is delayed due to functional safety, the control displays the text **Functional safety requires input**. When you select the **FS** button, the control switches to the **Functional safety** application.

Further information: "The Functional safety application", Page 464

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

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

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

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

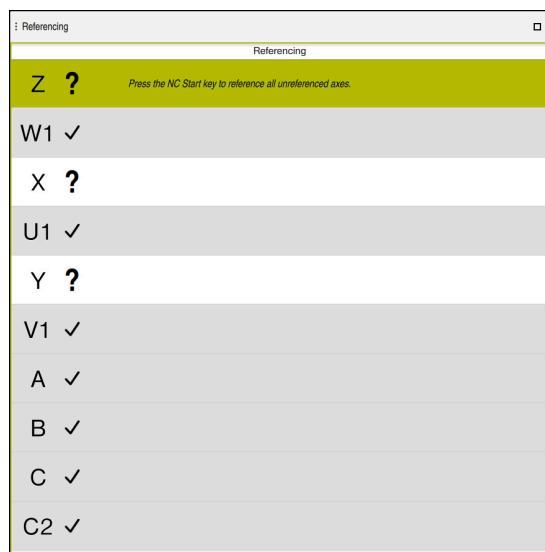
6.2 The Referencing workspace

Application

On machines with incremental linear and angle encoders, the control shows in the **Referencing** workspace which axes need to be referenced.

Description of function

The **Referencing** workspace is always open in the **Move to ref. point** application. If reference points are to be traversed when powering-on the machine, then the control opens this application automatically.



The **Referencing** workspace with axes to be referenced

The control displays a question mark behind all axes that need to be referenced.

Once all axes have been referenced, the control closes the **Move to ref. point** application and switches to the **Manual operation** application.

6.2.1 Axis reference run

To reference the axes in the prescribed sequence:



- ▶ Press the **NC start** key
- > The control moves to the reference points.
- > The control switches to the **Manual operation** application.

To reference the axes in any sequence:



- ▶ Press and hold the axis direction button for each axis until the reference point has been traversed
- > The control switches to the **Manual operation** application.

Notes

NOTICE

Danger of collision!

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

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

- You cannot switch to the **Program Run** operating mode as long as reference points still need to be traversed.
- If you intend only to edit or simulate NC programs, you can switch to the **Editor** operating mode without referencing the axes. You can still traverse the reference points at a later time.

Notes about traversing reference points in a tilted working plane

If the function **Tilt working plane** (#8 / #1-01-1) was active before the control was shut down, then the control automatically activates the function after restarting. This means that movements via the axis keys take place in the tilted working plane.

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

Further information: Programming and Testing User's Manual

6.3 Powering off

Application

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

Description of function

You can shut down the control in the **Start/Login** application of the **Home** operating mode.

If you select the **Shut down** button, the control opens the **Shut down** window. You choose whether to shut down the control or restart it.

If NC programs or contours contain any unsaved changes, the control displays the unsaved changes in the **Close file** window. You can save the changes, discard them, or cancel the shutdown.

6.3.1 Shutting down the control and powering-off the machine

To power-off the machine:



- ▶ Select the **Home** operating mode



- ▶ Select **Shut down**
- ▶ The control opens the **Shut down** window.



- ▶ Select **Shut down**
- ▶ If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
- ▶ If necessary, save unsaved NC programs with **Save** or **Save as**
- ▶ The control shuts down.
- ▶ After completion of the shutdown process, the control displays the text **Now you can switch off.**
- ▶ Switch off the main power switch of the machine

Notes

NOTICE

Caution: Data may be lost!

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

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

- Different machines have different power-off procedures.
Refer to your machine manual.
- Applications that are active on the control might delay the shutdown, such as a connection to **Remote Desktop Manager** (#133 / #3-01-1)
Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522

7

Manual operation

7.1 The Manual operation application

Application

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

Related topics

- Moving the machine axes
Further information: "Moving the machine axes", Page 162
- Incremental jog positioning of machine axes
Further information: "Incremental jog positioning of axes", Page 163

Description of function

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

- Document
- Positions
- Simulation
- Status

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

Button	Meaning
M	Define a miscellaneous function M or use the selection menu to choose one and activate it with the NC start key. Further information: Programming and Testing User's Manual The machine manufacturer uses the optional machine parameter forbidManual (no. 103917) to define which miscellaneous functions are allowed in the Manual operation application and are available in the selection menu.
S	Define the spindle speed S , activate it with the NC start key, and also switch on the spindle. Further information: Programming and Testing User's Manual
F	Define the feed rate F and activate it with the OK button. Further information: Programming and Testing User's Manual
T	Define a tool T or use the selection window to choose one and insert it with the NC start key. Further information: Programming and Testing User's Manual
3D ROT	The control opens a window for the 3D rotation settings (#8 / #1-01-1). Further information: Programming and Testing User's Manual
Active preset	The control opens the preset table in the Active preset window. Further information: "Preset table *.pr", Page 415
Q info	The control opens the Q parameter list window, where you can see and edit the current values and descriptions of the variables. Further information: Programming and Testing User's Manual
Manual cycles	The machine manufacturer can define manual cycles that you can use by means of this button.
DCM	The control opens the Dyna. Coll. Monitoring (DCM) window where you can activate or deactivate Dynamic Collision Monitoring (DCM (#140 / #5-03-2)). Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 228

Button	Meaning
Jog increment	Define the jog increment Further information: "Incremental jog positioning of axes", Page 163
Set the preset	Enter and set a preset Further information: "Preset management", Page 209
Tools	The control opens the Tool management application in the Tables operating mode. Further information: "Tool management ", Page 184
Handwheel	The control displays this toggle switch if a handwheel without display is configured on the control. If the handwheel is active, the operating mode's icon in the sidebar changes. Further information: "Electronic handwheel", Page 429
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. Further information: "Interrupting, stopping or canceling program run", Page 346

7.2 Moving the machine axes

Application

You can use the control to move the machine axes manually, such as pre-positioning for a manual touch probe function.

Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305

Related topics

- Programming traverse movements
Further information: Programming and Testing User's Manual
- Executing traverse movements in the **MDI** application
Further information: "The MDI application ", Page 301

Description of function

The control offers the following methods for moving axes manually:

- Axis-direction keys
- Incremental jog positioning with the **Jog increment** button
- Traversing with electronic handwheels

Further information: "Electronic handwheel", Page 429

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

Further information: "Status displays", Page 119

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

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

Further information: "Status overview of the TNC bar", Page 127

7.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:



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



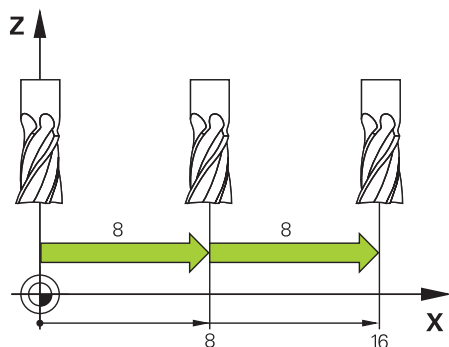
- ▶ Select an application (e.g., **Manual operation**)
- ▶ Press the axis key of the desired axis
- ▶ The control moves the axis as long as you press the key.



If you hold the axis key pressed down and simultaneously press the **NC Start** key, the control moves the axis at a continuous feed rate. You have to end traverse movement with the **NC Stop** key.
You can move more than one axis at a time.

7.2.2 Incremental jog positioning of axes

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



To position an axis incrementally:



- ▶ Select the **Manual** operating mode



- ▶ Select the **Manual operation** application



- ▶ Select **Jog increment**
 - The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.
 - ▶ Enter the jog increment for linear axes and rotary axes

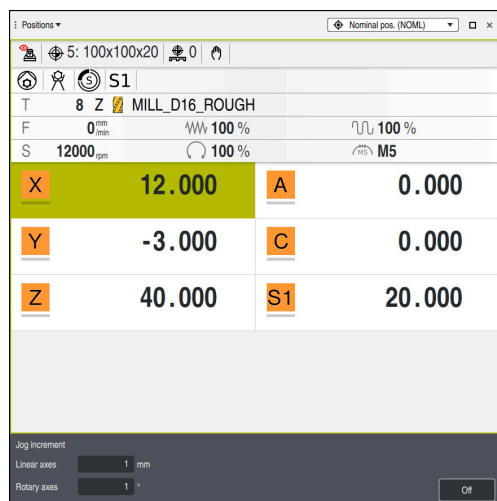


- ▶ Press the axis key of the desired axis
 - The control positions the axis in the selected direction by the defined jog increment.



- ▶ Select **Off**
 - The control ends incremental jog positioning and closes the **Jog increment** area in the **Positions** workspace.

i You can also end incremental positioning with the **Jog inc. on** button.



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

Note

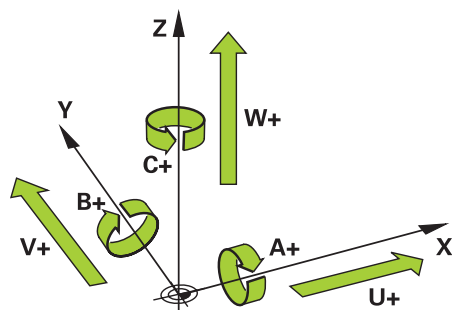
When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.

8

NC fundamentals

8.1 NC fundamentals

8.1.1 Programmable axes



The programmable axes of the control are in accordance with the axis definitions specified in DIN 66217.

The programmable axes are designated as follows:

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



Refer to your machine manual.

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

Your machine manufacturer can define further axes, such as PLC axes.



The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message.

If the axis position does not change, you can nevertheless program more than four axes.

8.1.2 Designation of the axes of milling machines

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

The axes are associated as follows:

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

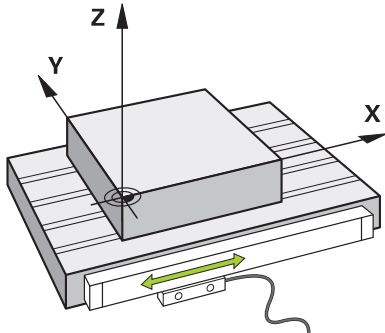


The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

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

8.1.3 Position encoders and reference marks

Fundamentals



The position of the machine axes is ascertained with position encoders. As a rule, linear axes are equipped with linear encoders. Rotary tables and rotary axes feature angle encoders.

The position encoders detect the positions of the tool or machine table by generating an electrical signal during movement of an axis. The control ascertains the position of the axis in the current reference system from this electrical signal.

Further information: "Reference systems", Page 196

Position encoders can measure these positions through different methods:

- Absolutely
- Incrementally

The control cannot determine the position of the axes while the power is interrupted. Absolute and incremental position encoders behave differently once power is restored.

Absolute position encoders

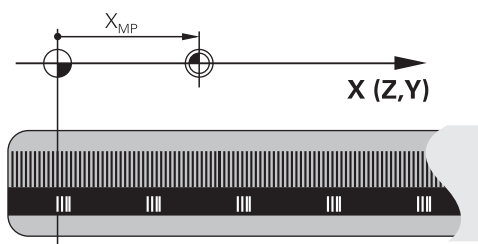
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

Incremental position encoders

Incremental position encoders need to find the distance between the current position and a reference mark in order to determine the actual position. Reference marks indicate a machine-based reference point. A reference mark must be traversed in order to determine the current position after a power interruption.

If the position encoders feature distance-coded reference marks, then you need to move the linear encoders of the axes by no more than 20 mm. On angle encoders this distance is no more than 20°.

Further information: "Axis reference run", Page 155



8.1.4 Presets in the machine

The following table contains an overview of the presets in the machine or on the workpiece.

Related topics

- Presets on the tool

Further information: "Presets on the tool", Page 171

Icon	Preset
	Machine datum The machine datum is a fixed point defined in the machine configuration by the machine manufacturer. The machine datum is the origin of the machine coordinate system M-CS. Further information: "Machine coordinate system M-CS", Page 198 If you program M91 in an NC block, the defined values are referenced to the machine datum. Further information: Programming and Testing User's Manual
 M92-ZP	M92 datum M92-ZP (zero point) The M92 datum is a fixed point defined relative to the machine datum by the machine manufacturer in the machine configuration. The M92 datum is the origin of the M92 coordinate system. If you program M92 in an NC block, the defined values are referenced to the M92 datum. Further information: Programming and Testing User's Manual
	Tool change position The tool change position is a fixed point defined relative to the machine datum by the machine manufacturer in the tool-change macro.
	Reference point The reference point is a fixed point for initializing position encoders. Further information: "Position encoders and reference marks", Page 167 If the machine has incremental position encoders, the axes must traverse the reference point after booting. Further information: "Axis reference run", Page 155
	Workpiece preset With the workpiece preset you define the origin of the workpiece coordinate system W-CS. Further information: "Workpiece coordinate system W-CS", Page 202 The workpiece preset is defined in the active row of the preset table. You determine the workpiece preset with a 3D touch probe, for example. Further information: "Preset management", Page 209 If no transformations are defined, the entries in the NC program refer to the workpiece preset.
	Workpiece datum You define the workpiece datum with transformations in the NC program, for example with TRANS DATUM or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset. If you tilt the working plane (#8 / #1-01-1), the workpiece datum is the point around which the workpiece is rotated.

9

Tools

9.1 Fundamentals

To use all of the control's functions, you must define the tools for the control using real data (e.g., the radius). This increase process reliability.

To add a tool to the machine and then be able to use it, follow the sequence below:

- Clamp the tool in an appropriate tool holder.
- To measure the tool dimensions, starting from the tool carrier preset, measure the tool (e.g., using a tool presetter). The control needs these dimensions for calculating the paths.

Further information: "Tool carrier reference point", Page 171

- Further parameters are needed to completely define the tool. One place to find these parameters is the manufacturer's tool catalog.

Further information: "Tool parameters", Page 175

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

Further information: "Tool management ", Page 184

- As needed, assign a tool carrier to the tool in order to achieve realistic simulation and collision protection.

Further information: "Tool carrier management", Page 186

- After finishing tool definition, program a tool call within an NC program.

Further information: Programming and Testing User's Manual

- If your machine is equipped with a chaotic tool changer system and a double gripper, the tool change time may be shortened by pre-selecting the tool.

Further information: Programming and Testing User's Manual

- If needed, perform a tool usage test before starting the program. This process checks if the tools are available in the machine and have sufficient remaining tool life.

Further information: "Tool usage test", Page 193

- After machining a workpiece and measuring it, you may correct the tools.

Further information: Programming and Testing User's Manual

9.2 Presets on the tool

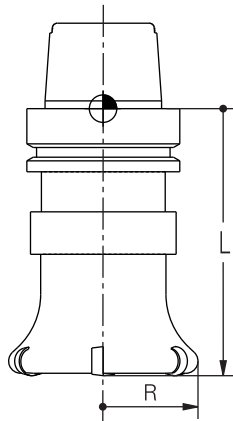
The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

Further information: "Presets in the machine", Page 167

9.2.1 Tool carrier reference point



Milling cutter

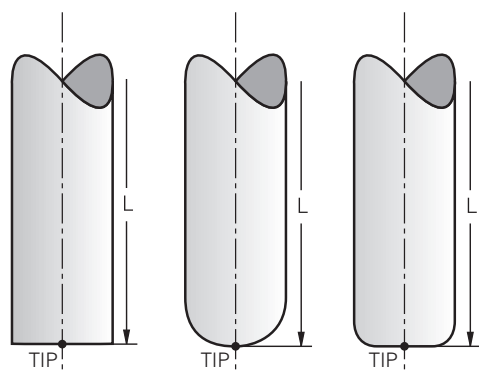
The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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

Further information: "Tool management ", Page 184

Further information: "Measuring the tool by scratching", Page 334

9.2.2 Tool tip TIP



Milling tools

Further information: "Tool coordinate system T-CS", Page 207

You define the position of the tool tip with the basic and delta values of the tool relative to the tool-carrier reference point.

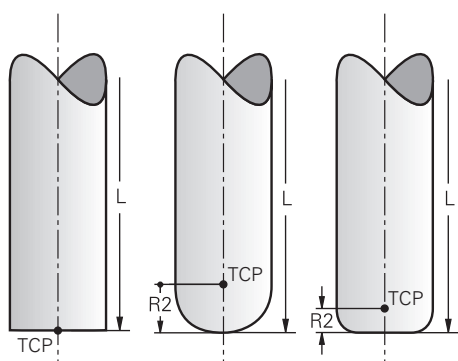
Further information: "Tool parameters", Page 175

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

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

Further information: "Tool location point (TLP, tool location point)", Page 173

9.2.3 Tool center point (TCP, tool center point)



Milling tools

The tool center point TCP is the center of the tool diameter. If a tool radius 2 **R2** is defined, the tool center point is offset from the tool tip by this value.

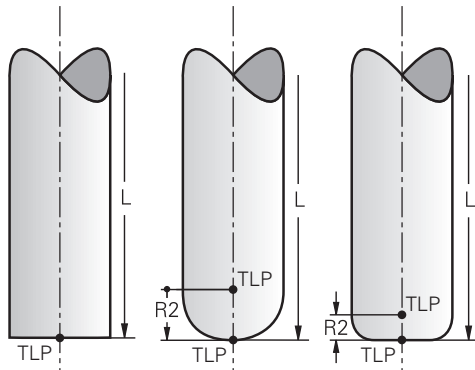
Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: "Tool management ", Page 184

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

Further information: "Tool location point (TLP, tool location point)", Page 173

9.2.4 Tool location point (TLP, tool location point)

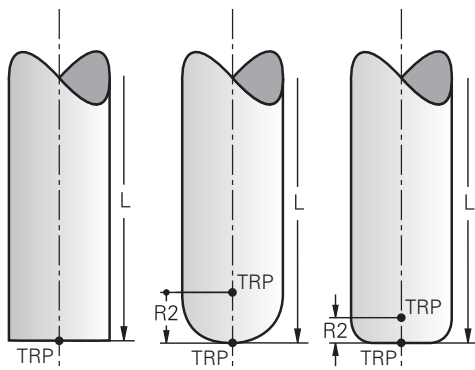


The control positions the tool on the tool location point TLP. By default, the tool location point is at the tool tip.

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

Further information: Programming and Testing User's Manual

9.2.5 Tool rotation point (TRP, tool rotation point)



When applying the tilting function with **MOVE** (#8 / #1-01-1), the control tilts around the tool rotation point TRP. By default, the tool center of rotation is at the tool tip.

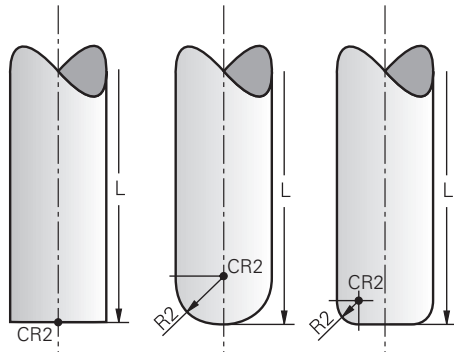
When selecting **MOVE** in **PLANE** functions, the syntax element **DIST** is used to define the relative position between the workpiece and the tool. The control shifts the tool rotation point from the tool tip by this value. When **DIST** is not defined, the control keeps the tool tip constant.

Further information: Programming and Testing User's Manual

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

Further information: Programming and Testing User's Manual

9.2.6 Tool radius 2 center (CR2, center R2)



The control uses the tool radius 2 center in conjunction with 3D tool compensation (#9 / #4-01-1). In the case of straight lines **LN**, the surface-normal vector points to that point and defines the direction of the 3D tool compensation.

Further information: Programming and Testing User's Manual

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

The tool radius 2 center is an auxiliary point for illustration purposes. The coordinates in the NC program reference the tool location point.

Further information: "Tool location point (TLP, tool location point)", Page 173

9.3 Tool parameters

Application

In the tool parameters you give the control all the information it needs to, for example, calculate the contours or run the simulation.

The parameters required depend, for example, on the tool type.

Related topics

- Editing parameters in tool management
Further information: "Tool management ", Page 184
- Tool types
Further information: "Tool types", Page 182
- Tool tables
Further information: "Tool tables", Page 392
- Tool table **tool.t**
Further information: "Parameters of the tool table tool.t", Page 393

Description of function

There are various possibilities for determining the parameters. For example:

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

The **Form** workspace in the **Tables** operating mode assists you when you enter the parameters. In the fillable form, the control filters the parameters in accordance with the selected tool types.

HEIDENHAIN recommends entering all known parameters in order to use the following functions to their full extent:

- Simulation
Further information: Programming and Testing User's Manual
- Machining or touch probe cycles
Further information: User's Manual for Machining Cycles
Further information: User's Manual for Machining Cycles
- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))
Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224

9.3.1 Tool ID number

Application

Each tool has a unique number which equals the row number of the tool management. Each tool ID number is unique.

Further information: "Tool management ", Page 184

Description of function

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

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

Further information: Programming and Testing User's Manual

9.3.2 Tool name

Application

A tool name can be assigned in addition to the tool ID number. Contrary to the tool ID number, a tool name is not unique.

Description of function

The tool name allows identifying tools easier within the tool management. To this end, key features can be defined such as the diameter or the type of machining (e.g., **MILL_D10_ROUGH**).

As tool names are not unique, assign names that clearly identify the tools.

A tool name may contain up to 32 characters.

Permitted characters

You can use the following characters for the tool name:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z 0 1 2 3 4 5 6 7 8 9 # \$ % & , - _ .

When entering lowercase letters, the control replaces them with uppercase letters upon saving.

In conjunction with AFC (#45 / #2-31-1), the following characters are not permitted in the tool name: # \$ % & , .

Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256

Note

- Assign unique tool names!

If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:

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



Refer to your machine manual.

If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

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

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

9.3.3 Database ID

Application

In a tool database for all machines, you can identify tools with unique database IDs (e.g., within one machine shop). This allows you to coordinate the tools of multiple machines more easily.

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

Related topics

- **DB_ID** column of tool management

Further information: "Tool table tool.t", Page 392

Description of function

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

For indexed tools, you can define the database ID either only for the physically existing main tool or as an ID for the data record at each index.

For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool.

Further information: "Indexed tool", Page 178

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

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

9.3.4 Indexed tool

Application

Using an indexed tool, several different parameters can be stored for one physically available tool. This feature enables indication of a certain point on the tool by means of the NC program which does not necessarily have to correspond with the maximum tool length.

Requirement

- Main tool has been defined

Description of function

Tools with multiple lengths and radii cannot be defined in one row of the tool management table. Additional table rows are required, specifying the full definitions of the indexed tools. The lengths of the indexed tools, starting from the maximum tool length, approach the tool carrier preset as the index increases.

Further information: "Tool carrier reference point", Page 171

Further information: "Creating an indexed tool", Page 179

Examples of an application of indexed tools:

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

Creating an indexed tool

To create an indexed tool:



- ▶ Select the **Tables** operating mode



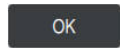
- ▶ Select the **Tool management** application



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



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



- ▶ Press **OK**
 - > The control adds table row **5**.
- ▶ Open the **Form** workspace
- ▶ Define all possible parameters in the form, including the maximum tool length

Further information: "Tool parameters", Page 175



- ▶ Select **Insert tool**
 - > The control opens the **Insert tool** pop-up window.
- ▶ Enable the **Index** check box
 - > The control adds the next free index number for the currently selected tool (e.g., **T5.1**).



- ▶ Press **OK**
 - > The control inserts table row **5.1** with the parameters of the main tool.
- ▶ Correct all deviating parameters in the form

Further information: "Tool parameters", Page 175



The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.

Further information: "Tool carrier reference point", Page 171

Notes

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

Further information: "Tool table tool.t", Page 392

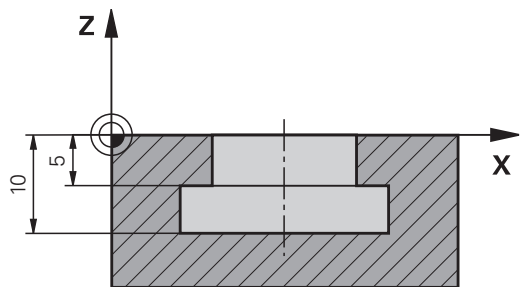
- When you create an indexed tool, the control will copy the parameters from the previous table row. The previous table row can either be the main tool or an existing indexed tool.
- Index numbers do not need to be sequential. It is possible, for example, to create the tools **T5**, **T5.1** and **T5.3**.
- If you delete a main tool, the control will delete all associated indexed tools as well.
- If you copy or cut indexed tools only, you can use **Append** to add the indices to the currently selected tool.

Further information: Programming and Testing User's Manual

- Up to nine indexed tools can be added to each main tool.
- If you define a replacement tool **RT**, this applies to the respective table row exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This means, for example, that the main tool can still be used.

Further information: Programming and Testing User's Manual

Example of T-slot milling cutter



In this example, you program a T-slot with dimensions referring to the top and bottom edges as viewed from the coordinates surface. The height of the T-slot is larger than the length of the cutting edge of the tool used. This requires two steps.

Two tool definitions are required for producing the T-slot.

- The main tool dimension refers to the lower point of the cutting edge, which equals the maximum tool length. This can be used for machining the bottom edge of the T-slot.
- The dimension of the indexed tool refers to the upper point of the cutting edge. This can be used for machining the top edge of the T-slot.



Please ensure that all required parameters are defined both for the main tool and for the indexed tool! In case of a rectangular tool, the radius remains identical in both table rows.

The T-slot is programmed in two machining steps:

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

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

9.3.5 Tool types

Application

The control uses the tool types to filter the parameters that you can edit in tool management.

Related topics

- Editing parameters in tool management
Further information: "Tool management ", Page 184
- Tool tables
Further information: "Tool tables", Page 392

Description of function

A number is additionally assigned to each tool type.

The following tool types can be selected using the **TYP** parameter of the tool management:

Icon	Tool type	Number
	Milling cutter (MILL)	0
	Rough cutter (MILL_R)	9
	Finishing cutter (MILL_F)	10
	Face mill (MILL_FACE)	14
	Ball-nose cutter (BALL)	22
	Toroid cutter (TORUS)	23
	Chamfer mill (MILL_CHAMFER)	24
	Side milling cutter (MILL_SIDE)	25
	Drill (DRILL)	1
	Tap (TAP)	2
	NC center drill (CENT)	4
	CAL_PIN	28
	Touch probe (TCHP) (#17 / #1-05-1)	21
	Reamer (REAM)	3
	Countersink (CSINK)	5
	Piloted counterbore (TSINK)	6
	Boring tool (BOR)	7

Icon	Tool type	Number
	Back boring tool (BCKBOR)	8
	Thread mill (GF)	15
	Thread miller with chamfer (GSF)	16
	Thread mill with single thread (EP)	17
	Thread mill with indexable insert (WSP)	18
	Thread drilling/milling cutter (BGF)	19
	Circular thread milling drill (ZBGF)	20

9.4 Tool management

Application

The control displays the tool definitions of all technologies as well as the tools currently present in the tool magazine in the **Tool management** application of the **Tables** operating mode.

The tool management allows you to add or delete tools, or to edit parameters.

Related topics

- Creating new tools
Further information: "Setting up a tool", Page 109
- Importing and exporting tool data (e.g., from a tool presetter or for the programming station)
Further information: "Importing and exporting of table contents", Page 387
- Table workspace
Further information: "The Table workspace", Page 376
- Form workspace
Further information: "The Form workspace for tables", Page 384

Description of function

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

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

- Tool table **tool.t**
Further information: "Tool table tool.t", Page 392
- Touch-probe table **tchprobe.tp** (#17 / #1-05-1)
Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401

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

Further information: "Pocket table tool_p.tch", Page 406

You can edit the parameters in the **Table** or **Form** workspaces. In the **Form** workspace the control shows the correct parameters for each tool type.

Further information: "Tool parameters", Page 175

Notes

- When creating a new tool, the length **L** and radius **R** parameters are empty at first. The control will not insert a tool whose length and radius are missing and will display an error message.
- If a tool has been saved to the pocket table, you can neither reset the corresponding row in tool management nor delete the tool. The tool must be removed from the magazine first.
- When editing parameters, bear in mind that the current tool may have been entered in the **RT** column as a replacement tool of another tool!
- Make sure to keep the tool table as short and clear as possible so that it does not impair the computing speed of your control. Use a maximum of 10,000 tool entries in tool management. For example, you can delete all unused tool numbers; tool numbers need not be sequential.
- In the **Form** workspace, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.
Further information: "Adding a column in the workspace", Page 386
- If the cursor is within the **Table** workspace and the **Edit** toggle switch is deactivated, a search using the keyboard can be started. The control opens a separate window with an input field and automatically searches for the entered string. If it finds a tool with the entered characters, the control selects this tool. If it finds several tools with this string of characters, you can scroll up and down in the window
- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.
Further information: Programming and Testing User's Manual
- With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters.
- Changing a tool parameter in the pocket table also changes that same parameter in the tool manager.
- In the **Table** workspace of the tool manager, the control displays the additional virtual **MAGAZINE** and **TOOL_LIFE** columns.
Further information: "Virtual columns", Page 383

9.5 Tool carrier management

Application

With tool carrier management, you can assign the 3D model of a tool carrier to a tool.

The tool carrier model will be used for the following functions:

- Representation in the **Simulation** workspace
- Consideration in Dynamic Collision Monitoring (DCM (#140 / #5-03-2))

Related topics

- The **Simulation** workspace
Further information: Programming and Testing User's Manual
- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))
Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
- Adding a tool model to the tool definition (#140 / #5-03-2)
Further information: "Tool model (#140 / #5-03-2)", Page 190
- Validating a 3D model for the tool carrier (#56-61 / #3-02-1*)
Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505

Requirements

- Kinematics description
The machine manufacturer creates the kinematics description.
- The insertion point is defined within the machine's kinematic description.
The machine manufacturer defines the insertion point for the tool carrier.
- Tool carrier model exists
You must save the tool carrier model in the **Toolkinematics** folder.
Path: **TNC:\system\Toolkinematics**
- The tool carrier model has been assigned to the tool
Further information: "Assigning a tool carrier", Page 187

Description of function

The tool carrier model must meet the following requirements:

- Use permitted characters for the file name
Further information: Programming and Testing User's Manual
- Use a supported format
 - CFG file
 - M3D file
 - STL file
 - Max. 20 000 triangles
 - Triangular mesh forms a closed shell**Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288

If you are using CFT or CFX files, you must edit the templates in the **ToolHolderWizard** window.

Further information: "Customizing tool carrier templates with ToolHolderWizard", Page 188

9.5.1 Assigning a tool carrier

To assign a tool carrier to a tool:



- ▶ Select the **Tables** operating mode



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



- ▶ Enable **Edit**

- ▶ If applicable, open the **Form** workspace
- ▶ In the **Additional geometry data** area, select the **KINEMATIC** parameter
- ▶ The control displays the available tool carriers in the **Tool-carrier kinematics** window.
- ▶ Select the desired tool carrier
- ▶ Select **OK**
- ▶ The control assigns the 3D model of the tool carrier to the tool.



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

Notes

- Sample files for tool carrier templates are available on the programming station in the **TNC:\system\Toolkinematics** folder.
- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.
Further information: Programming and Testing User's Manual
- On 3-axis machines with rectangular angle heads, tool carriers of angle heads are advantageous in connection with the tool axes **X** and **Y** because the control takes the dimensions of the angle heads into account.
HEIDENHAIN recommends using **Z** as the tool axis for machining. Using the Adv. Function Set 1 (#8 / #1-01-1) software option, you can tilt the machining plane to the angle of the exchangeable angle heads and thus continue working with the tool axis **Z**.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM (#140 / #5-03-2)). The tool carriers are thereby protected from colliding with fixtures or machine components.
Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

9.6 Customizing tool carrier templates with ToolHolderWizard



Many tool carriers only differ from others in terms of their dimensions, but their geometric shape is identical. HEIDENHAIN provides ready-to-use tool carrier templates for downloading. Tool carrier templates are 3D models with fixed geometries but editable dimensions.

They can be downloaded through the following link:

HEIDENHAIN NC solutions

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

If you would like to use a CFX or CFT file, you need to parameterize the tool carrier template (i.e., to define the required dimensions). The tool carrier templates can be parametrized in the **ToolHolderWizard** window.

Further information: "Parameterizing tool carrier templates", Page 189

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

Icon	Meaning
	Close the application
	Open file
	Switch between wire frame model and solid object view
	Switch between shaded and transparent view
	Show or hide Transformation vectors
	Show or hide Names of collision objects
	Show or hide Test points
	Show or hide Measuring points
	Redo (restore) the initial view
	Orientations (e.g., top view)

9.6.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:



- ▶ Select the **Files** operating mode



- ▶ Open the **TNC:\system\Toolkinematics** folder
- ▶ Double-tap or double-click desired tool carrier template with the ***.cft** extension
- > The control opens the **ToolHolderWizard** window.
- ▶ Define the dimensions in the **Parameter** area
- ▶ Define a name with the ***.cfx** extension in the **Output file** area
- ▶ Select **Generate file**
- > The control shows the message that the tool carrier template was successfully generated and saves the file in the folder **TNC:\system\Toolkinematics**.
- ▶ Select **OK**
- ▶ Select **Close the application**



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

Only use fully parameterized tool carriers and error-free STL or M3D files!

9.7 Tool model (#140 / #5-03-2)

Application

With the tool model, you can add to a tool definition (e.g., for forward or reverse deburring tools).

The tool model will be used in the following functions only:

- Representation in the **Simulation** workspace
- Consideration in Dynamic Collision Monitoring (DCM (#140 / #5-03-2))



The control will not use the tool model for path contours (e.g., for radius compensation or the **FUNCTION TCPM** function).

Related topics

- The **Simulation** workspace
- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))
Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
- Tool carrier management
Further information: "Tool carrier management", Page 186
- Validating 3D models with **OPC UA NC Server** (#56-61 / #3-02-1*)
Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505

Requirements

- Collision Monitoring v2 (#140 / #5-03-2) software option
- The tool has been defined in tool management
Further information: "Tool management ", Page 184
- A suitable tool model exists
 You must save the tool model in the **Toolshapes** folder.
 Path: **TNC:\system\Toolshapes**
Further information: "Tool model requirements", Page 191
- The tool model has been assigned to the tool
Further information: "Assigning a tool model", Page 192

Description of function

You can use the tool model for the following tool types:

- Milling tools
- Drilling tools
- Touch probes

Further information: "Tool types", Page 182

Tool model requirements

General requirements

The tool model must meet the following general requirements:

- Use permitted characters for the file name
Further information: Programming and Testing User's Manual
 - Use a supported format
 - M3D file
 - STL file
 - Max. 20 000 triangles
 - Triangular mesh forms a closed shell
- Further information:** "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288



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

Further information: "Options for fixture files", Page 232

Coordinate system requirements

The coordinate system of the tool model must meet the following requirements:

- The Z axis is the rotary axis of the tool model.
The control will align the tool model parallel to the tool coordinate system **T-CS**.
Further information: "Tool coordinate system T-CS", Page 207
- The coordinate origin of the 3D model must be identical to the measured point of the tool. If you measure the tool at the tool tip, you also need to set the coordinate origin of the 3D model to the tool tip.



If you measured a spherical cutter at the center of the sphere, you need to set the coordinate origin to the center of the sphere as well.

Further information: "Tool tip TIP ", Page 172

Further information: Programming and Testing User's Manual

9.7.1 Assigning a tool model

To assign a tool model to a tool:



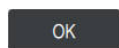
- ▶ Select the **Tables** operating mode



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



- ▶ Activate **Edit**
- ▶ If applicable, open the **Form** workspace
- ▶ In the **Additional geometry data** panel, select the **TSHAPE** parameter
- ▶ The control displays the available tool models in the **3D tool model** window.
- ▶ Select the desired tool model
- ▶ Select **OK**
- ▶ The control assigns the tool model to the tool.



The tool model will be taken into account only after the next tool call.

Notes

- The control will always take an assigned tool model into account (e.g., for the tool radius **R=0**). The simulation shows the correct shape of the tool model (e.g., in conjunction with a CAM output (center path)).
- When you delete a tool, make sure to remove the tool model from the **Toolshapes** folder as well. This way, you can avoid that the tool model is accidentally referenced for another tool.
- The **LCUTS** column of the tool table is independent of the datum of the tool model. The value is measured from the tool tip of the tool and is effective in the positive Z axis direction.

Further information: "Tool table tool.t", Page 392

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

9.8 Tool usage test

Application

The tool usage test allows checking the tools used in the NC program before starting the program. The control checks if the tools used are available in the machine magazine and have sufficient remaining tool life. Any missing tools can be stored in the machine or tools can be exchanged due to insufficient remaining tool life before starting the program. This avoids interruptions while the program is running.

Related topics

- Contents of the tool usage file
Further information: "Tool usage file", Page 409
- Tool usage test in Batch Process Manager (#154 / #2-05-1)
Further information: Programming and Testing User's Manual

Requirements

- To perform a tool usage test, you need a tool usage file
In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.
Further information: "Tool usage file", Page 409
- The **Generate tool-usage file** setting is set to **Once** or **Always**
Further information: "The Machine and Simulation areas", Page 475
- Use the same tool table for the simulation as for program run
Further information: Programming and Testing User's Manual

Description of function

Creating the tool usage file

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

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

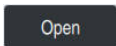
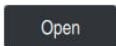
- Simulating the NC program completely
- Executing the NC program completely
- Select the **Refresh** icon in the **Tool usage** area of the **Tests** column
Further information: Programming and Testing User's Manual

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

Further information: "Tool usage file", Page 409

9.8.1 Performing the tool usage test

To perform a tool usage test:

-  ▶ Select the **Editor** operating mode
-  ▶ Select **Add**
-  ▶ Select the desired NC program
-  ▶ Select **Open**
-  ▶ The control opens the NC program in a new tab.
-  ▶ Open the **Tests** column
-  ▶ In the **Tool usage** area, select **Refresh**
-  ▶ The control generates a tool usage file and displays the tools used in the **Tool usage** area.
Further information: "Tool usage file", Page 409
-  ▶ In the **Tool check** area, select **Refresh**
-  ▶ The control performs the tool usage test.
-  ▶ The **Tool check** area shows whether all tools are available and have sufficient remaining tool life.

Notes

- If you double-tap or double-click a tool entry in the **Tool usage** or **Tool check** areas, the control switches to the tool selected in tool management. You can make modifications as needed.
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.
Further information: Programming and Testing User's Manual
- The control saves the tool usage file as a dependent file (*.dep).
Further information: "Tool usage file", Page 409
- In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.
Further information: Programming and Testing User's Manual
- The control displays the order of tool calls of the currently running NC program in the **T usage order** (#93 / #2-03-1) table.
Further information: "T usage order (#93 / #2-03-1)", Page 411
- An overview of all tool calls of the NC program active while executing the program is displayed by the control in the **Tooling list** table (#93 / #2-03-1).
Further information: "Tooling list (#93 / #2-03-1)", Page 413
- The function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- The function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. After **IDX** you define the pallet table row.
- The machine manufacturer uses the machine parameter **autoCheckPrg** (no. 129801) to define whether the control automatically generates a tool usage file upon selecting an NC program.
- The machine manufacturer uses the machine parameter **autoCheckPal** (no. 129802) to define whether the control automatically generates a tool usage file upon selecting a pallet table.

10

**Coordinate
transformation**

10.1 Reference systems

10.1.1 Overview

A control requires unambiguous coordinates in order to move an axis to a defined position correctly. For coordinates to be unambiguous, they not only require the values but also a reference system in which these values are valid.

The control differentiates between the following reference systems:

Abbrevia- tion	Meaning	Further information
M-CS	Machine coordinate system machine coordinate system	Page 198
B-CS	Basic coordinate system basic coordinate system	Page 200
W-CS	Workpiece coordinate system workpiece coordinate system	Page 202
WPL-CS	Working plane coordinate system working plane coordinate system	Page 204
I-CS	Input coordinate system input coordinate system	Page 206
T-CS	Tool coordinate system tool coordinate system	Page 207

The control uses different reference systems for different purposes. For example, this makes it possible to always exchange tools at the exact same position while maintaining the possibility of adapting an NC program to the workpiece position.

The reference systems build upon each other. The machine coordinate system **M-CS** is the fundamental reference system. The position and orientation of the following reference systems are determined by transformations of the M-CS.

Definition

Transformations

Translatory transformations each enable a shift along a number line. Rotatory transformations enable a rotation around a point.

10.1.2 Basics of coordinate systems

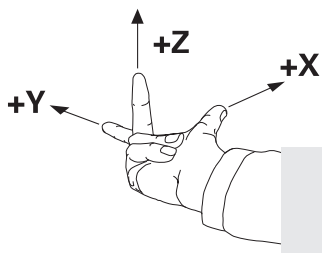
Types of coordinate systems

For coordinates to be unambiguous they must define one point in all axes of the coordinate system:

Axes	Function
One	In a one-dimensional coordinate system, one coordinate defines one point on a number line. Example: on a machine tool, a linear encoder represents a number line.
Two	In a two-dimensional coordinate system, two coordinates define one point in a plane.
Three	In a three-dimensional coordinate system, three coordinates define one point in space.

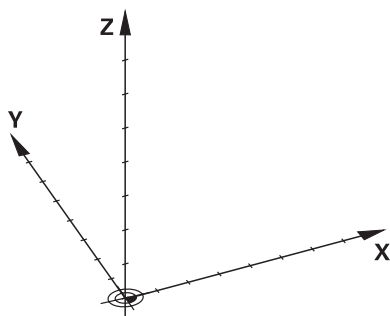
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

Using the right-hand rule you can recreate a three-dimensional Cartesian coordinate system. The fingertips point in the positive directions of the three axes.



Origin of the coordinate system

Unambiguous coordinates require a defined reference point to which the values refer, starting from zero. This point is the coordinate origin, which lies at the intersection of the axes for all three-dimensional Cartesian coordinate systems of the control. The coordinate origin has the coordinates **X+0**, **Y+0**, and **Z+0**.



10.1.3 Machine coordinate system M-CS

Application

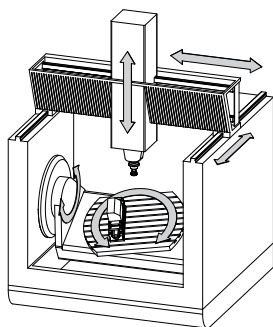
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

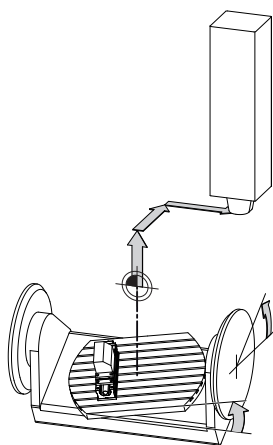
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate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

The machine manufacturer defines the position and orientation of the one-dimensional coordinate systems in the kinematics description.



The machine datum is the coordinate origin of the **M-CS**. The machine manufacturer defines the machine datum in the machine configuration.

The values in the machine configuration define the zero positions of the position encoders and the corresponding machine axes. The machine datum does not necessarily have to be located in the theoretical intersection of the physical axes. It can also be located outside of the traverse range.



Position of the machine datum in the machine

Transformations in the machine coordinate system M-CS

The following transformations can be defined in the **M-CS** machine coordinate system:

- Axis-specific shifts in the **OFFS** columns of the preset table

Further information: "Preset table *.pr", Page 415



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

- Axis-specific shifts in the rotary and parallel axes using the datum table

Further information: Programming and Testing User's Manual

- Axis-specific shifts in the rotary and parallel axes using the **TRANS DATUM** function

Further information: Programming and Testing User's Manual



The machine manufacturer can also define further transformations.

Further information: "Note", Page 199

Position display

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

- **Nominal reference position (RFNOML)**
- **Actual reference position (RFACTL)**
- **Nominal distance to go (REFDST)**

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

Programming coordinate entry in machine coordinate system M-CS

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

Further information: Programming and Testing User's Manual

Note

The machine manufacturer can define the following further transformations in the machine coordinate system **M-CS**:

- Additive axis shifts for parallel axes with the **OEM-offset**
- Axis-specific shifts in the **OFFS** columns of the pallet preset table

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

Example

This example illustrates the difference between traverse movements with and without **M91**. The example shows the behavior with a Y axis as oblique axis that is not arranged perpendicularly to the ZX plane.

Traverse movement without M91

```
11 L IY+10
```

You use the Cartesian input coordinate system **I-CS** for programming. The **ACTL.** and **NOML.** modes of the position display show only a movement of the Y axis in the **I-CS**.

The control uses the defined values to determine the required traverse paths of the machine axes. Since the machine axes are not arranged perpendicularly to each other, the control moves the axes **Y** and **Z**.

Since the machine coordinate system **M-CS** is a projection of the machine axes, the **RFACTL** and **RFNOML** modes of the position display show movements of the Y axis and Z axis in the **M-CS**.

Traverse movement with M91

```
11 L IY+10 M91
```

The control moves the machine axis **Y** by 10 mm. The **RFACTL** and **RFNOML** modes of the position display show only a movement of the Y axis in the **M-CS**.

In contrast to the **M-CS**, the **I-CS** is a Cartesian coordinate system; the axes of the two reference systems do not coincide. The **ACTL.** and **NOML.** modes of the position display show movements of the Y axis and Z axis in the **I-CS**.

10.1.4 Basic coordinate system B-CS

Application

In the basic coordinate system **B-CS** you define the position and orientation of the workpiece. You determine these values by using a 3D touch probe, for example. The control saves the values in the preset table.

Description of function

Properties of the basic coordinate system B-CS

The basic coordinate system **B-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the end of the kinematics description.

The machine manufacturer defines the coordinate origin and orientation of the **B-CS**.

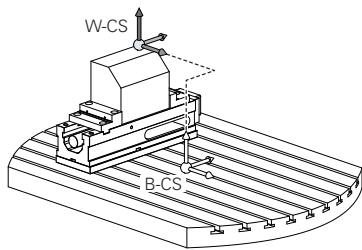
Transformations in the basic coordinate system B-CS

The following columns of the preset table have an effect in the basic coordinate system **B-CS**:

- X
- Y
- Z
- SPA
- SPB
- SPC

You determine the position and orientation of the workpiece coordinate system **W-CS** by using a 3D touch probe, for example. The control saves the determined values as basic transformations in the **B-CS** in the preset table.

Further information: "Preset management", Page 209



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

Further information: "Note", Page 201

Note

The machine manufacturer can define additional basic transformations in the pallet preset table.

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

10.1.5 Workpiece coordinate system W-CS

Application

In the workpiece coordinate system **W-CS** you define the position and orientation of the working plane. You do this by programming transformations and tilting the working plane.

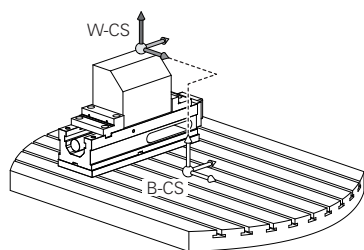
Description of function

Properties of the workpiece coordinate system W-CS

The workpiece coordinate system **W-CS** is a three-dimensional Cartesian coordinate system. Its coordinate origin is the active workpiece preset from the preset table.

Both the position and orientation of the **W-CS** are defined by basic transformations in the preset table.

Further information: "Preset management", Page 209



Transformations in the workpiece coordinate system (W-CS)

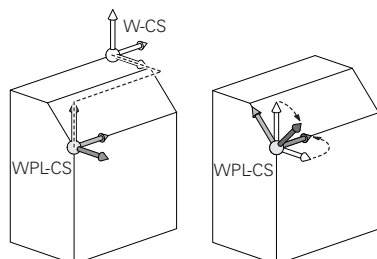
HEIDENHAIN recommends using the following transformations in the workpiece coordinate system **W-CS**:

- Axes **X, Y, Z** of the **TRANS DATUM** function before tilting the working plane
Further information: Programming and Testing User's Manual
- Columns **X, Y, Z** of the datum table before tilting the working plane
Further information: Programming and Testing User's Manual
- The **TRANS MIRROR** function or Cycle **8 MIRRORING** before tilting the working plane with spatial angles
Further information: Programming and Testing User's Manual
Further information: User's Manual for Machining Cycles
- **PLANE** functions for tilting the working plane (#8 / #1-01-1)
Further information: Programming and Testing User's Manual



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

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



NOTICE**Danger of collision!**

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

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



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

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

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

Further information: "Difference between spatial angles and axis angles", Page 216

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.
Further information: "Input coordinate system I-CS", Page 206
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.
Further information: "Working plane coordinate system WPL-CS", Page 204
- The result of transformations built upon each other depends on the programming sequence.

10.1.6 Working plane coordinate system WPL-CS

Application

In the working plane coordinate system **WPL-CS** you define the position and orientation of the input coordinate system **I-CS** and therefore the reference for the coordinate system in the NC program. You do this by programming transformations after having tilted the working plane.

Further information: "Input coordinate system I-CS", Page 206

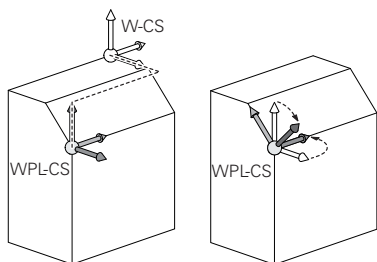
Description of function

Properties of the working plane coordinate system WPL-CS

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

Further information: "Workpiece coordinate system W-CS", Page 202

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

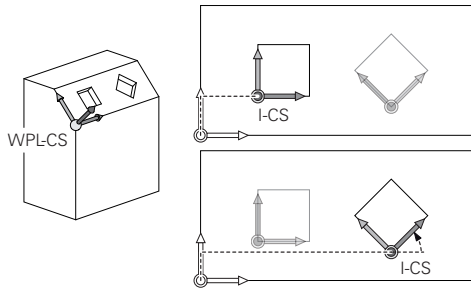


Transformations in the working plane coordinate system WPL-CS

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

- Axes **X, Y, Z** of the **TRANS DATUM** function
Further information: Programming and Testing User's Manual
- The **TRANS MIRROR** function or Cycle **8 MIRRORING**
Further information: Programming and Testing User's Manual
Further information: User's Manual for Machining Cycles
- The **TRANS ROTATION** function or cycle **10 ROTATION**
Further information: Programming and Testing User's Manual
Further information: User's Manual for Machining Cycles
- The **TRANS SCALE** function or cycle **11 SCALING FACTOR**
Further information: Programming and Testing User's Manual
Further information: User's Manual for Machining Cycles
- Cycle **26 AXIS-SPECIFIC SCALING**
Further information: User's Manual for Machining Cycles
- The **PLANE RELATIV** function (#8 / #1-01-1)
Further information: Programming and Testing User's Manual

With these transformations you modify the position and orientation of the input coordinate system **I-CS**.



NOTICE

Danger of collision!

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

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

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.
Further information: "Input coordinate system I-CS", Page 206
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.
- The result of transformations built upon each other depends on the programming sequence.
- As a **PLANE** function (#8 / #1-01-1), **PLANE RELATIV** is in effect in the workpiece coordinate system **W-CS** and orients the working plane coordinate system **WPL-CS**. The values of additive tilting always relate to the current **WPL-CS**.

10.1.7 Input coordinate system I-CS

Application

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

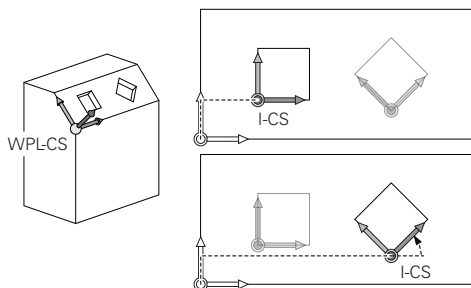
Description of function

Properties of the input coordinate system I-CS

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

Further information: "Working plane coordinate system WPL-CS", Page 204

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



Positioning blocks in the input coordinate system I-CS

In the input coordinate system **I-CS** you use positioning blocks to define the position of the tool. The position of the tool defines the position of the tool coordinate system **T-CS**.

Further information: "Tool coordinate system T-CS", Page 207

You can define the following positioning blocks:

- Paraxial positioning blocks
- Path functions with Cartesian or polar coordinates
- Straight lines **LN** with Cartesian coordinates and surface normal vectors (#9 / #4-01-1)
- Cycles

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

Position display

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

- **Nominal pos. (NOML)**
- **Actual pos. (ACT)**
- **Actual distance to go (ACTDST)**

Notes

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

Further information: "Working plane coordinate system WPL-CS", Page 204

10.1.8 Tool coordinate system T-CS

Application

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

Description of function

Properties of the tool coordinate system T-CS

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

By making entries in the tool management, you can define the tool tip relative to the tool carrier reference point. The machine manufacturer usually defines the tool carrier reference point on the spindle tip.

Further information: "Presets in the machine", Page 167

Use the following tool management parameters to define the tool tip relative to the tool carrier reference point:

- **L**
- **DL**

Further information: "Tool carrier reference point", Page 171

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

Further information: "Input coordinate system I-CS", Page 206

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

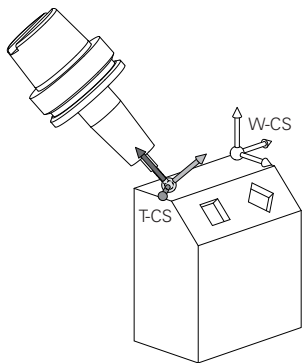
Further information: Programming and Testing User's Manual

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

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

- M function **M128** (#9 / #4-01-1)
- Function **FUNCTION TCPM** (#9 / #4-01-1)

Further information: Programming and Testing User's Manual



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

Further information: Programming and Testing User's Manual

11 L X+10 Y+45 A+10 C+0 R0 M128	; Straight line with miscellaneous function M128 and axis angles

You can also define a tool inclination in the working plane coordinate system **WPL-CS** using spatial angles (e.g., with the **FUNCTION TCPM** function or a straight line **LN**).

11 FUNCTION TCPM F TCP AXIS SPAT PATHCTRL AXIS	; FUNCTION TCPM with spatial angles
12 L A+0 B+45 C+0 R0 F2500	
11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 TX0 TY-0.34090025 TZ0.93600126 R0 M128	; Straight line LN with surface normal vector and tool orientation

Transformations in the tool coordinate system T-CS

The following tool compensations have an effect in the tool coordinate system **T-CS**:

- Compensation values from the tool management
Further information: Programming and Testing User's Manual
- Compensation values from the tool call
Further information: Programming and Testing User's Manual
- Values of the compensation tables ***.tco**
Further information: Programming and Testing User's Manual
- 3D tool compensation with surface normal vectors (#9 / #4-01-1)
Further information: Programming and Testing User's Manual

10.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include, for example, the position and misalignment of a workpiece in the preset table. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 167

Use the preset management in the following cases:

- To tilt the working plane of a machine with table or head rotation axes (#8 / #1-01-1)
- To work on a machine with a head change system
- To machine several workpieces that are clamped at different misaligned positions
- If REF-based datum tables were used on previous control models

Related topics

- Contents of preset table, write protection

Further information: "Preset table *.pr", Page 415

Description of function

Setting presets

Presets can be set in the following ways:

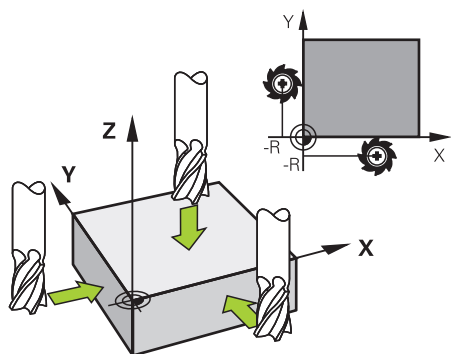
- Setting axis positions manually
Further information: "Setting a preset manually", Page 212
- Touch probe cycles in the **Setup** application
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
- Touch probe cycles in the NC program (#17 / #1-05-1)

If you try to write a value in a write-protected preset table row, the control cancels this process with an error message. Write-protection for this row must be rescinded first.

Further information: "Removing write protection", Page 420

Setting a preset with milling cutters

If no workpiece touch probe is available, the preset can also be set by using a milling cutter. In this case, the values are not obtained by probing, but by scratching.



When scratching with a milling cutter, the tool is slowly moved to the workpiece edge in the **Manual operation** application while the spindle is rotating.

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

Further information: "Setting a preset manually", Page 212

Activating presets

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

Presets can be activated in the following ways:

- Activating manually in the **Tables** operating mode
Further information: "Activating a preset manually", Page 213
- Cycle **247 PRESETTING**
Further information: User's Manual for Machining Cycles
- **PRESET SELECT** function
Further information: Programming and Testing User's Manual

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

- Datum shift with the **TRANS DATUM** function
- Mirroring with the **TRANS MIRROR** function or Cycle **8 MIRRORING**
- Rotation with the **TRANS ROTATION** function or cycle **10 ROTATION**
- Scaling with the **TRANS SCALE** function or cycle **11 SCALING FACTOR**
- Axis-specific scaling with Cycle **26 AXIS-SPECIFIC SCALING**

Tilting the working plane by using **PLANE** functions or Cycle **19 WORKING PLANE** will not be reset by the control.

Basic rotation and 3D basic rotation

The **SPA**, **SPB** and **SPC** columns define a spatial angle for orienting the workpiece coordinate system **W-CS**. This spatial angle defines the basic rotation or 3D basic rotation of the preset.

Further information: "Workpiece coordinate system W-CS", Page 202

When a rotation around the tool axis is defined, the preset contains a basic rotation (e.g., **SPC** for tool axis **Z**). If one of the remaining columns is defined, the preset contains a 3D basic rotation. If the workpiece preset contains a basic rotation or 3D basic rotation, the control takes these values into account when executing an NC program.

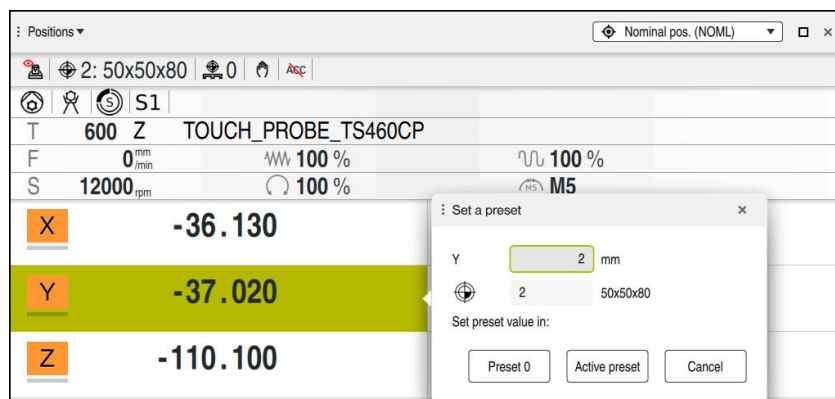
You can use the **3D ROT** (#8 / #1-01-1) button to define whether the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application.

Further information: Programming and Testing User's Manual

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

Further information: Programming and Testing User's Manual

10.2.1 Setting a preset manually



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

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

To set a preset manually in an axis:



- ▶ Select the **Manual operation** application in the **Manual** operating mode
- ▶ Open the **Positions** workspace
- ▶ Traverse the tool to the desired position (e.g., for scratching)
- ▶ Select the row of the desired axis
- ▶ Select **Set the preset**
 - The control opens the **Set a preset** window.
 - ▶ Enter the value of the current axis position, relating to the new preset (e.g., 0)
 - The control activates the **Preset 0** and **Active preset** buttons for selection.
- ▶ Select an option (e.g., **Active preset**)
 - The control saves the value in the selected preset table row and closes the **Set a preset** window.
 - The control updates the values in the **Positions** workspace.

Set the preset

Active preset



- When selecting **Preset 0**, the control automatically activates row 0 of the preset table as the workpiece preset.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

10.2.2 Activating a preset manually

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

To activate a preset manually:



- ▶ Select the **Tables** operating mode



- ▶ Select the **Presets** application
- ▶ Select the desired row

Activate
the preset

- ▶ Select **Activate the preset**
- > The control activates the preset.
- > The control displays the number and comment of the active preset in the **Positions** workspace and in the status overview.

Further information: "Description of function", Page 121

Further information: "Status overview of the TNC bar", Page 127

Notes

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.

Further information: Programming and Testing User's Manual

In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.

- When scratching a workpiece with the radius of a milling cutter, the radius value must be taken into account in the preset.
- Even if the current preset contains a basic rotation or a 3D basic rotation, the **PLANE RESET** function will position the rotary axes at 0° in the **MDI** application.

Further information: "The MDI application", Page 301

- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

Further information: Programming and Testing User's Manual

10.3 Tilting the working plane (#8 / #1-01-1)

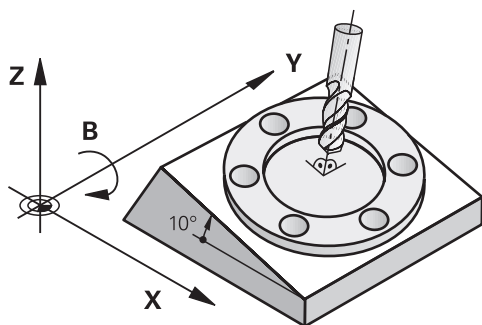
10.3.1 Fundamentals

Machines with rotary axes allow machining of, for example, several workpiece sides after one clamping process, by tilting the working plane. The tilting functions also allow aligning a workpiece clamped at an incorrect angle.

The working plane can be tilted only when tool axis **Z** is active.

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

Further information: "Working plane coordinate system WPL-CS", Page 204



Two functions are available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 218

- Tilting under program control with the **PLANE** functions in the NC program

Further information: Programming and Testing User's Manual



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

Notes concerning different machine kinematics

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

Further information: "Basic coordinate system B-CS", Page 200

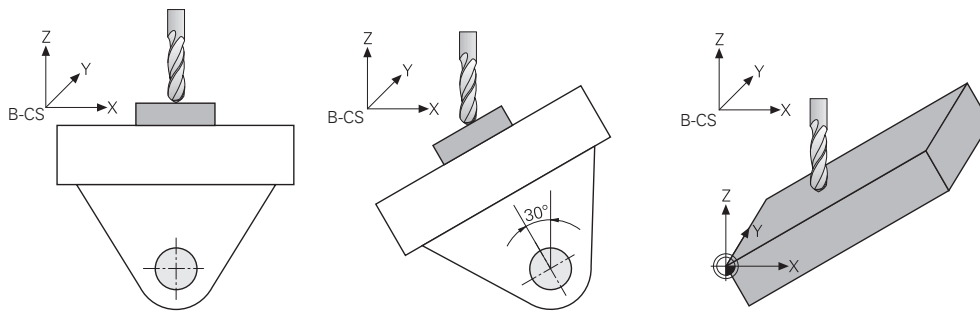
When tilting the working plane, the control moves the machine axes according to the kinematics.

Please observe the aspects below regarding the machine kinematics:

■ Machine with table rotary axes

With this kinematic model, the table rotary axes execute the tilting movement and the position of the workpiece in the work envelope changes. The linear machine axes move in the tilted working plane coordinate system **WPL-CS** just as they do in the non-tilted **B-CS**.

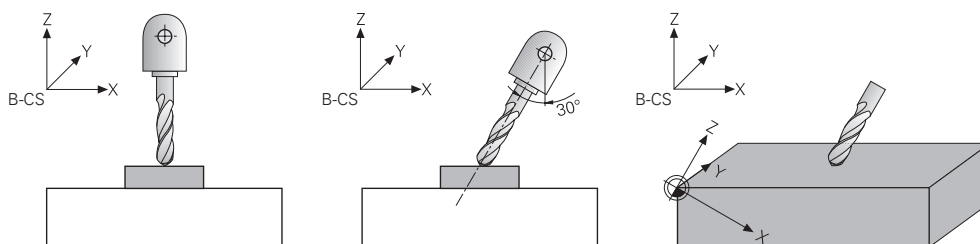
Further information: "Working plane coordinate system WPL-CS", Page 204



■ Machine with head rotary axes

With this kinematic model, the head rotary axes execute the tilting movement and the position of the workpiece in the work envelope remains the same. In the tilted **WPL-CS**, at least two linear machine axes no longer move in parallel with the non-tilted **B-CS**, depending on the rotary angle.

Further information: "Working plane coordinate system WPL-CS", Page 204



Difference between spatial angles and axis angles

Spatial angles

Using spatial angles, you can define the angle of the tool relative to the workpiece. While programming, there is no need to distinguish between head and table axes; often, the angles can be taken over directly from the drawing.

i When you program using spatial angles, the machine kinematics need not be taken into account. This means that you can program as if only the tool was moving, just like in the **Workpiece** mode in the simulation.

The control takes care of calculating the required axis positions. This means that NC programs with spatial angles can also be used for other machines that might have other rotary axes.

The control might approach a defined spatial angle using different axis angles (e.g., +90° or -270°). The solution selected by the control may vary, depending on the machine. You can predefine a solution (e.g., by pre-positioning or by defining **SYM**).

When tilting with spatial angles, the control considers an active basic rotation or 3D basic rotation.

The control performs calculations with spatial angles in the following NC functions:

- All **PLANE** functions except **PLANE AXIAL**
- Straight line **LN** with vectors (#9 / #4-01-1)
- **FUNCTION TCPM** with **AXIS SPAT** (#9 / #4-01-1) selected

Further information: Programming and Testing User's Manual

i HEIDENHAIN recommends using spatial angles because of their greater flexibility.

Axis angles

Using axis angles, you can define an unambiguous position for a rotary axis. You can configure only the axes that are actually present on the machine. When programming with axis angles, you always need to take into account whether the rotary axis is arranged in the head or in the table. The programmed positions must be located in the traverse range of the machine.

If you program axis angles, the control cannot account for the basic rotation or 3D basic rotation in its calculation. Use offsets to orient the workpiece.

Further information: "Comparison of offset and 3D basic rotation", Page 337

NC programs with axis angles can only be used for other machines that have the same rotary axes and matching traverse ranges.

Program axis angles in the following NC functions:

- **PLANE AXIAL**
- M function **M128** (#9 / #4-01-1)
- **FUNCTION TCPM** with **AXIS POS** (#9 / #4-01-1) selected

Further information: Programming and Testing User's Manual

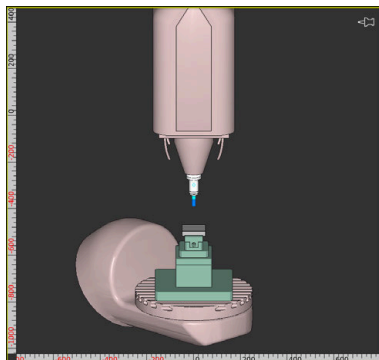
i Even if **M128** or **FUNCTION TCPM** is active with **AXIS POS**, the control will use spatial angles (#9 / #4-01-1) anyway when calculating straight lines **LN** with vectors.

Example: Spatial angles vs. axis angles

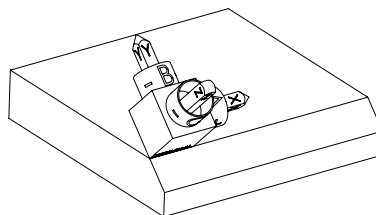
The following example illustrates the difference between spatial angles and axis angles for tilted machining.

For machining, a machine with the table rotary axes **B** and **C** is used. The B axis is not arranged perpendicularly, but at a 45° angle in the left rear corner of the machine.

To machine a 45° chamfer at the front edge of the workpiece, program a spatial angle with **PLANE SPATIAL**.



Machine kinematics (not tilted)



Workpiece with 45° chamfer

To mill a chamfer at the front edge, define the spatial angle **SPA+45**. The control calculates the required axis positions and rotates the B and C axes.

**11 PLANE SPATIAL SPA+45 SPB+0 SPC+0 MOVE
FMAX**

; Tilt the working plane using a spatial angle

Positions			
Nominal pos. (NOML)			
5: 100x100x20 0			
S1			
T	7	Z	MILL_D14_ROUGH
F	0 mm/min	100 %	100 %
S	5000 rpm	100 %	M3
X	115.000	B	-55.702
Y	-5.000	C	28.140
Z	0.000		

Position of rotary axes for **SPA+45**

In the **Positions** workspace, the control shows the positions for the **B** and **C** axes. If you program using axis angles, make sure to calculate and enter these axis positions.

10.3.2 The 3-D rotation window (#8 / #1-01-1)

Application

The **3-D rotation** window allows activating and deactivating tilting of the working plane for the **Manual** and **Program Run** operating modes. This allows restoring the tilted working plane and retracting the tool (e.g., after program cancellation in the **Manual operation** application).

Related topics

- Tilting the working plane in the NC program
Further information: Programming and Testing User's Manual
- Reference systems of the control
Further information: "Reference systems", Page 196

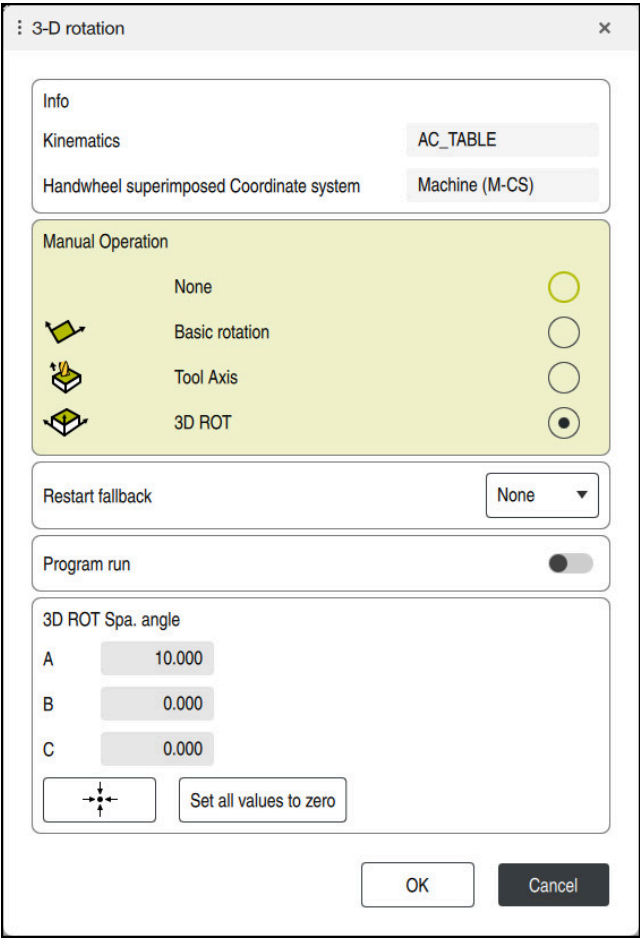
Requirements

- Machine with rotary axes
- Kinematics description
To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Software option Adv. Function Set 1 (#8 / #1-01-1)
- Function enabled by the machine manufacturer
In the machine parameter **rotateWorkPlane** (no. 201201), the machine manufacturer defines whether tilting the working plane is allowed on the machine.
- Tool with tool axis **Z**

Description of function

The **3-D rotation** window can be opened with the **3D ROT** button in the **Manual operation** application.

Further information: "The Manual operation application", Page 160



The **3-D rotation** window

The **3-D rotation** window contains the following information:

Area	Contents
Info	Information about the machine: ■ Name of the active machine kinematics ■ Coordinate system in which handwheel superimpositioning is active Further information: "Reference systems", Page 196 Further information: Programming and Testing User's Manual

Area	Contents
Manual Operation	Effect of the tilting function in the Manual operating mode: ■ None The control will ignore rotary axis positions not equal to 0 as well as columns SPA, SPB, and SPC of the preset table. Traverses are effective in the W-CS workpiece coordinate system. Further information: "Workpiece coordinate system W-CS", Page 202 ■ Basic rotation The control takes the columns SPA, SPB and SPC into account, but no rotary axis positions that are not equal to 0. Traverses take place in the W-CS workpiece coordinate system. Further information: "The Basic rotation selection item", Page 221 ■ Tool axis This is relevant only for head rotary axes. The traverses take place in the T-CS tool coordinate system. Further information: "The Tool axis selection item", Page 221 ■ 3D ROT The control will consider the positions of the rotary axes and the columns SPA, SPB, and SPC of the preset table. The control moves the axes in accordance with the current tilting situation in the working plane coordinate system WPL-CS. Further information: "The 3D ROT selection item", Page 221
Program run	Tilting function active or inactive in the Program Run operating mode and in the MDI application When activating the Tilt working plane function for the Program Run operating mode, the entered angle of rotation applies starting from the first NC block of the NC program to be run. If you use Cycle 19 WORKING PLANE or the PLANE function in the NC program, then the angular values defined there become active. The control will reset the entered angular values to 0.
3D ROT Spa. angle	Currently active angle for the 3D ROT selection item The machine manufacturer uses the machine parameter planeOrientation (no. 201202) to define whether the control calculates with spatial angles SPA, SPB and SPC or with the axis values of the existing rotary axes.

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

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

Further information: "The Positions workspace", Page 121

The Basic rotation selection item

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

Further information: "Basic rotation and 3D basic rotation", Page 211

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

Further information: "Workpiece coordinate system W-CS", Page 202

If the active workpiece preset contains a basic rotation or 3D basic rotation, the control additionally displays the corresponding icon in the **Positions** workspace.

Further information: "The Positions workspace", Page 121

With this selection item, the **3D ROT Spa. angle** area has no function.

The Tool axis selection item

If you select **Tool axis**, then you can move in the positive or negative direction of the tool axis. The control locks all other axes. This selection item makes sense only for machines with rotary head axes.

The traverse movement is active in the **T-CS** tool coordinate system.

Further information: "Tool coordinate system T-CS", Page 207

This selection item can be used, for example, in the following cases:

- When retracting the tool in the direction of the tool axis during an interruption of a 5-axis machining program.
- When traversing with the axis keys or the handwheel with a pre-positioned tool.

With this selection item, the **3D ROT Spa. angle** area has no function.

The 3D ROT selection item

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

You can use this selection item, for example, for manual positioning in a program interruption during tilted machining.

Further information: "Working plane coordinate system WPL-CS", Page 204

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

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



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

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
 - if a **PLANE** function was previously executed with **COORD ROT**
 - after **PLANE RESET**
 - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine manufacturer by the machine manufacturer



COORD ROT is only possible with a free rotary axis.

Further information: Programming and Testing User's Manual

- The control uses the **TABLE ROT** transformation type in the following situations:
 - if a **PLANE** function was previously executed with **TABLE ROT**
 - with corresponding configuration of the machine parameter **CfgRotWorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (#8 / #1-01-1). If the rotary axes are positioned differently than is defined in the **3-D rotation** window, then, by default, the control aborts with an error message.
In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- A tilted working plane will remain active even after a control restart.
Further information: "The Referencing workspace", Page 155
- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

11

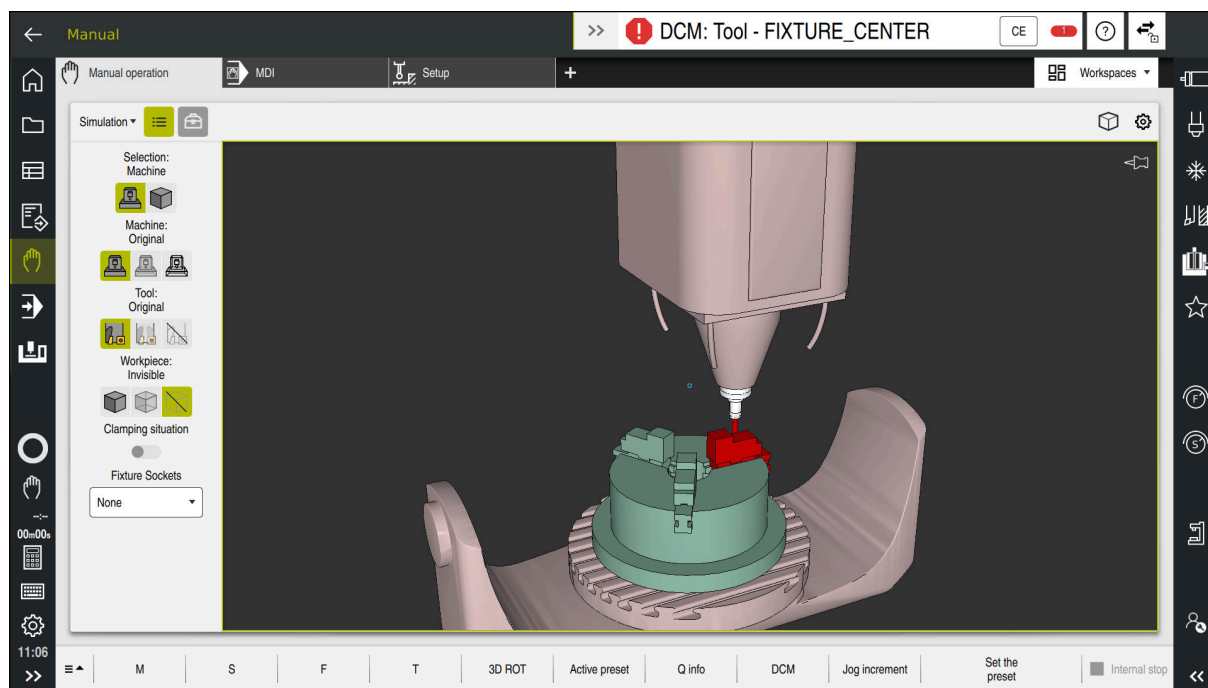
Collision monitoring

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

Fundamentals

Application

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



Dynamic Collision Monitoring (DCM) including collision warning

Related topics

- Fundamentals of fixture management
Further information: "Fixture management", Page 231
- Integrating fixtures in collision monitoring with the NC function **FIXTURE SELECT**
Further information: Programming and Testing User's Manual
- Extended tests in the simulation
Further information: Programming and Testing User's Manual
- Fundamentals of tool carrier management
Further information: "Tool carrier management", Page 186
- Reduce the minimum clearance between two collision objects (#140 / #5-03-2)
Further information: Programming and Testing User's Manual

Requirements

- The Collision Monitoring v2 (#140 / #5-03-2) software option
- Control prepared by the machine manufacturer
The machine manufacturer must define a kinematics model of the machine, insertion point for fixtures and the safety distance between collision objects.
Further information: "Fixture management", Page 231
- Fixture already loaded
For collision monitoring of fixtures, you must load the fixtures with the NC function **FIXTURE SELECT**.
Further information: Programming and Testing User's Manual
- Tools with a positive radius **R** and length **L**.
Further information: "Tool table tool.t", Page 392
- The values in the tool management equal the actual tool dimensions
Further information: "Tool management ", Page 184

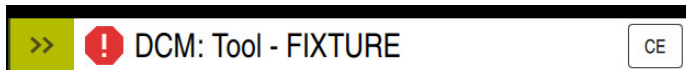
Description of function



Refer to your machine manual.

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

The machine manufacturer can define machine components and minimum distances to be monitored by the control during all machine movements. If two collision objects come closer to each other than a defined minimum distance, the control generates an error message and terminates the movement.



Error message for Dynamic Collision Monitoring (DCM)

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk of collision during all movements!

- ▶ Make sure to activate DCM whenever possible
- ▶ Make sure to always re-activate DCM immediately after a temporary deactivation
- ▶ Carefully test your NC program or program section in **Single Block** mode while DCM is deactivated

The control displays the collision objects graphically in the following operating modes:

- **Editor** operating mode
- **Manual** operating mode
- **Program Run** operating mode

The control also monitors the tools, as defined in tool management, for collision.

NOTICE**Danger of collision!**

Even if Dynamic Collision Monitoring (DCM) is active, the control will not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

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

Further information: Programming and Testing User's Manual

Dynamic Collision Monitoring (DCM) in the Manual and Program Run operating modes

Dynamic Collision Monitoring (DCM) is activated separately for the **Manual** and **Program Run** operating modes, using the **DCM** button.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 228

In the **Manual** and **Program Run** operating modes, the control stops the movement if two collision objects approach each other by less than a minimum clearance. In this case, the control displays an error message naming the two objects causing collision.



Refer to your machine manual.

The machine manufacturer can define the minimum distance between two collision-monitored objects.

Before the collision warning, the control dynamically reduces the feed rate of movements. This ensures that the axes stop in good time before a collision occurs. When the collision warning is triggered, the control displays the colliding objects in red in the **Simulation** workspace.



When a collision warning has been issued, machine movements via the axis direction keys or the handwheel are only possible if they increase the distance between the collision objects.

With active collision monitoring and a simultaneous collision warning, no movements are permitted that reduce the distance or leave it unchanged.

Dynamic Collision Monitoring (DCM) in the Editor operating mode

Dynamic Collision Monitoring (DCM) is activated for simulation in the **Simulation** workspace.

Further information: Programming and Testing User's Manual

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

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



If the **Advanced checks** function is activated, the control will display collisions during simulation (e.g., between workpiece and tool).

Further information: Programming and Testing User's Manual

To obtain a simulation result that is similar to the program run, the following aspects must match:

- Workpiece preset
- Basic rotation
- Offsets of each axis
- Tilting condition
- Active kinematic model

The active workpiece preset for the simulation must be selected. The active workpiece preset from the preset table can be adopted into the simulation.

Further information: Programming and Testing User's Manual

In a simulation, the following aspects may differ from the actual machine or may not be available at all:

- The simulated tool change position may differ from the tool change position in the machine.
- Changes in the kinematics may have a delayed effect in the simulation.
- PLC positioning movements are not displayed in the simulation.
- Handwheel override (#21 / #4-02-1) is not available
- Editing of job lists is not available
- Traverse range limits from the **Settings** application are not available.

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

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is deactivated, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a risk of collision during all movements!

- ▶ Make sure to activate DCM whenever possible
- ▶ Make sure to always re-activate DCM immediately after a temporary deactivation
- ▶ Carefully test your NC program or program section in **Single Block** mode while DCM is deactivated

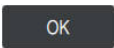
To activate Dynamic Collision Monitoring (DCM) for the **Manual** and **Program Run** operating modes:



- ▶ Select the **Manual** operating mode



- ▶ Select the **Manual** application
- ▶ Select **DCM**
- The control opens the **Dyna. Coll. Monitoring (DCM)** window.
- ▶ Activate DCM in the desired operating modes, using the toggle switches

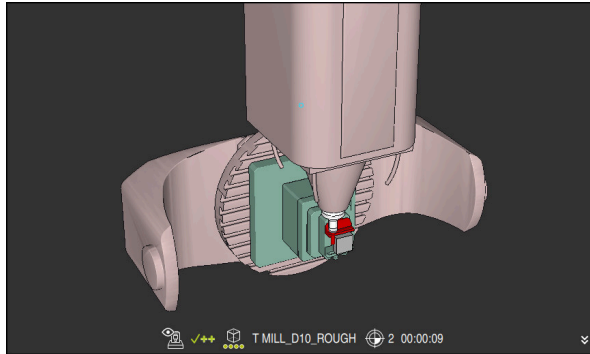


- ▶ Press **OK**
- The control activates DCM in the selected operating modes.



The control displays the status of Dynamic Collision Monitoring (DCM) in the **Positions** workspace. When deactivating DCM, the control displays an icon in the information bar.

Activating the graphic display of the collision objects



Simulation in the **Machine** mode

To activate the graphic display of the collision objects:



- ▶ Select an operating mode (e.g., **Manual**)
- ▶ Select **Workspaces**
- ▶ Select the **Simulation** workspace
- ▶ The control opens the **Simulation** workspace.



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

Changing the representation

To change the graphic display of the collision objects:

- ▶ Activate the graphic display of the collision objects



- ▶ Select the **Visualization options** column



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

Notes

NOTICE

Danger of collision!

Dynamic Collision Monitoring DCM does not detect collisions caused by the reciprocating stroke. Risk of collision!

- Carefully prove-out the NC program

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

11.2 Fixture management

11.2.1 Fundamentals

Application

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

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

Related topics

- Dynamic Collision Monitoring (DCM (#140 / #5-03-2))
Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
- Load fixtures in the NC program with the NC function **FIXTURE SELECT**
Further information: Programming and Testing User's Manual
- Combine fixtures in the **New Fixture** window.
Further information: "Combining fixtures in the New Fixture window", Page 252
- Combine fixtures in **KinematicsDesign**
Further information: "Editing CFG files with KinematicsDesign", Page 244
- Integrating an STL file as workpiece blank
Further information: Programming and Testing User's Manual

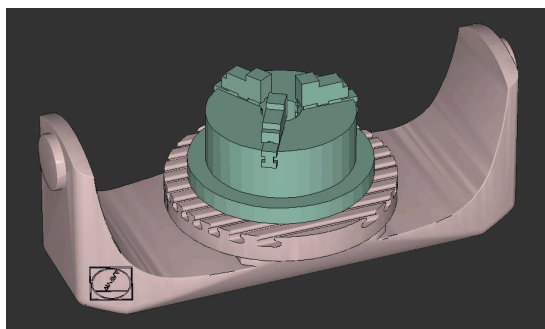
Requirements

- Kinematics description
 The machine manufacturer creates the kinematics description.
- The insertion point is defined within the machine's kinematic description.
 Refer to your machine manual.
 The machine manufacturer defines the insertion point within the machine's kinematic description. The insertion point defines the position at which the loaded fixtures are placed. The machine manufacturer can define multiple insertion points within a machine's kinematic description (e.g., on a rotary table and on an auxiliary fixture plate).
- Fixtures of suitable format:
 - STL file
 - Max. 20 000 triangles
 - Triangular mesh forms a closed shell
 - CFG file
 - M3D file
- If applicable, an insertion point is defined at the fixture
 To combine fixtures with each other in the NC program (#140 / #5-03-2), you must define insertion points in the fixture files. You can define the insertion points in the fixture file in **KinematicsDesign**.
Further information: "Defining an insertion point", Page 250

Description of function

To use fixture monitoring, the steps below are needed:

- Creating a fixture or loading it into the control
 - Further information:** "Options for fixture files", Page 232
- Fixture placement
 - The **Set up fixtures** function in the **Setup** (#140 / #5-03-2) application
 - Further information:** "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234
 - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program
 - Further information:** Programming and Testing User's Manual



Three-jaw chuck loaded as fixture

Options for fixture files

If you integrate the fixtures using the **Set up fixtures** function, you can use STL and CFG files (#140 / #5-03-2).

Alternatively, you can manually set up M3D files.

You can use the **3D mesh** function (#152 / #1-04-1) to create STL files from other file types and adapt STL files to the requirements of your control.

Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288

Fixtures from STL files

STL files allow you to map both individual components and entire assemblies as an immobile fixture. The STL format is useful, in particular, for datum clamping systems and recurring setups.

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

With the CAD Model Optimizer software option (#152 / #1-04-1) you can adapt STL files that do not meet the requirements and then use them as fixtures.

Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288

Fixtures from CFG files

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

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

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

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

Fixtures from M3D files

M3D is a file type designed by HEIDENHAIN. The paid M3D Converter software from HEIDENHAIN allows you to create M3D files from STL or STEP files.

In order to use an M3D file as a fixture, you need to use the M3D Converter software to create and check the file.

Notes

NOTICE

Danger of collision!

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

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

- When using a CAM system, use a postprocessor to output the fixture situation.
- Note the orientation of the coordinate system in the CAD system. Use the CAD system to adapt the orientation of the coordinate system to the desired orientation of the fixture in the machine.
- You can choose any orientation of the fixture model in the CAD system, and therefore the orientation does not always match the orientation of the fixture in the machine.
- Define the coordinate origin in the CAD system such that the fixture can be directly attached to the point of insertion within the machine kinematics.
- Create a central directory for your fixtures (e.g., **TNC:\system\Fixture**).
- When DCM is active, the control checks the fixtures for collisions during simulation or machining (#140 / #5-03-2).

By storing multiple fixtures, you can choose the appropriate fixture for your machining operation without needing to configure it.

- Example files for setups used in everyday manufacturing are provided in the NC database of the Klartext Portal:

HEIDENHAIN NC solutions

- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.
- In the **Simulation** workspace you can check for collisions between the tool (including its holder) and the workpiece or fixtures.

Further information: Programming and Testing User's Manual

11.2.2 Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)

Application

The **Set up fixtures** function determines the position of a 3D model in the **Simulation** workspace, matching the real fixture in the machine envelope. Once the fixture has been set-up, the control considers it in Dynamic Collision Monitoring (DCM).

Related topics

- The **Simulation** workspace
Further information: Programming and Testing User's Manual
- Dynamic Collision Monitoring (DCM)
Further information: "Dynamic Collision Monitoring DCM (#140 / #5-03-2)", Page 224
- Fixture monitoring
Further information: "Fixture management", Page 231
- Setting up a workpiece with graphical support (#159 / #1-07-1)
Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 326

Requirements

- If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.
Further information: "Calibrating the workpiece touch probe", Page 321
If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.
- Collision Monitoring v2 (#140 / #5-03-2) software option
- Kinematics description
The machine manufacturer creates the kinematics description.
- The insertion point is defined within the machine's kinematic description.
Refer to your machine manual.
The machine manufacturer defines the insertion point within the machine kinematics. The insertion point defines the position where loaded fixtures are placed.
- Workpiece touch probe
- Permitted fixture file matching the real fixture
Further information: "Options for fixture files", Page 232

Description of function

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

The **Set up fixtures** function determines the fixture position using various probing processes. First, one point on the fixture is probed in every linear axis. The position of the fixture is defined in this way. After probing one point in all linear axes, further points can be integrated in order to improve positioning accuracy. After defining the position in one axis direction, the control changes the status of that axis from red to green.

The error estimate diagram shows the estimated distance of the 3D model from the real fixture for each probing point.

Further information: "Error estimate diagram", Page 239

The scope of the **Set up fixtures** function depends on the Adv. Function Set 1 (#8 / #1-01-1) and Adv. Function Set 2 (#9 / #4-01-1) software options as follows:

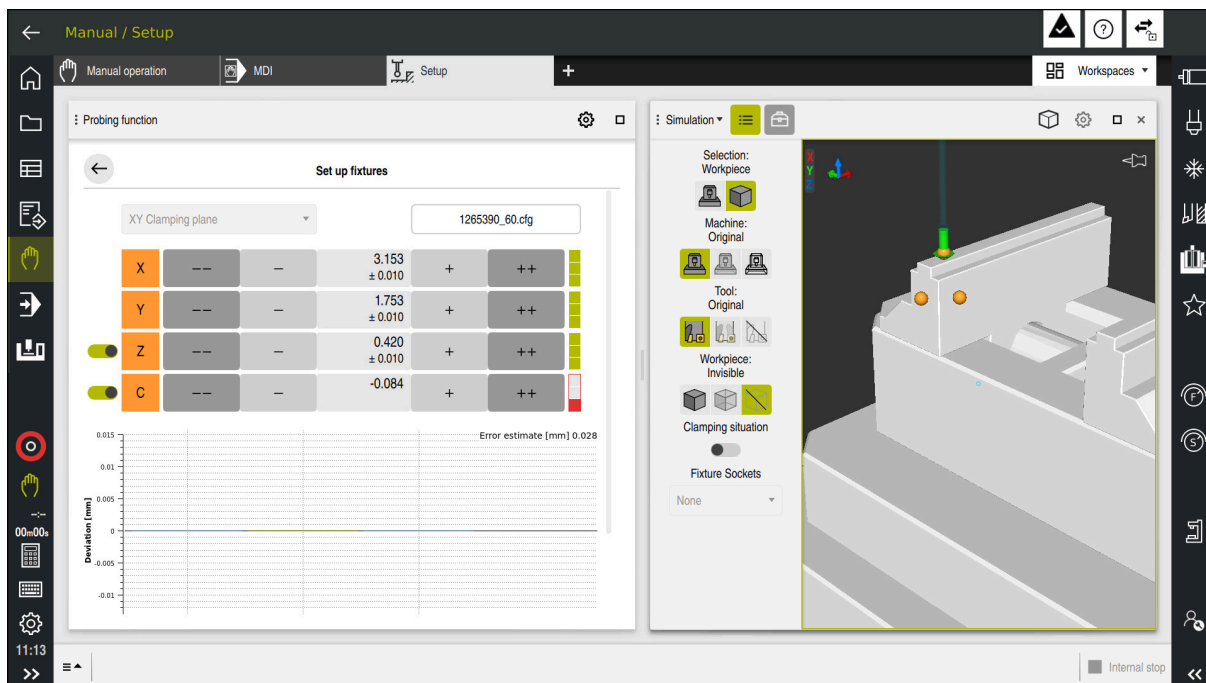
- Both software options enabled:
You can tilt before probing, and incline the tool while probing, in order to probe even complex fixtures.
 - Only Adv. Function Set 1 (#8 / #1-01-1) is enabled:
You can tilt before probing. The working plane must be consistent. If you move the rotary axes between the touch points, the control will display an error message.
-  If the current coordinates of the rotary axes and the defined tilt angles (**3D ROT** window) match, the working plane is consistent.
- None of the two software options is enabled:
You cannot tilt before probing. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 214

Further information: Programming and Testing User's Manual

Extension of the Simulation workspace

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



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

When the **Set up fixtures** function is active, the **Simulation** workspace shows the content below:

- Current position of fixture as viewed by the control
- Probed points on the fixture
- Possible direction of probing by means of an arrow:

- No arrow

Probing is not possible. The workpiece touch probe is too distant from the fixture or the workpiece touch probe is positioned within the fixture, as seen by the control.

In this case, you can adjust the position of the 3D model in the simulation, if applicable.

- Red arrow

Probing in the direction of the arrow is not possible.



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

- Yellow arrow

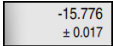
Probing in the direction of the arrow is possible under certain conditions. Probing is done in a deselected direction or might cause collisions.

- Green arrow

Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up fixtures** function contains the following icons and buttons:

Icon or button	Meaning
XY Clamping plane	This selection menu defines the plane in which the fixture is in contact with the machine. The control offers the following planes: ■ XY clamping plane ■ XZ clamping plane ■ YZ clamping plane i Depending on the selected clamping plane, the control displays the corresponding axis directions. In the XY Clamping plane, for example, the control displays the axes X, Y, Z and C.
	Name of fixture file The control automatically saves the fixture file in the initial folder. The fixture file name can be edited before saving.
	Shifts the position of the virtual fixture by 10 mm, 0.3937 inch, or 10° in the negative axis direction i Fixtures are shifted by mm or inch units in a linear axis and by degrees in a rotary axis.
	Shifts the position of the virtual fixture by 1 mm, 0.0394 inch, or 1° in the negative axis direction
	■ Enter the position of the virtual fixture directly ■ Value and estimated accuracy after probing
	Shifts the position of the virtual fixture by 1 mm, 0.0394 inch, or 1° in the positive axis direction
	Shifts the position of the virtual fixture by 10 mm, 0.3937 inch, or 10° in the positive axis direction
Status of the axis:	
	■ Dimmed The axis direction is deselected for this set-up process and will not be taken into account.
	■ Empty No probing points have been determined yet.
	■ Red The control cannot determine the fixture position in this axis direction.
	■ Yellow The position of the fixture in this axis direction already contains information. The information is not meaningful yet.
	■ Green The control can determine the fixture position in this axis direction.

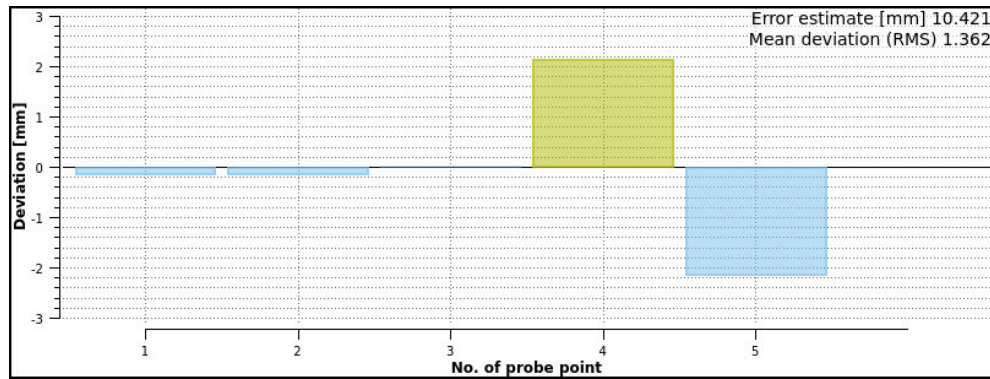
Icon or button	Meaning
Save and activate	This function saves all obtained data in a CFG file and activates the measured fixture in Dynamic Collision Monitoring (DCM).  When using a CFG file as the data source for the measuring process, the existing CFG file can be overwritten by Save and activate at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

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

Error estimate diagram

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

The error estimate diagram shows the estimated distance of the 3D model from the real fixture for each probing point.



Error estimate diagram of the **Set up fixtures** function with transparent columns

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

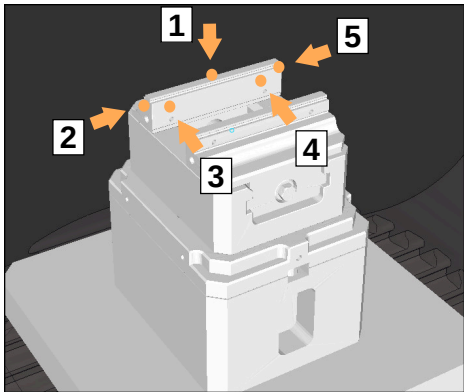
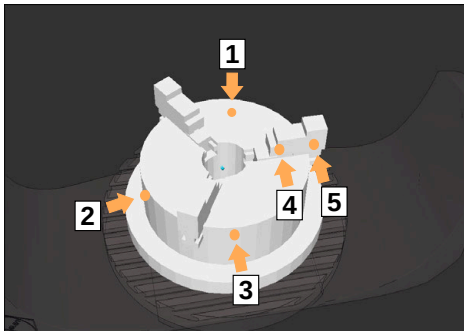
- **Error estimate [mm]**
This value indicates the greatest estimated distance between the 3D model and the fixture after each probing point.
- **Mean deviation (RMS)**
This value indicates the average of all measured distances between the 3D model and the fixture.
- **Deviation [mm]**
On this axis, you can see how great the estimated distance between the 3D model and the probing point at the fixture is.
- **No. of probe point**
This axis shows the number of probing points.
- **Columns**
If the status is not yet green for all axes, the control will display transparent columns.
After each probing point, the control will adjust the 3D model accordingly. This means that the previous values will change, too.
Once the columns in the error estimate diagram are no longer transparent and the **Error estimate [mm]** value displays the required precision, the set-up process is completed.

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

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

Example of sequence of fixture probing points

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

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

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

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

Further information: Programming and Testing User's Manual

Measuring the fixed-jaw vise



The desired 3D model must meet the requirements of the control.

Further information: "Options for fixture files", Page 232

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

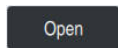
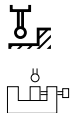
- ▶ Affix a real vise in the working space



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



This step makes the subsequent steps easier.



- ▶ Select the **Setup** application
- ▶ Select **Set up fixtures**
 - The control opens the **Set up fixtures** menu.
- ▶ Select a 3D model matching the real vise
- ▶ Select **Open**
 - The control opens the selected 3D model in the simulation.
- ▶ Pre-position the 3D model by using the buttons for the individual axes within the virtual working space



For pre-positioning the vise, use the workpiece touch probe as a point of reference.

At this point in time, the control does not know the precise position of the fixture, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe and by using, for example, the table's T-slots produces values close to the position of the real vise.

Even after recording the first measuring points, the shifting functions are still available for correcting the fixture position manually.

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



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

Do not touch directly near edges, chamfers and roundings.



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



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

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



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

Save and
activate

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

Notes

NOTICE

Danger of collision!

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

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

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

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

- If you cancel the **Set up fixtures** function, DCM will not monitor the fixture. In this case, any fixtures previously set up are also removed from the scope of monitoring. The control displays a warning.
- The control can measure fixtures only at one insertion point. However, you can measure any number of fixtures and save them for later use.
- When measuring a jaw chuck, the coordinates of the **Z**, **X**, and **Y** axes must be determined just as when measuring a vise. The rotation is determined from one or several jaws.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The saved fixture file can be integrated into the NC program with the **FIXTURE SELECT** function. This can be used for simulating and executing the NC program, considering the real setup situation.

Further information: Programming and Testing User's Manual

- Prior to measuring, check if the correct machine kinematics model is active.

Notes on combined fixtures

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

Further information: "Integrating 3D models", Page 249

Further information: "Combining fixtures in the New Fixture window", Page 252

- When combining fixtures with the **FIXTURE SELECT** function in the NC program, you can measure only the fixture at the insertion point of the machine.
You can measure a datum fixture system, for example. All fixtures loaded on this system will be oriented to this datum fixture system.

Further information: Programming and Testing User's Manual

11.2.3 Editing CFG files with KinematicsDesign

Application

KinematicsDesign allows editing CFG files in the control. In this process, **KinematicsDesign** displays the fixtures graphically and thus supports troubleshooting and removal of errors.

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

- Integrate the 3D model of the fixture in **KinematicsDesign** (e.g., STL file)
Further information: "Integrating 3D models", Page 249
- Save the fixture as a CFG file with the **Combine fixtures** function
Further information: "Combining fixtures in the New Fixture window", Page 252

Related topics

- Combine fixtures into complex clamping arrangements
Further information: "Combining fixtures in the New Fixture window", Page 252

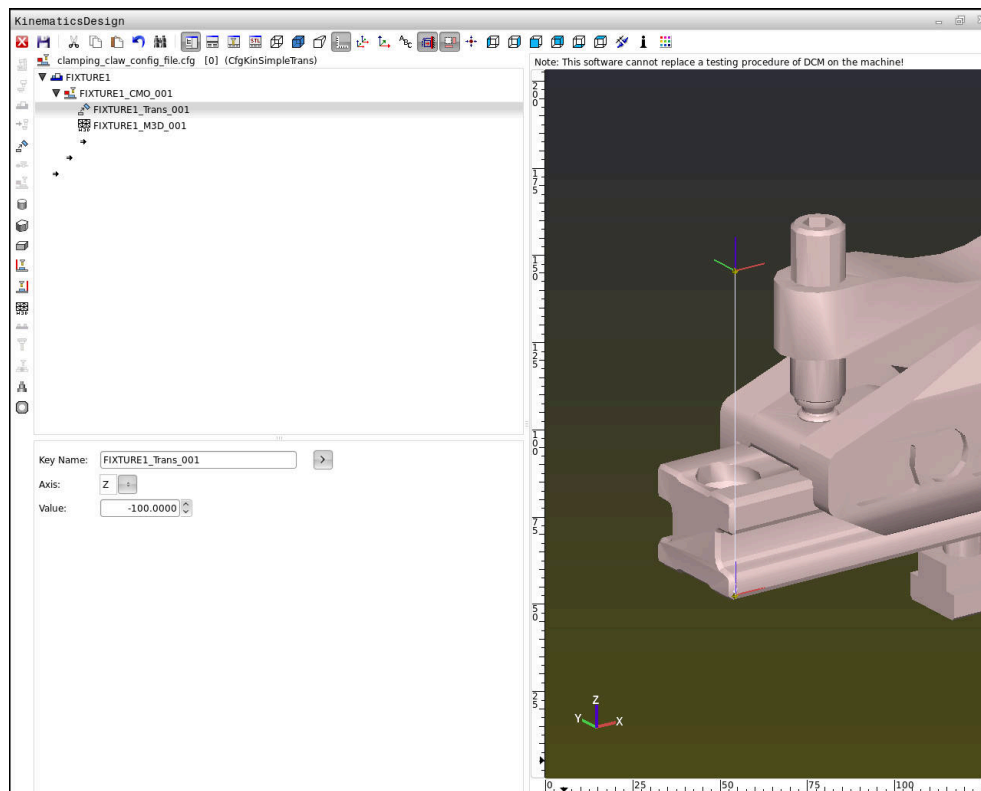
Description of function

When opening a CFG file in the control, the control makes **KinematicsDesign** available as a selection item.

KinematicsDesign offers the following functions:

- Editing of fixtures with graphic support
- Feedback in case of incorrect entries
- Integration of transformations
- Addition of new elements
 - 3D model (M3D or STL files)
 - Cylinder
 - Prism
 - Cuboid
 - Truncated cone
 - Hole
- Definition of insertion points for fixtures

You can integrate both STL files and M3D files into CFG files more than once.



Syntax in CFG files

The following syntax elements are used within the various CFG functions:

Syntax element	Meaning
<code>key:= ""</code>	Name of the function
<code>dir:= ""</code>	Direction of a transformation (e.g., X)
<code>val:= ""</code>	Value
<code>name:= ""</code>	■ Name displayed by the control in case of collision ■ Optional syntax element ■ Name of the insertion point
<code>filename:= ""</code>	File name
<code>vertex:= []</code>	Position of a cube
<code>edgeLengths:= []</code>	Dimensions of a cuboid
<code>bottomCenter:= []</code>	Center of a cylinder
<code>radius:= []</code>	Radius of a cylinder
<code>height:= []</code>	Height of a geometric object
<code>polygonX:= []</code>	Line of a polygon in X
<code>polygonY:= []</code>	Line of a polygon in Y
<code>origin:= []</code>	Starting point of a polygon

Each element is assigned its own **key**. A **key** must be unambiguous and unique, meaning that it must not occur more than once in the description of a fixture. Based on the **key**, the elements are referenced to each other.

The following functions are available if you wish to use CFG functions to describe a fixture in the control:

Function	Meaning
<code>CfgCMOMesh3D(key:="Fixture_body", filename:="1.STL", name:="")</code>	Definition of fixture component  You can also enter an absolute path for the defined fixture component (e.g., TNC:\nc_prog\1.STL)
<code>CfgKinSimpleTrans(key:="XShiftFixture", dir:=X, val:=0)</code>	Shift in X axis Inserted transformations, such as a shift or rotation, are effective for all of the elements following in the kinematic chain.
<code>CfgKinSimpleTrans(key:="CRot0", dir:=C, val:=0)</code>	Rotation in C axis

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

Geometric shapes

You can add simple geometric objects to your collision object either directly in the CFG file or by using **KinematicsDesign**.

All integrated geometric shapes are subelements of the higher-order **CfgCMO**, in which they are listed as **primitives**.

The following geometric objects are available:

Function	Description
<code>CfgCMOCuboid (key:="FIXTURE_Cub", vertex:= [0, 0, 0], edgeLengths:= [0, 0, 0], name:="")</code>	Definition of a cuboid
<code>CfgCMOCylinder (key:="FIXTURE_Cyl", dir:=Z, bottomCenter:= [0, 0, 0], radius:=0, height:=0, name:="")</code>	Definition of a cylinder
<code>CfgCMOPrism (key:="FIXTURE_Pris_002", height:=0, polygonX:=[], polygonY:=[], name:="", origin:= [0, 0, 0])</code>	Definition of a prism A prism is described by entering the height and several polygonal lines.

Creating a fixture entry with a collision object

The content below describes the procedure with **KinematicsDesign** opened.

To create a fixture entry with a collision object:



- ▶ Select **Insert chucking equipment**
- **KinematicsDesign** creates a new fixture entry within the CFG file.
- ▶ Enter a key name for the fixture (e.g., FIX_clamping jaw)
- ▶ Confirm your input
- **KinematicsDesign** applies the input to the fixture entry.
- ▶ Open the fixture entry



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

Defining geometric shapes

KinematicsDesign allows you to define various geometric shapes. You can construct simple fixtures by combining several geometric shapes.

To define a geometric shape:

- ▶ Create a fixture entry with a collision object



- ▶ Select **Insertion point** under Collision object
- ▶ Select the desired geometric shape (e.g., a cuboid)
- **KinematicsDesign** inserts the shape.
- ▶ Define the position of the cuboid (e.g., **X = 0, Y = 0, Z = 0**)
- ▶ Define the dimensions of the cuboid (e.g., **X = 100, Y = 100, Z = 100**)
- ▶ Confirm your input
- **KinematicsDesign** applies the input and displays the defined cuboid in the graphics.

Integrating 3D models

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

Further information: "Options for fixture files", Page 232

To integrate a 3D model as a fixture:

- ▶ Create a fixture entry with a collision object



- ▶ Select **Insertion point** under Collision object



- ▶ Select **Insert 3D model**
- ▶ **KinematicsDesign** opens the **Open File** window.
- ▶ Select the desired STL or M3D file
- ▶ Select **OK**
- ▶ **KinematicsDesign** integrates the selected file and displays the file in the graphic window.

Fixture placement

You can place the integrated fixture at any position (e.g., for correcting the orientation of an external 3D model). For this purpose, insert transformations for all axes you wish to use.

To position a fixture with **KinematicsDesign**:

- ▶ Creating a fixture entry with a collision object



- ▶ Select **Insertion point** under Collision object



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

Defining an insertion point

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

To define an insertion point:

- ▶ Open the fixture with **KinematicsDesign**



- ▶ Open the fixture entry



- ▶ Move the cursor down one level



- ▶ Open the collision object



- ▶ Select **Insertion point**



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



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

- **KinematicsDesign** inserts the transformation and displays it in the graphics.
 - ▶ Enter a key name for the transformation (e.g., Z shift)
 - ▶ Select the axis for the transformation (e.g., **Z**)
 - ▶ Enter the value for the transformation (e.g., **100**)
 - ▶ Confirm your input
- **KinematicsDesign** loads the input.
 - ▶ Select **Insert insertion point**
- **KinematicsDesign** inserts the insertion point and displays it in the graphics.
 - ▶ Enter the key name for the insertion point (e.g., FIX_1_GP)
 - ▶ Enter a name for the insertion point (e.g., insertion point _1_base plate)



Every insertion point must have an unambiguous name.

- **KinematicsDesign** applies the input and the insertion point has now been defined.
 - ▶ Insert further transformations if required (e.g., **X+**)
 - ▶ Define another insertion point as needed

Notes

- If one of the transformations contains the **?** character in the key, you can enter the value of the transformation within the **Combine fixtures** function. This allows easy positioning of clamping jaws, for example.

Further information: "Combining fixtures in the New Fixture window", Page 252

- As an alternative to using **KinematicsDesign**, you can also create fixture files directly from the CAM system or by using the appropriate code in a text editor.
- You can create a CFG file in the **Quick selection new file** workspace.

Further information: Programming and Testing User's Manual

Example

The example below describes the syntax of a CFG file for a vise with two movable jaws.

Files used

Various STL files are used to describe the vise. Since the jaws of the vise are dimensionally identical, they are defined using the same STL file.

Code	Explanation
<code>CfgCMOMesh3D (key:="Fixture_body", filename:="vise_47155.STL", name:="")</code>	Body of the vise
<code>CfgCMOMesh3D (key:="vise_jaw_1", filename:="vise_jaw_47155.STL", name:="")</code>	First jaw of the vise
<code>CfgCMOMesh3D (key:="vise_jaw_2", filename:="vise_jaw_47155.STL", name:="")</code>	Second jaw of the vise

Definition of jaw opening width

In this example, the opening width of the vise is defined using two mutually dependent transformations.

Code	Explanation
<code>CfgKinSimpleTrans (key:="TRANS_opening_width", dir:=Y, val:=-60)</code>	Jaw opening width of the vise in Y direction: 60 mm
<code>CfgKinSimpleTrans (key:="TRANS_opening_width_2", dir:=Y, val:=30)</code>	Position of the first jaw of the vise in Y direction: 30 mm

Positioning of the fixture within the working space

The defined fixture components are positioned using various transformations.

Code	Explanation
<code>CfgKinSimpleTrans (key:="TRANS_X", dir:=X, val:=0) CfgKinSimpleTrans (key:="TRANS_Y", dir:=Y, val:=0) CfgKinSimpleTrans (key:="TRANS_Z", dir:=Z, val:=0) CfgKinSimpleTrans (key:="TRANS_Z_vise_jaw", dir:=Z, val:=60) CfgKinSimpleTrans (key:="TRANS_C_180", dir:=C, val:=180) CfgKinSimpleTrans (key:="TRANS_SPC", dir:=C, val:=0) CfgKinSimpleTrans (key:="TRANS_SPB", dir:=B, val:=0) CfgKinSimpleTrans (key:="TRANS_SPA", dir:=A, val:=0)</code>	Positioning of the fixture components In this example, a rotation by 180° is inserted for rotating the defined jaw of the vise. This is necessary because the same initial model is used for both jaws of the vise. The rotation inserted applies to all subsequent components in the transformation chain.

Description of the fixture

You need to combine all objects and transformations in the CFG file in order to ensure that the fixture is correctly depicted in the simulation.

Code	Explanation
<pre>CfgCM0 (key:="FIXTURE", primitives:= ["TRANS_X", "TRANS_Y", "TRANS_Z", "TRANS_SPC", "TRANS_SPB", "TRANS_SPA", "Fixture_body", "TRANS_Z_vise_jaw", "TRANS_opening_width_2", "vise_jaw_1", "TRANS_opening_width", "TRANS_C_180", "vise_jaw_2"], active:=TRUE, name:="")</pre>	Combining the transformations and objects contained in the fixture

Fixture designation

You need to assign a designation to the combined fixture.

Code	Explanation
<pre>CfgKinFixModel (key:="FIXTURE1", kinObjects:=["FIXTURE"])</pre>	Designation of the combined fixture

11.2.4 Combining fixtures in the New Fixture window**Application**

The **New Fixture** window allows combining several fixtures and saving them as a new fixture. This enables realizing and monitoring complex clamping situations.

Related topics

- Fundamentals of fixtures
Further information: "Fundamentals", Page 231
- Integrating fixtures into the NC program
Further information: Programming and Testing User's Manual
- Set up fixtures (#140 / #5-03-2)
Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234

Requirement

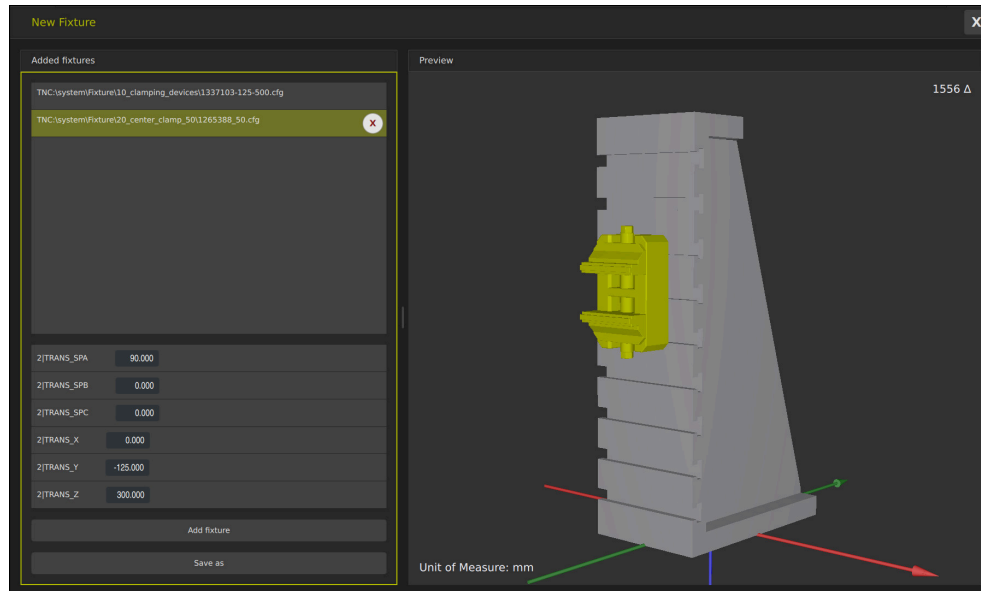
- Fixtures of suitable format:
 - STL file
 - Max. 20 000 triangles
 - Triangular mesh forms a closed shell
 - CFG file
 - M3D file

Description of function

To navigate to this function:

Tools ► Combine fixtures

The control also makes this function available as a selection option for opening CFG files.



Combined fixture with variable transformations

The **Add fixture** button selects all required fixtures one by one.

If one of the transformations contains the ? character in the key, you can enter the value of the transformation within the **Combine fixtures** function. This allows easy positioning of clamping jaws, for example.

The control displays a preview of the combined fixture and the total number of all triangles.

The **Save as** button saves the combined fixture as a CFG file.

Notes

- For optimum performance, HEIDENHAIN recommends that combined fixtures contain no more than 20,000 triangles.
- If the position or the size of a fixture must be adapted, use **KinematicsDesign**.
Further information: "Editing CFG files with KinematicsDesign", Page 244
- The **Combine fixtures** function allows saving individual fixtures of *.stl or *.m3d file types as CFG files. You need a fixture as a CFG file when you wish to define insertion points, for example.

Further information: "Defining an insertion point", Page 250

12

Control functions

12.1 Adaptive Feed Control (AFC) (#45 / #2-31-1)

12.1.1 Fundamentals

Application

Adaptive Feed Control (AFC) saves time when processing NC programs and reduces wear on the machine. The control regulates the contouring feed rate during program run depending on the spindle power. In addition, the control responds to overloading of the spindle.

Related topics

- Tables related to AFC

Further information: "Tables for AFC (#45 / #2-31-1)", Page 423

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- Enabled by the machine manufacturer
The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

To regulate the feed rate during program run with AFC:

- Define basic settings for AFC in the **AFC.tab** table
Further information: "Basic AFC settings in AFC.tab", Page 423
- Define settings for AFC for each tool in the tool management
Further information: "Tool table tool.t", Page 392
- Define AFC in the NC program
Further information: "NC functions for AFC (#45 / #2-31-1)", Page 259
- Define AFC in the **Program Run** operating mode with the **AFC** toggle switch.
Further information: "The AFC toggle switch in the Program Run operating mode", Page 260
- Prior to automatic control, determine the reference spindle power with a teach-in cut
Further information: "AFC teach-in cut", Page 262

If AFC is active in the teach-in cut or in control mode, the control displays an icon in the **Positions** workspace.

Further information: "The Positions workspace", Page 121

Detailed information about the function is provided by the control on the **AFC** tab of the **Status** workspace.

Further information: "The AFC tab (#45 / #2-31-1)", Page 131

Benefits of AFC

Adaptive feed control (AFC) has the following advantages:

- Optimization of machining time
By controlling the feed rate, the control tries to maintain the previously recorded maximum spindle power or the reference power specified in the tool table (**AFC-LOAD** column) during the entire machining time. It shortens the machining time by increasing the feed rate in machining zones with little material removal.
- Tool monitoring
If the spindle power exceeds the taught-in or specified maximum value, the control reduces the feed until the reference spindle power is reached. If the minimum feed rate is exceeded, the control executes a shutdown response. AFC can also use the spindle power to monitor the tool for wear and breakage without changing the feed rate.
Further information: "Monitoring tool wear and tool load", Page 264
- Protection of the machine's mechanical elements
Timely feed rate reduction and shutdown reactions help to avoid machine overload.

Tables related to AFC

The control offers the following tables in conjunction with AFC:

- **AFC.tab**
In the **AFC.tab** table, you define the feed-rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.
Further information: "Basic AFC settings in AFC.tab", Page 423
- ***.H.AFC.DEP**
With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.
Further information: "AFC.DEP settings file for teach-in cuts", Page 425
- ***.H.AFC2.DEP**
During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you are performing the teach-in cut.
In control mode, the control updates the data in this table and performs evaluations.
Further information: "Log file AFC2.DEP", Page 427

You can open and, if necessary, edit the tables for AFC during program run. The control provides only the tables of the active NC program.

Further information: "Editing the tables for AFC", Page 428

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as Adaptive Feed Control (AFC) is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g., due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This behavior applies regardless of how the function is deactivated. This feed acceleration may result in damage to the tool and/or the workpiece!

- ▶ If the feed rate is about to fall below the **FMIN** value, stop the machining operation (instead of deactivating the AFC function)
 - ▶ Define the overload response for cases in which the feed rate falls below the **FMIN** value
- If Adaptive Feed Control is active in **Control** mode, the control executes a shutdown response independent of the programmed overload response.
 - If, with the reference spindle load, the value falls below the minimum feed factor

The control executes the shutdown response from the **OVLD** column of the **AFC.tab** table.

Further information: "Basic AFC settings in AFC.tab", Page 423
 - If the programmed feed rate falls below the 30% threshold

The control executes an NC stop.
 - Adaptive Feed Control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.
 - Do not work with Adaptive Feed Control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.
 - In NC blocks containing **FMAX**, the adaptive feed control is **not active**.
 - In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.
- Further information:** Programming and Testing User's Manual

12.1.2 Activating and deactivating AFC

NC functions for AFC (#45 / #2-31-1)

Application

Adaptive Feed Control (AFC) is activated and deactivated from the NC program.

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- Control settings defined in the **AFC.tab** table
Further information: "Basic AFC settings in AFC.tab", Page 423
- Desired control setting defined for all tools
Further information: "Tool table tool.t", Page 392
- **AFC** toggle switch active
Further information: "The AFC toggle switch in the Program Run operating mode", Page 260

Description of function

The control provides several functions that enable you to start and stop AFC:

- **FUNCTION AFC CTRL:** The **AFC CTRL** function activates feedback control mode starting with this NC block, even if the learning phase has not been completed yet.
- **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3:** The control starts a sequence of cuts with active **AFC**. The changeover from the teach-in cut to feedback control mode begins as soon as the reference power has been determined in the teach-in phase, or once one of the **TIME**, **DIST** or **LOAD** conditions has been met.
- **FUNCTION AFC CUT END:** The **AFC CUT END** function deactivates AFC control.

Input

FUNCTION AFC CTRL

11 FUNCTION AFC CTRL	; Start AFC in control mode

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CTRL	Syntax initiator for the start of control mode

FUNCTION AFC CUT

**11 FUNCTION AFC CUT BEGIN TIME10
DIST20 LOAD80**

; Start AFC machining step, limit the duration of the teach-in phase

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CUT	Syntax initiator for an AFC machining step
BEGIN or END	Start or end machining step
TIME	End teach-in phase after the defined time in seconds Optional syntax element Only if BEGIN has been selected
DIST	End teach-in phase after the defined distance in mm Optional syntax element Only if BEGIN has been selected
LOAD	Enter the reference load of the spindle directly, max. 100% Optional syntax element Only if BEGIN has been selected

Notes

- The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.
- Execute the function **AFC CUT BEGIN** only after the starting rotational speed has been reached. If this is not the case, then the control issues an error message, and the AFC cut is not started.
- You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.
If you program both values, the control will use the value programmed in the NC program!

The AFC toggle switch in the Program Run operating mode**Application**

The **AFC** toggle switch allows you to activate or deactivate Adaptive Feed Control (AFC) in the **Program Run** operating mode.

Related topics

- Activating AFC in the NC program
Further information: "NC functions for AFC (#45 / #2-31-1)", Page 259

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- Enabled by the machine manufacturer
The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

The **AFC** toggle switch must be activated for the NC functions for AFC to have an effect.

If you do not specifically deactivate AFC using the toggle switch, AFC remains active. The control remembers the setting of the toggle switch even if the control is restarted.

If the **AFC** toggle switch is active, the control displays an icon in the **Positions** workspace. In addition to the current setting of the feed rate potentiometer, the control shows the controlled feed value as a percentage (%).

Further information: "The Positions workspace", Page 121

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as the AFC function is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g. due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This applies regardless of how the function is deactivated (e.g. feed rate potentiometer). This acceleration may result in damage to the tool or the workpiece!

- ▶ If the feed rate is about to fall below the **FMIN** value, stop the machining operation (instead of deactivating the **AFC** function)
 - ▶ Define the overload response for cases in which the feed rate falls below the **FMIN** value
- If Adaptive Feed Control is active in **Control** mode, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.
 - The feed rate override is the percentage value of the programmed feed rate that you usually set with the potentiometer. When the adaptive feed control is active in the **Control** mode, the control defines the feed rate override.
 - Turning the feed rate potentiometer will not influence the feed rate override.
 - If you reduce the feed override with the potentiometer by more than 10% in relation to the position at the start of the program, the control switches AFC off.
You can reactivate the controlling process with the **AFC** toggle switch.
 - Potentiometer values of up to 50% always have an effect, even with active control.
 - Mid-program startup is allowed during active feed control. The control takes the cutting number of the startup block in account.

12.1.3 AFC teach-in cut

Fundamentals

Application

With the teach-in cut, the control determines the reference power of the spindle for the machining step. Based on the reference power, the control adjusts the feed rate in control mode.

If you have already determined the reference power for a machining operation, you can specify the value for the machining operation. For this, the control provides the **AFC-LOAD** column in the tool management and the **LOAD** syntax element in the **FUNCTION AFC CUT BEGIN** function. In this case, the control no longer performs a teach-in cut, but uses the specified value immediately for control.

Related topics

- Enter the known reference power in the **AFC-LOAD** column in the tool management
Further information: "Tool table tool.t", Page 392
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function
Further information: "NC functions for AFC (#45 / #2-31-1)", Page 259

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- Control settings defined in the **AFC.tab** table
Further information: "Basic AFC settings in AFC.tab", Page 423
- Desired control setting defined for all tools
Further information: "Tool table tool.t", Page 392
- Desired NC program selected in the **Program Run** operating mode
- **AFC** toggle switch active
Further information: "The AFC toggle switch in the Program Run operating mode", Page 260

Description of function

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**.

Further information: "AFC.DEP settings file for teach-in cuts", Page 425

When you are performing a teach-in cut, the control shows the spindle reference power determined until this time in a pop-up window.

When the control has determined the control reference power, it ends the teach-in cut and switches to control mode.

Notes

- When you record a teach-in cut, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.
- During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.
- You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. If the programmed feed rate value is far too high and forces you to sharply decrease the feed rate override during the machining step, you will have to repeat the teach-in cut.
- If the determined reference load is greater than 2%, the control changes the status from teach-in (**L**) to controlling (**C**). Adaptive feed control is not possible for smaller values.

The AFC settings button

Application

The **AFC settings** button in the **Program Run** operating mode allows terminating a teach-in cut or opening the tables for AFC.

Related topics

- Fundamentals for the teach-in cut
Further information: "Fundamentals", Page 262
- Tables for AFC
Further information: "Tables for AFC (#45 / #2-31-1)", Page 423

Requirements

- Adaptive Feed Contr. (#45 / #2-31-1) software option
- Enabled by the machine manufacturer
The machine manufacturer uses the optional machine parameter **Enable** (no. 120001) to define whether you can use AFC.

Description of function

This button offers the following select options:

Button	Meaning
AFC.TAB	Editing the factory default settings When selecting this button, the control will open the AFC.TAB table in the Tables operating mode. Further information: "Basic AFC settings in AFC.tab", Page 423
AFC.DEP	Editing the settings file for teach-in cuts When selecting this button, the control will open the AFC.DEP table for the current NC program in the Tables operating mode. Further information: "AFC.DEP settings file for teach-in cuts", Page 425
AFC2.DEP	Editing the log file for evaluation When selecting this button, the control will open the AFC2.DEP table for the current NC program in the Tables operating mode. Further information: "Log file AFC2.DEP", Page 427
Stop Teach	Terminating a teach-in cut ■ The control terminates the teach-in cut and changes to control mode Further information: "AFC teach-in cut", Page 262 ■ In the AFC.DEP table, the control changes the status of the ST column from teaching-in (L) to controlling (C). Further information: "AFC.DEP settings file for teach-in cuts", Page 425 ■ In the Positions workspace, the control changes the icon for the teaching-in cut into the control mode icon. Further information: "The Positions workspace", Page 121



In a milling operation, you do not have to run the entire machining step in teaching-in mode. If the cutting conditions do not change significantly, you can switch to control mode immediately.

12.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear or breakage. To do this, use columns **AFC-OVLD1** or **AFC-OVLD2** in the tool management.

Related topics

- **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management

Further information: "Tool table tool.t", Page 392

Description of function

If the **AFC.TAB** columns **FMIN** and **FMAX** each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active.

Further information: "Basic AFC settings in AFC.tab", Page 423

Tool wear and tool breakage cannot be monitored at the same time. If the **AFC_OVLD2** column contains a value, the control will ignore the **AFC_OVLD1** column.

Tool wear monitoring

Activate cut-related tool wear monitoring by entering a value not equal to 0 in the **AFC-OVLD1** column in the tool table.

The overload response depends on the **AFC.TAB** column **OVLD**.

In conjunction with cut-related tool wear monitoring, the control only evaluates the options **M**, **E**, and **L** in the **OVLD** column. The following responses are possible:

- Pop-up window
- Lock current tool
- Insert replacement tool

Tool load monitoring

Activate cut-related tool load monitoring (tool breakage control) by entering a value not equal to 0 in the **AFC-OVLD2** column in the tool table.

As overload response, the control always executes a machining stop and locks the momentary tool.

Example

The entries in columns **AFC-OVLD1** and **AFC-OVLD2** are added to the feedback-control reference power **AFC-LOAD**.

Further information: "AFC teach-in cut", Page 262

Example input for tool wear and tool load monitoring:

Column	Input
AFC-LOAD	30%
AFC-OVLD1	5%
AFC-OVLD2	10%

In this example, the control adds the 5% and 10% to the 30% in each case.

As soon as a value is defined in column **AFC-OVLD1**, the tool will monitor tool wear. When the control used in the example reaches a spindle power of 35% in total, it executes the defined reaction.

12.2 Active Chatter Control (ACC) (#145 / #2-30-1)

Application

Chatter marks can be caused during heavy-duty machining, in particular. **ACC** reduces chattering, thereby reducing wear on the tool and machine. In addition, **ACC** increases metal removal rates.

Related topics

- **ACC** column in the tool table
Further information: "Tool table tool.t", Page 392

Requirements

- Software option Active Chatter Contr. (#145 / #2-30-1)
- Control adapted by the machine manufacturer
- **ACC** column in the tool management defined with **Y**
- Number of tool cutting edges defined in the **CUT** column

Description of function

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the machine can sometimes begin to **chatter**. This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

In order to reduce a machine's tendency to chatter, HEIDENHAIN offers an effective control function known as Active Chatter Control (**ACC**). The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. Depending on the type of machine, the metal-removal rate can often be increased by more than 25%. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

ACC was developed especially for roughing and heavy machining and is particularly effective in this area. You need to conduct appropriate tests to see whether ACC will also be advantageous on your machine and with your tool.

ACC is activated and deactivated using the **ACC** toggle switch in the **Program Run** operating mode or the **MDI** application.

Further information: "The Program Run operating mode", Page 340

Further information: "The MDI application ", Page 301

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

Further information: "The Positions workspace", Page 121

Notes

- ACC reduces or prevents vibrations in the range of 20 Hz to 150 Hz. If ACC does not appear to have an effect, the vibrations may be outside of this range.
- With the software option Machine Vibr. Contr. (#146 / #2-24-1), you can additionally improve the result.

13

CAD Viewer

13.1 Fundamentals

Application

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

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

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

Related topics

- Importing contours and positions with CAD Import from **CAD Viewer** (#42 / #1-03-1)
Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280
- Rectifying errors in 3D models with **3D mesh** (#152 / #1-04-1)
Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288
- Creating 2D sketches on the control
Further information: Programming and Testing User's Manual

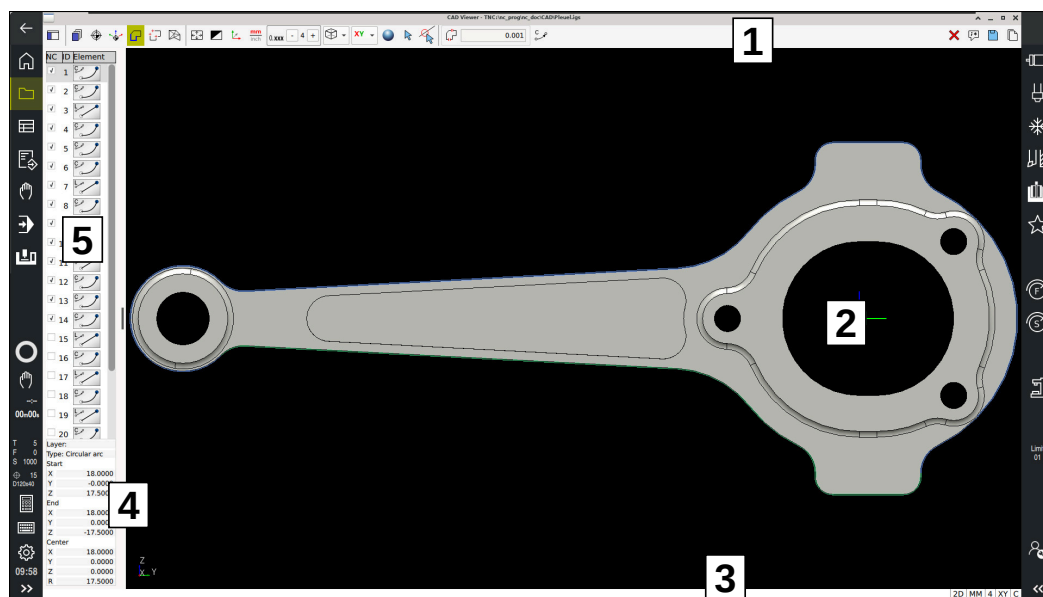
Description of function

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

CAD Viewer supports the following standard file types that can be opened directly in the control:

File type	Extension	Format
STEP	*.stp and *.step	■ AP 203 ■ AP 214
IGES	*.igs and *.iges	■ Version 5.3
DXF	*.dxf	■ R10 to 2015 ■ ASCII
STL	*.stl	■ Binary ■ ASCII

Screen layout



CAD file open in **CAD Viewer**

CAD Viewer contains the following areas:

- 1 Menu bar
Further information: "Menu bar icons", Page 270
- 2 Graphics area
In the Graphics area, the control displays the CAD model.
- 3 Status bar
In the status bar, the control displays the active settings.
- 4 Element information area
Further information: "The Element Information area", Page 272
- 5 List view area
The List View area displays information on the active function (e.g., available layers or the position of the workpiece preset).

Menu bar icons

The menu bar contains the following icons:

Icon	Meaning
	Show sidebar Show, enlarge or hide the List View and Element Information areas
	Display the layer Display the layer(s) in the List View area Further information: "Layer", Page 273
	Preset Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset Further information: "Workpiece preset in the CAD file", Page 275
	Datum Set the datum Datum has been set Further information: "Workpiece datum in the CAD file", Page 278
	Contour Select contour (#42 / #1-03-1) Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280
	Positions Select positions (#42 / #1-03-1) Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280
	3D mesh Create a 3D mesh (#152 / #1-04-1) Further information: "Generating STL files with 3D mesh (#152 / #1-04-1)", Page 288
	Show all Set the zoom to the largest possible view of the complete graphics
	Inverted colors Change the background color (black or white)
	Toggle between 2D and 3D modes
	Set the unit of measure (mm or inches) CAD Viewer always calculates internally with mm. If you select inches as the unit of measure, CAD Viewer will convert all values to inches. Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280

Icon	Meaning
	Number of decimal places Select decimal places: 3...7 for mm and 4...8 for inches The decimal places define the resolution and the number of segments during linearization. Further information: "Applying contours", Page 281 Default setting: 4 decimal places with mm, and 5 decimal places with inch as the unit of measure
	Set perspective Switch between various views of the model (e.g., Top)
	Axes Select the working plane: ■ XY ■ YZ ■ ZX ■ ZXØ If you take over a contour or position, the control outputs the NC program in the selected working plane. Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280
	Toggle a 3D model between a solid model and a wire-frame model.
	Select contour elements (e.g., to obtain element information) Further information: "The Element Information area", Page 272
	Add contour elements
	Remove contour elements The icon shows the current mode. Clicking the icon activates the next mode. Further information: "Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)", Page 280
	Activate or deactivate Hover If you swipe across the CAD model during touch operation, the controls display selectable options in orange. The function is comparable to hovering with a mouse. As soon as you stop the movement, the control will mark the current selection option. If this icon is active, you can no longer rotate the CAD model.
	Undo an action
	Delete entire list

Icon	Meaning
	Setting for whether comments are written to NC output files. Add comments with workpiece information to the NC program (#42 / #1-03-1) The control displays the icon if you select the Contour or Positions mode. Further information: "Selecting and saving a contour", Page 284
	H file with FMAX M99 at each position (for older NC controls) Add each position with rapid traverse FMAX and cycle call M99 to the NC program (#42 / #1-03-1) The control displays the icon if you select the Positions mode. Further information: "Applying positions", Page 282
	Save entire list content to a file
	Copy entire list contents to clipboard The control retains the content of the clipboard only as long as CAD Viewer is open.

The Element Information area

In the Element Information area, the following information is displayed for the selected element of the CAD file:

- Associated layer
- Element type
- Point type:
 - Point coordinates
- Line type:
 - Coordinates of the starting point
 - Coordinates of the end point
- Circular arc or circle type:
 - Coordinates of the starting point
 - Coordinates of the end point
 - Coordinates of the center point
 - Radius

The control always shows the **X**, **Y** and **Z** coordinates. In 2D mode, the Z coordinate is dimmed.

Layer

CAD files usually contain multiple layers. The designer uses these layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, hatching, and texts.

The CAD file to be processed must contain at least one layer. The control automatically moves all elements not assigned to a layer to the "anonymous" layer.

If the name of the layer is not shown completely in the List View area, you can use the **Show sidebar** icon to enlarge this area.

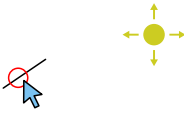
Use the **Display the layer** icon to display all the layers of the file in the List View area. Use the check box in front of the name to show and hide individual layers.

When you open a CAD file in **CAD Viewer**, all available layers are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Manipulating the graphics in CAD Viewer

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

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

Notes

- Before loading the file into the control, ensure that the name of the file contains only permitted characters.
Further information: Programming and Testing User's Manual
- When you select a layer in the List View area, you can press the spacebar to show and hide the layer.
- CAD Viewer** allows you to open CAD files consisting of any number of triangles.

13.2 Workpiece preset in the CAD file

Application

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

Related topics

- Presets in the machine

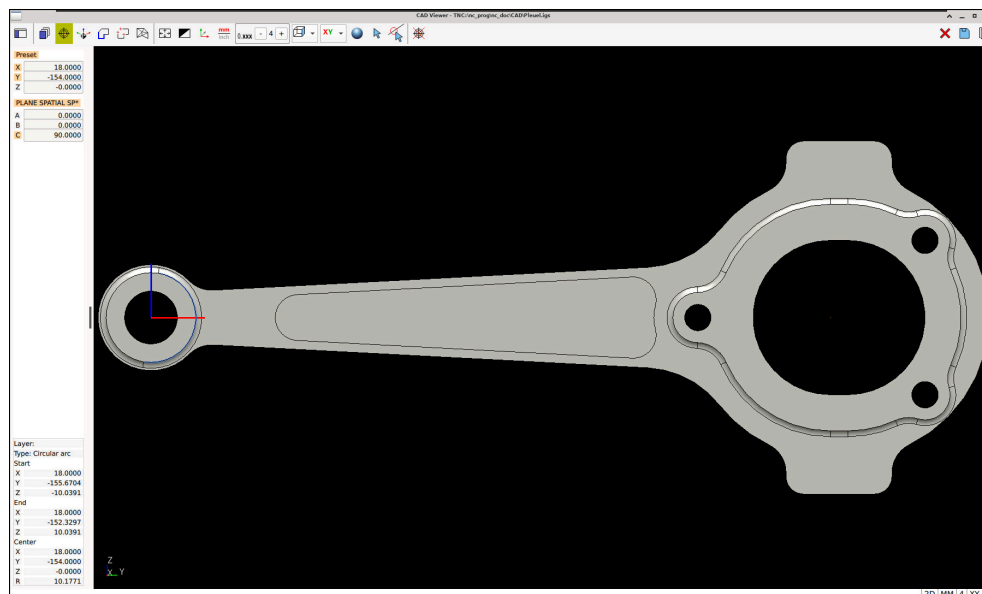
Further information: "Presets in the machine", Page 167

Description of function

When you select the **Preset** icon, the control displays the following information in the list view area:

- Distance between the defined preset and the drawing datum
- Orientation of the working plane

The control displays values not equal to 0 in orange.



Workpiece preset in the CAD file

You can position the preset at the following locations:

- By direct input of numerical values into the List View area
- For lines:
 - Starting point
 - Midpoint
 - End point
- For circular arcs:
 - Starting point
 - Midpoint
 - End point
- For full circles:
 - At the quadrant transitions
 - At the center

- At the intersection between:
 - Two lines, even if the point of intersection is actually on the extension of one of the lines
 - Line and circular arc
 - Line and full circle
 - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;origin = X... Y... Z...
```

```
5 ;origin_plane_spatial = SPA... SPB... SPC...
```

You can save the workpiece preset and workpiece datum information to a file or the clipboard even without the CAD Import software option (#42 / #1-03-1).



The control retains the content of the clipboard only as long as **CAD Viewer** is open.

You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.

13.2.1 Setting the workpiece preset or workpiece datum and orienting the coordinate system



- The following instructions also apply to the workpiece datum. In this case, start by selecting the **Datum** icon.
- The following instructions apply to the use of a mouse. You can also perform all steps with touch gestures.

Further information: "Manipulating the graphics in CAD Viewer",
Page 274

Setting the workpiece preset or workpiece datum on an individual element

To set the workpiece preset on an individual element:



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

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

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

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



- ▶ Select **Preset**
- ▶ Click the first element
- The control highlights the element in color.
- ▶ Click the second element
- The control sets the workpiece preset at the point of intersection of the two elements. The control marks the workpiece preset with a green symbol.
- ▶ Orient the working plane, if required



- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Orienting the working plane

The following requirements must be met in order to orient the working plane:

- Preset has been defined
- There are elements next to the preset that can be used for the desired orientation

To orient the working plane:

- ▶ Select an element in the positive direction of the X axis
- The control orients the X axis.
- The control changes the **C** angle in the List View area.
- ▶ Select an element in the positive direction of the Y axis
- The control orients the Y and Z axes.
- The control changes the **A** and **C** angles in the List View area.

13.3 Workpiece datum in the CAD file

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a working plane.

Related topics

- Presets in the machine

Further information: "Presets in the machine", Page 167

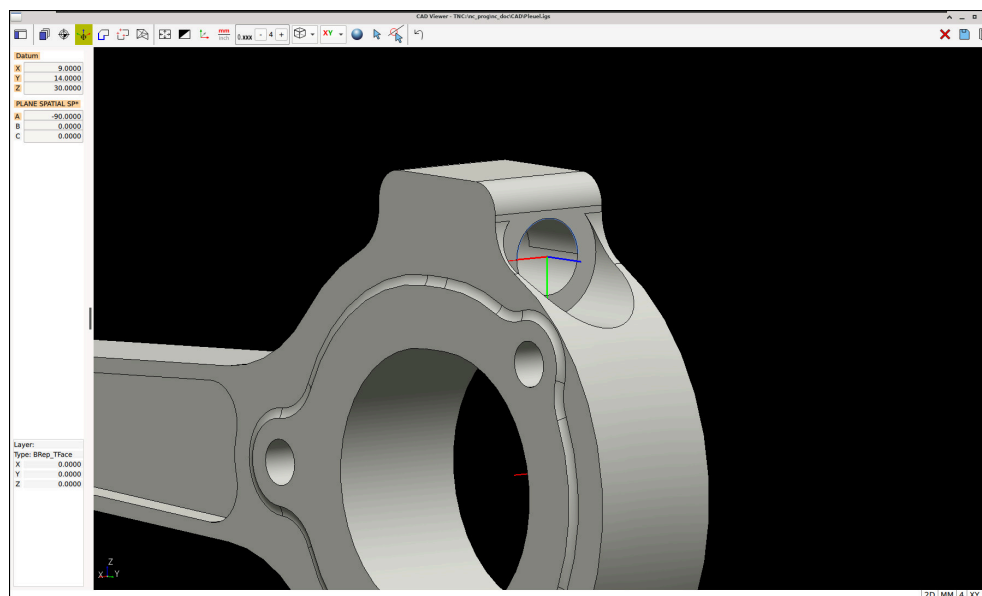
Description of function

When you select the **Datum** icon, the control displays the following information in the list view area:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the working plane

You can apply a workpiece datum set in CAD Viewer and shift it, if required, by entering values directly in the List View area.

The control displays values not equal to 0 in orange.



Workpiece datum for tilted machining

The datum with the orientation of the working plane can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD file", Page 275

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Setting the workpiece preset or workpiece datum and orienting the coordinate system", Page 276

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X+20 Y+25 Z-5
```

```
5 PLANE SPATIAL SPA+0 SPB+0 SPC+20 TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X+20 Y+25 Z-5
```

```
5 ;PLANE SPATIAL SPA+0 SPB+0 SPC+20 TURN MB MAX FMAX
```

You can save the workpiece preset and workpiece datum information to a file or the clipboard even without the CAD Import software option (#42 / #1-03-1).



The control retains the content of the clipboard only as long as **CAD Viewer** is open.

13.4 Loading contours and positions to NC programs with CAD Import (#42 / #1-03-1)

Application

You can open CAD files directly on the control to extract contours or machining positions from them. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs by default contain only **L** and **CC/C** blocks.

Related topics

- Using point tables

Further information: User's Manual for Machining Cycles

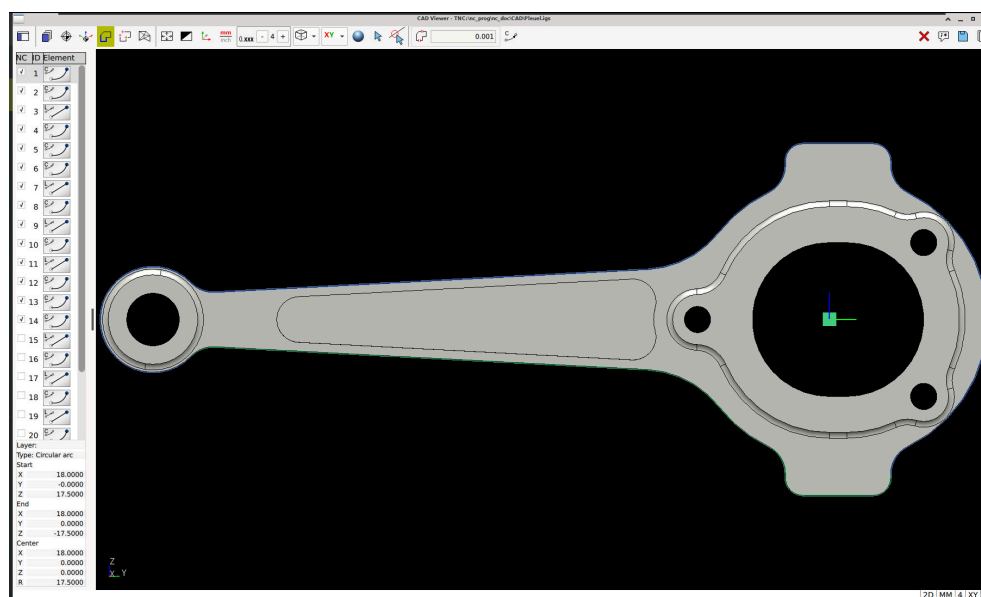
Requirement

- Software option CAD Import (#42 / #1-03-1)

Description of function

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to additional software tools (e.g., **Leafpad** or **Gnumeric**).

Further information: "Opening files with additional software", Page 594



CAD model with marked contour

Icons in the CAD Import

With the CAD Import, the control shows the following additional functions in the menu bar:

Icon	Meaning
	Set the transition tolerance The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred during drawing creation. The default setting is 0.001 mm.
 	C or CR You can select whether the control will output circular contours C or CR in the NC program.
	Show connections between two positions The control hides and displays the tool paths between the positions.
	Apply path optimization The control optimizes the tool traverse movement between the machining positions. When you select the icon again, the control will discard the optimization.
	Find circles according to diameter range. Load center coordinates to the position list The control opens the Find circle centers by diameter range window. You can filter by diameters as well as by depths.

Applying contours

The following elements can be selected as a contour:

- Line segment
- Full circle
- Pitch circle
- Polyline
- Any curves (e.g., splines, ellipses)

Linearization

CAD Viewer linearizes all of the contours that are not in the working plane.

During linearization, **CAD Viewer** subdivides a contour into individual segments. From these segments, CAD Import creates straight lines **L** and circular arcs **C** or **CR** that are as long as possible.

Thanks to linearization, it is also possible to import contours with CAD Import that cannot be programmed with the path functions of the control, such as splines.

The higher you define the resolution by specifying decimal places, the lower is the deviation from the imported contour. In any case, the deviation is less than 0.001 mm or 0.0001 inches.

Further information: "Screen layout", Page 269



You can prevent the linearization of, for example, circles that are not in the working plane. Select the working plane in which the circle has been defined.

Applying positions

You can also use the CAD Import to save positions (e.g., for holes).

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection
- Multiple selection within a range
- Multiple selection using search filters

Further information: "Select positions", Page 285

The following file types are available:

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

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

Example: active icon

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

Example: inactive icon

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



CAD Viewer also considers circles that consist of two semicircles to be one machining position.

Multi-selection filter settings

If you use the quick-selection function to mark positions, the **Find circle centers by diameter range** window opens. You can filter the diameter or depth values, referencing the workpiece datum, by means of the buttons below the displayed value. The control will only load the selected diameter or depth values.

The **Find circle centers by diameter range** window provides the following buttons:

Button	Meaning
<<	■ The control shows the smallest diameter found. ■ The control shows the smallest depth found. This filter is active by default.
<<	■ The control sets the filter for the largest diameter to the value selected for the smallest diameter. ■ The control sets the filter for the largest depth to the value selected for the smallest depth.
<	■ The control shows the next smaller diameter found. ■ The control shows the next smaller depth found.
>	■ The control shows the next larger diameter found. ■ The control shows the next larger depth found.
>>	■ The control sets the filter for the smallest diameter to the value selected for the largest diameter. ■ The control sets the filter for the smallest depth to the value selected for the largest depth.
>>	■ The control shows the largest diameter found. ■ The control shows the largest depth found. This filter is active by default.

13.4.1 Selecting and saving a contour



- The following instructions apply to the use of a mouse. You can also perform all steps with touch gestures.

Further information: "Manipulating the graphics in CAD Viewer", Page 274

- Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.

Selecting and saving a contour with existing contour elements

To select and save a contour with existing contour elements:



- ▶ Select **Contour**
- ▶ Place the cursor on the first contour element
- ▶ The control shows the suggested direction of rotation as a dashed line.
- ▶ If necessary, move the cursor towards the more distant end point.
- ▶ The control changes the suggested direction of rotation.
- ▶ Select the contour element
- ▶ The selected contour element is displayed in blue and is marked in the List View area.
- ▶ Other contour elements are shown in green.



The control suggests the contour that deviates least from the suggested direction. To change the suggested contour path, you can select paths independently of the existing contour elements.

- ▶ Select the last desired contour element
- ▶ All contour elements up to the selected element are shown in blue and are marked in the List View area.
- ▶ Activate the output of comments with workpiece information, if desired
- ▶ Select **Save entire list content to a file**
- ▶ The control opens the **Define file name for contour program** window.
- ▶ Enter the desired name
- ▶ Select the path to the storage location
- ▶ Select **Save**
- ▶ The selected contour is saved as an NC program.



- Alternatively, you can use the **Copy entire list contents to clipboard** icon to copy the selected contour to the clipboard and then paste it into an existing NC program.
- If you press the **CTRL** key and simultaneously select an element, the control deselects the element for export.

Selecting paths independently of existing contour elements

To select a path independently of existing contour elements:



- ▶ Select **Contour**



- ▶ Select **Select**, if necessary
- The icon changes, and the control activates the **Add** mode.
- ▶ Place the cursor relative to the desired contour element
- The control displays selectable points:
 - End point or center point of a line or curve
 - Quadrant transitions or center of a circle
 - Points of intersection between existing elements
- ▶ Select the desired point
- ▶ Select more contour elements



If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line. If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.

13.4.2 Select positions



- The following instructions apply to the use of a mouse. You can also perform all steps with touch gestures.
Further information: "Manipulating the graphics in CAD Viewer", Page 274
- Deselecting, deleting, and saving of elements works in the same way for applying contours and positions.
Further information: "Selecting and saving a contour", Page 284

Individual selection

To select individual positions (e.g., holes):



- ▶ Select **Positions**
- ▶ Position the cursor on the desired element
- The control shows the circumference and center point of the element in orange.
- ▶ Select the desired element
- The control highlights the selected element in blue and displays it in the List View area.
- ▶ Insert **FMAX** and **M99** in the NC program if necessary

FMAX
M99

Multiple selection within an area

To select multiple positions within an area:



- ▶ Select **Positions**
- ▶ Drag a box around the area while holding down the left mouse button
- ▶ The control opens the **Find circle centers by diameter range** window. The window shows the identified diameter and depth values.
- ▶ Change the filter settings as needed
- ▶ Select **OK**
- ▶ The control loads all positions within the selected diameter and depth ranges into the List View area.
- ▶ The control shows the traverse distance between the positions.
- ▶ Insert **FMAX** and **M99** in the NC program if necessary

FMAX
M99

Multiple selection by search filter

To select multiple positions using a search filter:



- ▶ Select **Positions**
- ▶ Select **Find circles according to diameter range. Load center coordinates to the position list**
- ▶ The control opens the **Find circle centers by diameter range** window. The window shows the identified diameter and depth values.
- ▶ Insert **FMAX** and **M99** in the NC program if necessary

FMAX
M99

Notes

- Set the correct unit of measure so that **CAD Viewer** shows the correct values.
- Ensure that the unit of measure used in the NC program matches with that used in the **CAD Viewer**. Elements that have been copied from the **CAD Viewer** to the clipboard do not contain any information about the unit of measure.
- The control retains the content of the clipboard only as long as **CAD Viewer** is open.
- **CAD Viewer** also considers circles that consist of two semicircles to be one machining position.
- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.
- CAD Import outputs the radii of the circular arcs as comments. At the end of the generated NC blocks, CAD Import displays the smallest radius to help you select the most suitable tool.

Notes on Contour Transfer

- If you double-click a layer in the List View area, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.
- Select the first contour element such that approach without collision is possible.
- You can even select a contour if the designer has saved it on different layers.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.
As an alternative, select the Remove mode.

13.5 Generating STL files with 3D mesh (#152 / #1-04-1)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective fixture and tool holder files, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

- Fixture management
- Exporting the simulated workpiece as an STL file
- Using an STL file as workpiece blank

Further information: Programming and Testing User's Manual

Requirement

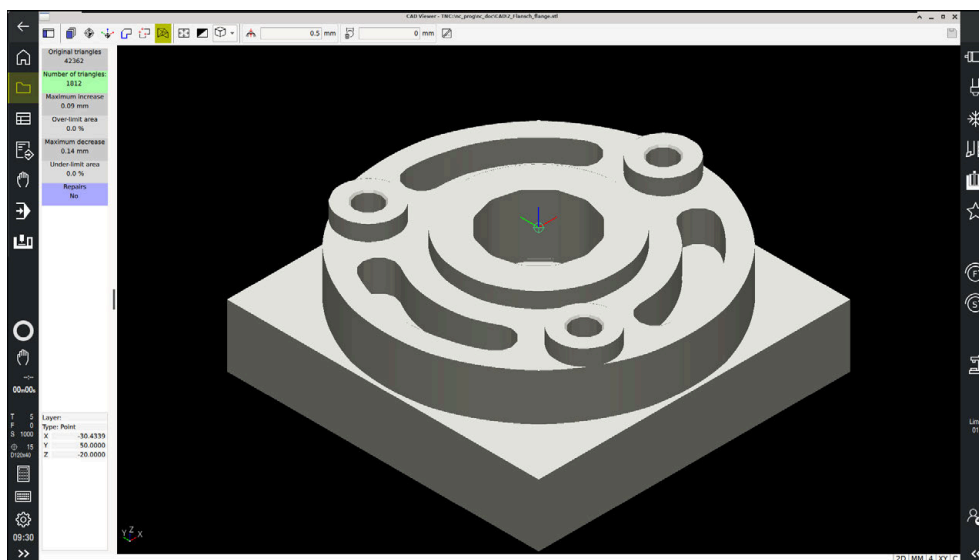
- Software option CAD Model Optimizer (#152 / #1-04-1)

Description of function

When you select the **3D mesh** icon, the control changes to **3D mesh** mode. The control covers the 3D model displayed in **CAD Viewer** with a mesh of triangles.

The control simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, for example as a workpiece blank with the **BLK FORM FILE** function.



3D model in **3D mesh** mode

The simplified model or parts of it may be smaller or larger than the original model. The result depends on the quality of the original model and the settings selected in **3D mesh** mode.

The List View area shows the following information:

Option	Meaning
Original triangles	Number of triangles in the original model
Number of triangles:	Number of triangles with active settings in the simplified model i If this option is highlighted in green, the number of triangles is in the optimum range. You can further reduce the number of triangles using the available functions. Further information: "Functions for the simplified model", Page 290
Maximum increase	Maximum increase of the triangle mesh
Over-limit area	Surface increase in percent compared to the original model
Maximum decrease	Maximum decrease of the triangle mesh compared to the original model
Under-limit area	Surface decrease in percent compared to the original model
Repairs	Indicates whether the original model has been repaired or not If it has been repaired, the control indicates the type of repair (e.g., Hole Int Shells). This indication consists of the following items: ■ Hole CAD Viewer closed holes in the 3D model. ■ Int CAD Viewer removed self-intersections. ■ Shells CAD Viewer joined multiple separate solids.
Time	Duration of simplification

In order to use STL files for control functions, the saved files must meet the following requirements:

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

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD Viewer provides the following functions:

Icon	Meaning
	Allowed simplification Use this function to simplify the output model by the specified tolerance. The higher the value, the more the surfaces may deviate from the original.
	Remove holes <= diameter Use this function to remove holes and pockets up to the specified diameter from the original model.
	Only optimized mesh shown The control shows the simplified model only.
	Original is displayed The control shows the simplified model, superimposed with the original mesh from the original file. You can use this function to evaluate deviations.
	Save Use this function to save the simplified 3D model with the selected settings as an STL file.

13.5.1 Positioning the 3D model for rear-face machining

To position an STL file for rear-face machining:

- ▶ Export the simulated workpiece as an STL file

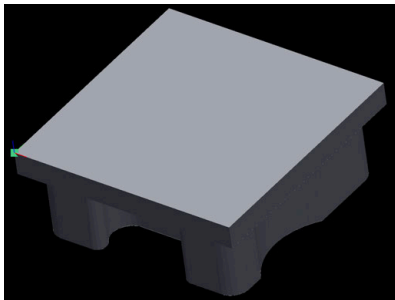
Further information: Programming and Testing User's Manual



- ▶ Select the **Files** operating mode



- ▶ Select the exported STL file
- ▶ The control opens the STL file in **CAD Viewer**.
- ▶ Select **Preset**
- ▶ The control displays information on the preset position in the List View area.
- ▶ Enter the value of the new preset in the **Preset** area (e.g., **Z-40**)
- ▶ Confirm your input
- ▶ Orient the coordinate system by specifying values under **PLANE SPATIAL SP*** (e.g., **A+180** and **C+90**)
- ▶ Confirm your input



- ▶ Select **3D mesh**
- ▶ The control opens the **3D mesh** mode and simplifies the 3D model using the default settings.
- ▶ Further simplify the 3D model using the **3D mesh** mode functions, if required.

Further information: "Functions for the simplified model", Page 290



- ▶ Select **Save**
- ▶ The control opens the **Define file name for 3D mesh** window.
- ▶ Enter the desired name
- ▶ Select **Save**
- ▶ The control saves the STL file positioned for rear-face machining.



The resulting file can then be used for rear-face machining with the **BLK FORM FILE** function.

Further information: Programming and Testing User's Manual

14

User aids

14.1 Virtual keyboard of the control bar

Application

You can use the virtual keyboard for entering NC functions, letters, and numbers, and for navigation.

The virtual keyboard offers the following modes:

- NC input
- Text input
- Formula entry

Description of function

The control opens NC input mode by default after the start procedure.

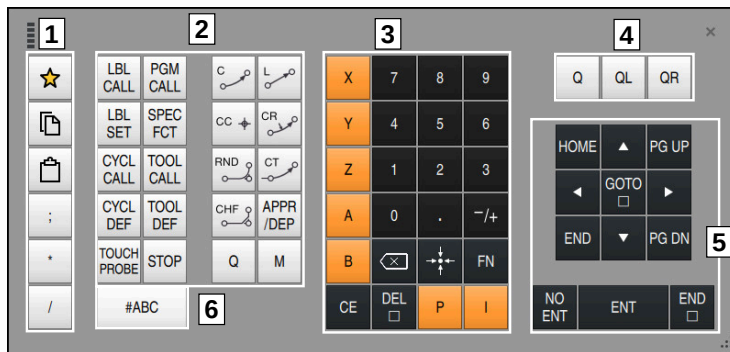
You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

The control remembers the position and mode of the virtual keyboard until it is shut down.

The **Keyboard** workspace provides the same functions as the virtual keyboard.

The **+**, **-**, *****, **/**, **(** and **)** keys permit calculations concerning numerical values in input fields and table rows.

NC input areas



Virtual keyboard in NC input mode

NC input mode contains the following areas:

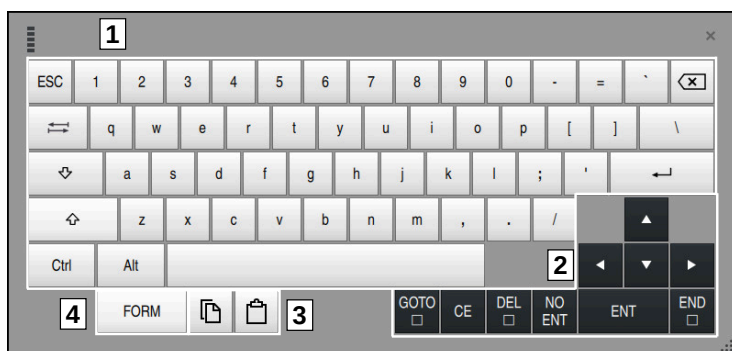
- 1 File functions
 - Define favorites
 - Copy
 - Paste
 - Add comment
 - Add structure item
 - Hide NC block
- 2 NC functions
- 3 Axis keys and numerical input
- 4 Q parameters
- 5 Navigation and dialog keys
- 6 Switch to text input



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

- **Q**
- **QL**
- **QR**

Text input areas

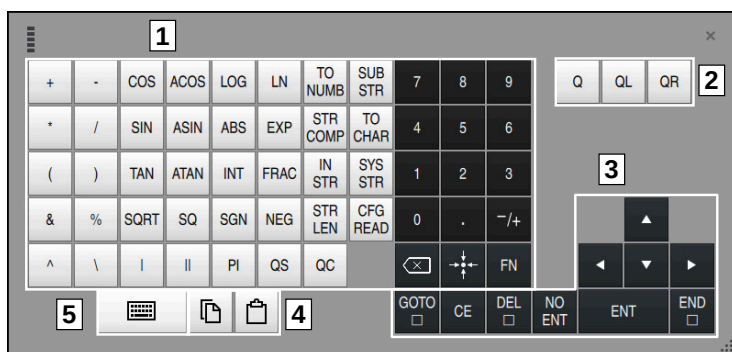


Virtual keyboard in text input mode

Text input mode contains the following areas:

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

Formula input areas



Virtual keyboard in formula input mode

Formula input mode contains the following areas:

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

14.1.1 Opening and closing the virtual keyboard

To open the virtual keyboard:



- Select the **virtual keyboard** on the control bar
- The control opens the virtual keyboard.

To close the virtual keyboard:



- Select the **virtual keyboard** when the virtual keyboard is open



- Or press **Close** in the virtual keyboard
- The control closes the virtual keyboard.

14.2 Message menu on the information bar

Application

In the message menu of the information bar, the control shows pending errors and notes. When opened, the control displays detailed information about the messages.

Description of function

The control uses the following symbols to differentiate between the types of messages:

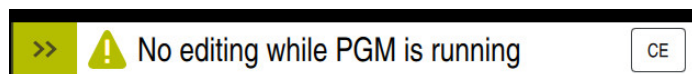
Symbol	Message type	Meaning
	Error Question type	The control displays a dialog with several options you can select from. You cannot clear this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.
	Error Reset type	The control must be restarted. This message cannot be cleared.
	Error Emergency-stop type	The control performs an emergency stop. An error message can only be cleared after the cause has been eliminated.
	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Information	You can continue without clearing the message. You can clear the information at any time.
	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
		No pending messages

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content



Collapsed message menu on the information bar

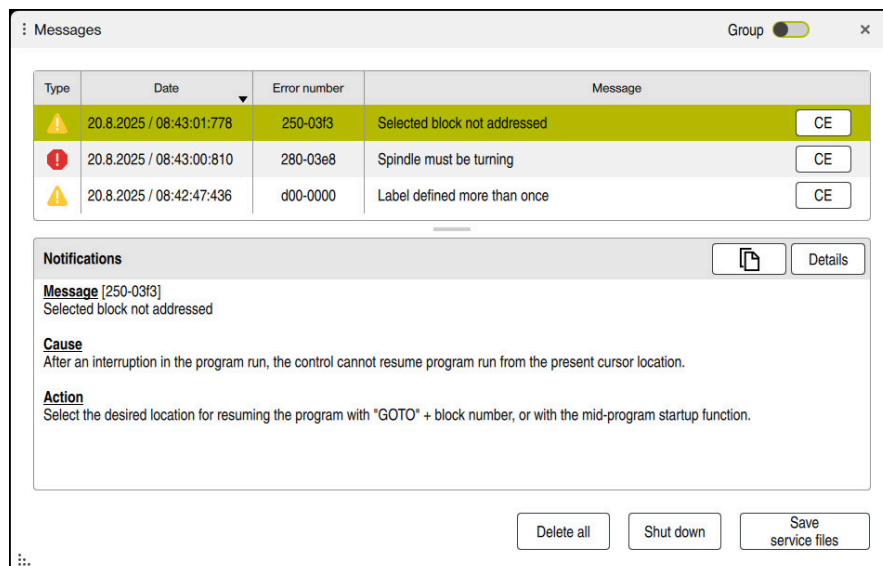
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.



Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

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

Deleting messages

Messages can be deleted in the following ways:

- **CE** key
- **CE** button in the message menu
- **Delete all** button in the message menu

Details

Press the **Details** button to show or hide internal information about the message. This information is of importance in case servicing is necessary.

Group

If you activate the **Group** toggle switch, the control displays all messages with the same error number in one row. This makes the list of messages shorter and easier to read.

Under the error number, the control displays the quantity of messages. Use **CE** to clear all messages of a group.

Service file

Click the **Save service files** button to open the **Save service files** window.

In the **Save service files** window, you can create service files in the following ways:

- If an error occurs, you can create a service file manually.
Further information: "Creating a service file manually", Page 299
- If an error occurs repeatedly, a service file can be created automatically by means of the error number. Once the respective error occurs, the control saves a service file.

Further information: "Creating a service file automatically", Page 300

Service files help service technicians in troubleshooting the problem. The control saves data that provide information about the current machine and operation status, such as active NC programs up to 10 MB, tool data, and keystroke logs.

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

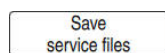
If you create multiple service files with the same name, the control saves a maximum of five files and then deletes the file with the oldest timestamp, if necessary. Make a backup of the service files you created (e.g., by moving them to a different folder).

14.2.1 Creating a service file manually

To create a service file manually:



- Expand the message menu



- Select **Save service files**
- > The control opens the **Save service file** window.
- Enter the file name



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

14.2.2 Creating a service file automatically

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

To specify a new error number:



Save
service files

Setting for autosave

- ▶ Expand the message menu
- ▶ Select **Save service files**
 - > The control opens the **Save service file** window.
- ▶ Select **Setting for autosave**
 - > The control opens a table of error numbers.
 - ▶ Enter the desired error number
 - ▶ Select the **Active** checkbox
 - > If the error occurs, the control automatically creates a service file.
 - ▶ Enter a comment, if applicable (e.g., to describe the problem)

15

The MDI application

Application

The **MDI** application allows you to execute individual NC blocks outside of the context of an NC program (e.g., **PLANE RESET**). When you press the **NC Start** key, the control will run the NC blocks separately.

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

Related topics

- Creating NC programs

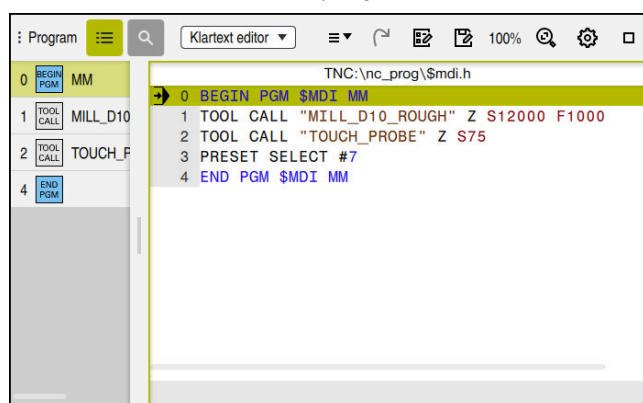
Further information: Programming and Testing User's Manual

- Running NC programs

Further information: "Program run", Page 339

Description of function

If you program using the millimeter unit of measurement, the control will use the NC program **\$mdi.h** by default. If you program using the inch unit of measurement, the control will use the NC program **\$mdi_inch.h**.



The **Program** workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **Document**

Further information: Programming and Testing User's Manual

- **Help**

Further information: Programming and Testing User's Manual

- **Positions**

Further information: "The Positions workspace", Page 121

- **Program**

Further information: Programming and Testing User's Manual

- **Simulation**

Further information: Programming and Testing User's Manual

- **Status**

Further information: "The Status workspace", Page 129

- **Keyboard**

Further information: "Virtual keyboard of the control bar", Page 294

Icons and buttons

In the **MDI** application, the function bar provides the following buttons:

Icon or button	Meaning
	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Insert NC function	The control opens the Insert NC function window. Further information: Programming and Testing User's Manual
Q info	The control opens the Q parameter list window, where you can see and edit the current values and descriptions of the variables. Further information: Programming and Testing User's Manual
GOTO block number	Mark an NC block to be run without considering any previous NC blocks Further information: Programming and Testing User's Manual
/ Skip block Off/On	Hide NC blocks with the / character. NC blocks hidden with a / character will be ignored during program run as soon as the Skip block toggle switch is active. Further information: Programming and Testing User's Manual
Skip block	If the toggle switch is active, the control does not execute NC blocks dimmed with a / character. If the toggle switch is active, then the control dims the NC blocks to be skipped. Further information: Programming and Testing User's Manual
; Comment Off/On	Insert or remove a ; character in front of an NC block. If an NC block begins with a ; character, then the block is a comment. Further information: Programming and Testing User's Manual
F LIMIT	Use this function to activate a feed-rate limit and define its value. Further information: "Feed rate limit F LIMIT", Page 345
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC (#145 / #2-30-1)). Further information: "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 266
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. Further information: User's Manual for Machining Cycles
Edit	The control opens the context menu. Further information: Programming and Testing User's Manual
Tools	The control opens the Tool management application in the Tables operating mode. Further information: "Tool management ", Page 184
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. Further information: "Interrupting, stopping or canceling program run", Page 346
Reset program	If you select Internal stop , the control activates this button. The control resets any modally active program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always execute NC blocks in **Single Block** mode. After the control has executed an NC block, program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 346

The block numbers of all NC blocks that you have successively run are shown in green.

The control saves the following data in this state:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

Notes

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
 - Cursor movement to another NC block
 - The jump command **GOTO** to another NC block
 - Editing an NC block
 - Modifying the values of variables by using the **Q parameter list** window
 - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

- In the **MDI** application, you can create and execute NC programs step by step. Then you can use **Save as** to save the current contents with a different file name.
- The following functions are not available in the **MDI** application:
 - Calling of an NC program with **PGM CALL**
 - Test run in the **Simulation** workspace
 - **Manual traverse** and **Approach position** functions while program run is interrupted
 - **Block scan** function
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

16

**Touch probe
functions in
the Manual
operating mode
(#17 / #1-05-1)**

16.1 Fundamentals

Application

The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Related topics

- Automatic touch probe cycles for the workpiece
Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Preset table
Further information: "Preset table *.pr", Page 415
- Datum table
Further information: Programming and Testing User's Manual
- Reference systems
Further information: "Reference systems", Page 196
- Preassigned variables
Further information: Programming and Testing User's Manual

Requirements

- Touch Probe Functions software option (#17 / #1-05-1)



If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.

- Calibrated workpiece touch probe
Further information: "Calibrating the workpiece touch probe", Page 321

Description of function

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

- Setting a preset
- Determine and compensate for workpiece misalignment
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- **Measure the tool**
- **Set up fixtures** (#140 / #5-03-2)
Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234
- **Set up the workpiece** (#159 / #1-07-1)
Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 326

Overview

The touch probe functions are structured in the following groups:

Probe the angle

The **Probe the angle** group contains the following touch probe functions:

Button	Function
Plane (PL) 	Use the Plane (PL) function to determine the solid angle of a plane. You then save the values in the preset table and align the plane, if necessary.
Plane over cylinder (PLC) 	Use the Plane over cylinder (PLC) function to probe one or two cylinders, each at two different heights. The control calculates the solid angle of a plane from the points probed. You then save the values in the preset table and align the plane, if necessary.
Rotation (ROT) 	Use the Rotation (ROT) function to determine the skew of a workpiece using a straight line. Then save the determined skew as a basic transformation or offset in the preset table. Further information: "Determining and compensating for the basic rotation of a workpiece", Page 318
Intersection point (P) 	Use the Intersection point (P) function to probe four probing objects. The probing objects can be either positions or circles. The control determines the intersection of the axes and the skew of the workpiece from the objects that have been probed. You can set the intersection point as a preset. You can transfer the determined skew to the preset table as a basic transformation or as an offset.



The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.

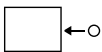
Further information: "Preset table *.pr", Page 415

You can compensate for the workpiece misalignment by rotating the table only if the machine is designed with a rotary table axis that is oriented perpendicularly with respect to the workpiece coordinate system **W-CS**.

Further information: "Comparison of offset and 3D basic rotation", Page 337

Probe the position

The **Probe the position** group contains the following touch probe functions:

Button	Function
Position (POS) 	You can use the Position (POS) function to probe a position in the X axis, Y axis or Z axis. Further information: "Setting a preset in a linear axis", Page 315
Circle (CC) 	The Circle (CC) function is used to determine the coordinates of a circle center point (e.g., for a hole or for a stud). Further information: "Setting the circle center point of a stud by means of the automatic probing method ", Page 316
Circle pattern (CPAT) 	The Circle pattern (CPAT) function is used to determine the center point coordinates of a circle pattern.
Centerline (CL) 	The Centerline (CL) function is used to determine the center point of a ridge or slot.

Additional functions

The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe 	The Calibrate touch probe function is used to determine the length and radius of a workpiece touch probe. Further information: "Calibrating the workpiece touch probe", Page 321
Measure tool 	The Measure tool function allows you to measure tools by touch-off. In this function, the control supports milling tools, drilling tools and turning tools. Further information: "Measuring the tool by scratching", Page 334
Set up fixtures 	The Set up fixtures function is used to determine the position of a clamping device in the working space using a workpiece touch probe (#140 / #5-03-2). Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234
Set up the workpiece 	The Set up the workpiece function is used to determine the position of a workpiece in the working space using a workpiece touch probe (#159 / #1-07-1). Further information: "Setting up the workpiece with graphical support (#159 / #1-07-1)", Page 326

Icons and buttons

General icons and buttons in the touch probe functions

The following icons and buttons are available, depending on the selected touch probe function:

Icon or button	Meaning
	Exit probing
	Select the workpiece preset and the pallet preset and edit values, if required The control shows the number of the active preset to the right of the icon. Further information: "The Change the preset window", Page 313 Further information: "Preset table *.pr", Page 415
	Display help graphics for the selected touch probe function
	Select the probing direction
	actual position capture
	Position Manually approach and probe positions on a straight surface You position the workpiece touch probe and start the probing process manually.
	Circle center A Automatically approach and probe positions on a stud or in a hole You manually pre-position the touch probe to the first probing point and define the parameters for the selected touch probe function. When you start the touch probe function, the control automatically positions and performs probing. After the defined number of touch points has been reached, the control determines the center of the stud or the hole. After the last probing process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.
	Circle center M Automatically approach and probe positions on a stud or in a hole You position the workpiece touch probe and contact the individual probing points manually. If you select the Assume measurement result button, the control determines the center of the stud or hole.
Tools	The control opens the Tool management application in the Tables operating mode. Further information: "Tool management ", Page 184

Icon or button	Meaning
Handwheel	The control displays this toggle switch if a handwheel without display is configured on the control. If the handwheel is active, the operating mode's icon in the sidebar changes. Further information: "Electronic handwheel", Page 429
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. Further information: "Interrupting, stopping or canceling program run", Page 346

Icons and buttons for calibration

The control offers the following functions for calibrating a 3D touch probe:

Icon or button	Meaning
	Calibrating the length of a 3D touch probe
	Calibrating the radius of a 3D touch probe
Apply calibration data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 321

You can calibrate a 3D touch probe by using a calibration standard, such as a calibrating ring.

The control provides the following options:

Icon	Meaning
	Measure the radius and the center offset using a calibration ring
	Measure the radius and the center offset using a stud or a calibration pin
	Measure the radius and the center offset using a calibration sphere

Buttons in the Working plane is inconsistent! window

If the positions of the rotary axes do not match the tilting situation in the **Manual operation** and **Setup** applications, the control opens the **Working plane is inconsistent!** window. The status of the tilting situation is shown in the **3-D rotation** window.

Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 218

The control offers the following functions in the **Working plane is inconsistent!** window:

Button	Meaning
3-D ROT Apply status	■ If the positions of the rotary axes match the tilting situation during program run (Program Run operating mode and MDI application), the control will apply this tilting situation. ■ Otherwise the control will apply the current rotary-axis angles as the tilting angles. The 3-D rotation window shows the applied spatial or axis angles. Further information: "The 3-D rotation window (#8 / #1-01-1)", Page 218
3-D ROT Ignore status	The control ignores the tilting situation and moves the axes as if the rotary axes were at their zero position.
Align the rotary axes	The control positions the rotary axes in a manner suitable to the current tilting situation. If, for example, no tilting function is active and the rotary axes have been moved, the control then returns the rotary axes to their zero position.

Buttons for touch-probe functions

The control offers these buttons in the touch-probe functions during or after probing:

Button	Meaning
Remove last measurement	The control resets the last touch point. Only with the Circle center M measuring method
Assume measurement result	The control determines the circle center of the probed stud or the probed hole. The control enters the circle center in the Measuring column. Only with the Circle center M measuring method after at least three touch points.
Compensate the active preset	The control sets the active workpiece preset to the entered nominal value. If the workpiece preset is locked, the control shows a window with a confirmation request. Further information: "Preset table *.pr", Page 415
Correct the datum	The control opens the Correct the datum window. In the Nominal value area, enter the desired distance of the measuring result from the workpiece datum. Based on this information, the control calculates the required table value relative to the active workpiece preset. The control enters the calculated value in the selected line of the datum table. Further information: Programming and Testing User's Manual
Align rotary table	The control moves the rotary axes to compensate for the measured misalignment.
Correct the pallet reference point	The control sets the active pallet datum to the entered nominal value. If the pallet preset is locked, the control shows a window with a confirmation request. Further information: Programming and Testing User's Manual

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

The Change the preset window

In the **Change the preset** window you can select a preset or edit the values of a preset.

Further information: "Preset management", Page 209

The **Change the preset** window provides the following buttons:

Icon or button	Meaning
	The control shows the preset table. Further information: Programming and Testing User's Manual
	The control shows the pallet preset table. Further information: Programming and Testing User's Manual
Reset basic rotation	The control resets the values from the columns SPA , SPB and SPC .
Reset offsets	The control resets the values from the columns A_OFFS , B_OFFS and C_OFFS .
Apply changes and delete existing probe objects	The control activates the selected preset and rejects the touch points used so far. Then the control closes the window.
Apply	The control saves the changes and the selected preset. Then the control closes the window.
Reset	The control cancels the changes and restores the initial condition.
Cancel	The control closes the window without saving.

i If you change a value, the control marks this value with a blue dot.
If you accept the changes and the active preset is locked, the control shows a window with a confirmation request.

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. The control indicates in the **Positions** workspace whether a pallet preset is active and if yes, which one. Since the values of the pallet preset table are neither visible nor editable outside the **Setup** application, there is a risk of collision during any movement!

- ▶ Refer to the machine manufacturer's documentation
- ▶ Use pallet presets only in conjunction with pallets
- ▶ Change pallet presets only after discussion with the machine manufacturer
- ▶ Check the pallet preset in the **Setup** application before you start machining

Log file of touch-probe functions

After executing any touch-probe function, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in the **TCHPRMAN.html** file.

If you have not defined a path in the machine parameter **FN16DefaultPath** (no. 102202), the control will store the TCHPRMAN.html file directly under **TNC:**.

If you run several touch probe functions in a row, the control stores the measured values below each other.

16.1.1 Setting a preset in a linear axis

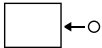
To probe the preset in any axis:



- ▶ Select the **Manual** operating mode



- ▶ Call the workpiece touch probe as a tool
- ▶ Select the **Setup** application



- ▶ Select **Position (POS)**
- The control opens the **Position (POS)** touch probe function.



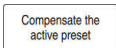
- ▶ If necessary, check the values of the active preset or choose a different preset
- ▶ Use the axis keys to position the workpiece touch probe at the desired probing position (e.g., above the workpiece in the workspace)



- ▶ Select the probing direction (e.g., **Z-**)



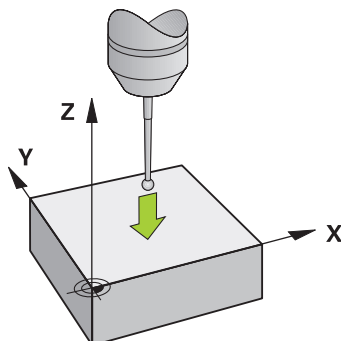
- ▶ Press the **NC Start** key
- The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- The control shows the measurement results.
- ▶ In the **Nominal value** area, enter the new preset of the probed axis (e.g., **1**)
- ▶ Select **Compensate the active preset**
- The control sets the active preset to the entered nominal value.



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



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



16.1.2 Setting the circle center point of a stud by means of the automatic probing method

To probe a circle center point:



- ▶ Select the **Manual** operating mode



- ▶ Call the workpiece touch probe as a tool
- ▶ Select the **Setup** application



- ▶ Select **Circle (CC)**
- ▶ The control opens the **Circle (CC)** touch probe function.



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



- ▶ Position the workpiece touch probe at the desired probing position in the workspace
- ▶ Select **Type of contour** (e.g., stud)
- ▶ Enter **Diameter** (e.g., 60 mm)
- ▶ Enter the **Safety clearance (min. value = SET_UP)** if required



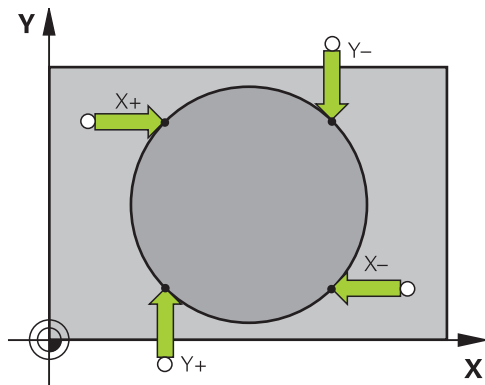
The control suggests the sum of the value in the **SET_UP** column of the touch probe table and the ball tip radius as a safety clearance.

- ▶ Enter **Starting angle** (e.g., -180°)
- ▶ Enter **Angular length** (e.g., 360°)
- ▶ Enter **No. of touch points** (e.g., 4)
- ▶ Position the workpiece touch probe at the desired probing position next to the workpiece and below the workpiece surface
- ▶ If necessary, turn the feed-rate potentiometer to zero
- ▶ Press the **NC Start** key
- ▶ If necessary, slowly turn up the feed-rate potentiometer
- ▶ The control executes the touch probe function based on the data entered.
- ▶ The control shows the measurement results.
- ▶ In the **Nominal value** area, enter the new preset of the probed axes (e.g., 0)
- ▶ Select **Compensate the active preset**
- ▶ The control sets the active preset to the entered nominal value.
- ▶ Select **Exit probing**
- ▶ The control closes the **Circle (CC)** touch probe function.



Compensate the active preset





16.1.3 Determining and compensating for the basic rotation of a workpiece

To probe the basic rotation of a workpiece:



- ▶ Select the **Manual** operating mode



- ▶ Call the workpiece touch probe as a tool
- ▶ Select the **Setup** application



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



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

- ▶ Position the workpiece touch probe at the desired probing position in the workspace



- ▶ Select the probing direction (e.g., **Y+**)

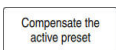


- ▶ Press the **NC Start** key
- The control executes the first probing operation and limits the subsequently selectable probing directions.

- ▶ Position the workpiece touch probe at the second probing position in the workspace



- ▶ Press the **NC Start** key
- The control executes the probing operation and then shows the measurement results.



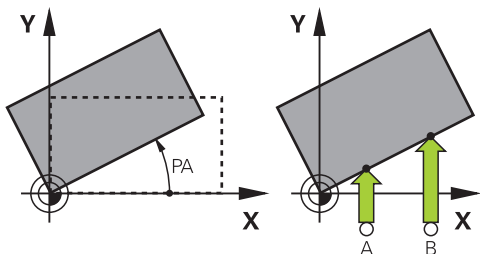
- ▶ Activate the **Basic rotation** toggle switch as needed
- ▶ Select **Compensate the active preset**
- The control transfers the determined basic rotation with respect to the tool axis to, for example, the **SPC** column of the active row of the preset table.



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



- ▶ Select **Exit probing**
- The control closes the **Rotation (ROT)** touch probe function.



Instead of **Basic rotation**, you can also select the **Table rotation** toggle switch. The control then transfers the values into columns **A_OFFS**, **B_OFFS**, or **C_OFFS** of the preset table.

16.1.4 Using touch probe functions with mechanical probes or dial gages

If your machine does not have an electronic 3D touch probe, you can also use mechanical probes or tool touch-off for all manual touch probe functions with manual probing methods.

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

To determine a basic rotation with a mechanical probe:



- ▶ Select the **Manual** operating mode



- ▶ Insert the tool (e.g., analog 3D probe or feeler lever gage)
- ▶ Select the **Setup** application



- ▶ Select **Rotation (ROT)**



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



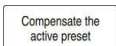
- ▶ Select the probing direction (e.g., **Y+**)
- ▶ Move the mechanical probe to the first position to be captured by the control.



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



- ▶ Select **Apply position**
 - > The control saves the current position.
 - ▶ Activate the **Basic rotation** toggle switch as needed



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



- ▶ Select **Exit probing**
 - > The control closes the **Rotation (ROT)** touch probe function.



Instead of **Basic rotation**, you can also select the **Table rotation** toggle switch. The control then transfers the values into columns **A_OFFS**, **B_OFFS**, or **C_OFFS** of the preset table.

Notes

- If you use a non-contacting tool touch probe (such as a laser touch probe), then you are using touch-probe functions from a third-party supplier. Refer to your machine manual.
- In the touch probe functions, you can switch between the measuring methods **Position**, **Circle center A** and **Circle center M** as desired. With the **Intersection point (P)** function, you can switch the measuring method after two completed measurements.
- The accessibility of the pallet preset table in the touch-probe functions depends on the machine manufacturer's configuration. Refer to your machine manual.
- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no workpiece touch probe is inserted, the actual position can be captured with **NC Start**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
 - Initial configuration
 - Broken stylus
 - Stylus replacement
 - Change in the probe feed rate
 - Irregularities caused, for example, when the machine heats up
 - Changes to the **TRACK** parameter
- If the touch point is not reached during the probing process, the control will display a warning. The probing process can be continued with **NC Start**.

Definition

Spindle tracking

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

16.2 Calibrating the workpiece touch probe

Application

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

Related topics

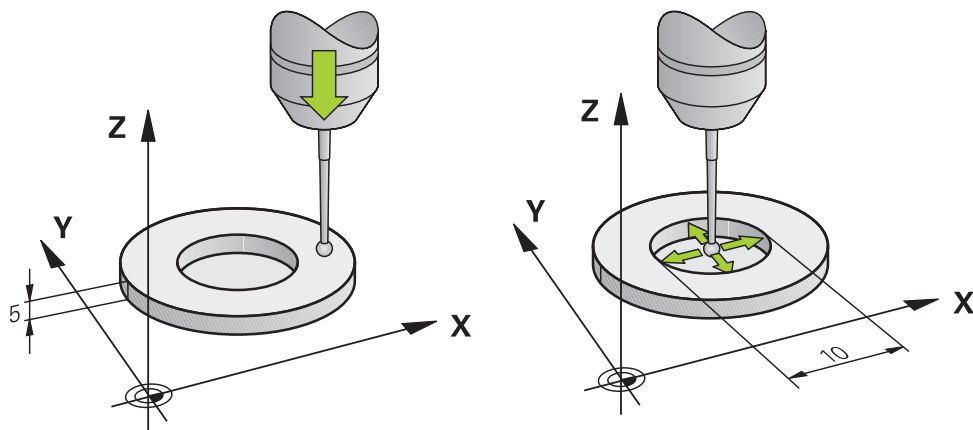
- Calibrate the workpiece touch probe automatically

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

- Touch probe table

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401

Description of function



During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a calibration ring or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 171

You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

Calibrating an L-shaped stylus

Before you calibrate an L-shaped stylus you first must define the parameters in the touch probe table. Based on these approximate values, the control can align the touch probe during the calibration and determine the actual values.

First, define the following parameters in the touch probe table:

Parameter	Value to be defined
CAL_OF1	Length of extension The extension is the angled length of the L-shaped stylus.
CAL_OF2	0
CAL_ANG	Spindle angle at which the extension is parallel to the main axis For this, manually position the extension in the positive direction of the main axis and read the value from the position display.

After the calibration, the control overwrites the previously defined values in the touch probe table with the determined values.

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401

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

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

Reversal measurement

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or pin (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

HEIDENHAIN touch probes are predefined as to whether or how a touch probe can be oriented. Other touch probes are configured by the machine manufacturer.

When calibrating the radius, up to three circular measurements can be taken depending on the possible orientation of the workpiece touch probe. The first two circular measurements determine the center offset of the workpiece touch probe. The third circular measurement determines the effective stylus tip radius. If orientation of the spindle is not possible or only a certain orientation is possible due to the workpiece touch probe, circular measurements are omitted.

16.2.1 Calibrating the length of the workpiece touch probe

To calibrate a workpiece touch probe using an overmilled surface in length:

- ▶ Measure the end milling cutter on the tool presetting device
- ▶ Store the measured end milling cutter in the tool magazine of the machine
- ▶ Enter the tool data of the end milling cutter in tool management
- ▶ Clamp the workpiece blank



- ▶ Select the **Manual** operating mode

- ▶ Replace the end milling cutter in the machine
- ▶ Switch on spindle (e.g., with **M3**)
- ▶ Use the handwheel to scratch the workpiece blank

Further information: "Setting a preset with milling cutters", Page 210

- ▶ Set preset in the tool axis (e.g., with **Z**)
- ▶ Position the end milling cutter next to the workpiece blank
- ▶ Set a small value in the tool axis (e.g., **-0.5 mm**)
- ▶ Overmill the workpiece blank using the handwheel
- ▶ Set the preset again in the tool axis (e.g., with **Z=0**)
- ▶ Switch off spindle (e.g., with **M5**)
- ▶ Insert the workpiece touch probe
- ▶ Select the **Setup** application



- ▶ Select **Calibrate touch probe**

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

- ▶ Select measuring method **Length**:

➢ The control displays the current calibration data.

- ▶ Enter a **Position of reference surface** (e.g., **0**)

- ▶ Position the workpiece touch probe close to the surface of the overmilled area



Check that the area to be probed is flat and free of chips before you start the touch probe function.



- ▶ Press the **NC Start** key

➢ The control performs the probing operation and then automatically retracts the workpiece touch probe to the starting point.

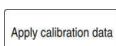
- ▶ Check results

- ▶ Select **Apply calibration data**

➢ The control transfers the calibrated length of the 3D touch probe to the tool table.

- ▶ Select **Exit probing**

➢ The control closes the **Calibrate touch probe** touch probe function.



16.2.2 Calibrating the radius of the workpiece touch probe

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

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



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



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



- ▶ Select the **Setup** application



- ▶ Select **Calibrate touch probe**



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



- ▶ Select measuring method **Radius**:



- ▶ Select calibration standard **Calibration ring**

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



- ▶ Press the **NC Start** key
- The workpiece touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

- ▶ Check results

Apply calibration data

- ▶ Select **Apply calibration data**

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



- ▶ Select **Exit probing**
- The control closes the **Calibrate touch probe** touch probe function.

Instructions for calibration

- In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.
- If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset in the touch probe table. The control uses the **TP_NO** parameter to link the data from the touch probe table with the data from the tool table.

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401

16.3 Setting up the workpiece with graphical support (#159 / #1-07-1)

Application

Use the **Set up the workpiece** function to determine the position and misalignment of a workpiece with only one touch-probe function and save it as a workpiece preset. During setup, you can probe curved surfaces.

The control supports you additionally by showing the setup situation and possible touch points in the **Simulation** workspace by means of a 3D model.

Related topics

- Touch-probe functions in the **Setup** application
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
- Generating an STL file of a workpiece
Further information: Programming and Testing User's Manual
- **Simulation** workspace
Further information: Programming and Testing User's Manual
- Setting fixtures with graphical support (#140 / #5-03-2)
Further information: "Measuring the position of a fixture with Set up fixtures (#140 / #5-03-2)", Page 234

Requirements

- Touch Probe Functions software option (#17 / #1-05-1)



If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.

- Model Aided Setup (#159 / #1-07-1) software option
- Touch probe properly defined in the tool management:
 - Sphere radius in parameter **R2**
Further information: "Parameters of the touch probe table tchprobe.tp", Page 402
- Workpiece touch probe calibrated
Further information: "Calibrating the workpiece touch probe", Page 321
- NC program with **BLK FORM** workpiece blank definition or STL file of the workpiece blank

Description of function

The control compares the positions of the probe points on the actual workpiece with a 3D model of the workpiece.

The more the 3D model corresponds to the actual workpiece, the higher the possible workpiece setup accuracy.

You have the following options for defining the 3D model:

- **BLK FORM** in the NC program
The control creates the 3D model using the workpiece blank definition.
- STL file of the workpiece with up to 300 000 triangles

The scope of the **Set up the workpiece** function depends on the Adv. Function Set 1 (#8 / #1-01-1) and Adv. Function Set 2 (#9 / #4-01-1) software options as follows:

- Both software options enabled:
You can tilt before setting up and incline the tool while setting up in order to probe even complex workpieces (e.g., shaped parts).
- Only Adv. Function Set 1 (#8 / #1-01-1) is enabled:
You can tilt before setting up. The working plane must be consistent. If you move the rotary axes between the touch points, the control will display an error message.



If the current coordinates of the rotary axes and the defined tilt angles (**3D ROT** window) match, the working plane is consistent.

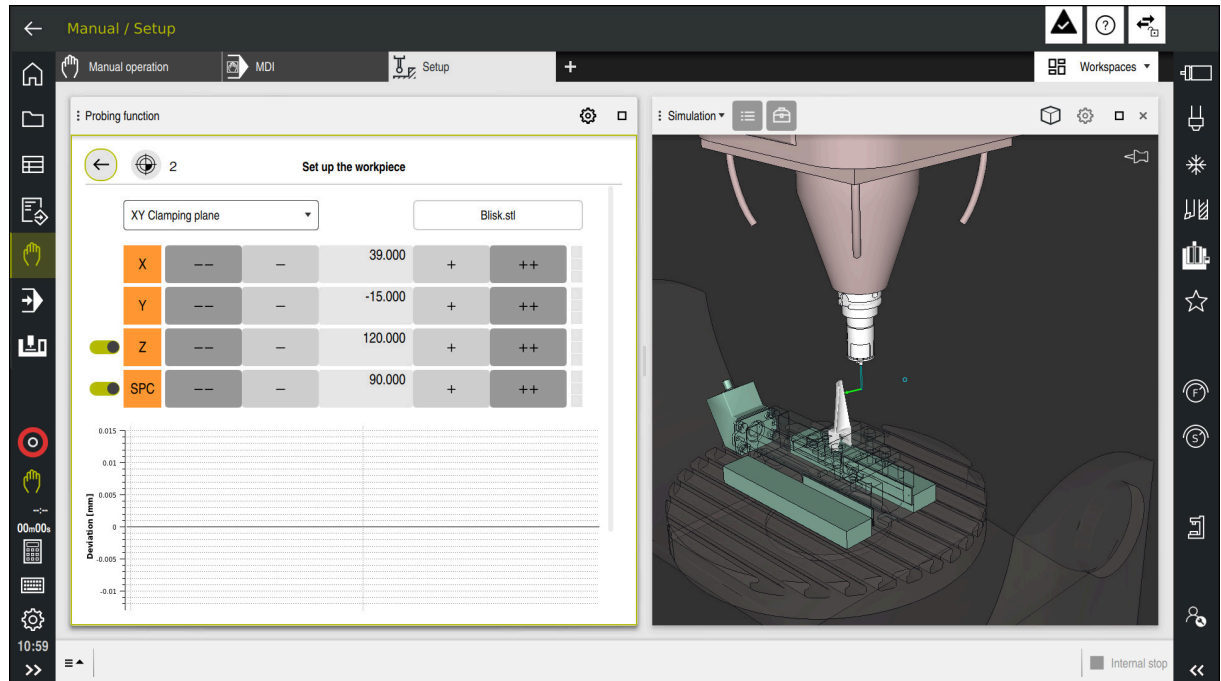
- None of the two software options is enabled:
You cannot tilt before setting up. If you move the rotary axes between the touch points, the control will display an error message.

Further information: "Tilting the working plane (#8 / #1-01-1)", Page 214

Further information: Programming and Testing User's Manual

Extension of the Simulation workspace

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



The **Set up the workpiece** function with the **Simulation** workspace open

When the **Set up the workpiece** function is active, the **Simulation** workspace displays the content below:

- Current position of workpiece as viewed by the control
- Probed points on the workpiece
- Possible direction of probing by means of an arrow:
 - No arrow
Probing is not possible. The workpiece touch probe is too distant from the workpiece or the workpiece touch probe is positioned within the workpiece, as seen by the control.

In this case you can correct the position of the 3D model in the simulation, if required.

- Red arrow
Probing in the direction of the arrow is not possible.

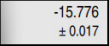


Probing on edges, corners or heavily curved workpiece areas fails to deliver precise measuring results. This is why the control blocks probing in these areas.

- Yellow arrow
Probing in the direction of the arrow is possible to a limited extent. The probing is performed in a deselected direction or could cause collisions.
- Green arrow
Probing in the direction of the arrow is possible.

Icons and buttons

The **Set up the workpiece** function contains the following icons and buttons:

Icon or button	Meaning
	Open the Change the preset window You can select the workpiece preset and the pallet preset and edit values if required. The control shows the number of the active preset to the right of the icon.  After the first point has been probed, the control dims the icon.
XY Clamping plane	Use this selection menu to define the probing mode. Depending on the probing mode, the control displays the respective axis directions and spatial angles. Further information: "Probing mode", Page 330
	File name of the NC program or 3D model
	Shifts the position of the virtual workpiece by 10 mm, 0.3937 inches, or 10° in the negative axis direction  Shifts the workpiece in mm or inches in a linear axis, and in degrees in a rotary axis.
	Shifts the position of the virtual workpiece by 1 mm, 0.0394 inches, or 1° in the negative axis direction
	■ Enter the position of the virtual workpiece directly ■ Value and estimated accuracy of the value after the probing
	Shifts the position of the virtual workpiece by 1 mm, 0.0394 inches, or 1° in the positive axis direction
	Shifts the position of the virtual workpiece by 10 mm, 0.3937 inches, or 10° in the positive axis direction
    	Status of the direction: ■ Dimmed The axis direction is deselected in this setup process and is not considered. ■ Empty No touch points have been determined yet. ■ Red The control cannot locate the workpiece position in this axis direction. ■ Yellow The position of the workpiece in this axis already contains information. The information is not meaningful yet. ■ Green The control can locate the workpiece position in this axis direction.
Compensate the active preset	The control saves the determined values in the active row of the preset table.

Probing mode

The following modes for probing the workpiece are available to you:

- **XY Clamping plane**
X, Y and Z axis directions as well as spatial angle **SPC**
- **XZ Clamping plane**
X, Y and Z axis directions as well as spatial angle **SPB**
- **YZ Clamping plane**
X, Y and Z axis directions as well as spatial angle **SPA**
- **6D**
X, Y and Z axis directions as well as spatial angles **SPA, SPB** and **SPC**

Depending on the probing mode, the control displays the respective axis directions and spatial angles. In the **XY**, **XZ** and **YZ** clamping planes a toggle switch allows you to deselect the respective tool axis and spatial angle, if required. The control will not take deselected axis directions into account in the setup process and positions the workpiece by considering the remaining axis directions only.

HEIDENHAIN recommends executing the setup process as follows:

- 1 Pre-position a 3D model in the machine's working space
By default, the control positions the 3D model to the active workpiece reference point. At this point in time, the control does not know the precise position of the workpiece, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe produces values close to the position of the real workpiece.
- 2 Set the first touch points in the **X**, **Y** and **Z** axis directions
If the control can determine the position in one axis direction, it will change the status of that axis to green.
- 3 Determine the spatial angle by setting further touch points
To achieve maximum accuracy when probing the spatial angles, the touch points should be as far apart from one another as possible.
- 4 Increase the accuracies by additional check points
Additional check points at the end of the measuring process improve the matching accuracy and minimize the misalignment between the 3D model and the real workpiece. Perform as many probing processes as necessary until the control displays the desired accuracy beneath the current value.

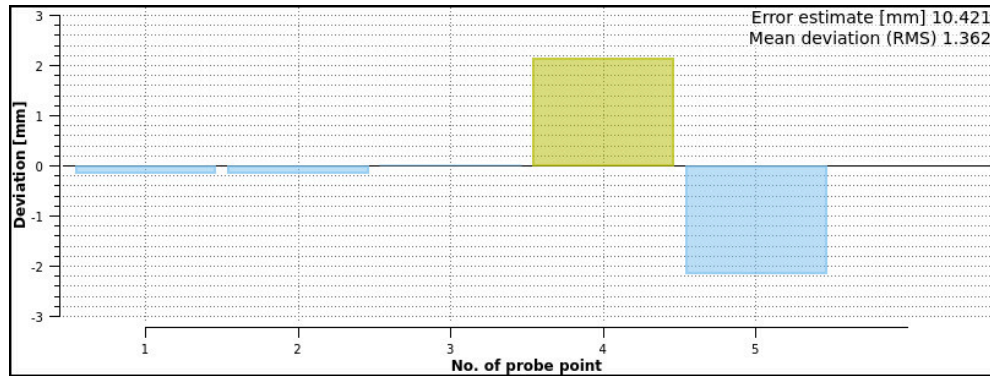
The error estimate diagram shows for each probe point the distance of the 3D model from the real workpiece.

Further information: "Error estimate diagram", Page 331

Error estimate diagram

Every additional touch point gradually restricts the possible positioning of the workpiece and puts the 3D model closer to the actual position in the machine.

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



Error estimate diagram in the **Set up the workpiece** function with transparent columns

The error estimate diagram of the **Set up the workpiece** function shows the following information:

- **Error estimate [mm]**
After each probe point, the value shows the greatest estimated distance between the 3D model and the workpiece.
- **Mean deviation (RMS)**
After each probe point, the value shows the average of all measured distances between the 3D model and the workpiece.
- **Deviation [mm]**
Using this axis you can determine how large the estimated distance between the 3D model and the probe point on the workpiece is.
- **No. of probe point**
This axis shows the numbers of the probe points so far.
- **Columns**
As long as the status of all axes is not green, the control shows transparent columns.
The control recalculates the 3D model after each probe point. This also changes the previous values.
When the columns of the error estimate diagram are no longer transparent, and **Error estimate [mm]** shows the desired accuracy, the setup process is complete.

The following factors influence the accuracy that can be achieved when measuring workpieces:

- Accuracy of workpiece touch probe
- Accuracy of the machine kinematic configuration
- Deviations of the 3D model from the real workpiece
- Condition of the actual workpiece (e.g., unmachined areas)

16.3.1 Setting up a workpiece

Use the **Set up the workpiece** function to set the preset:

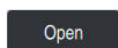
- ▶ Affix a real workpiece in the machine's working space



- ▶ Select the **Manual** operating mode
- ▶ Insert the workpiece touch probe
- ▶ Manually position the workpiece touch probe above the workpiece at a notable point (e.g., a corner)



This step makes the subsequent steps easier.



- ▶ Select the **Setup** application
- ▶ Select **Set up the workpiece**
 - The control opens the **Set up the workpiece** menu.
- ▶ Select NC program or 3D model
- ▶ Select **Open**
 - The control shows the 3D model in the simulation.
- ▶ If necessary, check the values of the active preset or choose a different preset
- ▶ Pre-position the 3D model by using the buttons for the individual axis directions within the virtual working space of the machine



For pre-positioning the workpiece, use the workpiece touch probe as a point of reference.
Even during the setup process, the shift functions are available for correcting the fixture position manually. Then, probe a new point.

- ▶ Specify the probing mode (e.g., **XY Clamping plane**)
- ▶ Position the workpiece touch probe until the control shows a green arrow pointing downward



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



- ▶ Press the **NC Start** key
 - The control probes in the direction of the arrow.
 - The control displays the status of the **Z** axis in green and shifts the workpiece to the probed position. The control marks the probed position with a point in the simulation.

- ▶ Repeat this process in axis directions **X+** and **Y+**
- ▶ The control changes the status of the axes to green.
- ▶ Probe another point in axis direction **Y+** for the basic rotation
- ▶ The control changes the status of the **SPC** spatial angle to green.
- ▶ Probe the check point in axis direction **X-**
- ▶ Select **Compensate the active preset**
- ▶ The control saves the determined values in the active row of the preset table.
- ▶ Exit the **Set up the workpiece** function

Compensate the active preset



Notes

NOTICE

Danger of collision!

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

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

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

16.4 Measuring the tool by scratching

Application

Not all machines are equipped with a tool touch probe for measuring a tool. The **Tool measured** touch probe function enables determining the tool dimensions by scratching a workpiece.

Related topics

- Touch probe functions in the **Setup** application
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
- Measuring the tool automatically with cycles
Further information: Measuring Cycles for Workpieces and Tools User's Manual

Requirement

- Touch Probe Functions software option (#17 / #1-05-1)



If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.

Description of function

You do not use a 3D touch probe for scratching, but the tool to be measured. In the scratching process, approach the tool carefully to a workpiece surface until you can see a thin chip being removed. The handwheel allows obtaining a higher accuracy. In the **X** or **Y** probing directions, the tool radius can be determined. When selecting probing direction **Z**, the tool length is measured.

Buttons in the Measure the tool function

The control offers the following options for writing the measured radius or length values into the tool table:

Button	Meaning
Write basic values	The control transfers the values into columns R or L . The control resets existing delta values in columns DR or DL .
Write delta values	The control enters the delta values in columns DR or DL .

Further information: "Tool tables", Page 392

16.4.1 Tool measurement by scratching

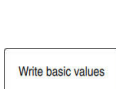
The dimensions of an end mill can be determined by using the **Tool measured** function as follows:



- ▶ Select the **Manual** operating mode
- ▶ Set the workpiece preset if required



Position the workpiece preset on the surfaces to be scratched in order to obtain a clear reference.



- ▶ Insert the tool to be measured
- ▶ Define the speed if required
- ▶ Start the tool spindle
- ▶ Select the **Setup** application
- ▶ Select the **Measure tool** touch probe function
- ▶ If necessary, check the values of the active preset or choose a different preset
- ▶ Scratch the workpiece in the desired axis direction (e.g., **X+**)
- ▶ Select the associated probing direction **X+**
- ▶ Select **actual position capture**
 - The control transfers the actual X axis position into the **Actual value** column.
 - The control shows the measurement results.
- ▶ Enter a **Nominal value** (e.g., **0**)
- ▶ Select **Write basic values**
 - The control transfers the value into column **R** of the tool table.
 - The control resets the existing delta value in the **DR** column.



When selecting **Write delta values**, the control will enter only one delta value in column **DR**.



- ▶ Scratch another axis direction if required (e.g., **Z-**)
- ▶ Select **Exit probing**
 - The control closes the **Measure tool** touch probe function.

16.5 Suppressing touch probe monitoring

Application

If you move a workpiece touch probe too close to the workpiece, you can accidentally deflect the workpiece touch probe. You cannot retract a deflected workpiece touch probe in the monitored state. You can retract a deflected workpiece touch probe by suppressing touch probe monitoring.

Description of function

If the control does not receive a stable signal from the probe, the button displays **Suppress touch probe monitoring**.

As long as touch-probe monitoring is switched off, the control displays the error message **The touch probe monitor is deactivated for 30 seconds**. This error message remains active only for 30 seconds.

16.5.1 Deactivating touch probe monitoring

To deactivate touch probe monitoring:



- ▶ Select the **Manual** operating mode
- ▶ Select **Suppress touch probe monitoring**
- ▶ The control disables touch-probe monitoring for 30 seconds.
- ▶ If required, move the touch probe so that the control receives a stable signal from it.

Notes

NOTICE

Danger of collision!

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

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

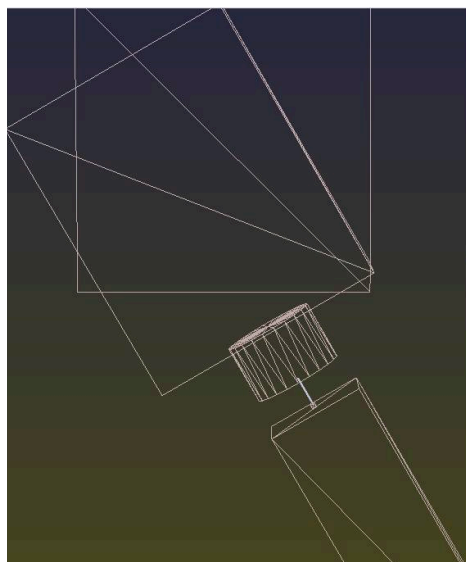
If the touch probe sends a stable signal within these 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

16.6 Comparison of offset and 3D basic rotation

The following example shows how the two functions differ.

Offset

Initial state



Position display:

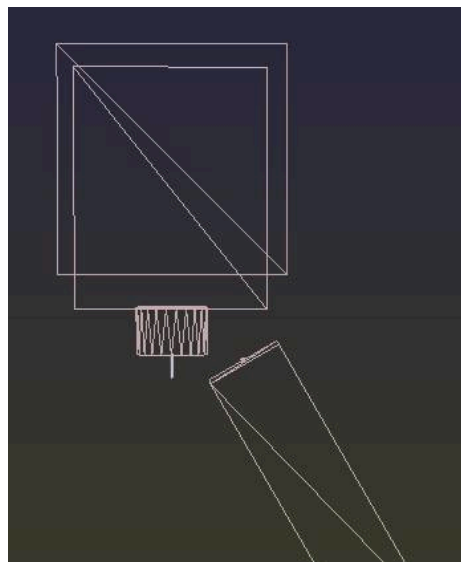
- Actual position
- **B** = 0
- **C** = 0

Preset table:

- **SPB** = 0
- **B_OFFS** = -30
- **C_OFFS** = +0

3D basic rotation

Initial state



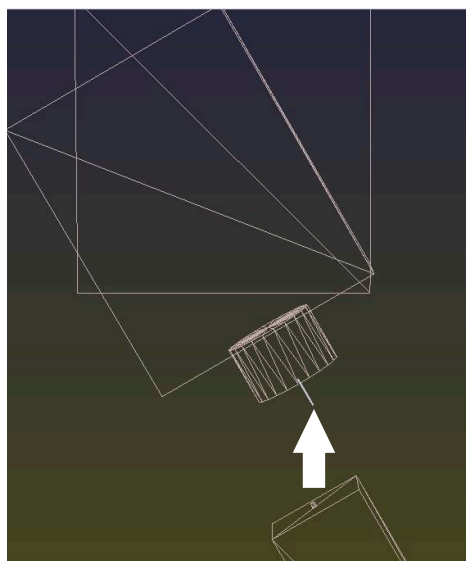
Position display:

- Actual position
- **B** = 0
- **C** = 0

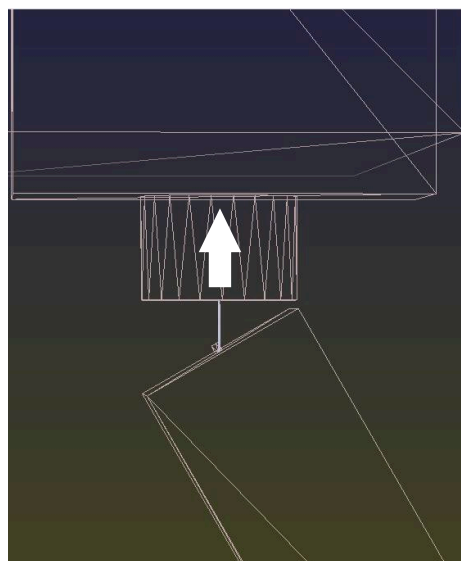
Preset table:

- **SPB** = -30
- **B_OFFS** = +0
- **C_OFFS** = +0

Movement in +Z without tilting

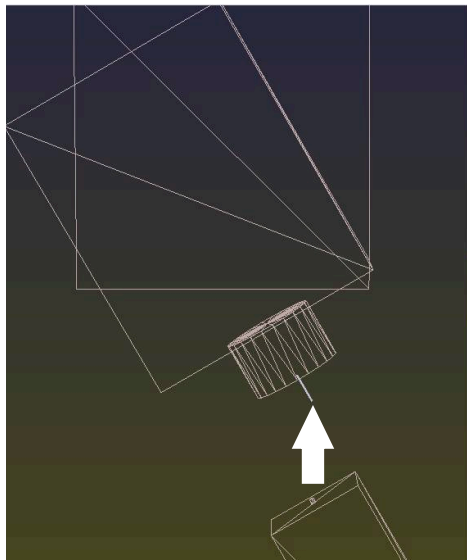


Movement in +Z without tilting



Offset

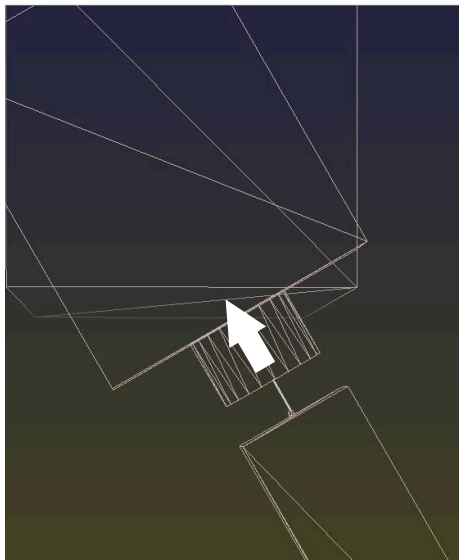
Movement in +Z with tilting

PLANE SPATIAL with **SPA+0 SPB+0 SPC+0**

> The orientation **is not correct!**

3D basic rotation

Movement in +Z with tilting

PLANE SPATIAL with **SPA+0 SPB+0 SPC+0**

> The orientation is correct!
> The next machining step will be **correct.**



HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

17

Program run

17.1 The Program Run operating mode

17.1.1 Fundamentals

Application

In the **Program Run** operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence.

You also execute pallet tables in this operating mode.

Related topics

- Executing individual NC blocks in the **MDI** application
Further information: "The MDI application ", Page 301
- Creating NC programs
Further information: Programming and Testing User's Manual
- Pallet tables
Further information: Programming and Testing User's Manual

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- Copy the NC program and all called files to the **TNC:** drive

NOTICE

Danger of collision!

When you edit NC programs outside the **Program** workspace, you have no control over whether the control will identify the changes. Undesirable machine movements or collisions may result.

- Edit NC programs in the **Program** workspace only

Description of function



The following information also applies to pallet tables and job lists.

When you select a new NC program or when an NC program has been completely executed, the cursor is at the beginning of the program.

If you want to start machining at a different NC block, you first need to select the desired NC block by using the **Block scan** function.

Further information: "Block scan for mid-program startup", Page 352

By default, the control runs NC programs in Full Sequence mode after the **NC Start** key has been pressed. In this mode, the control runs an NC program continuously up to its end, or up to a manual or programmed interruption.

In **Single Block** mode you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview of the TNC bar", Page 127

The **Program Run** operating mode provides the following workspaces:

- **Document**

Further information: Programming and Testing User's Manual

- **Positions**

Further information: "The Positions workspace", Page 121

- **Program**

Further information: Programming and Testing User's Manual

- **Simulation**

Further information: Programming and Testing User's Manual

- **Status**

Further information: "The Status workspace", Page 129

When opening a pallet table, the control displays the **Job list** workspace. You cannot modify this workspace.

Further information: Programming and Testing User's Manual

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
	Open File With Open File you can open a file (for example, an NC program). When you open a file, the control closes the file that was already open.
	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution.
Single Block	If this toggle switch is active, then you run each NC block separately with the NC Start key. If Single Block mode is selected, then the operating mode's icon in the control bar changes.
Q info	The control opens the Q parameter list window, where you can see and edit the current values and descriptions of the variables. Further information: Programming and Testing User's Manual
Compensation tables	The control opens a selection menu with the following tables: ■ D ■ T-CS ■ WPL-CS Further information: "Compensation during program run", Page 364
F LIMIT	Use this function to activate a feed-rate limit and define its value. Further information: "Feed rate limit F LIMIT", Page 345

Icon or button	Meaning
Program run options	When you select this button, the control opens the Program run options window with the following selection possibilities: ■ Settings for the override controller Further information: "The Program run options window", Page 450 ■ Perform conditional stop The control offers the following breakpoints: ■ Before switch to rapid traverse ■ Before switch to feed rate ■ Between two rapid traverses ■ Before tool call ■ Before tilting the working plane ■ Before cycle call ■ In cycle call Further information: "The Program run options window", Page 450 ■ Feed F LIMIT Use this function to activate a feed-rate limit and define its value. Further information: "Feed rate limit F LIMIT", Page 345 ■ Skip block If the toggle switch is active, the control does not execute NC blocks dimmed with a / character. If the toggle switch is active, then the control dims the NC blocks to be skipped. Further information: Programming and Testing User's Manual ■ Pause at M1 If the toggle switch is active, the control stops the execution at every NC block with M1. If the toggle switch is inactive, then the control dims the M1 syntax element. Further information: Programming and Testing User's Manual
GOTO Cursor	The control marks the table row currently selected for execution. This button is available when a pallet table is open. Further information: Programming and Testing User's Manual
AFC	Use this option to activate or deactivate Adaptive Feed Control (AFC (#45 / #2-31-1)). Further information: "The AFC toggle switch in the Program Run operating mode", Page 260
AFC settings	The control opens a selection menu with the following selection possibilities for AFC (#45 / #2-31-1): ■ AFC.TAB for AFC basic settings ■ AFC.DEP settings file for teach-in cuts of the active NC program ■ AFC2.DEP log file of the active NC program ■ Stop Teach Further information: "The AFC settings button", Page 263

Icon or button	Meaning
Skip block	If the toggle switch is active, the control does not execute NC blocks dimmed with a / character. If the toggle switch is active, then the control dims the NC blocks to be skipped. Further information: Programming and Testing User's Manual
Pause at M1	If the toggle switch is active, the control stops the execution at every NC block with M1. If the toggle switch is inactive, then the control dims the M1 syntax element. Further information: Programming and Testing User's Manual
Automatic program start	Starts machining at a defined time automatically Further information: "Automatic program start", Page 361
ACC	If this toggle switch is active, the control activates Active Chatter Control (ACC (#145 / #2-30-1)). Further information: "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 266
Edit	If this toggle switch is active, then you can edit the pallet table. This button is available if a pallet table is open. Further information: Programming and Testing User's Manual
GOTO block number	Mark an NC block to be run without considering any previous NC blocks Further information: Programming and Testing User's Manual
Manual traverse	While a program run is interrupted, you can move the axes manually. If Manual traverse is active, the operating mode's icon in the control bar changes. Further information: "Manual traverse during an interruption", Page 350
Handwheel	If Manual traverse is active, you can activate and deactivate the handwheel with this switch. The keys of the machine operating panel cannot be used for traversing while the handwheel is active. Further information: "Manual traverse during an interruption", Page 350
3D ROT	While a program run is interrupted, you can move the axes manually in the tilted working plane (#8 / #1-01-1). Further information: "Manual traverse during an interruption", Page 350
Approach position	Return to contour after manual traverse of the machine axes during an interruption Further information: "Returning to the contour", Page 360
Block scan	Use the Block scan function to start program run at any desired NC block. The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with M3. Further information: "Block scan for mid-program startup", Page 352
Tool Retract	If the NC program is stopped during a thread cycle, you can retract the tool. Further information: "The Retract application", Page 366
Open in the editor	The control opens the active NC program in the Editor operating mode and selects the currently selected NC block, even for called NC programs. This button is available when an NC program is open. Further information: Programming and Testing User's Manual

Icon or button	Meaning
Tools	The control opens the Tool management application in the Tables operating mode. Further information: "Tool management ", Page 184
Internal stop	For example, if an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run.
Reset program	If you select Internal stop , the control activates this button. The control places the cursor back to the beginning of the program and resets any modally active program information as well as the program run-time.

Feed rate limit F LIMIT

The **F LIMIT** button allows you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you have entered remains active across power cycles.

The **F LIMIT** button is available in the **MDI** application and in the **Editor** operating mode.

When you select the **F LIMIT** button in the function bar, the control will open the **Feed rate F LIMIT** window.

Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

If a feed rate limit is active, the control highlights the **F LIMIT** button in color and displays the defined value. In the **Positions** and **Status** workspaces, the feed rate is displayed in orange.

Further information: "Status displays", Page 119

You deactivate the feed rate limit by entering a value of 0 in the **Feed rate F LIMIT** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt program run (e.g., with the miscellaneous function **M1**)
- Stop program run (e.g., with the **NC Stop** key)
- Cancel program run (e.g., with the **NC stop** key and the **Internal stop** button)
- Terminate program run (e.g., with the miscellaneous functions **M2** or **M30**)

Upon major errors, the control automatically aborts program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 297

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview of the TNC bar", Page 127

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

- Select the **Manual** operating mode
- The **Manual traverse** function
- Checking variables and changing these if necessary using the **Q INFO** function
- Changing the setting for the optional programmed interruption with **M1**
- Changing the setting for the programmed skipping of NC blocks with **/**



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

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
 - Cursor movement to another NC block
 - The jump command **GOTO** to another NC block
 - Editing an NC block
 - Modifying the values of variables by using the **Q parameter list** window
 - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts program run in the NC block containing one of the following inputs:

- Programmed stop **STOP**
- Conditional stop **M1**

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

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

After an interruption of program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 352

Modally effective program information

The control saves the following data during a program interruption:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

The control uses the stored data for returning the tool to the contour (**Approach position** button).

Further information: "Returning to the contour", Page 360



The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE

Danger of collision!

Program cancellation, manual intervention, forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision.

- ▶ Rescind all programmed NC functions and transformations within the NC program
 - ▶ Run a simulation before executing an NC program
 - ▶ Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program
 - ▶ Carefully prove-out the NC programs in **Single Block** mode
- In the **Program Run** operating mode, the control marks active files with the status **M**, such as a selected NC program or tables. If you open such a file in another operating mode, the controls shows the status on the tab of the application bar.
 - When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.
 - You can adjust the feed rate and the spindle speed during program run with the potentiometers.
 - If you modify the workpiece preset during a program run interruption, you must re-select the NC block to resume.
Further information: "Block scan for mid-program startup", Page 352
 - HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
 - The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.

17.1.2 Navigation path in the Program workspace

Application

If you execute an NC program or a pallet table, or if you test it in the opened **Simulation** workspace, the control will display a navigation path in the file information bar of the **Program** workspace.

The control displays the names of all the NC programs used in the navigation path and opens the contents of all NC programs in the workspace. This makes it easier to keep an overview of the execution when calling programs and allows navigating between the NC programs when program run is interrupted.

Related topics

- Program call
Further information: Programming and Testing User's Manual
- The **Program** workspace
Further information: Programming and Testing User's Manual
- The **Simulation** workspace
Further information: Programming and Testing User's Manual
- Interrupted program run
Further information: "Interrupting, stopping or canceling program run",
 Page 346

Requirement

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

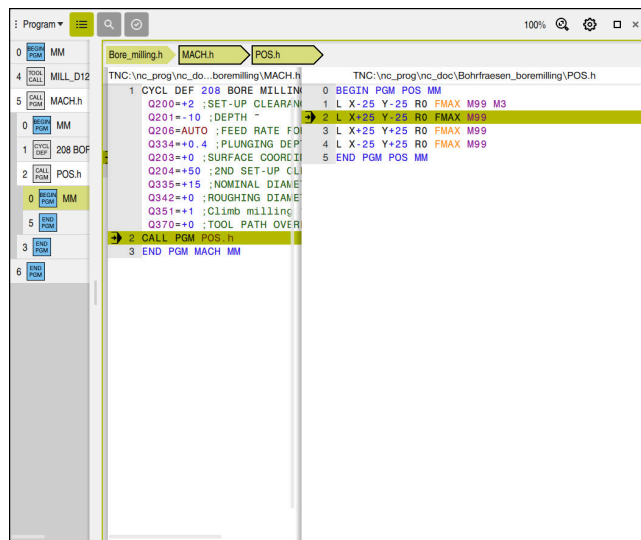
Description of function

The control shows the name of the NC program as a path element in the file information bar. As soon as the control calls a different NC program, the control adds a new path element with the name of the called NC program to the bar.

Additionally, the control displays the contents of the called NC program in a new pane in the **Program** workspace. The control displays as many NC programs side by side as the size of the workspace permits. If necessary, newly opened NC programs will cover previously opened NC programs. The control displays the covered NC programs in a narrow band at the left edge of the workspace.

When execution is interrupted, you can navigate between the NC programs. When you select the path element of an NC program, the control opens the content.

When you select the last path element, the control automatically marks the active NC block with the execution cursor. When you press the **NC Start** key, the control resumes execution of the NC program from this position.



Called NC programs in the **Program** workspace in the **Program Run** operating mode

Depiction of path elements

The control displays the path elements of the navigation path as follows:

Format	Meaning
Black frame	The NC program is visible in the Program workspace and is not covered by other NC programs.
Highlighted in green	The NC program at the current cursor position is active or is considered for program run. If, for example, the cursor is positioned in the called NC program, the calling NC program will be considered for program run.
Highlighted in gray	The NC program is active for execution but will not be considered for program run at the current cursor position. If, for example, you stop the execution and navigate into the calling NC program, the control displays the path element of the called NC program in gray.

Note

In the **Program Run** operating mode, the **Structure** column contains all structuring items, even those of the called NC programs. The control indents the structure of the called NC programs.

The structure items allow you to navigate into every NC program. The control displays the associated NC programs in the **Program** workspace. The navigation path always remains at the current point of execution.

Further information: Programming and Testing User's Manual

17.1.3 Manual traverse during an interruption

Application

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

Related topics

- Manual traverse of machine axes
Further information: "Moving the machine axes", Page 162
- Tilting the working plane manually (#8 / #1-01-1)
Further information: Programming and Testing User's Manual

Description of function

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

Further information: "Using axis keys to move the axes ", Page 162

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

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

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

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

Symbol	Function	Meaning
	M-CS machine	Traversing in the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
	W-CS workpiece	Traversing in the workpiece coordinate system W-CS Further information: "Workpiece coordinate system W-CS", Page 202
	WPL-CS working plane	Traversing in the working plane coordinate system WPL-CS Further information: "Working plane coordinate system WPL-CS", Page 204
	T-CS tool	Traversing in the tool coordinate system T-CS Further information: "Working plane coordinate system WPL-CS", Page 204

When you select one of the functions, the control will display the associated icon in the **Positions** workspace. The control additionally shows the active coordinate system on the **3D ROT** button.

Notes

NOTICE

Danger of collision!

During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). Selecting an incorrect **3D ROT** setting or moving the tool in the wrong direction involves risk of collision!

- ▶ It is better to use the **T-CS** function
- ▶ Check the direction of movement
- ▶ Move at slow feed rate

- Refer to your machine manual.
On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.
- Using the optional machine parameter **userControl** (no. 144101), the machine manufacturer defines if the control will display the **Handwheel** toggle switch or not.
If the **Handwheel** toggle switch is available, the control displays an additional area in its machine settings. You can define whether the machine operating panel or the handwheel is active by default.
Further information: "The Machine Settings menu item", Page 475
If this machine parameter has been defined with **FALSE** or not at all, the machine manufacturer determines the user control that is active by default.
- While the **Handwheel** toggle switch is active, you cannot move the axes with the machine operating panel.
- Refer to your machine manual.
Depending on the machine, the control indicates that a handwheel is active (for example, with a handwheel symbol next to the selected axis in the **Positions** workspace or via a message in the information bar).

17.1.4 Block scan for mid-program startup

Application

The **Block scan** function allows you to start an NC program at any desired NC block. The control factors workpiece machining up to this NC block into the calculations. For example, the control will switch on the spindle before the start.

Related topics

- Creating NC programs
Further information: Programming and Testing User's Manual
- Pallet tables and job lists
Further information: Programming and Testing User's Manual

Requirement

- The function must be enabled by your machine manufacturer.
The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function



The following information also applies to pallet tables and job lists.

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

- The **Internal stop** button
- Emergency stop
- Power failure

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

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary
Further information: "Performing a single-level block scan", Page 355
- Multi-level block scan in subprograms and touch probe cycles
Further information: "Performing a multi-level block scan", Page 356
- Block scan in a point table
Further information: "Block scan in point tables", Page 357
- Block scan in pallet programs
Further information: "Block scan in pallet tables", Page 358

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During the block scan you can activate or deactivate **Single Block** mode.

The Block scan window

The **Block scan** window with saved interruption point and open **Advanced** area

The **Block scan** window provides the following data:

Row	Meaning
Row number	Row number in the pallet table
	Row number in the pallet table at the time of interruption
Program	Path of the active NC program
	Path of the NC program that is active at the time of interruption
Block number	Number of the NC block at which program run should start The control proposes the currently selected NC block. You can open the Search column from the Block scan window.
	Number of the NC block that was active at the time of interruption
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat or a program loop.
	Number of the repetition at the time of interruption if the interruption was located within a program-section repeat or program loop.
Point table	Path of the point table Selection by means of a selection window
	Path of the active point table at the time of interruption
Point number	Row in the point table
	Active row in the point table at the time of interruption

Select the interruption point by using the **Select last** button in the **Advanced** area.

Performing a single-level block scan

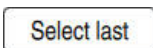
To start in an NC program by using a single-level block scan:



- ▶ Select the **Program Run** operating mode



- ▶ Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- ▶ Enter the **Program** if required
- ▶ Enter **Block number**
- ▶ Enter the **Repetitions** if required
- ▶ If applicable, open the **Extended** area



- ▶ If required, use **Select last** to start at a saved interruption point



- ▶ Press the **NC Start** key
- The control starts the block scan and calculates until the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.



- ▶ Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.



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



You can also position the axes individually in your own selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 361



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

Performing a multi-level block scan

If you, for example, start in a subprogram that is called several times by the main program, then use the multi-level block scan. For this, you first go to the desired subprogram call and then continue the block scan. The same procedure is used for called NC programs.

To start in an NC program by using a multi-level block scan:



- ▶ Select the **Program Run** operating mode



- ▶ Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.

- ▶ Perform a block scan to the first start-up point

Further information: "Performing a single-level block scan", Page 355



- ▶ Activate the **Single Block** toggle switch as needed



- ▶ Press the **NC Start** key to execute individual NC blocks as needed



- ▶ Select **Continue block scan**



- ▶ Define the NC block where you wish to start

- ▶ Press the **NC Start** key

- The control starts the block scan and calculates until the entered NC block.

- If you have changed the machine status, the control displays the **Restore machine status** window.



- ▶ Press the **NC Start** key

- The control restores the machine status (e.g., **TOOL CALL** or M functions).

- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.



- ▶ Press the **NC Start** key

- Using the displayed positioning logic, the control moves to the required positions.



You can also position the axes individually in your own selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 361



- ▶ Select **Continue block scan** again as needed

- ▶ Repeat the steps



- ▶ Press the **NC Start** key

- The control resumes execution of the NC program.

Block scan in point tables

To start in a point table:



- ▶ Select the **Program Run** operating mode



- ▶ Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.



- ▶ Open the **Extended** area



- ▶ **Point table:** Select the point table

- ▶ **Point number:** Select the row number of the point table for mid-program startup



- ▶ Press the **NC Start** key
- The control starts the block scan and calculates until the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.



- ▶ Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.



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



You can also position the axes individually in your own selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 361



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

Further information: User's Manual for Machining Cycles

Block scan in pallet tables

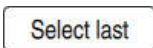
To start in a pallet table:



- ▶ Select the **Program Run** operating mode



- ▶ Select **Block scan**
- The control opens the **Block scan** window.
- ▶ **Row number.** Enter the row number of the pallet table
- ▶ Enter the **Program** if required
- ▶ Enter **Block number**
- ▶ Enter the **Repetitions** if required
- ▶ If applicable, open the **Extended** area



- ▶ If required, use **Select last** to start at a saved interruption point



- ▶ Press the **NC Start** key
- The control starts the block scan and calculates until the entered NC block.
- If you have changed the machine status, the control displays the **Restore machine status** window.



- ▶ Press the **NC Start** key
- The control restores the machine status (e.g., **TOOL CALL** or M functions).
- If you have changed the axis positions, the control displays the **Axis sequence for return to contour:** window.



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



You can also position the axes individually in your own selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 361



If program run of a pallet table has been canceled, the control will suggest the most recently selected NC block of the most recently executed NC program as a point of interruption.

Notes

NOTICE**Danger of collision!**

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

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

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

If program run is interrupted within a called NC program, the control always offers the first call of this NC program as a point of interruption for mid-program startup. If program run was interrupted in a later call, the control might then execute parts of the program that have already run.

- ▶ Use multi-level block scan to manually navigate to the point of interruption

NOTICE**Danger of collision!**

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

- ▶ Use the **Block scan** function at multiple levels

- The TNC7 basic can move up to four axes simultaneously. If an NC block commands movement of more than four axes, the control displays an error message. When the control reads such an NC block during the **Block scan**, it will also display an error message.
 - The control only displays the dialogs required by the process in the pop-up window.
 - Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
 - If you use the block scan to start in a pallet table, the control will always execute the chosen row in the pallet table as a workpiece-oriented process. After the pallet table line selected in the **Block scan**, the control resumes machining according to the defined machining method.
- Further information:** Programming and Testing User's Manual
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.
- Further information:** "The LBL tab", Page 136
- The **Block scan** function must not be used in conjunction with the following functions:
 - Touch probe cycles **0**, **1**, **3**, and **4** during the block scan search phase
 - HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.

17.1.5 Returning to the contour

Application

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

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the **INTERNAL STOP** function.
- Return to the contour after a block scan (e.g., after an interruption with **INTERNAL STOP**)
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption

Related topics

- Manual traverse during program run interruptions
Further information: "Manual traverse during an interruption", Page 350
- The **Block scan** function
Further information: "Block scan for mid-program startup", Page 352

Description of function

If you have selected the **Manual traverse** button, this button will change to **Approach position**.

When you select **Approach position**, the control will open the **Axis sequence for return to contour:** window.

The Axis sequence for return to contour: window

Actual pos. (ACT)	Target	Current	Δ Remaining
Approach logic	X	✓	366.820
	Y	-300.000	

Execute with NC Start key

The **Axis sequence for return to contour:** window

In the **Axis sequence for return to contour:** window, the control displays all of the axes that are not yet located at the correct position for program execution.

The control suggests a positioning logic for the sequence of the traversing movements. If the tool is located in the tool axis below the position to be approached, then the control offers the tool axis as the first traverse direction. You can also traverse the axes in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 361

If manual axes are included in the axes to be returned to the contour, then the control will not suggest a positioning logic. As soon as you have correctly positioned the manual axis, the control will suggest a positioning logic for the remaining axes.

Further information: "Positioning manual axes", Page 361

Positioning the axes in a self-selected sequence

To position the axes in a self-selected sequence:



- ▶ Select **Approach position**
- The control displays the **Axis sequence for return to contour:** window and the axes to be positioned.
- ▶ Select the desired axis (e.g., **X**)
- ▶ Press the **NC Start** key
- The control moves the axis to the required position.
- When the axis has reached the correct position, the control shows a check mark for **Target**.
- ▶ Position the remaining axes
- When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:



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

Note

In the machine parameter **restoreAxis** (no. 200305), the machine manufacturer defines in which sequence of axes the control approaches the contour again.

Definition

Manual axis

Manual axes are non-driven axes that need to be positioned by the machine operator.

17.1.6 Automatic program start

Application

With the **Automatic program start** function you define a specific time at which the control will start to execute an NC program, such as a warm-up program for the machine. No machine operator needs to be present.

Requirements

- The function must be enabled by your machine manufacturer.
In the machine parameter **autoStartEnabled** (no. 100701), the machine manufacturer defines whether an automatic program start is possible.
- Machine is switched on
- No NC program is currently running on the machine
If a program is already running at the specified time, the control will not execute an automatic program start.
- Machine is prepared for the NC program (e.g., correct workpiece preset is active)



HEIDENHAIN recommends activating the workpiece preset in the NC program.

Further information: Programming and Testing User's Manual

Description of function

Use this function to execute NC programs and pallet tables.

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

The **Automatic program start** window

The control offers the following possibilities in the **Automatic program start** window:

- 1 Defining the date and time
- 2 Selecting an NC program via a selection window
- 3 Activating automatic program start
- 4 Deactivating automatic program start

If an automatic program start is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 99

Notes

WARNING

Caution: hazard to the user!

If automatic program start is active, the machine starts an NC program on its own and moves the axes. There is an increased risk of injury on machines without housing.

- ▶ Clearly indicate on the machine whether **Automatic program start** is active
- ▶ If necessary, restrict access to the machine

NOTICE

Caution: Significant property damage!

If automatic program start is active, the machine starts an NC program without a machinist and moves the axes. Collisions can occur if the machine was not set up correctly, or if there have been changes in the meantime. There might be no machinist present in order to abort program run.

- ▶ Set up the machine appropriately for such machining, e.g.:
 - Install the correct workholding equipment
 - Measure the tools used in the NC program and stock them in the magazine
 - ▶ Optimize the NC program for automation, e.g.:
 - Automatic presetting
 - Activating the most recently set preset
 - Calling the correct workholding equipment
- In the machine parameter **closeDialogOnOK** (no. 100702), the machine manufacturer defines whether the control closes the **Automatic program start** window after activation.
 - In the machine parameter **useLastStartData** (no. 100703), the machine manufacturer defines whether the control offers the last defined settings when opening the **Automatic program start** window.
 - Ensure that the tools being used have sufficient service life remaining. If necessary, define a replacement tool.

17.2 Compensation during program run

Application

During program run, you can open the selected compensation tables and the active datum table, and edit the values.

Related topics

- Using compensation tables
Further information: Programming and Testing User's Manual
- Editing compensation tables in the NC program
Further information: Programming and Testing User's Manual
- Contents and creation of compensation tables
Further information: Programming and Testing User's Manual
- Contents and creation of a datum table
Further information: Programming and Testing User's Manual
- Activating a datum table in the NC program
Further information: Programming and Testing User's Manual

Description of function

The control opens the selected tables in the **Tables** operating mode.

The changed values do not take effect until the compensation or the datum has been activated again.

17.2.1 Opening tables from within the Program Run operating mode

To open the compensation tables from within the **Program Run** operating mode:

Compensation
tables

- ▶ Select **Compensation tables**
- > The control displays a selection menu.
- ▶ Select the desired table
 - **D:** Datum table
 - **T-CS:** Compensation table ***.tco**
 - **WPL-CS:** Compensation table ***.wco**
- > The control opens the selected table in the **Tables** operating mode.

Notes

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- ▶ Make sure to confirm any changes made to the table immediately (e.g., by pressing the **ENT** key)
- ▶ Activate the datum or compensation value in the NC program again
- ▶ Carefully prove-out the NC program after changing the table values

- When opening a table in the **Program Run** operating mode, the control will display the **M** status in the table tab. This status means that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.

Further information: "Status overview of the TNC bar", Page 127

17.3 The Retract application

Application

The **Retract** application allows you to disengage the tool from the workpiece after an interruption in power (e.g., retraction of a tap engaged in the workpiece).

You can also retract a tool when the working plane is tilted or retract an inclined tool.

Requirement

- This application must be enabled by your machine manufacturer.
The machine parameter **retractionMode** (no. 124101) allows the machine manufacturer to define whether the control will display the **Retract** toggle switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**
Further information: "The Retract workspace", Page 367
- **Positions**
Further information: "The Positions workspace", Page 121
- **Status**
Further information: "The Status workspace", Page 129

The **Retract** application provides the following buttons in the function bar:

Button	Meaning
Retract	Retract the tool with the axis keys or the electronic handwheel
End retraction	Close the Retract application The control opens the End retraction? window and prompts you to answer a confirmation request.
Start values	Reset the entries in the A, B, C, and Thread pitch fields to their original values

You select the **Retract** application by using the **Retract** toggle switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- The **Move to ref. point** application

If you have activated a feed rate limit before the power failure occurred, this feed rate limit will still be active. When you select the **Retract** button, the control will display a pop-up window: This window allows you to deactivate the feed rate limit.

Further information: "Feed rate limit F LIMIT", Page 345

The Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: ■ Machine axes: Move in the machine coordinate system M-CS ■ Tilted system: Move in the working plane coordinate system WPL-CS (#8 / #1-01-1) ■ Thread: Move in the tool coordinate system T-CS with compensating movements of the spindle Further information : "Reference systems", Page 196
Kinematics	Name of the active machine kinematics
A, B, C	Current position of the rotary axes Effective in the Tilted system traverse mode
Thread pitch	Thread pitch from the PITCH column of tool management Effective in the Thread traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: ■ Right-hand thread ■ Left-hand thread Effective in the Thread traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective in the Tool axis traverse mode

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

A power failure during the machining operation can cause uncontrolled "coasting" or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last saved axis values as the current position, which can deviate from the actual position. Thus, subsequent traverse movements do not correspond to the movements prior to the power failure. If the tool is still in effect during the traverse movements, then the tool and the workpiece can sustain damage through tension!

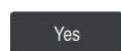
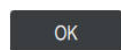
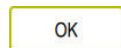
- ▶ Use a low feed rate
- ▶ Please keep in mind that the traverse range monitoring is not available for non-referenced axes

- Use the +, -, *, /, (, and) keys for calculations in the numerical input fields.

Example

The power failed while a thread cutting cycle in the tilted working plane was being performed. You have to retract the tap:

- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > In the **Start** workspace, the control provides the **Power interrupted** column.



- ▶ Activate the **Retract** toggle switch
- ▶ Select **OK**
- > The control compiles the PLC program.
- ▶ Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit
- > The control opens the **Retract** application and displays the **Assume position values?** window.
- ▶ Compare the displayed position values with the actual position values
- ▶ Select **OK**
- > The control closes the **Assume position values?** window
- ▶ Select the **Thread** traverse mode as needed
- ▶ Enter the thread pitch as needed
- ▶ Enter the direction of rotation as needed
- ▶ Select **Retract**
- ▶ Retract the tool with the axis keys or the handwheel
- ▶ Select **End retraction**
- > The control opens the **End retraction?** window and prompts you to answer a confirmation request.
- ▶ If the tool was correctly retracted, select **Yes**
- > The control closes the **End retraction?** window and the **Retract** application.

18

Tables

18.1 The Tables operating mode

Application

In the **Tables** operating mode you can open various tables and edit them as necessary.

Description of function

If you select **Add**, the control displays the **Quick selection new table** and **Open File** workspaces.

In the **Quick selection new table** workspace you can create a new table and open some tables directly.

Further information: Programming and Testing User's Manual

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: Programming and Testing User's Manual

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application. The status of the active application is highlighted in color and for the remaining applications in gray.

You can open the **Table, Form and Document** workspaces in every application.

Further information: "The Table workspace", Page 376

Further information: "The Form workspace for tables", Page 384

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

Further information: Programming and Testing User's Manual



Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Buttons

In the **Tables** operating mode, the function bar contains the following buttons that can be used for any table:

Button	Meaning
Undo	The control undoes the last change.
Redo	The control restores the change that was undone.
GOTO record	The control opens the GOTO jump instruction window. The control selects the line number that you defined.
Edit	If the toggle switch is active, you can edit the table.
Mark row	The control marks the currently selected row.

Depending on the selected table, the control provides the following additional buttons in the function bar:

Button	Meaning
Insert rows	The control opens the Insert rows window where you can insert one or more new rows. If you enable the Append checkbox, the control will insert the rows after the last table row.
Reset row	The control resets all data contained in the row.
Delete rows	The control deletes the currently selected row.
Insert tool	The control opens the Insert tool window where you can define the following: ■ Type: Further information: "Tool types", Page 182 ■ Line number (Tool number?) ■ Number of rows ■ Index Further information: "Indexed tool", Page 178 ■ Append Append rows at the end of the table Further information: "Tool management ", Page 184
Delete tool	The control deletes the tool selected in the tool management. You cannot delete any tools that have been entered into the pocket table. The button is dimmed. Further information: "Tool management ", Page 184
Import	The control imports table contents from a CSV file (e.g., tool data). Further information: "Importing and exporting of table contents", Page 387
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.
Activate the preset	The control activates the currently selected row of the preset table as preset. Further information: "Preset table *.pr", Page 415
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes. Further information: "Write-protection for table rows", Page 418



Refer to your machine manual.

If necessary, the machine manufacturer adapts the buttons.

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

- **Cut**
- **Copy**
- **Reset**
Only for tables that also provide the **Reset row** button
- **Delete**
- **Select all**
- **Cancel marking**
- **Export**
Export data of the selected lines as a CSV file
Further information: "Importing and exporting of table contents", Page 387
- **Cancel**

18.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell



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

i To edit a table content, you can also double-tap or double-click the table cell. The control displays the **Editing disabled. Enable?** window. You can enable the values for editing or cancel the process.

i If the **Edit** toggle switch is enabled, you can edit the contents both in the **Table** workspace and in the **Form** workspace.

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control enables you to transfer tables from previous controls to the TNC7 basic and to adapt them automatically, if needed.
- When you open a table where columns are missing, for example in case of a tool table from a previous control, the control will display the **Incomplete table layout** window.

When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode.

In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.

- If you have, for example, processed tables in a text editor, the control provides the **Update TAB / PGM** function. Use this function to complete an incorrect table format.

i Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., format errors).

Notes about machine parameters

Refer to your machine manual.

- Using the optional machine parameter **CfgTableCellCheck** (no. 141300), the machine manufacturer can define rules for table columns. The machine parameter allows you to define columns as mandatory fields or reset them automatically to a default value. If this rule is not fulfilled, the control will display an information symbol.
- The machine manufacturer uses the machine parameter **CfgTableCellLock** (no. 135600) to define in which cases individual table cells will be blocked or write-protected. On some machines, you cannot change the tool type once a tool has been inserted into the machine.

18.2 The Create new table window

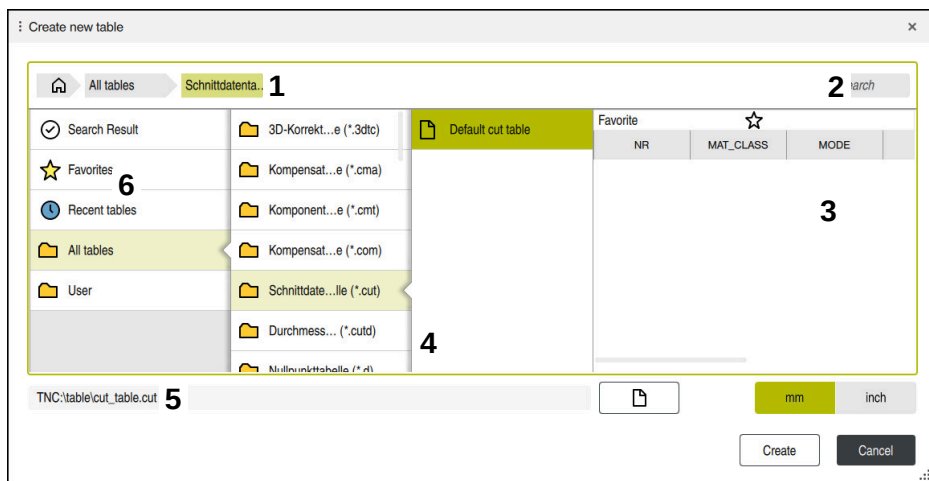
Application

You can create tables using the **Create new table** window in the **Quick selection new table** workspace.

Related topics

- The **Quick selection new table** workspace
Further information: Programming and Testing User's Manual
- Available file types for tables
Further information: Programming and Testing User's Manual

Description of function



The **Create new table** window

The **Create new table** window shows the following areas:

- 1 Navigation path
 In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level. You can change the path or use the history to open a previous path.
- 2 Content columns
 The control shows a folder and the available prototypes for each table type.
- 3 Searching
 You can search for any strings. The control displays the results under **Search Result**.
- 4 The control shows the following information and functions:
 - Add or remove a favorite
 - Preview
- 5 Unit of measure (mm or inches)
- 6 Path of the table to be created

7 Navigation column

The navigation column offers the following possibilities for navigation:

- **Search Result**

- **Favorites**

The control displays all folders and prototypes that you have marked as favorites.

- **Last functions**

The control shows the eleven most recently used prototypes.

- **All functions**

The control shows all available table types in the folder structure.

Notes

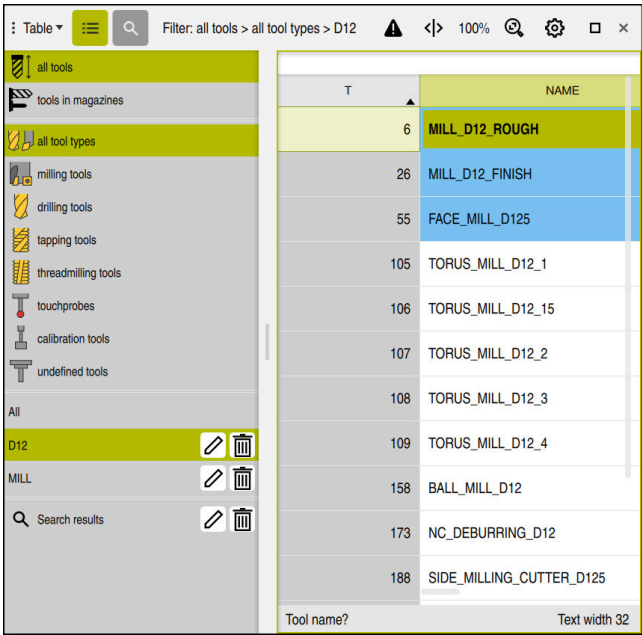
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., +). Due to SQL commands, these characters can cause problems when data are input or read.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer can provide additional areas in the navigation column (e.g., tables for the user).
- With the optional machine parameter **dialogText** (no. 105506), the machine manufacturer can define other names for the table types (e.g., tool table instead of **t**).

18.3 The Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. You can search in all tables and filter the table content.

Description of function



The **Table** workspace

In the **Tables** operating mode, the **Table** workspace is open in every application by default.

The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column in ascending or descending order.

If the table allows it, you can also edit the table contents in this workspace.

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

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

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

Icons and shortcuts

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

Icon or shortcut	Meaning
	Open or close the Filter column Further information: "The Filter column in the Table workspace", Page 378
 [CTRL] + [F]	Open or close the Search column Further information: "The Search column in the Table workspace", Page 380
	■ Activate or deactivate "Rules not met" filter The control displays only the rows that do not meet the rules defined by the machine manufacturer in CfgTable-CellCheck (no. 141300). ■ Open the Data record consistency violations window In these rows, the control shows the icon at the start of the row, even if the filter is inactive. The control displays a window showing how the row violates the rules.
	Edit table characteristics Further information: Programming and Testing User's Manual
100%	Current size of the content Open or close the Scale selection menu
	Reset scaling Set the font size of the table to 100%
	Open or close settings in the Tables window Further information: "Settings in the Table workspace", Page 381
	Open the Search column and edit the selected filter Only in the Filter column Further information: "The Search column in the Table workspace", Page 380
	Delete the selected filter Only in the Filter column Further information: "The Filter column in the Table workspace", Page 378
[CTRL] + [A]	Mark all rows
[CTRL] + [SPACE]	Mark the active row or end the marking function
[SHIFT] + [UP]	Additionally mark the row above
[SHIFT] + [DOWN]	Additionally mark the row below

The Filter column in the Table workspace

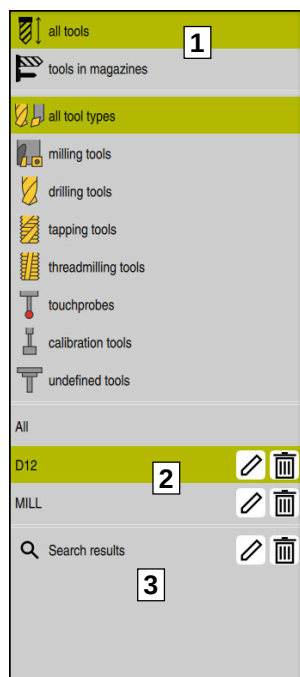
The control provides default filters for the following tables:

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

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

Further information: "User-defined filters", Page 380

The **Filter** column in the **Table** workspace is divided into several filter groups. The control separates the filter groups with a white double line.



The control provides the following filter groups:

- 1 **Default filters**
Filters that are available by default for the respective table
In the **Tool management** application, the default filters comprise two filter groups
- 2 **User-defined filters**
Search processes saved as filters
Further information: "User-defined filters", Page 380
- 3 **Search results**
Results of the **Search** column
Further information: "The Search column in the Table workspace", Page 380

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

When you double-tap or click a filter, the control activates the selected filter in addition to the active filters.

Further information: "Connecting conditions and filters", Page 380

The control highlights active filters green.



Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Filters in the Tool management

The control provides the following default filters in the **Tool management**:

- All tools
- Magazine tools
- All types
- Milling cutters
- Drills
- Taps
- Thread cutters
- Touch probes (#17 / #1-05-1)
- Calibration tools
- Undefined tools

Filters in the Pocket table

The control provides the following default filters in the **Pocket table**:

- All magazines
- spindle
- main magazine
- all pockets
- empty pockets
- occupied pockets
- Locked pockets

Filters in the Presets table

The control provides the following default filters in the **Presets** table:

- Show all
- Basic transformations
- Offsets

User-defined filters

You can additionally create user-defined filters by saving a search.

Further information: "The Search column in the Table workspace", Page 380

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

Further information: "The Filter column in the Table workspace", Page 378

Connecting conditions and filters

The control connects the filters as follows:

- AND operation for several requirements within one filter
You create, for example, a user-defined filter that contains the requirements **R = 8** and **L > 150**. The control filters the table rows when you activate this filter. The control displays only the table rows that meet both requirements at the same time.
- OR operation between filters of the same filter groups
When you activate the default filters **Drills** and **Taps**, for example, the control filters the table rows. The control displays only the table rows that meet at least one of the conditions. The table row must contain either a drill or a tap.
- AND operation between filters of different filter groups
You create, for example, a user-defined filter that contains the requirement **R > 8**. When you activate this filter and the default filter **Milling cutters**, the control filters the table rows. The control displays only the table rows that meet both requirements at the same time.

The Search column in the Table workspace

The control offers a search function in all tables.

You can define multiple search conditions in the **Search** function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**
Use the **Search in** selection menu to select the column.
- Operator if applicable (e.g., **Contains** or **Equal to (=)**)
Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field



If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use Add to add several conditions. The conditions will have a combined effect when you perform the search. You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a user-defined filter. You can assign any name to the filter. Further information: "User-defined filters", Page 380

An unsaved search works like a user-defined filter. If an unsaved search is active, the control highlights the **Search results** filter group green in the **Filter** column.

Further information: "The Filter column in the Table workspace", Page 378



Refer to your machine manual.

This User's Manual describes the basic functions of the control. The machine manufacturer can adapt, enhance or restrict the control functions to the machine. The machine manufacturer can also change the colors in the control's GUI.

Settings in the Table workspace

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

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

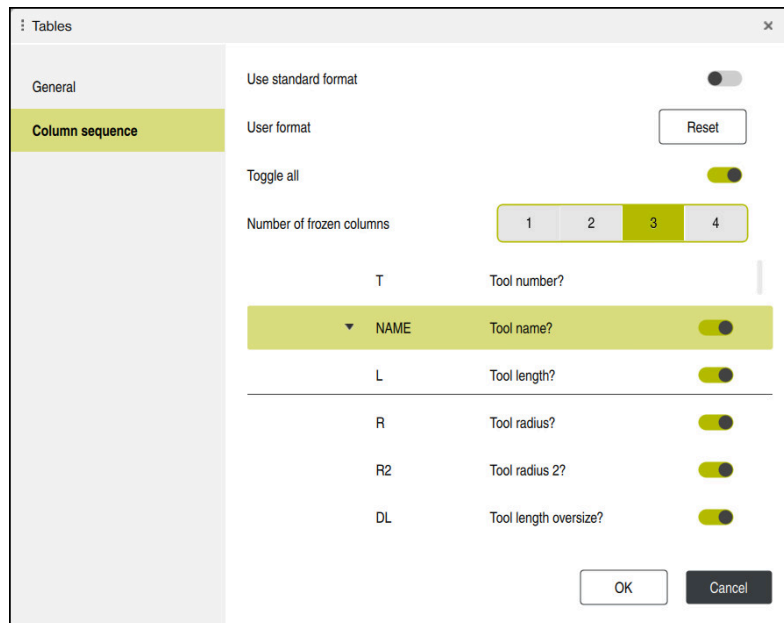
- **General**
- **Column sequence**

The General area

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

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

The Column sequence area



The **Tables** window

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

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

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

Virtual columns

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

The control contains the following virtual columns:

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



Refer to your machine manual.

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

Notes

- If you insert the contents of the clipboard while editing is inactive, the control displays the **Editing disabled. Enable?** prompt. If you select **Yes**, the control inserts the copied contents.
- Refer to your machine manual.
Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.
- Since the values in virtual columns are not included in the table file, you cannot read these values with functions such as:
 - NC functions (e.g., **TABDATA READ**)
 - **HEIDENHAIN DNC** (#18 / #3-03-1)
 - **OPC UA NC Server** (#56-61 / #3-02-1*)

18.4 The Form workspace for tables

Application

In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.

Description of function

The **Form** workspace in the **Favorites** view

The control displays the following information for each parameter:

- An icon, if applicable
- Name
- Text width or input range
Only in the dialog bar
- Unit of measure as needed
Only in the dialog bar
- Description
- Current value

The control displays the contents of specific tables in groups within the **Form** workspace.



Refer to your machine manual.

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

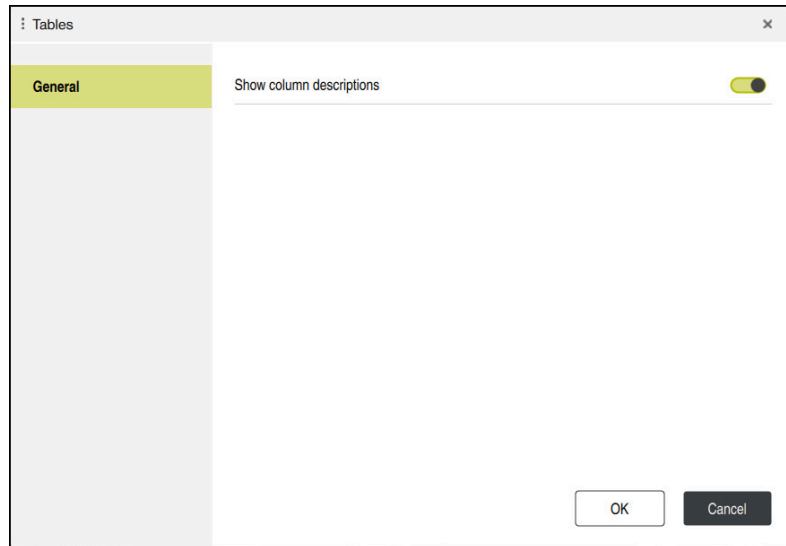
Icons

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

Icon or shortcut	Meaning
	Configure the layout You can make the following layout adaptations: ■ Add or remove areas to the Favorites view ■ Rearrange areas using the gripper ■ Add or remove columns
	Favorites In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites. If the icon is not active, the control shows all areas.
^ [SHIFT] + [UP] ▼ [SHIFT] + [DOWN]	Navigate Navigate between table rows
100%	Current size of the content Open or close the Scale selection menu
	Reset scaling Set font size of the content to 100%
	Settings ■ Open the settings in the Tables window Further information: "Settings in the Form workspace", Page 386 ■ Change the size of the graphic in the Tool Icon area
+	Add The control only shows this icon when you are adapting the layout. With this icon you can add the following elements: ■ Column You can divide the workspace into several columns. Further information: "Adding a column in the workspace", Page 386 ■ Area In the Favorites view you can add another area.
—	Remove The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the parameter descriptions. The selected setting is modally effective.



18.4.1 Adding a column in the workspace

To add a column:



- ▶ Select **Configure the layout**
- > The control enables all functions for adapting the layout of the workspace.



- ▶ In the workspace, swipe to the left



- ▶ Select **Add**
- > The control adds a new column.
- ▶ Move the areas if required



- ▶ Select **Configure the layout**
- > The control saves your changes.

Notes

- In the tool manager, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.
 - The control displays an icon of the selected tool type in the **Tool Icon** area.
- Further information:** "Tool types", Page 182
- Refer to your machine manual.

Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.

18.5 Importing and exporting of table contents

Application

You can import the contents of specific tables into the control and export them from the control. This avoids manual editing efforts and possible typing errors.

Importing tool data is particularly useful in connection with a tool presetter. Via exported contents you can, for example, transfer the tools and presets of the control to the CAM system.

The control uses CSV files for exporting and importing of table contents.

Related topics

- Overview of file types

Further information: Programming and Testing User's Manual

- Fundamentals of the **Tool management** application

Further information: "Tool management ", Page 184

Description of function

Importing of data is possible with all tables that contain the **Import** button in the function bar.

To export contents, you have to mark all desired table rows. If a table does not permit exporting, the **Export** button is grayed out in the action bar.

The CSV file has the following structure:

- The first line contains the column names of the table being exported.
- The other lines contain the exported contents. The contents are arranged in the same sequence as the first line. Decimal numbers are separated by a point.

For compatibility with earlier controls, the control provides the following formatting options for exporting:

- **TNC7 (semicolon-separated)** writes the values between double quotation marks and separates the values by semicolons.

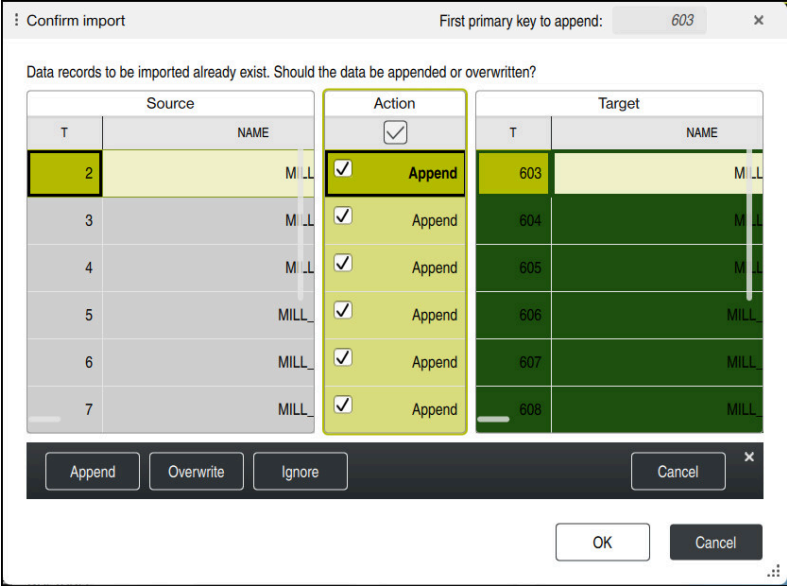
Most table calculation programs use this formatting.

- **iTNC 530 / TNC 640 (comma-separated)** writes the values partly between brackets and separates the values by commas

The control is able to import and export data in both formats.

The Confirm import window

When you select a CSV file for importing, the control will display the **Confirm import** window.



The **Confirm import** window with table rows to be appended

The window contains the following areas:

Area	Meaning															
Source	The control shows the values of the CSV file that are to be imported.															
Action	If you double-tap or double-click a row, the following actions for this row can be selected: ■ Append Append row at end of table ■ Overwrite Import row with line number of the CSV file ■ Ignore Do not import row of the CSV file Clicking the checkbox in the header selects all table rows. You can select the desired action for all rows that have been marked in the action bar.															
Target	The control shows the table values that will result from the currently selected action after importing. Depending on the selected action, the control shows the cells in the following colors: <table><tr><th>Color</th><th>Meaning</th><th>Action</th></tr><tr><td>Yellow</td><td>The content of the source differs from the content of the target table.</td><td>Ignore</td></tr><tr><td>Dark gray</td><td>The table will not contain a row with this number after importing.</td><td>Ignore</td></tr><tr><td>Dark green</td><td>The rows will be added in the target file.</td><td>Append</td></tr><tr><td>Red</td><td>The current table content will be overwritten after importing.</td><td>Overwrite</td></tr></table>	Color	Meaning	Action	Yellow	The content of the source differs from the content of the target table.	Ignore	Dark gray	The table will not contain a row with this number after importing.	Ignore	Dark green	The rows will be added in the target file.	Append	Red	The current table content will be overwritten after importing.	Overwrite
Color	Meaning	Action														
Yellow	The content of the source differs from the content of the target table.	Ignore														
Dark gray	The table will not contain a row with this number after importing.	Ignore														
Dark green	The rows will be added in the target file.	Append														
Red	The current table content will be overwritten after importing.	Overwrite														

The control displays an input field in the title bar of the window. If **Append** is selected, you can optionally enter a row number that will be the start number of the imported table rows. This allows you to define new tool numbers, for example.

18.5.1 Exporting table contents

To export table contents:



- ▶ Select the **Tables** operating mode



- ▶ Select the table (e.g., **Tool management**)



- ▶ Select the row to be exported

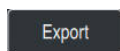


Mark row

- ▶ Select **Mark row**

> The control marks the selected line and opens the action bar.

- ▶ Select further rows, if required



Export

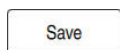
- ▶ Select **Export**

> The control opens the **Save as** window.

- ▶ Select a path

▶ Enter the file name

- ▶ Select the formatting



Save

- ▶ Select **Save**

> The control saves the CSV file using the defined path.



If you export tools from the **Tooling list** (#93 / #2-03-1) table, you have to use the context menu instead of the **Mark row** button. The exported CSV file contains all contents of the tool, and not just the columns of the tooling list. This allows you to import this CSV file to the tool management.

Further information: "Tooling list (#93 / #2-03-1)", Page 413

18.5.2 Importing table contents

To import table contents:



- ▶ Select the **Tables** operating mode



- ▶ Select the table (e.g., **Tool management**)
- ▶ Activate **Edit**
- > The control enables the table for editing.



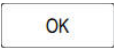
- ▶ Select **Import**
- > The control opens a selection window.
- ▶ Select the desired CSV file



- ▶ Select **Import**
- > The control opens the **Confirm import** window.



- ▶ Click the checkbox in the header of the **Action** area
- > The control marks all rows to be imported and opens the action bar.
- ▶ Select an action:



- **Append**: the control adds the contents as new rows at the end of the table.
 - **Overwrite**: the control overwrites the initial contents with the contents from the CSV file.
 - **Ignore**: the control does not import the contents.
- ▶ Select **OK**
- > The control imports the contents with the selected action.

 If you double-tap or double-click each cell in the **Action** area, the following actions can be selected separately for each row.

NOTICE

Caution: Data may be lost!

When overwriting existing table values with the **Overwrite** function, the control will permanently delete the initial values!

- ▶ Use this function only with table files that are no longer needed

Notes

NOTICE

Caution: Possible material damage!

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

- ▶ Check whether the column names are correct
 - ▶ After importing, check the tool data and adapt them if required.
- The CSV file contains information on the table that the export comes from. Only tables of the same file type allow importing.
 - By default, the control saves the export under **TNC:\system\tooltab**.
 - To be able to import the CSV file, the column with the line numbers must be available.
 - In the **Presets** application, the **Confirm import** window does not include an input field in the title bar. Within the preset table, you can append rows only using the next line number.
 - If rows cannot be overwritten when importing CSV files, the control will display an information symbol. If you select the information symbol, the control displays a pop-up window showing the reason for the problem. The control provides a button with the information symbol. It allows you to filter the contents to find all rows with this problem.

18.6 Tool tables

18.6.1 Overview

This chapter contains information about the tool tables of the control.

- Tool table **tool.t**

Further information: "Tool table tool.t", Page 392

- Touch probe table **tchprobe.tp** (#17 / #1-05-1)

Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 184

18.6.2 Tool table tool.t

Application

The tool table **tool.t** contains the parameters specific to drilling and milling tools. The tool table also contains all parameters that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

- Editing parameters in tool management

Further information: "Tool management ", Page 184

- Tool parameters

Further information: "Tool parameters", Page 175

Description of function

The file name of the tool table is **tool.t** and this table must be stored in the folder **TNC:\table**.

Parameters of the tool table tool.t

The **Tool_management** application shows the following parameters of the tool table:



The **tool.t** file contains, if applicable, additional table columns that are not shown in the **Tool_management** application. These additional columns are not relevant for the TNC7 basic.

Parameter	Meaning
T	Tool number? Row number in the tool table The tool number allows you to identify each tool unambiguously (e.g., for calling a tool). Further information: Programming and Testing User's Manual You can define an index after the period. Further information: "Indexed tool", Page 178 This parameter applies to all tools, regardless of technology.
NAME	Tool name? The tool name identifies a tool, for example when calling it. Further information: Programming and Testing User's Manual You can define an index after a period (i.e., name.index). Further information: "Indexed tool", Page 178 This parameter applies to all tools, regardless of technology.
L 	Tool length? Length of tool, with respect to the tool carrier reference point Further information: "Tool carrier reference point", Page 171
R 	Tool radius? Tool radius, with respect to the tool carrier reference point Further information: "Tool carrier reference point", Page 171
R2 	Tool radius 2? Corner radius for the exact definition of the tool for three-dimensional radius compensation, graphic representation and collision monitoring of, for example, ball-nose cutters or toroid cutters. Further information: Programming and Testing User's Manual
DL 	Tool length oversize? Delta value of tool length as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: Measuring Cycles for Workpieces and Tools User's Manual Is added to the parameter L

Parameter	Meaning
DR 	Tool radius oversize? Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: Measuring Cycles for Workpieces and Tools User's Manual Is added to parameter R
DR2 	Tool radius oversize 2? Delta value of tool radius 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: Measuring Cycles for Workpieces and Tools User's Manual Is added to parameter R2
TL 	Tool locked? Tool is enabled or locked for machining: ■ No value entered: Enabled ■ L: Locked The control locks the tool after exceeding maximum tool age TIME1 , maximum tool age 2 TIME2 or after exceeding one of the parameters for automatic tool measurement. This parameter applies to all tools, regardless of technology.
RT 	Replacement tool? Number of the replacement tool If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool. If M101 is active and the current tool age CUR_TIME exceeds the TIME2 value, the control locks the tool and inserts the replacement tool at a suitable location. Further information: Programming and Testing User's Manual If the replacement tool is not available or locked, the control inserts the replacement tool of the replacement tool. You can define an index after the period. Further information: "Indexed tool", Page 178 If you define the value 0, the control will not use a replacement tool. This parameter applies to all tools, regardless of technology. Selection by means of a selection window
TIME1 	Maximum tool age? Maximum tool age in minutes If the current tool age CUR_TIME exceeds the TIME1 value, the control locks the tool and displays an error message when the tool is called the next time. The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology.

Parameter	Meaning
TIME2 	Max. tool age for TOOL CALL? Maximum tool age 2 in minutes The control inserts a replacement tool in the cases below: ■ When the current tool age CUR_TIME exceeds the TIME2 value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool RT is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message. ■ If M101 is active and the current tool age CUR_TIME exceeds the TIME2 value, the control locks the tool and inserts the replacement tool RT at a suitable location. Further information: Programming and Testing User's Manual The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology.
CUR_TIME 	Current tool age? The current tool age equals the time during which the tool is cutting a workpiece. The tool is cutting a workpiece when the spindle is switched on and the control moves the tool at the machining feed rate. The control counts this time automatically and enters the current tool age in minutes. You can edit the tool age of an active tool during program run after you have inserted an indexable insert, for example. The control will directly apply the value to tool life monitoring. The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program. This parameter applies to all tools, regardless of technology.
TYP	Tool type? Depending on the selected tool type, the control displays the suitable parameters in the Form workspace of the tool management. Further information: "Tool types", Page 182 Further information: "Tool management ", Page 184 This parameter applies to all tools, regardless of technology. Selection using a selection menu
DB_ID	ID for central tool management The database-ID allows you to identify a tool (e.g., within a tool management system by using client applications). Further information: "Database ID", Page 177 For indexed tools, HEIDENHAIN recommends that you assign the database ID to the main tool. Further information: "Indexed tool", Page 178 This parameter applies to all tools, regardless of technology.
DOC	Tool description This parameter applies to all tools, regardless of technology.
PLC	PLC status? Tool information for the PLC Refer to your machine manual. This parameter applies to all tools, regardless of technology.

Parameter	Meaning
LCUTS 	Tooth length in the tool axis? Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring.
LU 	Usable length of the tool? Usable length of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., of necks of end mills).
RN 	Neck radius of the tool? Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of, for example, necks of end mills or side milling cutters. The tool can contain a neck radius RN only if the useful length LU is longer than the LCUTS length of the cutting edge.
R_TIP 	Radius at the tip Radius at the tool tip for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of tools such as countersinks.
ANGLE 	Maximum plunge angle? Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles.
CUT 	Number of teeth? Number of teeth of the tool for automatic tool measurement or cutting data calculation. Further information: Measuring Cycles for Workpieces and Tools User's Manual Further information: Programming and Testing User's Manual
TMAT 	Tool material? Tool material from the tool material table TMAT.tab for cutting data calculation. Further information: Programming and Testing User's Manual Selection by means of a selection window
CUTDATA 	Cutting data table? Select the cutting data table with the *.cut or *.cutd file extension for cutting data calculation. Further information: Programming and Testing User's Manual Selection by means of a selection window
LTOL 	Wear tolerance: length? Permitted tool length deviation in wear detection for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the TL parameter.

Parameter	Meaning
RTOL 	Wear tolerance: radius? Permitted tool radius deviation in wear detection for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the TL parameter.
R2TOL 	Wear tolerance: Radius 2? Permitted tool radius 2 deviation in wear detection for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the TL parameter.
DIRECT  	Cutting direction? Cutting direction for exact definition of a tool for graphical representation, automatic tool measurement, and calculation of traverse movements. For milling tools you indicate the rotational direction of the tool spindle with which the tool cuts. ■ -: M3 ■ +: M4
R-OFFS 	Tool offset: radius? Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual
L-OFFS 	Tool offset: length? Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual Is added to the machine parameter offsetToolAxis (no. 122707)
LBREAK 	Breakage tolerance: length? Permitted tool length deviation in breakage detection for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the TL parameter.
RBREAK 	Breakage tolerance: radius? Permitted tool radius deviation in breakage detection for automatic tool measurement. Further information: Measuring Cycles for Workpieces and Tools User's Manual If the entered value is exceeded, the control locks the tool in the TL parameter.
NMAX 	Maximum speed [rpm] Limitation of spindle speed for the programmed value including control by the potentiometer.

Parameter	Meaning
LIFTOFF 	Lift-off allowed? Allow automatic tool lift-off with active M148 or FUNCTION LIFTOFF : ■ Y: Activate LIFTOFF ■ N: Deactivate LIFTOFF Further information: Programming and Testing User's Manual
TP_NO	Number of the touch probe Number of touch probe in the touch probe table tchprobe.tp Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401 Selection by means of a selection window
T-ANGLE 	Point angle Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of drilling tools, for example. Further information: User's Manual for Machining Cycles
PITCH 	Tool thread pitch? Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread. Further information: User's Manual for Machining Cycles
AFC	Feedback-control strategy Control setting for adaptive feed control (AFC (#45 / #2-31-1)) from the AFC.tab table Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256 Selection by means of a selection window
AFC-LOAD	Reference power for AFC [%] Tool-dependent reference power for AFC (#45 / #2-31-1). The input in percent refers to the rated spindle power. The control immediately uses the value given for feedback control, meaning a teach-in cut is dropped. Calculate the value beforehand with a teach-in step. Further information: "AFC teach-in cut", Page 262
AFC-OVLD1	AFC overload warning level [%] Cut-related tool wear monitoring for AFC (#45 / #2-31-1). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. Further information: "Monitoring tool wear and tool load", Page 264
AFC-OVLD2	AFC overload switch-off level [%] Cut-related tool load monitoring for AFC (#45 / #2-31-1). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. Is this parameter contains a value, the control ignores the AFC-OVLD1 parameter. Further information: "Monitoring tool wear and tool load", Page 264

Parameter	Meaning
LAST_USE 	Date/time of last tool usage The time at which the tool was last used The control updates the value cyclically during NC program run, as well as during a tool call and at the end of the program. This parameter applies to all tools, regardless of technology.
PTYP	Tool type for pocket table? Tool type for evaluation in the pocket table Further information: "Pocket table tool_p.tch", Page 406 Refer to your machine manual. This parameter applies to all tools, regardless of technology.
ACC	ACC active? Activate or deactivate active chatter control (ACC (#145 / #2-30-1)): ■ Y: Activate ■ N: Deactivate Further information: "Active Chatter Control (ACC) (#145 / #2-30-1)", Page 266
KINEMATIC 	Tool-carrier kinematics Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring. Further information: "Tool carrier management", Page 186 Selection by means of a selection window This parameter applies to all tools, regardless of technology.
TSHAPE 	3D tool model Assigning a 3D model for exact definition of the tool for graphical simulation and collision monitoring. Further information: "Tool model (#140 / #5-03-2)", Page 190 Selection by means of a selection window
OVRTIME 	Tool life expired Time in minutes during which the tool may be used beyond the tool life defined in the TIME2 parameter. The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control will use the parameter when searching for tool names. Refer to your machine manual. This parameter applies to all tools, regardless of technology.
RCUTS 	Width of the indexable insert Front-face width of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring (e.g., for indexable inserts).

Notes

- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

Further information: "Creating a tool table in inches", Page 405

- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control changes only the position of the tool in the simulation.
- Assign unique tool names!

If you define identical tool names for multiple tools, the control will look for the tool in the following sequence:

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



Refer to your machine manual.

If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

- Tool that is defined in the tool table but is currently not in the magazine
If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with the least remaining tool life.
- In the machine parameter **offsetToolAxis** (no. 122707), the machine manufacturer defines the distance between the upper edge of the tool touch probe and the tool tip.
The parameter **L-OFFS** is added to this defined distance.
- In the machine parameter **zeroCutToolMeasure** (no. 122724), the machine manufacturer defines whether the control takes the parameter **R-OFFS** into account for automatic tool measurement.
- With the optional machine parameter **resetOnTypeChange** (no. 125304), you can define how the control will react when the tool type is changed. The machine manufacturer enables this parameter. If this machine parameter has been set to **TRUE** and you change the tool type, the control will first display a confirmation prompt and then reset all tool parameters.

18.6.3 Touch probe table **tchprobe.tp** (#17 / #1-05-1)

Application

In the touch probe table **tchprobe.tp** you define the parameters of the touch probe for the touching process, such as the probing feed rate. If you use several touch probes, you can save separate parameters for each touch probe.

Related topics

- Editing parameters in tool management
Further information: "Tool management ", Page 184
- Tool parameters
Further information: "Tool parameters", Page 175
- Touch probe functions
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305
- Calibrating touch probe cycles for the workpiece touch probe
Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Calibrating touch probe cycles for the tool touch probe
Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Automatic touch probe cycles for the workpiece
Further information: Measuring Cycles for Workpieces and Tools User's Manual
- Automatic touch probe cycles for the tool
Further information: Measuring Cycles for Workpieces and Tools User's Manual

Description of function

The file name of the touch probe table is **tchprobe.tp** and this table must be stored in the folder **TNC:\table**.

Parameters of the touch probe table tchprobe.tp

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	Sequential number of touch probe Enter this number in the TP_NO parameter of the tool management. The control links the data of the touch probe table to the tool management.
TYPE	Selection of the touch probe?   The TS 642 touch probe makes the following values available: ■ TS642-3: The touch probe is activated by a conical switch. This mode is not supported. ■ TS642-6: The touch probe is activated by an infrared signal. Select this mode. Selection using a selection menu
STYLUS	Shape of the stylus ■ SIMPLE: Straight stylus ■ L-TYPE: L-shaped stylus If you do not define the parameter, the control uses SIMPLE Selection using a selection menu
CAL_OF1	TS center misalignmt. ref. axis? [mm] Depending on the selection of the STYLUS parameter, this parameter has the following function: ■ SIMPLE: Offset of the touch probe axis to the spindle axis in the main axis ■ L-TYPE: Length of extension on an L-shaped stylus Required when ON is selected in parameter TRACK The control describes this value in connection with the calibration cycle.
CAL_OF2	TS center misalignmt. aux. axis? [mm] Offset of the touch probe axis to the spindle axis in the secondary axis Required when ON is selected in parameter TRACK The control describes this value in connection with the calibration cycle.
CAL_ANG	Spindle angle for calibration? Depending on the selection of the STYLUS parameter, this parameter has the following function: ■ SIMPLE: Prior to calibrating or probing, the control orients the touch probe with this spindle angle (if possible). ■ L-TYPE: The control orients the extension using the spindle angle. Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible). Required when ON is selected in parameter TRACK
F	Probing feed rate? [mm/min] In the machine parameter maxTouchFeed (no. 122602), the machine manufacturer defines the maximum probing feed rate. If F is greater than the maximum probing feed rate, then the maximum probing feed rate will be used.

Parameter	Meaning
FMAX 	Rapid traverse in probing cycle? [mm/min] Feed rate at which the control pre-positions the touch probe and positions it between the measuring points
DIST 	Maximum measuring range? [mm] If the stylus is not deflected in a probing process within the defined value, the control will display an error message.
SET_UP 	Set-up clearance? [mm] Distance of touch probe from the defined touch point when pre-positioning The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value.
F_PREPOS 	Pre-position at rapid? ENT/NOENT Speed for pre-positioning: ■ FMAX_PROBE: Pre-position at the speed from FMAX ■ FMAX_MACHINE: Pre-position at machine rapid traverse Selection using a selection menu
TRACK 	Probe oriented? Yes=ENT/No=NOENT Orienting the infrared touch probe in each probing process: ■ ON: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy. ■ OFF: The control will not orient the touch probe. When selecting L-TYPE in the STYLUS parameter, ON must be selected. If you change the TRACK parameter, you must recalibrate the touch probe.
SERIAL 	Serial number? The control automatically edits this parameter of touch probes with an EnDat interface.
REACTION 	Reaction? EMERGSTOP=ENT/NCSTOP=NOENT As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. Reaction to resetting the ready signal: ■ NCSTOP: Interrupting an NC program ■ EMERGSTOP: Emergency stop, quick braking of the axes Selection using a selection menu

Editing the touch probe table

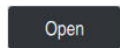
To edit the touch probe table:



- ▶ Select the **Tables** operating mode



- ▶ Select **Add**
- > The control opens the **Quick selection** and **Open File** workspaces.
- ▶ Select the **tchprobe.tp** file in the **Open File** workspace



- ▶ Select **Open**
- > The control opens the **Touch probes** application.



- ▶ Activate **Edit**
- ▶ Select the desired value
- ▶ Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

18.6.4 Creating a tool table in inches

To create a tool table in inches:



- ▶ Select the **Manual** operating mode



- ▶ Select **T**



- ▶ Select the tool **T0**

- ▶ Press the **NC Start** key

- The control removes the current tool and does not insert a new tool.



- ▶ Restart the control

- ▶ Do not acknowledge **Power interrupted**

- ▶ Select the **Files** operating mode



- ▶ Open the **TNC:\table** folder

- ▶ Rename the original file (e.g., **tool.t** as **tool_mm.t**)

- ▶ Select the **Tables** operating mode



- ▶ Select **Create new table**

- The control opens the **Create new table** window.

- ▶ Select a folder with the corresponding table type (e.g., **t**)

- ▶ Select INCH as the unit of measure if necessary

- ▶ Select the desired prototype



- ▶ Select a path

- The control opens the **Save as** window.

- ▶ Select the **table** folder

- ▶ Enter a name (e.g., **tool**)

- ▶ Select **Create** twice

- The control opens the **Tool table** tab in the **Tables** operating mode.



- ▶ Restart the control



- ▶ Acknowledge **Power interrupted** with the **CE** key



- ▶ Select the **Tool table** tab in the **Tables** operating mode

- The control uses the newly created table as a tool table.



To use the **Tool management** application you have to create all existing tool tables in inches.

18.7 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. The control needs the pocket table in order to change the tool.

Related topics

- Tool call
Further information: Programming and Testing User's Manual
- Tool table
Further information: "Tool table tool.t", Page 392

Requirement

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

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	Pocket number? Pocket number of the tool in the tool magazine
T	Tool number? Row number of the tool from the tool table With the machine parameter deleteLoadedTool (no. 125301) you define whether you are allowed to edit the T column. The machine manufacturer enables this parameter. Further information: "Tool table tool.t", Page 392 Selection by means of a selection window
RSV	Reserve pocket? When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: ■ No value entered: Pocket is not reserved ■ R: Pocket is reserved
ST	Special tool? Define the tool as a special tool (e.g., with oversize tools): ■ No value entered: No special tool ■ S: Special tool
F	Fixed pocket? Always return the tool to the same pocket in the tool magazine (e.g., with special tools) To define a fixed pocket for the tool: ■ No value entered: No fixed pocket ■ F: Fixed pocket

Parameter	Meaning
L	Locked pocket? To lock a pocket for tools (e.g., the pockets next to special tools): ■ No value entered: Do not lock ■ L: Lock
NAME	Tool name? Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. Further information: "Tool table tool.t", Page 392
DOC	Tool comment? Tool comment from the tool table When you define the tool number, the control will automatically load the tool comment. Further information: "Tool table tool.t", Page 392
PLC	PLC status? Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.
P1 ... P5	Value? The machine manufacturer defines the function of this parameter. Refer to your machine manual.
PTYP	Tool type for pocket table? Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual.
LOCKED_ABOVE	Lock pocket above? Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual.
LOCKED_BELOW	Lock pocket below? Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual.
LOCKED_LEFT	Lock pocket at left? Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual.
LOCKED_RIGHT	Lock pocket at right? Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual.
S1	S1 Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.
S2	S2 Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual.

Notes

- If a tool has been saved to the pocket table, you can neither reset the corresponding row in tool management nor delete the tool. The tool must be removed from the magazine first.
- Changing a tool parameter in the pocket table also changes that same parameter in the tool manager.
- In the **Table** workspace of the pocket table, the control displays the additional virtual **MAGAZINE** and **TOOL_LIFE** columns.

Further information: "Virtual columns", Page 383

18.8 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test
Further information: "Tool usage test", Page 193
- Working with a pallet table
Further information: Programming and Testing User's Manual
- Tool data from the tool table
Further information: "Tool table tool.t", Page 392

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
In the machine parameter **createUsageFile** (no. 118701), the machine manufacturer defines whether the **Generate tool-usage file** function will be enabled.
Further information: "Creating the tool usage file", Page 193
- The **Generate tool-usage file** setting is set to **Once** or **Always**
Further information: "The Machine and Simulation areas", Page 475

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file
TOKEN	In the TOKEN column, the control uses one word to show which information is contained in the respective row: ■ TOOL: Data per tool call; listed in chronological order ■ TTOTAL: All data of a tool; listed in alphabetical order ■ STOTAL: Called NC programs; listed in chronological order ■ TIMETOTAL: Total tool usage time of an NC program ■ TOOLFILE: Path of the tool table This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table tool.t
TNR	Tool number If the control has not yet inserted a tool, the column contains the value -1.
IDX	Tool index
NAME	Tool name
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements)

Parameter	Meaning
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece
RAD	Sum of the tool radius R and the delta radius DR from the tool table
BLOCK	NC block number of the tool call
PATH	Path of the NC program, the pallet table, or the tool table
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value -1 .
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value 100 .
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value -1 .
NAMEPRG	Type of tool definition during a tool call: ■ 0: The tool number is programmed ■ 1: The tool name is programmed
LINENR	Row number of the pallet table in which the NC program is defined

Note

The control saves the tool usage file as a dependent file (***.dep**).

In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: Programming and Testing User's Manual

18.9 T usage order (#93 / #2-03-1)

Application

In the **T usage order** table, the control displays the tool call sequence in an NC program. Before starting the program, you can see, for example, when a manual tool change will take place.

Requirements

- Ext. Tool Management (#93 / #2-03-1) software option
- Tool-usage file has been created
 - Further information:** "Creating the tool usage file", Page 193
 - Further information:** "Tool usage file", Page 409

Description of function

When you select an NC program in the **Program Run** operating mode, the control will automatically create the **T usage order** table. The control displays the table in the **T usage order** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot edit the table.

The **T usage order** table provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows
T	Number of the tool used, including an index as needed Further information: "Indexed tool", Page 178 May differ from the programmed tool (e.g., when a replacement tool is used)
NAME	Name of the tool used, including an index as needed Further information: "Indexed tool", Page 178 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ OK: Tool is in order ■ Locked: Tool is locked ■ Not found: Tool is not defined in the pocket table Further information: "Pocket table tool_p.tch", Page 406 ■ T no. missing: Tool is not defined in the tool management Further information: "Tool management ", Page 184
T PROG	Number or name of the programmed tool, including an index as needed Further information: "Indexed tool", Page 178
USAGE	Total tool usage time from the WTIME column of the tool usage file (in seconds) Total time between the tool changes, during which the tool is cutting a workpiece Further information: "Tool usage file", Page 409
TOOL TIME	Estimated time of tool change

Parameter	Meaning
M3/M4 TIME	Tool usage time from the TIME column of the tool usage file (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Further information: "Tool usage file", Page 409
MIN OVRD	Minimum value of the feed-rate potentiometer during program run (in percent)
MAX OVRD	Maximum value of the feed-rate potentiometer during program run (in percent)
NC PGM	Path of the NC program in which the tool is programmed
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. Further information: "Pocket table tool_p.tch", Page 406

18.10 Tooling list (#93 / #2-03-1)

Application

In the **Tooling list** table, the control displays information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine.

Requirements

- Ext. Tool Management (#93 / #2-03-1) software option
- Tool-usage file has been created
 - Further information:** "Creating the tool usage file", Page 193
 - Further information:** "Tool usage file", Page 409

Description of function

When you select an NC program in the **Program Run** operating mode, the control will automatically create the **Tooling list** table. The control displays the table in the **Tooling list** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in numerical order. You cannot edit the table.

The **Tooling list** table provides the following parameters:

Parameter	Meaning
T	Number of the tool used, including an index as needed Further information: "Indexed tool", Page 178 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: ■ OK: Tool is in order ■ Locked: Tool is locked ■ Not found: Tool is not defined in the pocket table Further information: "Pocket table tool_p.tch", Page 406 ■ T no. missing: Tool is not defined in the tool management Further information: "Tool carrier management", Page 186
T PROG	Number or name of the programmed tool, including an index as needed Further information: "Indexed tool", Page 178
M3/M4 TIME	Tool usage time from the TIME column of the tool usage file (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Further information: "Tool usage file", Page 409
MAGAZINE	In this column, the control writes whether the tool is currently in the magazine or in the spindle. This column remains empty if the tool is a zero tool or not defined in the pocket table. Further information: "Pocket table tool_p.tch", Page 406

Note

When you export the contents of the **Tooling list** table, the CSV file contains all contents of the tool, and not just the columns of the Tooling list. This allows you to import this CSV file to the tool management.

Further information: "Importing and exporting of table contents", Page 387

18.11 Preset table *.pr

Application

The **preset.pr** preset table allows you to manage presets, such as the position and misalignment of a workpiece in the machine. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 167

Related topics

- Setting and activating presets

Further information: "Preset management", Page 209

Description of function

By default, the preset table has the name **preset.pr**, and is saved in the **TNC:\table** directory. The preset table is open in the **Tables** operating mode by default.



Refer to your machine manual.

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

In the optional machine parameter **basisTrans** (no. 123903), the machine manufacturer defines a specific preset table for each range of traverse.

Icons and buttons of the preset table

The preset table contains the following icons:

Icon	Meaning
	Active row
	Write-protected row

When you define a preset, the control opens a window with the following input options:

Button	Function
Set a preset	The control interprets the entered value as desired display value for the actual position. The control calculates the required table value from this. The entered value is active in the basic coordinate system B-CS. Further information: "Basic coordinate system B-CS", Page 200 When you activate the edited preset, the control displays the entered value as actual position in the position display.
Correct	The control offsets the entered value against the actual table value. You can enter either a positive or a negative value. The entered value is active incrementally in the basic coordinate system B-CS.

Button	Function
Edit	The control accepts the entered value unchanged as table value. The entered value refers to the coordinate origin of the basic coordinate system B-CS.

Parameters of the preset table

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row
DOC	Comment
X	X coordinate of preset Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 200
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 200
Z	Z coordinate of preset Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 200
SPA	Spatial angle of preset in the A axis Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 200 Has the effect of a 3D basic rotation for tool axis Z Further information: "Basic rotation and 3D basic rotation", Page 211
SPB	Spatial angle of preset in the B axis Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 200 Has the effect of a 3D basic rotation for tool axis Z Further information: "Basic rotation and 3D basic rotation", Page 211
SPC	Spatial angle of preset in the C axis Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 200 Has the effect of a basic rotation for tool axis Z Further information: "Basic rotation and 3D basic rotation", Page 211
X_OFFS	Position of the X axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
Y_OFFS	Position of the Y axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
Z_OFFS	Position of the Z axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198

Parameter	Meaning
A_OFFS	Axis angle of the A axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
B_OFFS	Axis angle of the B axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
C_OFFS	Axis angle of the C axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
U_OFFS	Position of the U axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
V_OFFS	Position of the V axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
W_OFFS	Position of the W axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 198
ACTNO	Active workpiece preset The control automatically enters 1 in the active row.
LOCKED	Write-protection of the table row



Refer to your machine manual.

In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Basic transformation and offset

The control interprets the basic transformations **SPA**, **SPB** and **SPC** as basic rotation or 3D basic rotation in the workpiece coordinate system **W-CS**. During program execution, the control moves the linear axes in accordance with the basic rotation without any change in the workpiece position.

Further information: "Basic rotation and 3D basic rotation", Page 211

The control interprets all offsets for each respective axis as a shift in the machine coordinate system **M-CS**. The effect that offsets have is contingent on the kinematics.

Further information: "Machine coordinate system M-CS", Page 198



HEIDENHAIN recommends using 3D basic rotation because of its greater flexibility.

Application example

Use the **Rotation (ROT)** touch probe function to determine workpiece misalignment. You can transfer the result to the preset table either as a basic transformation or as an offset.

Further information: "Determining and compensating for the basic rotation of a workpiece", Page 318

Calculated results	Actual value	Nominal value
☒ Basic rotation ☐ Table rotation	-360.00000	-360.00000 °
	0.00000	0.00000 °
Compensate the active preset Align rotary table Correct the pallet reference point		

Results of the **Rotation (ROT)** probing function

If you activate the **Basic rotation** toggle switch, the control interprets the misalignment as a basic transformation. When using the **Compensate the active preset** button, the control saves the result in the columns **SPA**, **SPB** and **SPC** of the preset table. The **Align rotary table** button has no function in this case.

If you activate the **Table rotation** toggle switch, the control interprets the misalignment as an offset. When using the **Compensate the active preset** button, the control saves the result in the columns **A_OFFS**, **B_OFFS** and **C_OFFS** of the preset table. To move the rotary axes to the position of the offset, use the **Align rotary table** button.

Write-protection for table rows

The **Lock record** button allows protecting any rows of the preset table against overwriting. The control enters the value **L** in the **LOCKED** column.

Further information: "Protecting table rows without a password", Page 419

Alternatively, the row can be protected with a password. The control enters the value **###** into the **LOCKED** column.

Further information: "Protecting table rows with a password", Page 419

The control displays an icon ahead of write-protected rows.



If the control displays the value **OEM** in the **LOCKED** column, this column has been locked by the machine manufacturer.

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

- ▶ Protecting table rows without a password is recommended
- ▶ Note down your passwords

18.11.1 Activating write protection

Protecting table rows without a password

To protect a table row without a password:



- ▶ Activate the **Edit** toggle switch



- ▶ Select the desired row
- ▶ Activate the **Lock record** toggle switch
- > The control enters the value **L** in the **LOCKED** column.



- > The control activates write-protection and displays an icon ahead of the row.

Protecting table rows with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

- ▶ Protecting table rows without a password is recommended
- ▶ Note down your passwords

To protect a table row with a password:



- ▶ Activate the **Edit** toggle switch



- ▶ Double-tap or double-click the **LOCKED** column of the desired row
- ▶ Enter the password
- ▶ Confirm your input
- > The control enters the value **###** in the **LOCKED** column.
- > The control activates write-protection and displays an icon ahead of the row.

18.11.2 Removing write protection

Unlocking table rows that are protected without a password

To unlock a table row that is protected without a password:



- ▶ Activate the **Edit** toggle switch



- ▶ Deactivate the **Lock record** toggle switch
- The control removes the value **L** from the **LOCKED** column.
- The control deactivates the write protection and removes the icon ahead of the row.

Unlocking table rows that are protected with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This would lock the protected rows permanently.

- ▶ Protecting table rows without a password is recommended
- ▶ Note down your passwords

To unlock a table row that is protected with a password:



- ▶ Activate the **Edit** toggle switch
- ▶ Double-tap or double-click the **LOCKED** column of the desired row
- ▶ Delete **###**
- ▶ Enter the password
- ▶ Confirm your input
- The control deactivates write-protection and removes the icon ahead of the row.

18.11.3 Creating a preset table in inches

If you define inches as the unit of measure in the **Machine Settings** menu item, the unit of measure of the preset table will not be adjusted automatically.

Further information: "The Machine Settings menu item", Page 475

To create a preset table in inches:

	▶ Restart the control
	▶ Do not acknowledge Power interrupted
	▶ Select the Files operating mode
	▶ Open the TNC:\table folder
	▶ Rename the original file preset.pr (e.g., as preset_mm.pr)
	▶ Select the Tables operating mode
	▶ Select Create new table
	> The control opens the Create new table window.
	▶ Select the pr folder
	▶ Select INCH as the unit of measure if necessary
	▶ Select the desired prototype
	▶ Select a path
	> The control opens the Save as window.
	▶ Select the table folder
	▶ Enter the name preset.pr
	▶ Select Create twice
	> The control opens the Presets tab in Tables operating mode.
	▶ Restart the control
	▶ Acknowledge Power interrupted with the CE key
	▶ Select the Presets tab in Tables operating mode
	> The control uses the newly created table as a preset table.
	> The control shows INCH as the unit of measure in the dialog bar of the workspaces.

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept. If the previous value is kept, there is a danger of collision!

- ▶ Before activating a preset, check whether all columns contain values.
- ▶ For undefined columns, enter values (e.g., **0**)
- ▶ As an alternative, have the machine manufacturer define **0** as the default value for the columns

- To optimize the file size and the processing speed, keep the preset table as short as possible.
- New rows can be inserted only at the end of the preset table.
- If you edit the value of the **DOC** column, then the preset must be reactivated. Only then does the control apply the new value.

Further information: "Activating presets", Page 210

- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

Further information: Programming and Testing User's Manual

- If a manual probing process or an NC program is interrupted or stopped, you cannot edit the preset table. When you double-tap or double-click a table cell the control shows the **Editing not possible. Perform internal stop?** window. If you select **Yes**, the control may lose touch points or modally active program information.

Notes about machine parameters

- In the optional machine parameter **initial** (no. 105603), the machine manufacturer defines a default value for every column of a new row.
- If the unit of measure of the preset table does not match the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the dialog bar of the **Tables** operating mode.
- The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offsets in the following NC functions:
 - **FUNCTION PARAXCOMP**
 - **POLARKIN** (#8 / #1-01-1)
 - **FUNCTION TCPM** or **M128** (#9 / #4-01-1)

18.12 Tables for AFC (#45 / #2-31-1)

18.12.1 Basic AFC settings in AFC.tab

Application

In the **AFC.tab** table, you define the feed-rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

Related topics

- Programming AFC

Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256

Requirement

- Adaptive Feed Contr. (#45 / #2-31-1) software option

Description of function

The data in this table are default values that, during a teach-in cut, are copied into an associated dependent file of the relevant NC program. The values are the basis for feedback control.

Further information: "Description of function", Page 426



If you define a tool-specific reference power in the **AFC-LOAD** column in the tool table, the control will create the associated dependent file for the respective NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Parameter

The **AFC.tab** table provides the following parameters:

Parameter	Meaning
NR	Row number in the table
AFC	Name of the control setting Enter this name in the AFC tool management column. It specifies the assignment of the control parameters to the tool.
FMIN	Feed rate at which the control will perform an overload response Enter the value in percent of the programmed feed rate If the AFC.TAB columns FMIN and FMAX each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. Further information: "Monitoring tool wear and tool load", Page 264
FMAX	Maximum feed rate within the material up to which the control can automatically increase the feed rate Enter the value in percent of the programmed feed rate If the AFC.TAB columns FMIN and FMAX each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. Further information: "Monitoring tool wear and tool load", Page 264
FIDL	Feed rate at which the control will traverse the tool outside of the material Enter the value in percent of the programmed feed rate
FENT	Feed rate at which the control will move the tool into and out of the material Enter the value in percent of the programmed feed rate
OVLD	Desired reaction of the control to overload: ■ M: Execution of a macro defined by the machine manufacturer ■ S: Immediate NC stop ■ F: Execute NC stop when the tool is no longer in the material ■ E: Just display an error message on the screen ■ L: Disable active tool ■ -: No overload reaction If the maximum spindle power is exceeded for more than one second and the feed rate falls below the defined minimum while feedback control is active, the control will conduct an overload reaction. In conjunction with the cut-related tool wear monitoring function, the control will evaluate only the options M, E, and L! For tool-load monitoring with the column AFC_OVLD2, this parameter has no function.
POUT	Spindle power at which the control will detect that the tool exits the workpiece Enter the value in percent of the learned reference load Recommended input value: 8%
SENS	Sensitivity (aggressiveness) of feedback control 50 is for slow feedback control, 200 for a very aggressive feedback control. An aggressive feedback control responds quickly and significantly changes the values, but it tends to overshoot.

Parameter	Meaning
PLC	Value that the control will transfer to the PLC at the beginning of a machining step The machine manufacturer defines whether and which function will be performed by the control.

Notes

- If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a permanently defined, internal control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends using the AFC.TAB table in order to ensure safe and well-defined operation.
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., +). Due to SQL commands, these characters can cause problems when data are input or read.

Further information: Programming and Testing User's Manual

18.12.2 AFC.DEP settings file for teach-in cuts

Application

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. In addition, the control measures the maximum spindle power consumed during the teach-in cut and saves this value to the table.

Related topics

- AFC basic settings in the table **AFC.tab**
Further information: "Basic AFC settings in AFC.tab", Page 423
- Setting up and using AFC
Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256

Requirement

- Adaptive Feed Contr. (#45 / #2-31-1) software option

Description of function

Each row in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file for optimization purposes. If you have optimized the values from the AFC.TAB table, the control places a * in front of these control settings in the AFC column.

Further information: "Basic AFC settings in AFC.tab", Page 423

In addition to the contents from the **AFC.tab** table, the **AFC.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed (not editable)
	 In conjunction with AFC (#45 / #2-31-1), the following characters are not permitted in the tool name: # \$ & , .
IDX	Index of the tool with which the machining step was performed (not editable)
N	Difference for tool call: ■ 0: Tool was called by its tool number ■ 1: Tool was called by its tool name
PREF	Reference load of the spindle. The control measures the value in percent with respect to the rated spindle power
ST	Status of the machining step: ■ L: In the next program run, a teach-in cut is recorded for this machining step. The control overwrites any existing values in this row ■ C: The teach-in cut was completed successfully. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

Notes

- Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.

The control does not remove the editing lock until one of the following functions has been executed:

- **M2**
- **M30**
- **END PGM**
- In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: Programming and Testing User's Manual

18.12.3 Log file AFC2.DEP

Application

The control stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP** file. The string **<name>** is identical to the name of the NC program for which you have recorded the teach-in cut. During feedback control, the control updates the data and performs various evaluations.

Related topics

- Setting up and using AFC

Further information: "Adaptive Feed Control (AFC) (#45 / #2-31-1)", Page 256

Requirement

- Adaptive Feed Contr. (#45 / #2-31-1) software option

Description of function

The **AFC2.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed
IDX	Index of the tool with which the machining step was performed
SNOM	Nominal spindle speed [rpm]
SDIFF	Maximum difference of the spindle speed in % of the nominal speed
CTIME	Machining time (tool in effect)
FAVG	Average feed rate (tool in effect)
FMIN	Smallest occurring feed factor. The control shows the value as a percentage of the programmed feed rate
PMAX	Maximum recorded spindle power during machining. The control shows the value as a percentage of the spindle's rated power
PREF	Reference load of the spindle. The control shows the value as a percentage of the spindle's rated power
OVL	Overload reaction performed by the control: ■ M: A macro defined by the machine manufacturer has been run ■ S: Immediate NC stop was conducted ■ F: NC stop was performed once the tool was no longer in the material ■ E: An error message was displayed ■ L: The current tool was locked ■ -: There was no overload response
BLOCK	Block number at which the machining step begins



During feedback control, the control determines the current machining time as well as the resulting time saving in percent. The control enters the results of the evaluation between the key words **total** and **saved** in the last line of the log file. Where the time balance is positive, the percentage value is also positive.

Note

In the settings of the **Files** operating mode, you can specify whether the control displays dependent files in the file management.

Further information: Programming and Testing User's Manual

18.12.4 Editing the tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control provides only the tables of the active NC program.

To open a table for AFC:



AFC settings

- ▶ Select the **Program Run** operating mode
- ▶ Select **AFC settings**
- > The control displays a selection menu. The control displays all the tables available for this NC program.
- ▶ Select a file (e.g., **AFC.TAB**)
- > The control opens the file in the **Tables** operating mode.

19

**Electronic
handwheel**

19.1 Fundamentals

Application

The electronic handwheel allows you to traverse the axes without needing to stand at the operating panel. You can also use the handwheel to perform control functions, such as setting up the machine or influencing program run.



Refer to your machine manual.

The machine manufacturer defines which functions are available on the handwheel and how the control evaluates these functions. This chapter describes the standard features of the handwheel.

Related topics

- Incremental jog positioning
Further information: "Incremental jog positioning of axes", Page 163
- Handwheel superimpositioning with **M118** (#21 / #4-02-1)
Further information: Programming and Testing User's Manual
Further information: Programming and Testing User's Manual
- Touch probe functions in the **Manual** operating mode (#17 / #1-05-1)
Further information: "Touch probe functions in the Manual operating mode (#17 / #1-05-1)", Page 305

Overview

The control supports the following handwheels:

Handwheel	Meaning	Further information
HR 130, HR 180	Handwheel mounted on the operating panel	
HR 510, HR 510 FS	Handwheel without display	Page 432
HR 520, HR 520 FS HR 550 FS	Handwheel with display with wireless transmission	Page 435

Notes

DANGER

Caution: hazard to the user!

Unsecured connections, defective cables, and improper use are always sources of electrical dangers. The hazard starts when the machine is powered up!

- ▶ Devices should be connected or removed only by authorized service technicians
- ▶ Only switch on the machine via a connected handwheel or a secured connection

- The machine manufacturer defines which axes you can move with the handwheel. Your machine manufacturer can also place the virtual axis **VT** on an axis key.
- If the handwheel is active, the control shows an icon for the selected axis in the **Positions** workspace. The icon indicates whether you can move the axis with the handwheel.

Further information: "The Positions workspace", Page 121

19.2 Handwheel without display

Application

This chapter contains supplementary information about the HR 510 and HR 510 FS handwheels without display.

Related topics

- Overview of the available handwheels
Further information: "Overview", Page 430
- Handwheels with display
Further information: "Handwheel with display", Page 435

Description of function

Operating elements



A handwheel without display features the following operating elements:

- 1 Axis keys
- 2 The **actual position capture** key
Further information: "Creating an NC block with the current position", Page 434
- 3 Keys for the speed level
Further information: "Speed levels", Page 433
- 4 Traverse direction keys
- 5 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** or **NC stop**
- 6 Handwheel permissive buttons
Further information: "Activating and deactivating a handwheel", Page 433
- 7 Dial
- 8 **Emergency stop** key

Speed levels

The handwheel offers three keys with pre-defined speed levels that you can choose from.

The speed level influences the following values:

- Distance that the control moves an axis by when you rotate the wheel by one detent stop
- Feed rate at which the control moves the axis when you press an axis-direction key

 You use a key to specify both the feed rate as well as the speed level for the distance moved. However, the control uses different, independent values, depending on whether you rotate the wheel or press a key.

Key	Meaning
	Low speed level Example: Distance of 0.001° or 0.001 mm/inch
	Medium speed level Example: Distance of 0.01° or 0.01 mm/inch
	High speed level Example: Distance of 0.1° or 0.1 mm/inch

 Refer to your machine manual.
The machine manufacturer defines the values of the speed levels for each axis.

19.2.1 Activating and deactivating a handwheel

To activate a handwheel without display:

- 
- ▶ Press the **Handwheel** key on the control
 - The NC control activates the handwheel and changes the symbol in the **Manual** operating mode.
 - The control displays a handwheel symbol next to the currently selected axis in the **Position** workspace.

To deactivate a handwheel without display:

- 
- ▶ Press the **Handwheel** key on the control

 As an alternative to the **Handwheel** key, you can also use the **Handwheel** toggle switch in the function bar of the control.

19.2.2 Creating an NC block with the current position



Refer to your machine manual.

The machine manufacturer can assign any keys on the handwheel. Your handwheel therefore might not have an **actual position capture** key.

To create an NC block using the handwheel:



- ▶ Select the **Manual** operating mode



- ▶ Select the **MDI** application
- ▶ If necessary, select the NC block after which you want to insert the NC block



- ▶ Activate the handwheel
- ▶ Press the **actual position capture** key
- The control inserts a straight line **L** with the actual positions of all defined axes.



You use the **actPosAxes** machine parameter (no. 105415) to define the axes used by the **actual position capture** key to create a straight line **L**.

19.2.3 Incremental jog positioning using handwheels without display

Incremental jog positioning allows you to move the selected axis by a defined value each time you press a key. In order to perform incremental jog positioning using a handwheel without display, you must first define the jog increment on the control.



Incremental jog positioning works only for traverse movements using the axis-direction keys.

To perform incremental jog positioning using a handwheel without display:



- ▶ Select the **Manual** operating mode



- ▶ Select the **Manual operation** application



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



- ▶ Enter the jog increment for linear axes and rotary axes
- ▶ Press the **Handwheel** key on the control
- The control activates the handwheel.



- ▶ Press an axis key



- ▶ Press a traverse direction key
- The control moves the axis by the defined jog increment.

19.3 Handwheel with display

Application

This chapter contains information specific to the HR 520, HR 520 FS and HR 550 FS handwheels with display. Handwheels with display can perform more functions than handwheels without display.

Related topics

- Overview of handwheels

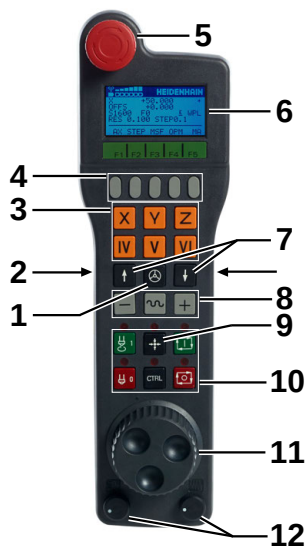
Further information: "Overview", Page 430

- Handwheels without display

Further information: "Handwheel without display", Page 432

Description of function

Operating elements



A handwheel with display provides the following operating elements:

- 1 Handwheel activation key

Further information: "Activating and deactivating a handwheel with display", Page 440

- 2 Handwheel permissive buttons on the sides

- 3 Axis keys

- 4 Soft-key selection keys

Further information: "Handwheel soft keys", Page 437

- 5 **Emergency stop** key

- 6 Display

Further information: "Contents of display", Page 436

- 7 Speed level

Further information: "Speed levels", Page 438

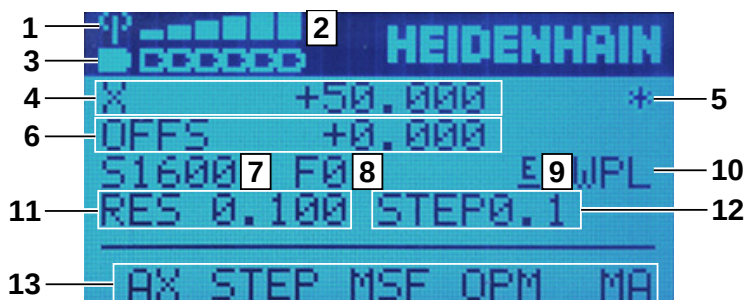
- 8 Keys for traverse direction and rapid traverse

- 9 The **actual position capture** key

Further information: "Creating an NC block with the current position", Page 434

- 10 Keys which the machine manufacturer can assign, such as Spindle ON, **NC Start** or **NC stop**
- 11 Dial
- 12 Potentiometers for spindle speed and feed rate

Contents of display



The display of a handwheel consists of the following areas:

- 1 Handwheel is in the docking station or radio mode is active
Only with HR 550 FS wireless handwheel
- 2 Field strength, up to six bars
Only with HR 550 FS wireless handwheel
- 3 Charge status of the rechargeable battery, up to six battery symbols
Only with HR 550 FS wireless handwheel
- 4 Selected axis and current position
- 5 Control-in-operation
Program run has been started or axis is in motion
- 6 Handwheel superimpositioning from **M118** (#21 / #4-02-1)
Further information: Programming and Testing User's Manual
- 7 Current speed of the active spindle
- 8 Current feed rate of the selected axis
During program run: current contouring feed rate
- 9 Pending error message
- 10 Active setting in the **3-D rotation** window:
 - **VT: Tool axis** function
 - **WP: Basic rotation** function
 - **WPL: 3D ROT** function**Further information:** "The 3-D rotation window (#8 / #1-01-1)", Page 218
- 11 Traverse per wheel detent stop
Further information: "Speed levels", Page 438
- 12 Incremental jog active or inactive, and jog increment
Further information: "Incremental jog positioning", Page 442
- 13 Handwheel soft keys
Further information: "Handwheel soft keys", Page 437

Handwheel soft keys



You can use the handwheel soft keys to select the following functions:

Soft key	Key	Meaning
AX	F1	Select machine axis
STEP	F2	Active or deactivate incremental jog positioning, and select the jog increment Further information: "Incremental jog positioning", Page 442
MSF	F3	Define cutting data, miscellaneous functions, and presets Further information: "Defining the spindle speed S", Page 441
OPM	F4	Select operating mode
MA	F5	Execute machine-specific functions (such as switching magazine pockets)
MOP	F3	Select manual options Only if program run was interrupted by an NC stop

Operating modes

Press **OPM** to choose between the following operating modes:

Soft key	Key	Meaning
MAN	F1	Manual operating mode
MDI	F2	MDI application in the Manual operating mode
RUN	F3	Program Run operating mode
SGL	F4	Single Block mode in the Program Run operating mode

Functions during program run

Press **MOP** to choose between the following functions:

Soft key	Key	Meaning
MAN	F1	Manual traverse
STOP	F4	Internal stop

Press **MAN** at this level to choose between the following functions:

Soft key	Key	Meaning
REPO	F1	Return to the contour
3D	F2	Settings of the Tilt working plane function

Speed levels

You use the up and down arrow keys to choose the speed level.

The handwheel shows the value of the selected level in the display after **RES**. If you rotate the wheel by one detent stop, the control moves the axis by the value in the unit of measure for that axis.

Key	Meaning
	Increase speed level
	Decrease speed level



On handwheels with display the speed level defines only the distance per detent stop; it does not define the feed rate for traverse movements with axis-direction keys.

You use the **MSF** soft key to define the feed rate.

Further information: "Defining the handwheel feed rate F", Page 442

Special features of the HR 550 FS wireless handwheel

With the HR 550 FS wireless handwheel you can move farther away from the machine operating panel than with cable-connected handwheels. The HR 550 FS wireless handwheel thus provides an important benefit, in particular for large machines.

The HRA 551 FS handwheel holder and the HR 550 FS handwheel together form one functional unit.



HR 550 FS handwheel



HRA 551 FS handwheel holder

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

The HR 550 FS can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you aren't using the HR 550 FS, always place it in the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.



Always dock an HR 550 FS in its HRA 551 FS when not in use. This has the following benefits:

- The safety self-test can be performed:
- The NiMH rechargeable battery of the handwheel is always fully charged.
- An unexpected emergency stop reaction due to an empty NiMH rechargeable battery is avoided.
- Confusion between different wireless handwheels is avoided.



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

You can configure wireless handwheels in the **TNCdiag** application.

Further information: "Setting up a wireless handwheel", Page 444

If the control has triggered an emergency stop, you must reactivate the handwheel.

If you happen to get close to the limit of the transmission range, the HR 550 FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

19.3.1 Activating and deactivating a handwheel with display

NOTICE

Caution: Possible damage to the workpiece!

When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.

- ▶ Retract the tool before switching
- ▶ Before switching, check whether the settings of the feed-rate potentiometers match

To activate a handwheel with display:



- ▶ Press the **Handwheel** key on the handwheel
- The NC control activates the handwheel and changes the symbol in the **Manual** operating mode.
- The control displays a handwheel symbol next to the currently selected axis in the **Position** workspace.
- ▶ If necessary, adjust the setting of the feed-rate potentiometer

To deactivate a handwheel with display:



- ▶ Press the **Handwheel** key on the handwheel



The control also activates or deactivates the feed-rate potentiometer.

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

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

19.3.2 Creating an NC block with the current position



Refer to your machine manual.

The machine manufacturer can assign any keys on the handwheel. Your handwheel therefore might not have an **actual position capture** key.

To create an NC block using the handwheel:



- ▶ Select the **Manual** operating mode



- ▶ Select the **MDI** application
- ▶ If necessary, select the NC block after which you want to insert the NC block



- ▶ Activate the handwheel
- ▶ Press the **actual position capture** key
- > The control inserts a straight line **L** with the actual positions of all defined axes.



You use the **actPosAxes** machine parameter (no. 105415) to define the axes used by the **actual position capture** key to create a straight line **L**.

19.3.3 Defining the spindle speed S

To define the spindle speed **S** of the active spindle when using a handwheel with display:

- ▶ Press the **MSF** soft key on the handwheel
- ▶ Press the **S** soft key on the handwheel
- ▶ Select the desired spindle speed by pressing the **F1** or **F2** keys
- > The handwheel shows the defined spindle speed in the display after **S**.
- ▶ Press the **NC Start** key
- > The control activates the defined spindle speed.



If you press and hold the **F1** or **F2** key, the handwheel counts the value up or down. The longer you press the key, the greater the counting step.

If you additionally press the **CTRL** key, the handwheel starts with a larger counting step.

19.3.4 Defining the handwheel feed rate F

To define the feed rate **F** when using a handwheel with display:

- ▶ Press the **MSF** soft key on the handwheel
- ▶ Press the **F** soft key on the handwheel
- ▶ Select the desired feed rate by pressing the **F1** or **F2** keys
- ▶ Press **OK**
- ▶ Confirm the new feed rate by pressing the **OK** soft key on the handwheel



- ▶ Press an axis key



- ▶ Press a traverse direction key
- The control moves the axis at the defined feed rate.



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

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

19.3.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a defined value each time you press a key.



- Incremental jog positioning works only for traverse movements using the axis-direction keys.
- The control compares the settings for incremental jog positioning defined for the handwheel and the control.

To perform incremental jog positioning using a handwheel with display:

- ▶ Press the **STEP** soft key on the handwheel
- ▶ Press the **ON** soft key on the handwheel
- The control activates incremental jog positioning.
- ▶ Select the desired jog increment by pressing the **F1** or **F2** keys
- The handwheel shows the defined jog increment in the display after **STEP**.



The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches).

- ▶ Confirm the jog increment by pressing the **OK** soft key on the handwheel



- ▶ Press an axis key



- ▶ Press a traverse direction key
- The control moves the axis by the defined jog increment.



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

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

Notes on wireless handwheels

DANGER

Caution: hazard to the user!

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- ▶ Check the radio connection of the handwheel for possible overlapping with other wireless devices
- ▶ Switch off the handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test the next time it is restarted (only for handwheels 598515-03, 606622-03 and holder 731928-02)
- ▶ If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- ▶ If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

NOTICE

Caution: Danger to the tool and workpiece!

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

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

- The control displays a warning if you connect a wireless handwheel with an already selected radio channel.

19.4 Setting up a wireless handwheel

Application

You can configure the HR 550 FS wireless handwheel in the **Setup for wireless handwheel** application.

Related topics

- Electronic handwheel
Further information: "Electronic handwheel", Page 429
- HR 550 FS wireless handwheel
Further information: "Special features of the HR 550 FS wireless handwheel", Page 439
- TNCdiag
Further information: "TNCdiag", Page 537

Requirements

- Machine with a handwheel holder
The machine manufacturer installs the handwheel holder on the machine.

Description of function

To navigate to this function:

Home ► Settings ► Machine Settings ► Set Up Wireless Handwheel

Einrichtung des Funkhandrads Einrichtbetrieb beenden

SN: 0058241184

Status **1** +

Spektrum **2** +

Konfiguration **3** -

Seriennummer des benutzbaren Handrads 0058241184

Seriennummer des Handrads in der Basisstation 0058241184

Zustand des Handrads in der Basisstation ● Handrad paaren **4**

Benutzer Funkkanal der Funkverbindung 11 Bitte auswählen

Sendeleistung Mittel Bitte auswählen

Verbindungszustand Leerlauf Handrad starten

TNCdiag within the **Setup for wireless handwheel** application

TNCdiag shows the following areas:

1 Status

Information about the transmission quality

If the reception quality of the wireless connection is poor, the control triggers an emergency stop. Safe stopping of the axes cannot be ensured when the reception quality is poor.

2 Spectrum

Frequency of the individual radio channels

The radio channel with the shortest column has the least amount of radio traffic. The radio channel recommended for the wireless handwheel is marked with a green circle.

3 Configuration

■ Pair handwheel

Assign the wireless handwheel to the handwheel holder

■ Channel used for radio connection

In the selection menu, choose **Best channel** to select the recommend radio channel for the wireless handwheel

■ Transmitter power

Select Transmitter power in the selection menu. The lower the transmission power, the smaller the range of the wireless handwheel.

■ Connection status

Once **TNCdiag** continuously shows **Active** as the connection status, configuration is complete.

4 Information

Each area shows the **Information** icon. When you select the icon, **TNCdiag** displays the description of the settings.

19.4.1 Setting up a new wireless handwheel

To set up a new wireless handwheel:

- Place the handwheel in the handwheel holder



- Select the **Home** operating mode



- Select the **Settings** application



- Select **Machine Settings**



- Double-tap or double-click **Set Up Wireless Handwheel**
- The control opens the **Setup for wireless handwheel** application in **TNCdiag**.
- Select **Pair handwheel**
- **TNCdiag** briefly shows **Active** under **Connection status**.
- In the **Channel used for radio connection** selection menu, choose **Best channel** for the channel
- Select the transmitter power in the **Transmitter power** selection menu (e.g., **Medium**)
- Select **Start handwheel**
- **TNCdiag** activates the handwheel.
- **TNCdiag** dims the **Channel used for radio connection** and **Transmitter power** selection menus.



If an error occurs during configuration, **TNCdiag** colors the **Configuration** area red.

In order to read the error details, switch to an operating mode, such as the **Home** operating mode.

20

Override controller

Application

The override controller is an operating element with additional functions compared to a usual override potentiometer.

In conjunction with the override controller, the control gives you the following possibilities:

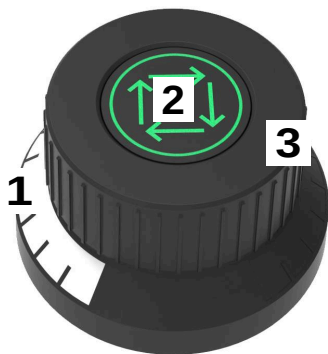
- Use the dial to manipulate the feed rate and/or rapid traverse
- Start NC programs with the integrated **NC Start** button
- Receive tactile responses through vibrations
- Use breakpoints to define conditional stops
- Resume the NC program by increasing the override

Requirements

- Override controller OC 310
The availability of the override controller depends on the machine.
Refer to your machine manual.
- Control is fully booted
The control only detects the override controller once the machine control voltage has been acknowledged.
- Tool inspection has been performed
Further information: Programming and Testing User's Manual

Description of function

Elements of the override controller



The override controller consists of the following elements:

- 1 Override scale
The override scale is illuminated in color up to the current override value.
Further information: "Visual feedback from the override controller",
Page 449

2 The **NC Start** button

The **NC Start** button starts the NC program.

Depending on the setting in the **Program run options** window, the NC program can be continued with the **NC Start** button.

3 Dial

Use the dial to change the override for the feed rate and/or rapid traverse.

Depending on the setting in the **Program run options** window, the NC program can be continued with the Override.

Visual feedback from the override controller

The override controller uses the following visual feedback:

Status	Override scale
Override Controller not active (e.g., because of an emergency stop)	Not illuminated
Override value of 0%	Not illuminated
Override value between 0% and 99.5%	White
Override value of 100%	Green
Override value greater than 100.5%	Blue

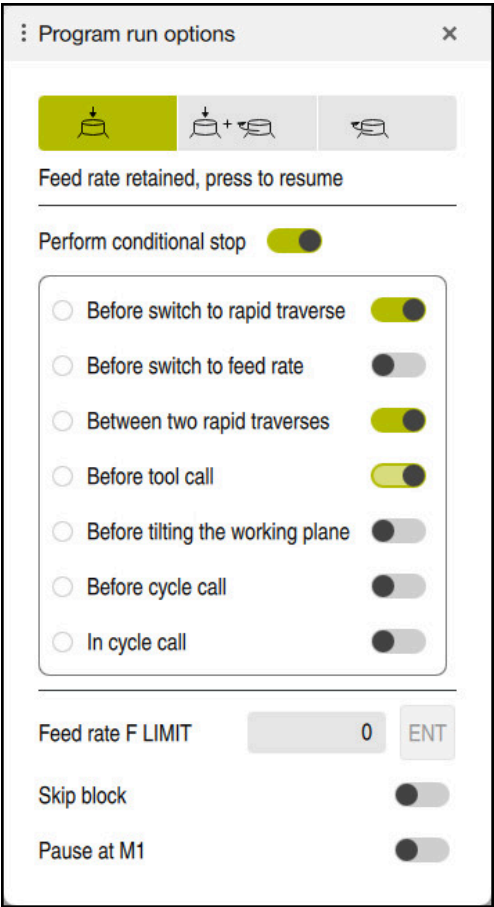
The **NC Start** button lights up green. The actual color may differ, depending on the machine.

Tactile feedback from the override controller

The override controller uses the following tactile feedback:

Status	Acknowledgment
Minimum or maximum override value	The override controller vibrates as soon as the minimum or maximum override value is reached.
Override value of 100%	The override controller vibrates as soon as the override value is at 100%.
Stop at the breakpoint	The override controller vibrates as soon as the control stops at a breakpoint.

The Program run options window



The **Program run options** window

You can open the **Program run options** window as follows:

- In the **Program Run** operating mode with the **Program run options** button
Further information: "Icons and buttons", Page 342
- In the **Simulation** workspace with the **Program run options** toggle switch in the **Visualization options** column

Further information: Programming and Testing User's Manual

The following settings of the **Program run options** window are relevant for the override controller:

Icon or button	Meaning
	Feed rate retained, press to resume When this button is active, the control does not change the override value when stopping at a breakpoint. Continue the NC program by pushing the NC Start button.
	Feed rate set to 0%, press and turn to resume When this button is active, the control changes the override value to 0% when stopping at a breakpoint. Continue the NC program by pushing the NC Start button and increasing the override value.

Icon or button	Meaning
	Feed rate set to 0%, turn to resume When this button is active, the control changes the override value to 0% when stopping at a breakpoint. Continue the NC program by increasing the override value.  Refer to your machine manual. The machine manufacturer uses the optional machine parameter resumeByTurning (no. 141801) to define if this button is available.
Perform conditional stop	Toggle switch for activating and deactivating breakpoints Further information: "Breakpoints", Page 451

- i** The following functions are available also without the override controller:
- **Feed rate F LIMIT**
Further information: "Feed rate limit F LIMIT", Page 345
 - **Skip block**
Further information: Programming and Testing User's Manual
 - **Pause at M1**
Further information: Programming and Testing User's Manual

Breakpoints

The control offers the following breakpoints:

Breakpoint	Meaning
Before switch to rapid traverse	The control stops at each change from the feed rate F to rapid traverse FMAX .
Before switch to feed rate	The control stops at each change from rapid traverse FMAX to the feed rate F .
Between two rapid traverses	The control stops between two directly sequential FMAX rapid traverse movements.
Before tool call	The control stops before every physical tool call with TOOL CALL. i The control does not stop, for example, before a TOOL CALL that simply changes the spindle speed.
Before tilting the working plane	The control stops before NC blocks with the following syntax elements: ■ PLANE functions (#8 / #1-01-1) ■ M128 (#9 / #4-01-1) ■ FUNCTION TCPM (#9 / #4-01-1) ■ Cycle 19 WORKING PLANE (#8 / #1-01-1) i You can still execute NC programs from earlier controls that contain Cycle 19 WORKING PLANE.

Breakpoint	Meaning
Before cycle call	The control stops before NC blocks with the following syntax elements: ■ M89 The control stops before each machining position. ■ M99 ■ CYCL CALL ■ CYCL CALL POS ■ CYCL CALL PAT The control stops before each machining position. ■ Cycles 220 POLAR PATTERN, 221 CARTESIAN PATTERN, 224 DATAMATRIX CODE PATTERN The control stops before each machining position.
In cycle call	Stop before the first infeed In the cycles below, the control stops before the first infeed: ■ Cycles for drilling and thread machining ■ Cycles for cylinder surface machining (#8 / #1-01-1) Further information: User's Manual for Machining Cycles Stop before every infeed In the cycles below, the control stops before every infeed: ■ Milling cycles Further information: User's Manual for Machining Cycles No stop The control will not stop in the following cycles: ■ Programmable touch probe cycles (#17 / #1-05-1) Further information: Measuring Cycles for Workpieces and Tools User's Manual ■ Cycles for monitoring

The control displays active breakpoints on the **PGM** tab of the **Status** workspace.

Further information: "The PGM tab", Page 137

Displaying breakpoints

The control displays breakpoints with the following icons:

Icon	Meaning
	Active stop The control has detected a breakpoint and stops program run or the simulation at this point.
	Inactive stop The control has detected a breakpoint but does not stop program run or the simulation at this point. In order to stop before this NC block, you must first activate the corresponding toggle switch in the Program run options window. Further information: "The Program run options window", Page 450

The control displays the icons for breakpoints in the NC program before the block number as soon as at least one conditional stop is active in the **Program run options** window.

When you select an icon, the control displays the name of the associated breakpoint.

Notes

NOTICE

Caution: Possible material damage!

Incorrect operation may cause mechanical damage to the override controller, for example when using it as a handle for moving the operating panel.

- ▶ Use the override controller only for the following purposes:
 - Use the dial to manipulate the feed rate and/or rapid traverse
 - Start NC programs with the integrated **NC Start** button
 - Receive tactile responses through vibrations
 - Use breakpoints to define conditional stops
 - Resume the NC program by increasing the override
- The override controller is also effective as a feed rate and/or rapid traverse override in the **Manual** operating mode.
- If the NC program contains breakpoints, the control displays a check mark in the **Perform conditional stop** area of the **Tests** column.
Further information: Programming and Testing User's Manual
- If you turn the override controller down with a sudden jerk, the control will automatically set the feed-rate override to 0%, even if the controller itself did not reach 0%.
 The control will resume the NC program if you turn the override controller up again. You don't need to press the **NC Start** key for this. This behavior is independent of the settings in the **Program run options** window.
 After turning the controller down with a sudden jerk, must turn the feed rate back up to 100% from 0%.
- When the execution cursor reaches a breakpoint, the two icons overlap so you can see why the control stops.
- If the **Feed rate set to 0%, turn to resume** button is active, the control reacts as follows:
 - You can continue the NC program only following a conditional stop and by increasing the override value. Otherwise an **NC Start** is necessary (e.g., when starting a program).
 - When the NC program includes two subsequent conditional stops, the 0% override value cannot be changed for 0.3 seconds. This way, the control ensures that you will not continue beyond both conditional stops by just one movement of the override controller.
 - After a conditional stop with a manual tool change you must press the **NC Start** button. You can't continue the NC program by increasing the override value.

Notes about machine parameters

Refer to your machine manual.

- The machine manufacturer defines the maximum override value for rapid traverse. If the maximum override value is, for example, 100% and you enter a rapid-traverse override value greater than 100%, the control still calculates with 100%. If you turn the dial down in this case, then there is no immediate effect. Only once the override controller actually reaches 100% will the control change the override value.
- The machine manufacturer can use the optional machine parameter **ocWaitTime** (no. 103412) to define whether a waiting time will be effective in the cases below:
 - When the program is continued at 0 % after a breakpoint
 - When 100% of the override value is reached

21

**Embedded
Workspace
and Extended
Workspace**

21.1 Embedded Workspace (#133 / #3-01-1)

Application

You use Embedded Workspace to operate a Windows PC and display its screen contents on the control's user interface. You use Remote Desktop Manager to connect the Windows PC (#133 / #3-01-1).

Related topics

- Remote Desktop Manager (#133 / #3-01-1)
Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522
- Using Extended Workspace to operate a Windows PC through an additional connected monitor
Further information: "Extended Workspace", Page 460

Requirements

- Established RemoteFX connection to the Windows PC through Remote Desktop Manager (#133 / #3-01-1)
- Connection defined in the machine parameter **CfgRemoteDesktop** (no. 133500)
In the optional machine parameter **connections** (no. 133501), the machine manufacturer enters the name of the RemoteFX connection.
Refer to your machine manual.

21.2 Extended Workspace

Application

With Extended Workspace you can use an additional attached monitor as a second screen of the control. That way you can use the additional monitor independently of the control's user interface and also to show the control's applications.

Related topics

- Using Embedded Workspace to operate a Windows PC within the control's user interface (#133 / #3-01-1)

Further information: "Embedded Workspace (#133 / #3-01-1)", Page 458

- ITC hardware expansion

Further information: "Hardware enhancements", Page 81

Requirement

- Additional attached monitor configured by the machine manufacturer as Extended Workspace
Refer to your machine manual.

Description of function

Here are some functions you can perform with Extended Workspace:

- Opening files from the control (e.g., drawings)
- Opening windows from HEROS functions in addition to the control's user interface

Further information: "HEROS menu", Page 579

- Displaying and operating computers connected through Remote Desktop Manager (#133 / #3-01-1)

Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522

22

**Integrated
functional safety
(FS)**

Application

The safety design of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety design automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety design.

Requirements

- On controls with **SIK**:
 - Software option 160 (Integrated FS: Basic) or software option 161 (Integrated FS: Full)
 - Software options 162 to 166 (Add. FS Ctrl. Loop) or software option 169 (Add. FS Full) as needed

Whether you need these software options depends on the machine's number of motors.
- On controls with **SIK2**:
 - Integrated FS: Basic software option (#6-30-1)
 - Integrated FS: Full software option as needed (#6-30-2*)

If your control is equipped with **SIK2**, software option #6-30-1 will enable four safe axes. You can order software option #6-30-2* multiple times and thus enable up to six additional safe axes.
- The machine manufacturer must adapt the FS safety design to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g., guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "The Axis status menu item", Page 465

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of motors using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the motors
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the motors
SLS	Safely Limited Speed	Safely limited speed. Prevents the motors from exceeding the specified speed limits when the guard door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:



Refer to your machine manual.

The machine manufacturer must adapt the safety-related operating modes to each machine.

Icon	Safety-related operating mode	Short description
SOM ₁	Operating mode SOM_1	Safe operating mode 1: Automatic mode, production mode
SOM ₂	Operating mode SOM_2	Safe operating mode 2: Setup mode
SOM ₃	Operating mode SOM_3	Safe operating mode 3: Manual intervention; only for qualified users
SOM ₄	Operating mode SOM_4 This function must be enabled and adapted by the machine manufacturer.	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users

Functional safety (FS) in the Positions workspace

On a control with functional safety (FS), the monitored operating states of the speed **S** and feed rate **F** are displayed in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

Further information: "Axis display and position display", Page 122

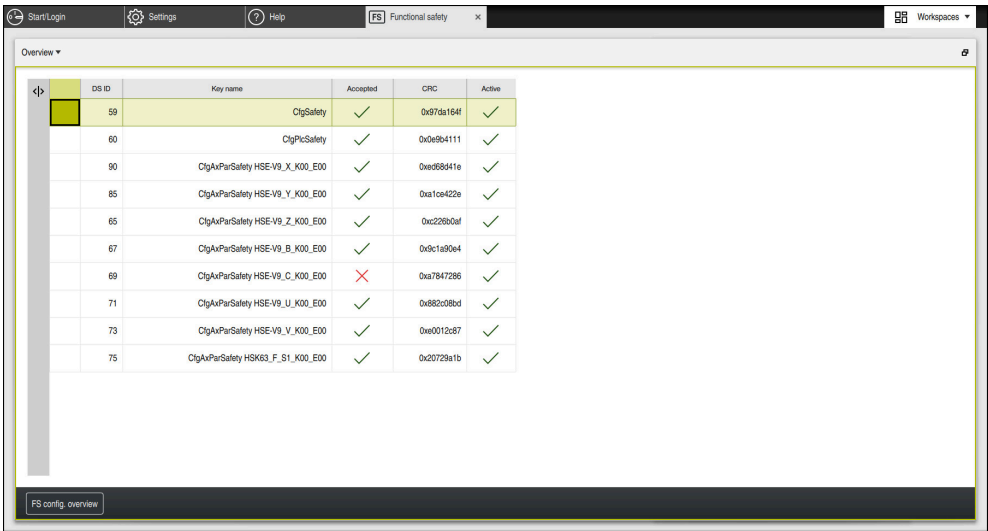
The Functional safety application



Refer to your machine manual.

The machine manufacturer configures the safety functions in this application.

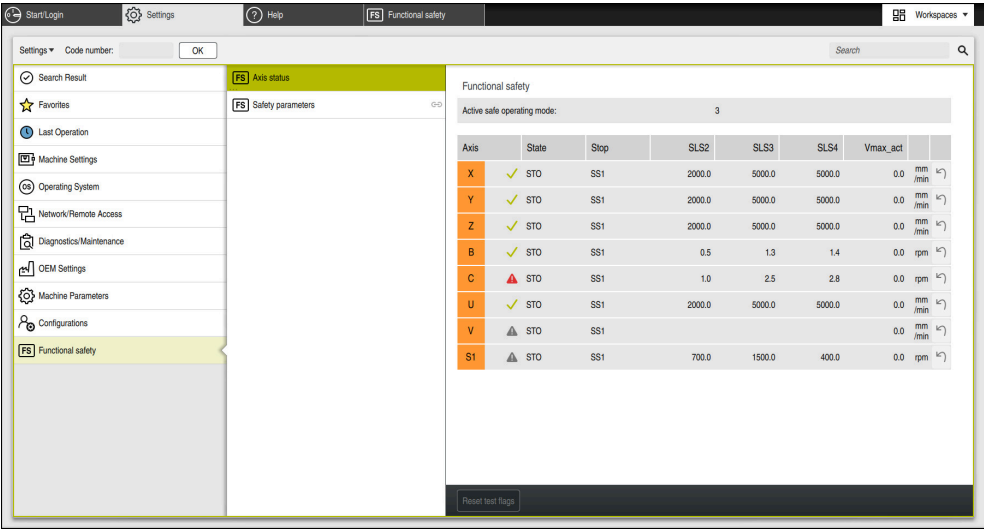
In the **Functional safety** application in the **Home** operating mode, the control provides information about the status of the individual safety functions. In this application you can see whether individual safety functions are active and have been accepted by the control.



DS ID	Key name	Accepted	CRC	Active
59	CltgSafety	✓	0x97da164f	✓
60	CltgPrcSafety	✓	0x0e9b4111	✓
90	CltgAxParSafety HSE-V9_X_K00_E00	✓	0xc66641e	✓
85	CltgAxParSafety HSE-V9_Y_K00_E00	✓	0xa1ae422e	✓
65	CltgAxParSafety HSE-V9_Z_K00_E00	✓	0xc226b0af	✓
67	CltgAxParSafety HSE-V9_B_K00_E00	✓	0x9c1a90e4	✓
69	CltgAxParSafety HSE-V9_C_K00_E00	✗	0xa7847286	✓
71	CltgAxParSafety HSE-V9_U_K00_E00	✓	0x882c08bd	✓
73	CltgAxParSafety HSE-V9_V_K00_E00	✓	0xe0012cd7	✓
75	CltgAxParSafety HSK63_F_S1_K00_E00	✓	0x20729a1b	✓

The **Overview** workspace in the **Functional safety** application

The Axis status menu item



The **Axis status** menu item in the **Settings** application

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction Further information: "Functional safety (FS) in the Positions workspace", Page 464
SLS2	Maximum speed or feed-rate values for SLS in the SOM_2 operating mode
SLS3	Maximum speed or feed-rate values for SLS in the SOM_3 operating mode
SLS4	Maximum speed or feed-rate values for SLS in the SOM_4 operating mode This function must be enabled and adapted by the machine manufacturer.
Vmax_act	Currently valid speed or feed-rate limit These are either values from the SLS settings or from the SPLC If values are greater than 999 999, the control displays MAX .

The control shows the following icons and buttons:

Icon or button	Meaning
	Reset the test status of the selected axis
Reset test flags	Reset the test statuses of all axes



- Resetting of the test status is a function for the Service department. Use this function only if instructed by HEIDENHAIN or the machine manufacturer.
- To reset the test status of axes, you need the NC.ApproveFsAxis right. This right is only available if user administration is active.

Further information: "User administration", Page 549

Further information: "User administration roles and rights", Page 603

Test status of the axes

In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.

If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 467

The control indicates the test status of the individual axes with the following icons:

Icon	Meaning
	The axis has been tested or does not need to be tested.
	The axis has not been tested, but must be tested to ensure safe operation. Further information: "Checking axis positions manually", Page 467
	The axis is not monitored by functional safety (FS) or is not configured as a safe axis. The axis is monitored by functional safety (FS), but the SLP safety function is deactivated. In machine parameter safeAbsPosition (no. 403130), the machine manufacturer defines whether the SLP safety function is activated for an axis.

22.1 Checking axis positions manually



Refer to your machine manual.

This function must be adapted by your machine manufacturer.

The machine manufacturer defines the test position.

To check the position of an axis:



- ▶ Select the **Manual** operating mode



- ▶ Select the **Move to ref. point** application



- ▶ Select **Axis position** in the **Referencing** workspace
- ▶ Select the desired axis
- ▶ Press the **NC start** key
- > The axis moves to the test position.
- > After the test position has been reached, the control issues a message.
- ▶ Press the **permissive button** on the machine operating panel
- > The control displays the axis as a tested axis.

NOTICE

Danger of collision!

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

- ▶ If necessary, move to a safe position before approaching the test positions
- ▶ Watch out for possible collisions



In the **Referencing** workspace, you can switch as desired between the **Referencing** and **Axis position** modes.

Notes

- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- The control performs repeated self-tests to detect defective cables, for example. The machine manufacturer defines at what intervals the control will perform the self-tests. When a self-test of the control is active, the control displays an icon in the information bar. The control cannot perform any axis movements while a self-test is active.

23

**The Settings
application**

23.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Icon	Menu item
	Machine Settings		Machine Settings Further information: "The Machine Settings menu item", Page 475
			General Information Further information: "The General Information menu item", Page 478
			SIK Further information: "The SIK menu item", Page 480
			Machine Times Further information: "The Machine Times menu item", Page 483
			Overview of touch probes Further information: "Overview of touch probes menu item", Page 484
			Adjustment of analog voltage offset Further information: "The Adjustment of analog voltage offset menu item", Page 487
			Set Up Wireless Handwheel Further information: "Setting up a wireless handwheel ", Page 444
	Operating System		Date/Time Further information: "The Adjust system time window", Page 488
			Language/Keyboards Further information: "Conversational language of the control", Page 489
			About HeROS Further information: "Information on licensing and use", Page 75
			SELinux Further information: "SELinux security software", Page 491
			UserAdmin Further information: "The User administration window", Page 558
			Current User Further information: "The Active user window", Page 559
			Touchscreen Configuration You can select the touchscreen sensitivity and define whether touch points should be shown or hidden.

Icon	Category	Icon	Menu item
	Network/Remote Access		Shares Further information: "Network drives on the control", Page 492
			Network Further information: "Ethernet interface", Page 496
			PKI Admin Manage certificates for the control (e.g., for OPC UA NC Server) Further information: "PKI Admin", Page 503
			OPC UA Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505
			DNC Further information: "The DNC menu item", Page 513
			Embedded Workspace Show the connection status Further information: "Embedded Workspace (#133 / #3-01-1)", Page 458
			Printer Further information: "Printers", Page 516
		vnc	VNC Further information: "The VNC menu item", Page 519
			Remote Desktop Manager Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522
		vnc 	Real VNC Viewer Connect to a remote device via a VNC server Available only to network specialists
			Firewall Further information: "Firewall", Page 528

Icon	Category	Icon	Menu item
	Diagnostics/Maintenance		Terminal program Enter and execute console commands
			HeLogging Define settings for internal diagnostic files
			Portscan Further information: "Portscan", Page 533
			perf2 Check processor load and process load
			TNCdiag Further information: "TNCdiag", Page 537
			TNCscope Available only to authorized specialists
			NC/PLC Backup Further information: "Backup and restore", Page 534
			NC/PLC Restore Further information: "Backup and restore", Page 534
			Touchscreen Cleaning The control disables the touchscreen for input for 90 seconds.
			Update the documentation Further information: "Update the documentation", Page 538
			Expanded logging If this function is active, the control will save graphics journal data. These data may be needed by the HEIDENHAIN Customer Service for diagnostic purposes. If you activate this function, you need to restart the control. Deactivate this function after the fault has been fixed to reduce the amount of memory used.
	OEM Settings		Settings for the machine manufacturer
	Machine Parameters		The group contains machine parameters that can be edited, depending on your rights (e.g., MPs for setters). Further information: "Machine parameters", Page 540
	Configurations		Configurations Further information: "Configuring the control's user interface", Page 547

Icon	Category	Icon	Menu item
FS	Functional safety	FS	Axis status Further information: "The Axis status menu item", Page 465
		FS	Safety parameters Further information: "The Functional safety application", Page 464

23.2 Code numbers

Application

The top part of the **Settings** application contains the **Code number:** input field. This input field is accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Meaning
123	Editing machine-specific user parameters Further information: "Machine parameters", Page 540
	When this code number is active, the control displays a different icon for the Home operating mode.
555343	Special functions for programming with variables Further information: Programming and Testing User's Manual Special functions defining the machine behavior Further information: Programming and Testing User's Manual
0	Resetting active code numbers

 The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

23.3 The Machine Settings menu item

Application

In the **Machine Settings** menu item of the **Settings** application, you can define the settings for simulation and program run.

Related topics

- Graphic settings for simulation

Further information: Programming and Testing User's Manual

Description of function

To navigate to this function:

Settings ► Machine Settings ► Machine Settings

The Unit of Measure area

In the **Unit of Measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10 before entering it.

The Machine and Simulation areas

Channel Settings

You can define the following settings:

Setting	Meaning
Active Kinematics	Use the Active Kinematics function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, have been programmed for other machines. The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose. The control displays the active kinematics model in the Machine mode of the Simulation workspace.
Generate tool-usage file	The control uses the tool-usage file to check tool usage. Further information: "Tool usage test", Page 193 You select when the control should generate a tool-usage file: ■ Never The control does not generate a tool-usage file. ■ Once The next time you simulate or run an NC program, the control will generate a tool-usage file once. ■ Always When you simulate or run an NC program, the control will generate a tool-usage file each time.

Oper. station for program interruption

The control displays this function only in the **Machine** area.



Refer to your machine manual.
The machine manufacturer uses the optional machine parameter **userControl** (no. 144101) to define if this area is available.

You can define the following settings for the **Manual traverse** during program run:

Setting	Meaning
Standard operating station	You select the operating element to be used for moving the axes: ■ MB machine operating panel ■ HR handwheel
Remember last operating station	If the toggle switch is active, the control will not consider the selected Standard operating station . The control remembers the state of the Handwheel toggle switch from the last manual traverse, and activates the associated operating element.

Further information: "Manual traverse during an interruption", Page 350

Traverse Limits

The control displays this function only in the **Machine** area.

Use the **Traverse Limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse Limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents Valid or Invalid .
Lower Limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper Limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

- The lower limit must be smaller than the upper limit.
- The upper and lower limit may not both equal 0.

Other conditions apply to traverse limits for modulo axes.

Further information: Programming and Testing User's Manual

Notes

NOTICE

Danger of collision!

You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision!

- ▶ Use the **Active Kinematics** function for the simulation only
 - ▶ Use the **Active Kinematics** function for selecting the active machine kinematics only if required
-
- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active Kinematics** function can be selected.
 - Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
 - You can open the tool-usage file in the **Tables** operating mode.
Further information: "Tool usage file", Page 409
 - If the control generated a tool-usage file for an NC program, the **T usage order** and **Tooling list** tables contain data (#93 / #2-03-1).
Further information: "T usage order (#93 / #2-03-1)", Page 411
Further information: "Tooling list (#93 / #2-03-1)", Page 413

23.4 The General Information menu item

Application

In the **General Information** menu item of the **Settings** application, the control provides information about the control and the machine. If, for example, the HEIDENHAIN service department requires the NC software number, you will find that number in this area.

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ General Information

The Version Information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	■ Control Model Designation of the control
	■ NC-SW Number of the NC software
	■ NCK Version of the software
	Is managed by HEIDENHAIN
PLC	PLC-SW Number or name of the PLC software
	Is managed by the machine manufacturer
Operating System	■ HEROS version
	■ Yocto version Is managed by HEIDENHAIN

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

If the machine manufacturer has defined a version number for at least one OEM cycle, the control displays this information in the **OEM cycles** sub-area.

The Info about machine manufacturer area

The control shows the contents of the optional machine parameter **CfgOemInfo** (no. 131700). The control displays this area only if the machine manufacturer defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 506

The Machine information area

Sub-area	Meaning
Hardware	RAM RAM of the main computer

The control also shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 506

23.5 The SIK menu item

Application

Use the **SIK** menu item of the **Settings** application to view control-specific information (e.g., the serial number and the available software options).

Related topics

- Software options on the control
 Further information: "Software options", Page 69

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ SIK

The **SIK** menu item consists of the following areas:

Area	Contents
SIK Information	The control displays the following information: ■ Serial Number ■ ID number ■ Control Model ■ Performance Class ■ Features ■ Status The control shows whether it is equipped with a SIK or SIK2. ■ Temporarily enable options / Disable options
Machine manufacturer key	The machine manufacturer can define a manufacturer-specific password for the control.
General key Only with SIK	The machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing). The control indicates the status of the general key: ■ NONE The general key has not yet been used for this software version. ■ dd.mm.yyyy Date up to which all software options will be available. Once the general key has expired, it cannot be used again. ■ EXPIRED The general key has expired for this software version.
Enabling of options Only with SIK2	You can select a key file in order to enable software options. You can obtain the key file from your machine manufacturer. Further information: "Enabling software options", Page 482
Software Options	The control displays all available options in a table. Further information: "The Software Options area", Page 481

The Software Options area

The table with the available software options contains the following columns:

Column	Meaning
#	Number of the software option
Option	Name of the software option On controls with SIK2, the part number and the name of the software option are displayed. The control indicates the status of the software option by means of the following symbols: ■ No symbol: The software option is not enabled. ■ Checkmark: The software option is enabled permanently with all functions. ■ Clock symbol: The software option has been enabled for a limited period of time or can be ordered again on controls with SIK2. ■ Padlock: The software option has been locked by the machine manufacturer.
Expiration Date or Status	The control displays the following information on the status of the software option: ■ Enabled ■ YYYY-MM-DD If a software option has been enabled for a limited period of time, the control shows the date up to which it will be available. ■ X of X On controls with SIK2, the control shows how often the software option has been enabled.
Details	Detailed information for the machine manufacturer
Config.	Function that the machine manufacturer can use to lock software options

23.5.1 Viewing of software options

To view enabled software options on the control:



- ▶ Select the **Home** operating mode



- ▶ Select the **Settings** application
- ▶ Select **Machine Settings**
- ▶ Select **SIK**
- ▶ Navigate to the **Software Options** area
- For enabled software options, the control displays the text **Enabled**.

23.5.2 Enabling software options

Control equipped with SIK

To enable a software option for a control with **SIK**, you need an enabling code (from the machine manufacturer, for example).

To enable a software option with **SIK**:

- ▶ Switch on the power supply of the control and the machine
- > The control starts the operating system and displays the **Power interrupted** message.
- ▶ Navigate to the **SIK** menu item
- ▶ Scroll to the **Software Options** area



- ▶ Select **Set** in the **Option** column
- > The control opens the **Unlock the option** window.
- ▶ Select the desired software option in the selection menu as needed
- ▶ Enter the activation code



- ▶ Select **Set**
- > The control restarts and activates the software option.

Control equipped with SIK2

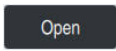
To enable a software option for a control with **SIK2**, you need a key file (from the machine manufacturer, for example).

To enable a software option with **SIK2**:

- ▶ Switch on the power supply of the control and the machine
- > The control starts the operating system and displays the **Power interrupted** message.
- ▶ Navigate to the **SIK** menu item



- ▶ In the **Enabling of options** area, select the **Select** button
- > The control opens the **Select key file for options (SIK2_xxx.txt)** window.
- ▶ Navigate to the folder containing the key file



- ▶ Select **Open**
- > The control reads the key file and opens the **Restart** window.



- ▶ Select **Restart**
- > The control restarts and activates the software option.



If a key file is present in the **TNC:\SIK2** folder, the control will read that file automatically during the next startup.

Definition

Abbreviation	Definition
SIK (System Identification Key)	SIK is the designation of the plug-in board for the control hardware. Each control can clearly be identified by the serial number of the SIK. The software options are saved on the SIK. The TNC7 basic can be equipped with a SIK or SIK2 plug-in board. Depending on which one is used, the numbers of the software options differ.

23.6 The Machine Times menu item

Application

In the **Machine Times** menu item of the **Settings** application, the control shows the run times since being put into service.

Related topics

- Date and time of the control

Further information: "The Adjust system time window", Page 488

Description of function

To navigate to this function:

Settings ► Machine Settings ► Machine Times

The control displays the following machine times:

Machine time	Meaning
Control On	Run time of the control since being put into service
Machine On	Run time of the machine tool since being put into service
Program Run	Run time of all program runs since being put into service



Refer to your machine manual.

The machine manufacturer can define up to 20 additional run times.

23.7 Overview of touch probes menu item

Application

The **Overview of touch probes** menu item of the **Settings** application allows you to create and manage all workpiece touch probes and tool touch probes of the control.

Related topics

- Touch probe table
Further information: "Touch probe table tchprobe.tp (#17 / #1-05-1)", Page 401
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)
Further information: "Machine parameters", Page 540

Description of function

To navigate to this function:

Settings ▶ Machine Settings ▶ Overview of touch probes

The control displays one table for the **SE** transceivers, one table for the **TS** workpiece touch probes, and one table for the **TT** tool touch probes.

The tables contain the following information:

- Model
- Number
Only for **TS** and **TT**
- Serial number
- **Add** button
Only for **TS** and **TT**

 The control shows the table for transceivers only if you use touch probes with radio transmission.

Transceiver unit

If you use touch probes with radio transmission, the control displays the following information for **Transceiver unit**:

Display	Meaning
Status	The transceiver unit is active or inactive
SE	Select the SE transceiver
Dimension	Select or change the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or handwheels with radio transmission.

Details

If you use touch probes with radio transmission, the control displays the following information for **Details**:

Display	Meaning
Signal strength	The signal strength in the bar chart The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision detected
Battery status	If the battery charge falls below the plotted limit, the control displays a warning.

For touch probes with infrared transmission, the control displays the following information for **Details**:

There are no further diagnostic data and functions available for this touch probe.

Buttons

The control displays the following buttons:

Button	Meaning
+	Add The control adds a new row to the respective table. You define a workpiece touch probe in the Touch probe table and a tool touch probe in the machine parameter CfgTT (no. 122700).
Connect	Connect a radio touch probe to a transceiver
Switch-on	Switch touch probe on
Switch off	Switch touch probe off
Change	Change the radio channel used for communication by the touch probe and the transceiver
TNCdiag	The control opens TNCdiag. Further information: "TNCdiag", Page 537
Touch probe table	The control opens the Touch probe table.
Delete	The control deletes the highlighted table row.

23.7.1 Connecting a new touch probe with radio transmission

To connect a touch probe with radio transmission:



- ▶ Select the **Settings** application
- ▶ Select **Machine Settings**
- ▶ Select **Overview of touch probes**
- ▶ Select the desired transceiver unit (e.g., SE 661)
- ▶ Select **Connect new touch probe**
- The control opens the **Connecting... Please insert the batteries in the touch probe** window.
- ▶ Insert the battery into the touch probe
- The control closes the window once the touch probe has been connected.
- ▶ Select the newly connected touch probe
- ▶ Select **Switch-on**
- The status of the touch probe changes to **On**.

23.7.2 Changing the radio channel

Before changing the radio channel, ensure that the touch probe is switched off and that the desired transceiver unit is not connected to any active touch probe.

To change the radio channel:



- ▶ Select the **Settings** application
- ▶ Select **Machine Settings**
- ▶ Select **Overview of touch probes**
- ▶ Select the desired transceiver unit (e.g., SE 661)
- ▶ Select **Change**
- The control opens the **Change** window.
- ▶ Select a new channel number
- The control closes the window and displays the new channel number.

23.8 The Adjustment of analog voltage offset menu item

Application

The **Adjustment of analog voltage offset** menu item of the **Settings** application allows you to correct the voltage offsets of all analog axes that are present.



Only use this function after consultation with your machine manufacturer.

Requirement

- Machine with analog axes
- If user administration is active, the NC.Setter role

Further information: "List of roles", Page 603

Description of function

To navigate to this function:

Settings ► Machine Settings ► Adjustment of analog voltage offset

If there are any analog axes, the control displays a table with the following columns:

Column	Meaning
Axis	All analog axes
In closed control loop	Axis is in a closed control loop or is not in a closed control loop. If the axis is in a closed control loop, the control shows a green checkmark.
Current offset	Currently stored voltage offset
New offset	Currently ascertained voltage offset If permitted by the axis configuration, you can adjust the voltage offset here.
Deviation	Deviation of the ascertained voltage offset from the stored offset



If you adjust the voltage offsets in the **New offset** column, the control synchronizes the optional machine parameter **analogOffset** (no. 402810) with the current voltage offsets.

23.9 The Adjust system time window

Application

In the **Adjust system time** window, you can set the time zone, date and time manually or by means of NTP server synchronization.

Related topics

- Run times of the machine tool
Further information: "The Machine Times menu item", Page 483

Description of function

To navigate to this function:

Settings ► Operating System ► Date/Time

The **Adjust system time** window contains the following areas:

Area	Function
Set the time manually	Activate this check box to define the following data: ■ Year ■ Month ■ Day ■ Time
Synchronize the time over NTP server	If you activate this check box, the control will automatically synchronize the system time with the defined NTP server. You can add a server with a host name or a URL.
Time zone	You can select your time zone from a list.

23.10 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control

Further information: "Machine parameters", Page 540

Description of function

To navigate to this function:

Settings ► Operating System ► Language/Keyboards

You can't define two different conversational languages for the operating system and control.

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

Area	Function
Language	Choose the HEROS conversational language from a selection menu Only if the machine parameter applyCfgLanguage (no. 101305) is defined as FALSE .
Keyboards	Select the language layout of the keyboard for HEROS functions

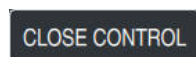
23.10.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:



- ▶ Select the **Settings** application
- ▶ Enter the code number 123
- ▶ Select **OK**
- ▶ Select **Machine Parameters**
- ▶ Double-tap or double-click **MPs for setters**
- > The control opens the **MPs for setters** application.
- ▶ Navigate to the machine parameter **ncLanguage** (no. 101301)
- ▶ Select the desired language
- ▶ Select **Save**
- > The control opens the **Configuration data changed. All changes.** window.
- ▶ Select **Save**
- > The control opens the notification menu and displays a "Question type" error.
- ▶ Select **CLOSE CONTROL**
- > The control restarts.
- > Once the control has restarted, the NC conversational language and the HEROS conversational language are changed.



Notes

- Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.
 - **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.
Further information: "Changing the language", Page 490
 - **FALSE:** The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.
- Use the optional machine parameter **noRebootDialog** (no. 101306) to define whether, after a change of the conversational language, the control displays a restart message.

23.11 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions (such as viruses and other malicious software).

The machine manufacturer defines the **SELinux** settings in the **Security Policy Configuration** window.

Related topics

- Security settings with firewall
Further information: "Firewall", Page 528

Description of function

To navigate to this function:

Settings ► Operating System ► SELinux

By default, **SELinux** access control is implemented as follows:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Safety-relevant files, such as **SELinux** system files or HEROS boot files, may only be modified using explicitly selected programs.
- New files created by other programs may not be run.
- USB data carriers can be deselected.
- Only two processes can run new files:
 - Software update: A software update from HEIDENHAIN can replace or modify system files.
 - SELinux configuration: The configuration of **SELinux** in the **Security Policy Configuration** window is usually protected by a password defined by the machine manufacturer. Please refer to the machine manual.

Note

HEIDENHAIN recommends using **SELinux** as additional protection against attacks from outside the network.

Definition

Abbreviation	Definition
MAC (mandatory access control)	MAC means that the control performs only explicitly permitted actions. SELinux is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can be performed only if the standard functions and access control of SELinux permit it.

23.12 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management
Further information: Programming and Testing User's Manual
- Network settings
Further information: "Ethernet interface", Page 496

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► Shares

You can define any number of network drives, but only seven can be connected at a time.

The Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive The control selects the check box in the Mount column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting. The control selects the check box in the Auto column if an active automatic connection exists.
Add	Define a new connection Further information: "The Mount assistant window", Page 494
Remove	Delete an existing connection
Copy	Copy connection Further information: "The Mount assistant window", Page 494
Edit	Edit the connection settings Further information: "The Mount assistant window", Page 494
Private network drive	User-specific connection if user administration is active The control selects the check box in the Privat column if a user-specific connection exists.

The Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

The Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	■ Drive name: Network drive name in the file management of the control The names must be all uppercase letters, terminated by a colon (:). ■ Volume ID: Currently no function ■ Private network drive With user administration active, the connection is only visible to the user who created it.
Share type	Transfer protocol ■ Windows share (CIFS/SMB) or Samba server ■ UNIX share (NFS)
Server and Share	■ Server name: Server name or IP address ■ Share name: Directory accessed by the control
Automount	Connect automatically (not possible with the "Ask for password?" option) The control connects the network drive automatically during the starting process.
User name and password (only with Windows share)	■ Single Sign On With user administration active, the control automatically connects an encrypted network drive when the user logs in. ■ Windows user name: ■ Ask for password? (not possible with the "Connect automatically" option) Select whether a password is required upon connecting. ■ Password ■ Password verification
Mounting options	Parameters for mount option "-o": Auxiliary parameters for the connection Further information: "Examples of Mounting options", Page 495
Check	The control displays a summary of the defined settings. You can check the settings and save them with Apply .

Examples of Mounting options

Enter options without a space, only separated by a comma

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to include the domain in the user name, but rather specify it as an option.
vers=3.1.1	Protocol version
sec=ntlmssp	Authentication method ntlm Use this option if the control displays the Permission denied error message upon connecting.

Options for NFS

Example	Meaning
rsize=8192	Packet size in bytes for data reception Input: 512...8192
wsize=4096	Packet size in bytes for data transmission Input: 512...8192
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will try to connect again
nfsvers=2	Protocol version



If you use the CIMCO NFS software, you must enter the option nfsvers = 2. CIMCO NFS supports NFS only up to version 2.

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

23.13 Ethernet interface

Application

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

Related topics

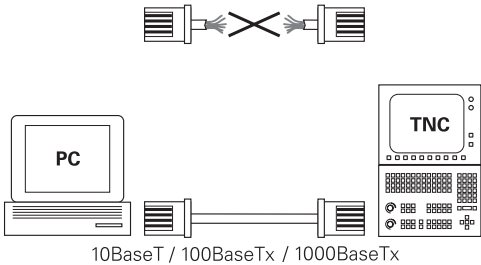
- Firewall settings
Further information: "Firewall", Page 528
- Network drives on the control
Further information: "Network drives on the control", Page 492
- External access
Further information: "The DNC menu item", Page 513

Description of function

The control transfers data via the Ethernet interface using the following protocols:

- **CIFS** (common internet file system) or **SMB** (server message block)
The control supports versions 2, 2.1 and 3 of these protocols.
- **NFS** (network file system)
The control supports versions 2 and 3 of this protocol.

Connection options



You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

i The maximum cable length permissible between the control and a node depends on the quality grade of the cable, the sheathing, and the type of network.

Ethernet connection icon

Icon	Meaning
	Ethernet connection The control displays the icon at the bottom right in the taskbar. Further information: "Taskbar", Page 582 When you click the icon, the control opens a pop-up window. The pop-up window contains the following information and functions: ■ Connected networks You can disconnect the network connection. Select the network name to reconnect. ■ Available networks ■ VPN connections Currently no function

Notes

- Protect your data and the control by running the machines in a secure network.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

23.13.1 The Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

 Have a network specialist configure the control.

Related topics

- Network configuration
Further information: "Network configuration with Advanced Network Configuration", Page 596
- Firewall settings
Further information: "Firewall", Page 528
- Network drives on the control
Further information: "Network drives on the control", Page 492

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► Network

Network settings

Status

Interfaces

DHCP server

Ping/Routing

SMB share

Computer name

Default gateway

10.3.56.254 on eth0

☐ Use proxy

Address:Port

Interfaces

Name	Connection	Connection status	Configuration Name	Address
eth0	X26	Activated	DHCP	10.3.56.18
eth1	X116	Activated	DHCP-HostOnly	10.3.56.17

DHCP client

Name	IP address	MAC address	Type	Valid up to

OK

Apply

OEM authorization

Export configuration

Import the configuration

HEIDENHAIN default

Cancel

The **Network settings** window

The Status tab

The **Status** tab contains the following information and settings:

Area	Information or Setting
Computer name	The control displays the name under which the control is visible in the company network. You can change the name. Further information: "Notes", Page 502
Default gateway	The control shows the default gateway and the Ethernet interface being used.
Use proxy	You can define the address and the port of a proxy server in the network.
Interfaces	The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty. The control displays the following information in the table: ■ Name (e.g., eth0) ■ Connection (e.g., X26) ■ Connection status (e.g., CONNECTED) ■ Configuration Name (e.g., DHCP) ■ Address (e.g., 10.7.113.10) Further information: "The Interfaces tab", Page 500
DHCP client	The control displays an overview of the devices that have received a dynamic IP address in the machine network. If there are no connections to other network components of the machine network, the table is empty. The control displays the following information in the table: ■ Name Host name and connection status of the device. The control shows the following connection status: ■ Green: Connected ■ Red: No connection ■ IP address Dynamically assigned IP address of the device ■ MAC address Physical address of the device ■ Type Type of connection The control displays the following connection types: ■ TFTP ■ DHCP ■ Valid up to Time until which the IP address is valid without being renewed The machine manufacturer can make settings for these devices. Refer to your machine manual.

The Interfaces tab

The control displays the available Ethernet interfaces on the **Interfaces** tab.
The **Interfaces** tab contains the following information and settings:

Column	Information or Setting
Name	The control displays the name of the Ethernet interface. You can activate or deactivate the connection by means of a toggle switch.
Connection	The control displays the number of the network connection.
Connection status	The control displays the connection status of the Ethernet interface. The following connection statuses may be displayed: ■ CONNECTED Connected ■ DISCONNECTED Connection separated ■ CONFIGURING The IP address is being fetched from the server ■ NOCARRIER No cable present
Configuration Name	You can execute the following functions: ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: ■ DHCP-LAN: Settings for the standard interface for a standard company network ■ MachineNet: Settings for the second, optional Ethernet interface; for configuration of the machine network Further information: "Network configuration with Advanced Network Configuration", Page 596 ■ Reconnect the Ethernet interface with Reconnect ■ Edit the selected profile Further information: "Network configuration with Advanced Network Configuration", Page 596



- If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.
- The control exclusively supports the **Ethernet** connection type.

The DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network (e.g., with industrial computers).
Refer to your machine manual.

The Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.

The **Ping/Routing** tab contains the following information and settings:

Area	Information or Setting
Ping	Address:Port and Address: You can enter the IP address of the computer and possibly the port number for checking the network connection. Entry: Four numerical values separated by dots and, if necessary, a port number separated by a colon (e.g., 10.7.113.10:22) As an alternative, you can enter the name of the computer whose connection you want to check. Starting and stopping the test ■ Start button: starts the test The control displays status information in the ping field. ■ Stop button: stops the test
Routing	The control displays status information of the operating system about the current routing for network administrators.

The SMB share tab

The **SMB share** tab is included only in connection with a VBox programming station. When the check box is active, the control releases areas or partitions protected by a code number for the Explorer of the Windows PC used, e.g. **PLC**. You can activate or deactivate the check box only by using the machine manufacturer code number.

In the **TNC VBox Control Panel**, select a drive letter within the **NC share** tab for displaying the selected partition and then connect the drive with **Connect**. The host displays the partitions of the programming station.



Further information: Programming station for milling controls

You download the documentation together with the programming station software.

Exporting and importing a network profile

To export a network profile:

- ▶ Open the **Network settings** window
- ▶ Select **Export configuration**
- > The control opens a window.
- ▶ Select the storage location for the network profile (e.g., **TNC:/etc/sysconfig/net**)
- ▶ Select **Open**
- ▶ Select the desired network profile
- ▶ Select **Export**
- > The control saves the network profile.



You can't export **DHCP** or **eth1** profiles.

To import an exported network profile:

- ▶ Open the **Network settings** window
- ▶ Select **Import the configuration**
- > The control opens a window.
- ▶ Select the storage location of the network profile
- ▶ Select **Open**
- ▶ Select the desired network profile
- ▶ Press **OK**
- > The control opens a window with a prompt.
- ▶ Press **OK**
- > The control imports and activates the selected network profile.
- ▶ You might need to restart the control



The **HEIDENHAIN default** button allows you to import the default values of the network settings.

Notes

- Preferably restart the control after making changes in the network settings.
- If you change the computer name of the control, you must reconfigure the LDAP database of user administration.

Further information: "Local LDAP database", Page 560

You must also regenerate the server certificates of **OPC UA NC Server** (#56-61 / #3-02-1*).

Further information: "Login options", Page 507

- The HEROS operating system manages the **Network settings** window. You must restart the control in order to change the HEROS conversational language.

Further information: "Conversational language of the control", Page 489

23.14 PKI Admin

Application

With **PKI Admin**, you can manage the server and client certificates of **OPC UA NC Server** (#56-61 / #3-02-1*) on the control. To define access rights to the control, you can classify the certificates as trusted or not trusted, for example.

PKI Admin has no functionality without the OPC UA NC Server (#56-61 / #3-02-1*) software option.

Related topics

- Quickly and easily connecting the OPC UA client application to the control (#56-61 / #3-02-1*)

Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 511

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► PKI Admin

The **Administration of the PKI Infrastructure** window provides the following tabs:

Tab	Function
Own certificates	The control provides the following areas: ■ Check the configuration The control checks the validity of the server certificates. ■ Use self-created certificate: ■ Optional certificate settings The control adds static IP addresses to the server certificates. You can select the IP address of the eth0 or eth1 interface or specify the required IP addresses. ■ Recreate certificate The control recreates the server's chain of trust. After the next restart of the control, it will use the new certificate. ■ Export certificate chain The control saves the server's chain of trust that you import into the client application. ■ Use customer-specific certificate: ■ Load certificate You can import a customized certificate. Please note the requirements for self-created certificates for OPC UA (#56-61 / #3-02-1*). Further information: "Login options", Page 507 ■ Existing certificates of server The control displays the available certificates and revocation lists. You can export the selected certificate or the selected revocation list, show its details, or delete it.

Tab	Function
Trusted	The server knows the certificate and trusts it after successful validation. For connection to the server, the client certificate must have been specified on this tab. For an OPC UA connection (#56-61 / #3-02-1*), you also need to assign an OPC UA license to the certificate. Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 512
Issuers	On this tab, you can specify the issuer of the trusted certificates. The server uses the issuer's information to validate the certificate.
Rejected	On this tab, the control specifies client certificates whose connection attempt to the OPC UA NC Server (#56-61 / #3-02-1*) failed. Connection failures can occur in the following situations: ■ The client certificate is unknown and has not been classified as trusted. If you want to connect the client application to the server, you can use the Move function to move the certificate to the Trusted tab. ■ A trusted client certificate has expired.
Revocation lists	On this tab, you can specify CRL files that list untrusted certificates. The server prohibits connections that use these certificates. In the Settings for revocation lists area, you can permit connections of applications with certificates in a multi-level certificate chain even if no associated CRL files exist.

Definition

PKI

PKI (public key infrastructure) is the management structure for digital certificates that are required for secure communication. A digital certificate has the same purpose as an identity card or passport. With a digital certificate, its owner can encrypt, sign and authenticate the communication.

23.15 OPC UA NC Server (#56-61 / #3-02-1*)

23.15.1 Fundamentals

Open Platform Communications Unified Architecture (OPC UA) describes a collection of specifications. These specifications are used to standardize machine-to-machine communication (M2M) in the field of industrial automation. OPC UA enables the data exchange across operating systems between products from different manufacturers, e.g. between a HEIDENHAIN control system and third-party software. Thus, OPC UA has become the data exchange standard for secure, reliable, manufacturer- and platform-independent industrial communication over the last years.

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to **OPC UA**. The security analysis was updated in 2022. The specification analysis performed by the BSI determined that **OPC UA** provides a high level of security as compared to most other industrial protocols.

HEIDENHAIN follows the BSI recommendations and provides SignAndEncrypt, which exclusively features up-to-date IT security profiles. For this purpose, OPC UA-based industrial applications and the **OPC UA NC Server** exchange certificates for authentication. In addition, any transferred data is encrypted. This effectively prevents messages between the communication partners from being intercepted or altered.

Application

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The **OPC UA NC Server** allows you to access the data and functions of the HEIDENHAIN NC information model exposed in the server address space.



Pay attention to the interface documentation of the **OPC UA NC Server** as well as the documentation of the client application.

Related topics

- **Information Model** interface documentation with the specification of the **OPC UA NC Server** in English
ID: 1309365-xx or **OPC UA NC Server Interface Documentation**
- Quickly and easily connecting the OPC UA client application to the control
Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 511
- User roles and user rights for OPC UA
Further information: "User administration roles and rights", Page 603
- Comparison of the transmission duration of different protocols
Further information: "Example: Transmission duration of different transmission types", Page 593

Requirements

- OPC UA NC Server (#56-61 / #3-02-1*) software options

For OPC UA-based communication, the HEIDENHAIN control provides the **OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 to 61).

If your control features the **SIK2**, you can order this software option multiple times and enable up to ten connections.
- Firewall configured

Further information: "Firewall", Page 528
- The OPC UA client supports the **security policy** and authentication method of **OPC UA NC Server**:
 - **Security Mode: SignAndEncrypt**
 - **Algorithm:**
 - **Basic256Sha256**
 - **Aes128Sha256RsaOaep**
 - **Aes256Sha256RsaPss**
 - **User authentication:**
 - **X509 certificates**
 - User name and password
- For logon with the user name and password:
 - Permitted by the machine manufacturer
 - User administration is active
 - NC.OpcUaPwAuth or NC.OpcUaPwAuthOnlyMachineNet right

Description of function

The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- Creation of service files
- Read and write tool data (the corresponding right is required)
- Read from and write to the counter (the corresponding right is required)
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (the corresponding right is required)
- Validation of 3D models for tool carriers

Further information: "Tool carrier management", Page 186

- Validate 3D models for tools (#140 / #5-03-2)

Further information: "Tool model (#140 / #5-03-2)", Page 190

Machine parameters in conjunction with OPC UA

The **OPC UA NC Server** enables OPC UA client applications to query general machine information, such as the year of construction of the machine or its location.

The following machine parameters are available for the digital identification of your machine:

- For users: **CfgMachineInfo** (no. 131700)

Further information: "The Machine information area", Page 479
- For the machine tool manufacturer: **CfgOemInfo** (no. 131600)

Further information: "The Info about machine manufacturer area", Page 478

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives. The following actions are permitted:

- Creating and deleting folders
- Reading, editing, copying, moving, creating, and deleting files

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. As long as the operating system is active, you can create and transmit service files, for example.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

- System-relevant files must be edited only by authorized specialists

Login options

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them (the application instance certificates) in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions. As an alternative to the user certificate, the **OPC UA NC Server** also permits login with a user name and password.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function. All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

Further information: "PKI Admin", Page 503

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

- ▶ Open the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- ▶ Select **SSH keys and certificates**
- ▶ Select **Import certificate**
- > The control opens the **Import certificate** window.
- ▶ Select the certificate
- ▶ Select **Open**
- > The control imports the certificate.
- ▶ Select **Use for OPC UA**
- > The control uses the certificate for **OPC UA**.

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
 - File format: *.der
 - Signature with hash SHA256
 - Validity period of at most 5 years is recommended
- Client certificates
 - Host name of the client
 - Application URI of the client
- Server certificates
 - Host name of the control
 - Application URI of the server according to the following structure:
urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
 - Validity period of 20 years maximum

Login with user name and password

The machine manufacturer can permit login with a user name and password, for example for client applications that do not support login with a user certificate.

For this type of login, a user with NC.OpcUaPwAuth or NC.OpcUaPwAuthOnlyMachineNet rights must exist while user administration is active.

In the **OPC UA** menu item of the **Settings** application, the control indicates the options available to the current user for logging in.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 510

Notes

- OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.
- Refer to your machine manual.
The machine manufacturer can create additional function users (for example, to enable client applications in order to access specific machine data when user administration is active).

Further information: "User administration", Page 549

23.15.2 The OPC UA (#56-61 / #3-02-1*) menu item

Application

In the **OPC UA** menu item of the **Settings** application, you can set up the connections to the control and check the status of the **OPC UA NC Server**.

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► OPC UA

The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether the OPC UA NC Server is active: ■ Green icon OPC UA NC Server is active ■ Gray icon: OPC UA NC Server is not active or software option not enabled You can manually start or restart the OPC UA NC Server as required. Further information: "Manually starting the OPC UA NC Server", Page 510
OPC UA connection assistant	Open the OPC UA NC Server connection assistant window Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 511
OPC UA license settings	Open the OPC UA NC Server - License Settings window Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 512
PKI Admin	Open the Administration of the PKI Infrastructure window Further information: "PKI Admin", Page 503
Login options of the current user	The control shows whether the options are available for the current user: ■ User certificate ■ User name and password Possible only if user administration is active
Host computer operation	Activate or deactivate host computer operation with a toggle switch Further information: "The DNC area", Page 513

When a connection is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 99

Manually starting the OPC UA NC Server

You can manually start or restart the **OPC UA NC Server** as required. Thus, you can apply changes made to the machine parameters or the certificates, which are relevant to the server, without having to shut down the control.

While an OPC UA connection is active, the control displays a confirmation prompt before the restart. During the restart, the control will disconnect active connections automatically.

For this function, you need the HEROS.SetNetwork permission.

Further information: "User administration roles and rights", Page 603

23.15.3 The OPC UA connection assistant function (#56-61 / #3-02-1*)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection assistant** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning the OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-10 using the **OPC UA NC Server - License Settings** window
Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 512
- Managing certificates with the **PKI Admin** menu
Further information: "PKI Admin", Page 503

Description of function

Use the **OPC UA** menu item to open the **OPC UA NC Server connection assistant** window.

Further information: "The OPC UA (#56-61 / #3-02-1*) menu item", Page 510

The assistant provides the following steps:

- 1 Export **OPC UA NC Server** certificates
- 2 Import the certificates of the OPC UA client application
- 3 Assign each of the available OPC UA NC Server software options to an OPC UA client application
- 4 Select the type of user logon: certificate or password
 If you select the user logon with a password, the control will skip the following steps up to the firewall.
- 5 Import the user certificates
- 6 Assign the user certificates to users
- 7 Configure the firewall
- 8 Connect the client application to the control

If at least one software option is active for the OPC UA NC Server, the control will generate the server certificate as a part of a self-generated certificate chain during the first start-up. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. The user name and the password are defined in the user administration. Please contact your IT department.

Note

The **OPC UA NC Server connection assistant** also helps you create test or sample certificates for users and the OPC UA client application. Do not use the user and client application certificates created at the control for other purposes than development at the programming station.

23.15.4 The OPC UA license settings function (#56-61 / #3-02-1*)

Application

You can use the **OPC UA NC Server - License Settings** window to assign an OPC UA client application to a software option 56 to 61 or #3-02-1 to #3-02-10.

Related topics

- Setting up the OPC UA client application with the **OPC UA connection assistant** function

Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 511

- Managing certificates with **PKI Admin**

Further information: "PKI Admin", Page 503

Requirement

- Certificate has been added to the **Trusted** category in **PKI Admin**

Description of function

Use the **OPC UA** menu item to open the **OPC UA license settings** window.

After using the **OPC UA connection assistant** or the **PKI Admin** menu item to import a certificate of an OPC UA client application, you can choose the certificate from a selection window.

If you enable the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

23.16 The DNC menu item

Application

The **DNC** menu item allows you to permit or block access to the control (e.g., connections via a network or TNCremo).

Related topics

- Connecting network drives
Further information: "Network drives on the control", Page 492
- Setting up a network
Further information: "Ethernet interface", Page 496
- TNCremo
Further information: "PC software for data transfer", Page 586
- Remote Desktop Manager (#133 / #3-01-1)
Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► DNC

The **DNC** area contains the following icons:

Icon	Meaning
	Add a computer-specific connection
	Edit a computer-specific connection
	Delete a computer-specific connection

When a connection is active, the control displays an icon in the information bar.

Further information: "Icons on the control's user interface", Page 99

The DNC area

In the **DNC** area you use toggle switches to activate the following functions:

Switch	Meaning
DNC access permitted	Permit or block all accesses to the control through a network
TNCopt full access allowed	Permit or block full access via TNCopt Only for testing
Host computer operation	Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation If host computer operation is active, the control displays the Host computer is active message in the information bar. You cannot use the Manual and Program Run operating modes. You cannot activate host computer operation while running an NC program.

Secure connections

The control displays general and customized settings for **Secure connections**. You can activate the following functions:

Row	Meaning
Fingerprint of the host key	Pressing the Show button tells the control to show a unique ASCII image that is equivalent to a fingerprint. When setting up a secure connection, you can compare this ASCII image to an image within the client application. That way you can ensure that you are connecting to the right control.
Setup permitted	If the toggle switch is active, client applications can establish a secure connection for the current user. Activate this toggle switch only while you are setting up a connection.
Key management	In this row, you open the Certificate and keys window. Further information: "SSH-secured DNC connection", Page 572

Computer-specific connections

If the machine manufacturer has defined the optional machine parameter **CfgAccessControl** (no. 123400), then in the **Connections** area you can permit or block access for up to 32 connections defined by you.

The control shows the defined information in a table:

Column	Meaning
Name	Host name of the external computer
Description	Additional information
IP address	Network address of the external computer
Access	■ Permit The control permits network access without confirmation. ■ Inquire The control asks for confirmation upon a network access attempt. You can choose whether to permit or block access once or always. ■ Deny The control does not permit network access.
Type	■ Com1 Serial interface 1 ■ Com2 Serial interface 2 ■ Ethernet Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Overview of ports for DNC connections

A DNC connection can use the following protocols:

- SSH (secure connection) with RPC-Secure or LSV2-Secure
- RPC
- LSV2



When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.

Further information: "SSH-secured DNC connection", Page 572

Depending on the protocol being used, the connection is established via the following ports:

Port	Control or client PC	Protocol	Firewall
22	Control and client PC with secure connection	RPC-Secure LSV2-Secure	Allow SSH
19000	Control	LSV2	Allow DNC
19003	Control	RPC	Allow DNC
19010-19034	Client PC	RPC	Allow port range

For the client PC, an RPC-Secure connection via an SSH tunnel also uses port 19036. You don't need to allow port 19036, since the secure connection is established via port 22.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** toggle switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

23.17 Printers

Application

You add and manage printers through the **Printer** menu item in the **Heros Printer Manager** window.

Related topics

- Using the **FN 16: F-PRINT** function for printing
Further information: Programming and Testing User's Manual

Requirement

- PostScript-capable printer
The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting PostScript emulation in the printer menu.
Further information: "Note", Page 518

Description of function

To navigate to this function:

Settings ▶ Network/Remote Access ▶ Printer ▶ Heros Printer Manager

You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: Programming and Testing User's Manual

Once you have added a printer, the control shows the **PRINTER:** drive in the file management. The drive contains one folder for each defined printer.

Further information: "Creating a printer", Page 518

There are various methods to start printing:

- Copying the file to be printed to the **PRINTER:** drive
The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.
You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.
- Using the **FN 16: F-PRINT** function

Icons and buttons

The **Heros Printer Manager** window provides the following icons and buttons:

Icon or button	Meaning
	Print a test page Prints a test page on the selected printer
	Delete Deletes the selected printer
CREATE	Creates a printer

Icon or button	Meaning
Copy	Creates a copy of the selected printer setting At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
Status	Displays the status information of the selected printer

The Change the printer window

You open the window by double-tapping or double-clicking the desired printer.
For each printer, the following properties can be set:

Area	Meaning
Name of the printer	Customizes the printer name
Connection	■ USB: The control automatically displays the name ■ Network: Network name or IP address of the printer Port for the network printer (default: 9001) ■ Printer %1 not connected ■ Timeout Delays the printing process The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in PRINTER:. Use this setting if the file to be printed is populated with FN functions (e.g., when probing). ■ Standard printer Select the default printer The control automatically assigns this setting to the first printer added.

Area	Meaning
Settings for printing of text	■ Paper size ■ Number of copies ■ Job name ■ Font size ■ Header ■ Printing options ■ Black/white ■ Color ■ Duplex
Orientation	■ Portrait ■ Landscape
Expert options	Available only to authorized specialists

23.17.1 Creating a printer

- To create a new printer:
- ▶ Enter the printer name in the input field
 - ▶ Select **CREATE**
 - > The control creates a new printer.
 - ▶ Double-tap or double-click the printer
 - > The control opens the **Change the printer** window.
 - ▶ Define the properties
 - ▶ Select **Save**
 - > The control applies the settings and displays the defined printer in the list.

23.17.2 Copying a printer

- To copy the printer properties of an available printer:
- ▶ Select the desired printer
 - ▶ Enter the name of the new printer in the input field
 - ▶ Select **Copy**
 - > The control creates a new printer with the settings of the selected printer.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

23.18 The VNC menu item

Application

VNC is software that allows you to access the control from remote devices, such as from an additional ITC operating station. **VNC** enables you to transmit screen contents, mouse movements, and keystrokes between devices.

Related topics

- Firewall settings
Further information: "Firewall", Page 528
- Remote Desktop Manager (#133 / #3-01-1)
Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► VNC

When an ITC is connected and **Enabling VNC focus** is set, the control displays an icon.

Further information: "The VNC Focus Settings area", Page 521

Further information: "Icons on the control's user interface", Page 99

Icons and buttons

The **VNC settings** window provides the following buttons:

Button	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view Required with connection attempts during which the dialog is open.
Set standard values	Reset the settings to their default values
Set preferred owner of the focus	Select the Preferred owner of the focus check box

The VNC participant settings area

In the **VNC participant settings** area, the control shows a list of all clients.
The control displays the following contents:

Column	Contents
Computer name	IP address or computer name
VNC	Connection of the client to the VNC viewer
VNC Focus	The client participates in the focus assignment
Type	■ Manual Manually entered client ■ Denied This client is not permitted to connect. ■ Enable TeleService and IPC Client via a TeleService connection ■ DHCP Other computer that retrieves an IP address from this computer.

The Firewall warning area

If the firewall blocks **VNC**, the control displays the **Firewall warning** area.
Further information: "Firewall", Page 528

The Global settings area

In the **Global settings** area, you can define the following settings:

Function	Meaning
Enable RemoteAccess and IPC	If the check box is selected, the connection is always permitted.
Password verification	Client must enter a password for verification The control opens a window when you select the check box. In this window you define the password for this client. The client must enter the password when establishing the connection.

The Enabling other VNC area

In the **Enabling other VNC** area, you can define the following settings:

Function	Meaning
Deny	Other VNC clients are not permitted.
Inquire	A dialog opens when another VNC client wants to connect. You must grant permission for this connection.
Permitted	Other VNC clients are permitted.

The VNC Focus Settings area

In the **VNC Focus Settings** area, you can define the following settings:

Function	Meaning
Enabling VNC focus	Enables focus assignment for this system When the check box is inactive, the focus owner actively gives away the focus by using the focus symbol. The remaining clients can request the focus only after it was given away.
Reset the CapsLock key when changing the focus	When the check box is active and the focus owner has activated the CapsLock key, the CapsLock key is deactivated if the focus changes. Only if the Enabling VNC focus check box is enabled
Enable Concurrency VNC Focus	When the check box is active, every client can request the focus at any time. The focus owner does not need to give away the focus before to enable that. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the Enabling VNC focus check box is enabled
Timeout Concurrency VNC Focus	Period of time after requesting the focus during which the focus owner can object to the focus change (at most 60 seconds). This period of time is set by moving a slider. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the Enabling VNC focus check box is enabled



Select the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for this purpose (e.g., ITC industrial computers).

Notes

- The machine manufacturer defines the procedure for assigning the focus with multiple clients or operating units. Focus assignment depends on the setup and operating situation of the machine tool.
Refer to your machine manual.
- The control displays a message if the firewall settings of the control do not permit the VNC protocol for all clients.

Definition

Abbreviation	Definition
VNC (virtual network computing)	VNC is software that allows you to control other devices over a network connection.

23.19 The Remote Desktop Manager window (#133 / #3-01-1)

Application

With Remote Desktop Manager you can display external computer units on the control screen that are connected via Ethernet, and operate them through the control. You can also shut down a Windows computer together with the control.

Related topics

- External access
Further information: "The DNC menu item", Page 513

Requirements

- Remote Desk. Manager (#133 / #3-01-1) software option
- Existing network connection
Further information: "Ethernet interface", Page 496

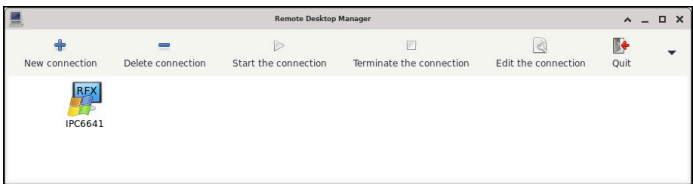
Description of function

To navigate to this function:

Settings ► Network/Remote Access ► Remote Desktop Manager

Remote Desktop Manager grants the following connection options:

- **Windows Terminal Service (RemoteFX):** Display the desktop of an external Windows computer on the control
Further information: "Windows Terminal Service (RemoteFX)", Page 523
- **VNC:** Display the desktop of an external Windows, Apple or Unix computer on the control
Further information: "VNC", Page 523
- **Switch-off/restart of a computer:** Automatically shut down a Windows computer together with the control
- **World Wide Web:** Available only to authorized specialists
- **SSH:** Available only to authorized specialists
- **User-defined connection:** Available only to authorized specialists



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

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

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

Buttons

Remote Desktop Manager contains the following buttons:

Button	Function
New connection	Create a new connection in the Edit the connection window Further information: "Establishing and starting a connection", Page 526
Delete connection	Delete the selected connection
Start the connection	Start the selected connection Further information: "Establishing and starting a connection", Page 526
Terminate the connection	Terminate the selected connection
Edit the connection	Edit the selected connection in the Edit the connection window Further information: "Connection settings", Page 524
Exit	Close Remote Desktop Manager
Import connections	Restore the selected connection Further information: "Exporting and importing connections", Page 527
Export the connections	Back-up the selected connection Further information: "Exporting and importing connections", Page 527

Windows Terminal Service (RemoteFX)

You don't need any additional software on a computer for a RemoteFX connection, but you might need to change some settings on the computer.

Further information: "Configuring an external computer for Windows Terminal Service (RemoteFX)", Page 526

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

With RemoteFX, a separate window opens for the screen of the external computer. The active desktop on the external computer is then locked and the user logged off. This prevents two users from accessing the control simultaneously.

VNC

You need an additional **VNC** server for your external computer when connecting through VNC. Install and configure the VNC server (e.g., TightVNC server) before establishing the connection.

VNC mirrors the screen of the external computer. The active desktop on the external computer is not locked automatically.

With a **VNC** connection you can shut down the external computer through the Windows menu. The computer cannot be restarted through the connection.

Additional settings for VNC

The control offers the following additional connection settings for the **VNC** option:

Setting	Meaning	Usage
Full-screen mode or User-defined window size:	Size of the connection window on the control	Required
Permit further connections (share)	Additionally grant other VNC connections access to the VNC server	Required
View only	In display mode, the external computer cannot be operated.	Required

Additional settings for Switch-off/restart of a computer

The control offers the following additional connection settings for the **Switch-off/restart of a computer** option:

Setting	Meaning	Usage
User name	User name with which the connection should log in.	Required
Windows domain:	If required, domain of the target computer	Optional
Max. waiting time (seconds):	A shutdown of the control causes the Windows computer to shut down as well. Before the control displays the Now you can switch off. message, it waits for the number of seconds defined here. While waiting, the control checks whether the Windows computer is still accessible (port 445). If the Windows computer is switched off before the defined number of seconds have expired, the control will wait no longer.	Required
Additional waiting time:	Waiting time after the Windows computer has stopped being accessible. Windows applications may delay the shutdown of the computer after port 445 has been closed.	Required
Force	Close all programs on the Windows computer, even if dialogs are still open. If Force is not selected, Windows waits up to 20 seconds. This delays the shutdown process or the Windows computer is switched off before Windows has shut down.	Required
Restart	Restart the windows computer	Required
Run during restart	When the control restarts, restart the Windows computer as well. Effective only if the control is restarted using the shutdown icon at the bottom right in the taskbar or if it is restarted as a result of a change in the system settings (e.g. network settings).	Required
Run during switch-off	Shut down the Windows computer (no restart) when shutting down the control. This is the default behavior. Even the END key will then not trigger a restart.	Required

23.19.1 Configuring an external computer for Windows Terminal Service (RemoteFX)

To configure the external computer (e.g., in Windows 10 operating systems):

- ▶ Press the Windows key
- ▶ Select **Control Panel**
- ▶ Select **System and Security**
- ▶ Select **System**
- ▶ Select **Remote Settings**
- > The computer opens a pop-up window.
- ▶ Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
- ▶ In the **Remote Desktop** area, enable **Allow Remote connections to this computer**
- ▶ Press **OK** to confirm your settings

23.19.2 Establishing and starting a connection

To establish and start a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select **New connection**
- > The control displays a selection menu.
- ▶ Select a connection option
- ▶ Under **Windows Terminal Service (RemoteFX)**, select the operating system
- > The control opens the **Edit the connection** window.
- ▶ Define the connection settings
- Further information:** "Connection settings", Page 524
- ▶ Press **OK**
- > The control saves the settings and closes the window.
- ▶ Select connection
- ▶ Select **Start the connection**
- > The control starts the connection.

23.19.3 Exporting and importing connections

To export a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the desired connection
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Export the connections**
- > The control opens the **Select export file** window.
- ▶ Define the name of the saved file
- ▶ Select the target file
- ▶ Select **Save**
- > The control saves the connection data under the name defined in the window.

To import a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Import connections**
- > The control opens the **Select file to import** window.
- ▶ Select file
- ▶ Select **Open**
- > The control creates the connection under the name that was defined originally in **Remote Desktop Manager**.

Notes

NOTICE

Caution: Data may be lost!

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

- ▶ Configure the automatic shutdown of the Windows computer

- When you edit an existing connection, the control will automatically delete all impermissible characters from the name.

Notes in connection with the IPC 6641

- HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.
- If you use the computer name **IPC6641.machine.net** to connect an IPC 6641, it is important to enter **.machine.net**.

With this entry, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**; this reduces the time needed for access.

23.20 Firewall

Application

The control provides a firewall to allow or reject incoming network traffic, depending on the sender and service.

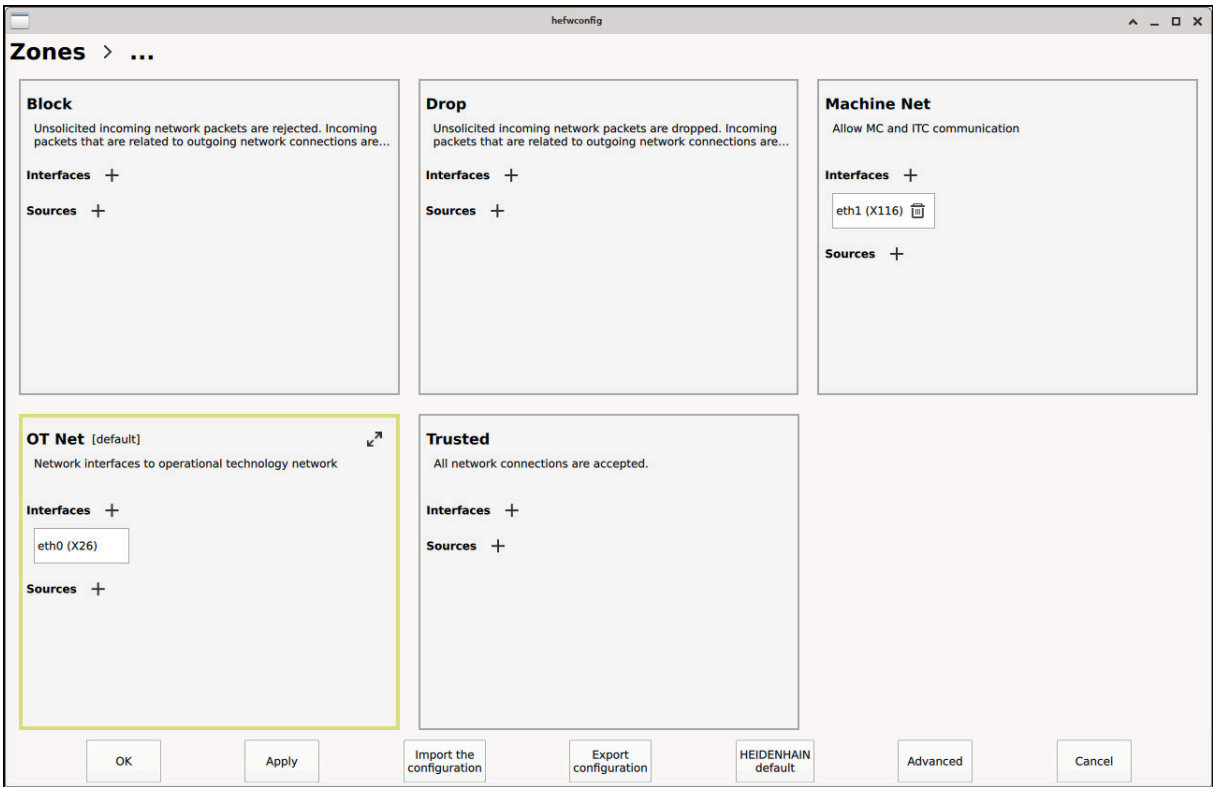
Related topics

- Existing network connection
Further information: "Ethernet interface", Page 496
- SELinux security software
Further information: "SELinux security software", Page 491
- Ports required for a DNC connection
Further information: "Overview of ports for DNC connections", Page 515
- Comparison of the transmission duration of different protocols
Further information: "Example: Transmission duration of different transmission types", Page 593

Description of function

To navigate to this function:

Settings ► Network/Remote Access ► Firewall



Overview of zones

The **Firewall** window always shows the **OT Net** zone after opening. If you select **Zones** in the breadcrumb navigation, the control opens an overview of the zones.

Five zones are displayed on the default overview screen. The overview shows which interfaces and sources have been added to the respective zones.

Each zone has its own default configuration.

Further information: "Zones", Page 530

You can edit the configuration of the zones.

Further information: "Settings of the zones", Page 531



For example, network specialists can make the following changes:

- Add and remove zones
- Rename zones
- Edit the zone description
- Edit the default target of the zone

If a network specialist has made changes to the firewall, the firewall of your machine might differ from the default configuration.

Icons and buttons

The **Firewall** window provides the following icons and buttons:

Icon or button	Meaning
	Maximize Open the selected zone
	Reduce Close the open zone and return to the overview screen
	Add
	Clear
	Edit Edit comprehensive rule
Logging	Activate or deactivate logging of rejected packets in the service file Further information: "Service file", Page 299 This information might be needed by HEIDENHAIN Customer Service for diagnostic purposes. Deactivate this function after the fault has been fixed, so as not to impair the performance.
OK	Save the changes and close the window
Apply	Save the changes
Import the configuration	Import the configuration and overwrite the present configuration
Export configuration	Export the configuration of all zones
HEIDENHAIN default	Reset the settings to their default values If the machine manufacturer has stored standard values, the control displays the OEM default button
Advanced	Open the Firewall Configuration window Available only to network specialists
Cancel	Discard the changes that have not been saved and close the window

Default targets

Each zone has a default target. The default target defines how the firewall will handle incoming network connections. The firewall provides the following default targets:

Default target	Meaning
ACCEPT	Accept all incoming network connections This corresponds to the deactivation of the firewall.
DROP	Discard the incoming network connections You can add or remove exceptions.
REJECT	Reject the incoming network connections You can add or remove exceptions.

Further information: "Settings of the zones", Page 531

Zones

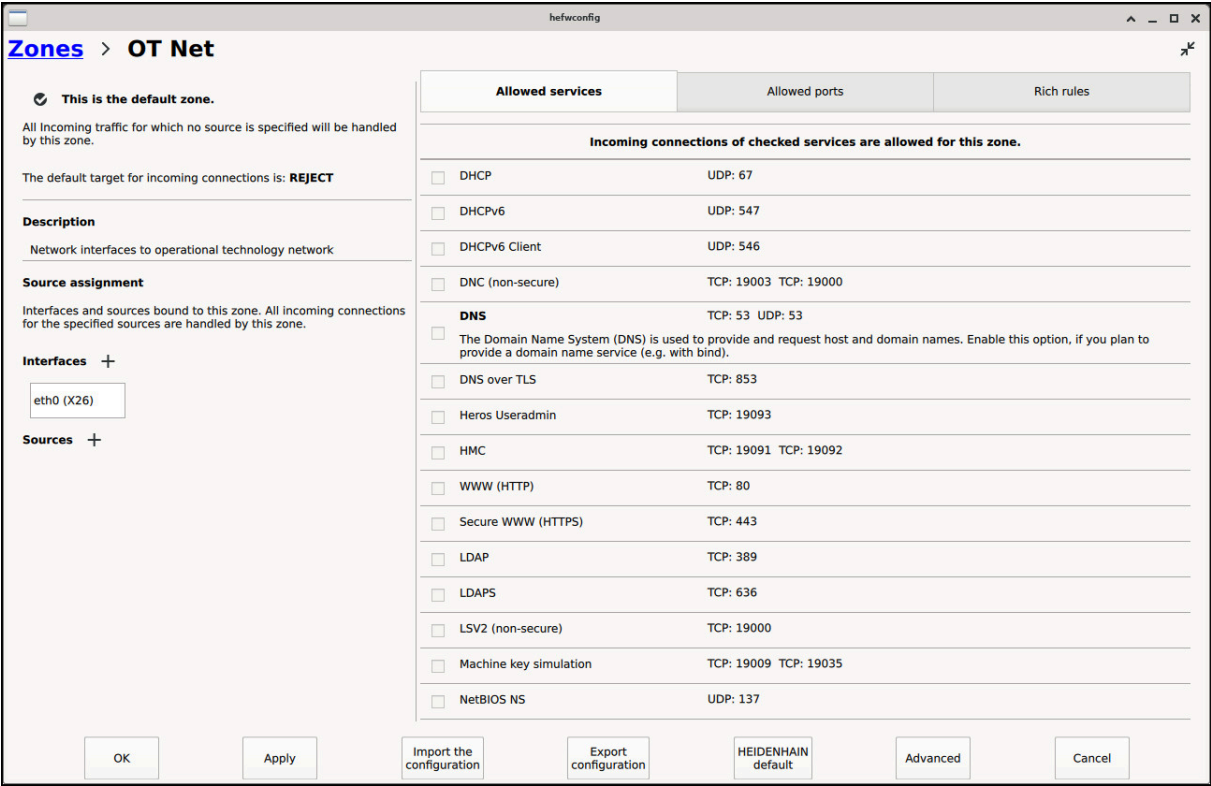
The following table shows the available zones and the default configuration:

Zone	Meaning
Block	Default target: REJECT This zone rejects all incoming connections.
Drop	Default target: DROP This zone discards all incoming connections.
Machine Net	Default target: REJECT with exceptions This zone accepts all the services needed for connections between the control and an additional ITC operating station (e.g., VNC or DNS). The eth1 interface is assigned to this zone.
OT Net	This zone is the default zone. Default target: REJECT with exceptions This zone accepts the SSH service. The eth0 interface is assigned to this zone.
PNC Net	This zone is the default zone on a PNC7. Default target: REJECT with exceptions This zone accepts the services SSH and VNC . On a PNC the eth0 interface is assigned to this zone.
Trusted	Default target: ACCEPT This zone accepts all incoming connections.



On programming stations, the **eth1** interface is assigned to the additional zone **Programmingstation Network** by default.

Settings of the zones



OT Net zone with description of the DNS service

When you open a zone, the control displays the following settings:

Setting	Meaning
Default zone	In this area, the control shows whether the zone is the default zone. If the zone is not the default zone, you can define this zone as the default zone by selecting the check box. The control automatically assigns all the unassigned interfaces and sources to the default zone.
Source assignment	In this area, the control shows the interfaces and sources assigned to this zone. You can add or delete interfaces and sources.
Allowed services	On the Allowed services tab, the control displays all available services and the related ports. Use the check boxes to allow or reject services. If the check box is selected, the service is allowed. When you select a service, the control displays the appropriate description.  HEIDENHAIN recommends that you add or delete exceptions only in the OT Net zone.
Allowed ports	On the Allowed ports tab, you can allow the TCP or UDP protocol. When you select the Add button, the control displays a window. Select TCP or UDP and define the port or the range of ports.

Setting	Meaning
Rich rules	On the Rich rules tab, you can define the exceptions for sources, services, and ports in more detail. When you create a comprehensive rule, the control provides the following selection options: ■ Action ■ Accept Accept the selected element ■ Reject Reject the selected element ■ Drop Discard the selected element ■ Source IP address or MAC address You can also define a rule using the element Service, TCP, or UDP without specifying a source. ■ Element ■ All You must specify a source. The selected action applies to all services and ports. ■ Service The control provides a selection menu containing all available services. ■ TCP The control provides an input field for the port or the range of ports. ■ UDP The control provides an input field for the port or the range of ports.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.
- You must save all changes by using the **Apply** button; the control will discard the changes that have not been saved.
- You can also open a zone by double-tapping or double-clicking the zone.
- You can assign the interfaces or sources to different zones. A zone will be active once an interface or a source has been assigned to it.
- You can also add or delete interfaces and sources on the overview screen of the zones.
- If you delete an interface or source from a zone, the control will always assign this interface or source to the default zone. You cannot delete any interfaces or sources from the default zone.

23.21 Portscan

Application

With the **Portscan** function, the control checks all open, incoming TCP and UDP listen ports at defined intervals or when commanded. The control shows a message if a port is not listed.

Related topics

- Firewall settings
Further information: "Firewall", Page 528
- Network settings
Further information: "Network configuration with Advanced Network Configuration", Page 596

Description of function

To navigate to this function:

Settings ► Diagnostics/Maintenance ► Portscan

The control searches for all open, incoming TCP and UDP listen ports on the system and compares them to the following whitelists:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with machine-manufacturer-specific functions: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

Each whitelist contains the following information:

- Port type (TCP/UDP)
- Port number
- Offering program
- Comments (optional)

Start the portscan manually by selecting the **Start** button in the **Manual Execution** area. In the **Automatic Execution** area, you can use the **Automatic update on** function to specify that the control will perform the portscan automatically in the selected interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists may be open. The control shows a message window if a port is not listed.

23.22 Backup and restore

Application

The **NC/PLC Backup** and **NC/PLC Restore** functions allow you to back up and restore individual folders or the complete **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

- File management, **TNC:** drive

Further information: Programming and Testing User's Manual

Description of function

To navigate to this function:

Settings ► Diagnostics/Maintenance ► NC/PLC Backup

Settings ► Diagnostics/Maintenance ► NC/PLC Restore

The backup function creates a ***.tncbck** file. The restore function can restore these files as well as files from existing TNCbackup programs. If you double-tap or double-click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: Programming and Testing User's Manual

Within the backup function you can choose between the following types of backups:

- **Back up the “TNC:” partition**
Back-up all data on the **TNC:** drive
- **Back up the directory tree**
Back-up the selected folders and their subfolders on the **TNC:** drive
- **Back up the machine configuration**
Only for the machine manufacturer
- **Complete backup (TNC: and machine configuration)**
Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **Continue** and **Back** buttons.

23.22.1 Backing up data

To back-up the data of the **TNC:** drive:



- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Backup**
- The control opens the **Back up the “TNC:” partition** window.
- ▶ Specify the type of backup
- ▶ Select **Continue**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select any predefined exclusion rules or ones you have defined yourself
- ▶ Select **Continue**
- The control generates a list of files for backing up.
- ▶ Check list
- ▶ Deselect files if necessary
- Further information:** "Selecting or deselecting multiple files",
 Page 536
- ▶ Select **Continue**
- ▶ Enter the name of the backup file
- ▶ Select the storage path
- ▶ Select **Continue**
- The control generates the backup file.
- ▶ Confirm with **OK**
- The control concludes the backup process and restarts the NC software.

23.22.2 Restoring data

NOTICE

Caution: Data may be lost!

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

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

To restore data:



- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Restore**
- > The control opens the **Restore data - %1** window.
- ▶ Select the archive to be restored
- ▶ Select **Continue**
- > The control generates a list of files for restoring.
- ▶ Check list
- ▶ Deselect files if necessary
- Further information:** "Selecting or deselecting multiple files", Page 536
- ▶ Select **Continue**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select **Extract archive**
- > The control restores the files.
- ▶ Confirm with **OK**
- > The control restarts the NC software.

23.22.3 Selecting or deselecting multiple files

To select or deselect multiple files at a time:

- ▶ Select the first of the range of desired files.
- ▶ Select **Multiple selection**
- > The control activates multiple selection.
- ▶ Select the last of the range of desired files
- > The control highlights all files starting from the first selected file.
- ▶ Select the check box for selecting or deselecting
- > The control selects or deselects all the highlighted files.
- ▶ Select **Multiple selection**
- > The control deactivates multiple selection.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

23.23 TNCdiag

Application

TNCdiag displays status and diagnostic information of HEIDENHAIN components.

Description of function

To navigate to this function:

Settings ► Diagnostics/Maintenance ► TNCdiag



Only use **TNCdiag** after consultation with your machine manufacturer, unless you want to use it for setting up wireless handwheels.

Further information: "Setting up a wireless handwheel ", Page 444



For general information, please refer to the **TNCdiag** documentation.

23.24 Update the documentation

Application

The **Update the documentation** function can be used, for example, to install or update the integrated **TNCguide** product aid.

Related topics

- Integrated product aid **TNCguide**

Further information: "User's Manual as integrated product aid: TNCguide", Page 54

- Product aid on the HEIDENHAIN website

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

Description of function

To navigate to this function:

Settings ► Diagnostics/Maintenance ► Update the documentation

The file manager is located in the **Update the documentation** area. You can select and install the desired documentation from the file manager.

Further information: "Transferring TNCguide", Page 539

The control shows all available documents in the **Help** application.



In the **Update the documentation** area, you can install all HEIDENHAIN-specific documents (e.g., NC error messages).

23.24.1 Transferring TNCguide

To find and transfer the desired **TNCguide** version:

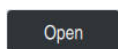
- ▶ Select the link to the HEIDENHAIN website
https://content.heidenhain.de/doku/tnc_guide/html/de/index.html
- ▶ Select **TNC controls**
- ▶ Select **TNC7 Series**
- ▶ Select the NC software number
- ▶ Navigate to the **product aid (HTML)**
- ▶ Select **TNCguide** in the desired language
- ▶ Select the path for saving the file
- ▶ Select **Save**
- > The download starts.
- ▶ Transfer the downloaded file to the control



- ▶ Select the **Home** operating mode



- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Select **Update the documentation**
- > The control opens the **Update the documentation** area.
- ▶ Select the desired file with the extension ***.tncdoc**



- ▶ Select **Open**
- > The control reports in a window whether installation was successful or failed.



- ▶ Select the **Help** application



- ▶ Select **Home**
- > The control displays all available documentation.

23.25 Machine parameters

Application

You can configure the behavior of the control with machine parameters. For this purpose, the control provides the **MPs for Users** and **MPs for setters** applications. You can open the **MPs for Users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics



Overview of the Machine Parameters, Error Numbers and System Data

The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:

- Machine parameters of the **MPs for setters** application
- Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
- System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions

ID 1445456-xx

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

TNCguide

Requirements

- Code number 123
Further information: "Code numbers", Page 474
- The contents of the **MPs for setters** application have been defined by the machine manufacturer

Description of function

To navigate to this function:

Settings ► Machine Parameters ► MPs for setters

In the **Machine Parameters** group the control shows only those menu items that you can choose with the current access rights.

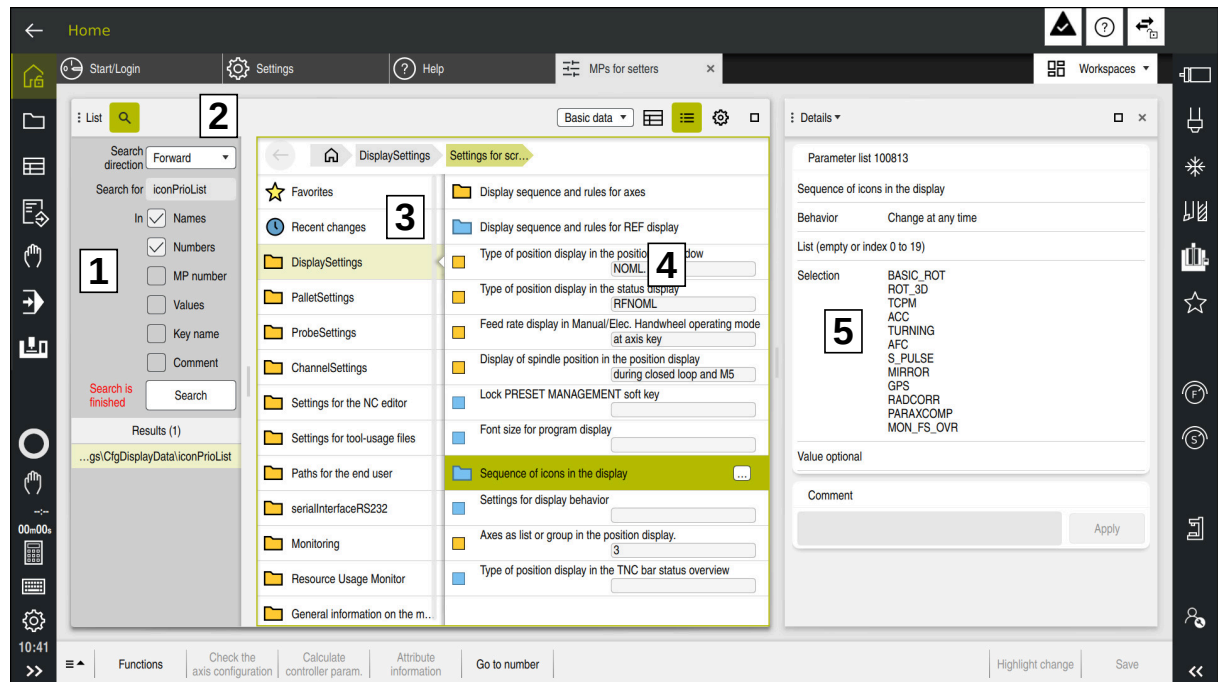
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- Details**
- Document**
- List**

You cannot close the **List** workspace.

The configuration editor areas



MPs for setters application with selected machine parameter

The configuration editor shows the following areas:

1 The **Search** column

You can search forward or backward with the following characteristics:

- **Name**
This is the language-neutral name used for machine parameters in the User's Manual.
- **Number**
This is the unique number used for machine parameters in the User's Manual.
- **MP number of the iTNC 530**
- **Value**
- **Key name**
Machine parameters for axes or channels exist more than once. In order to avoid ambiguity, each axis and each channel is identified with a key name (e.g., **X1**)
- **Comment**

The control displays the results.

2 Title bar of the **List** workspace

The title bar of the **List** workspace includes the following functions:

- Open or close the **Search** column
- Filter contents using a selection menu
- Toggle between structure and table views
Further information: "The contents displayed in table view", Page 544
- Open or close the **Details** workspace
Further information: "The Details workspace", Page 545
- Open or close the **Configuration** window
Further information: "The Configuration window", Page 545

- 3 Navigation column
- The control provides the following options for navigation:
- Navigation path

■ Favorites

■ 21 most recent changes

■ Structure of the machine parameters
- 4 Content column
- In the content column the control displays objects, machine parameters, or changes that you select using the search function or navigation column.
- 5 The **Details** workspace
- The control displays information on the selected machine parameter or the most recent change you made.
- Further information:** "The Details workspace", Page 545

Icons, buttons and shortcuts

The configuration editor provides the following icons, buttons, and shortcuts:

Icon, button, or shortcut	Meaning
	Activate or deactivate the table view The control toggles between structure and table views. Further information: "The contents displayed in table view", Page 544
	Active or deactivate the Change column width window You can adjust the width of the currently selected column. Only if table view is active
	Open or close the Details workspace Further information: "The Details workspace", Page 545
	Open or close the Configuration window Further information: "The Configuration window", Page 545
	Expand
	Collapse Only if the Display in tree view toggle switch is active Further information: "The Configuration window", Page 545
	Select Recent changes
	Object exists ■ Data object ■ Directory ■ Parameter list
	Object empty
	Open context menu Further information: Programming and Testing User's Manual
	Machine parameter exists

Icon, button, or shortcut	Meaning
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter not readable and not editable
	Changes to the machine parameter not yet saved
Functions	Open the context menu Further information: Programming and Testing User's Manual
[CTRL] + [N]	Create a new object in a list of data objects or parameters
Check the axis configuration	Only for the machine manufacturer
Calculate controller param.	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Go to number	The control opens the Enter number, then select from list window and suggests the number of the currently selected object. You can enter the number of a machine parameter and navigate directly to this parameter.
Highlight change	The control marks the selected machine parameter as changed, even though the value remains the same. For machine manufacturers only
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

The contents displayed in table view

The screenshot shows a software interface titled 'List' with a search bar and a 'Basic data' dropdown. Below is a table with two columns: 'TT140_1' and 'TT140_2'. The table contains several rows of machine parameters. Annotations with numbered boxes identify specific elements: '1' points to the column header 'TT140_2', '2' points to the entity name 'CfgTTRectStylus', '3' points to the attribute name 'centerPos', and '4' points to the index '[1]' in the row for 'centerPos'. The value '0' in the 'TT140_2' column for 'centerPos [1]' is highlighted in yellow.

	TT140_1	TT140_2
CfgTTRectStylus		
centerPos		
[0]	0	0
[1]	0	0
[2]	0	0
safetyDistToolAx	15	15
safetyDistStylus	11	11

The **List** workspace in table view

The table view of the **List** workspace displays the following information:

- 1 Group name (key)
The key is displayed in the header of the table. If no key is available, the header is empty.
- 2 Object name (entity)
The entity is displayed left-aligned in the first column. The name of the entity starts with **Cfg**.
- 3 Name (attribute) of the machine parameter
The attribute is displayed right-aligned in the first column.
- 4 Index of the machine parameter
If indexes are available, the index numbers are enclosed within square brackets.

The table view allows you to compare the configurations of different keys, for example.

The Configuration window

The control provides the following toggle switches in the **Configuration** window:

- **Show MP descriptive texts**

If the toggle switch is active, the control displays a description of the machine parameter in the active conversational language.

If the toggle switch is not active, the control displays the language-neutral name of the machine parameter.

- **Display in tree view**

If the toggle switch is active, the control displays the machine parameters in a tree view.

If the toggle switch is not active, the control displays the machine parameters in the structure view.

- **Do not show attributes without values**

If the toggle switch is active, the control hides optional machine parameters and parameter lists.

- **Do not show empty folders**

If the toggle switch is active, the control hides empty data objects and data-object lists.

The Details workspace

If you select contents from the favorites or the structure, the control will display information in the **Details** workspace, such as:

- Type of object, such as data object list or parameter
- Descriptive text of machine parameter
- Permitted or required input
- Prerequisite for the change (e.g., program run blocked)
- Number of the machine parameter on the iTNC 530
- Machine parameter optional

This information is included if a machine parameter can be enabled optionally.

If you select contents from the most recent changes you made, the control will display the following information in the **Details** workspace:

- Sequential number of the last change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Permitted or required input

Notes

- The **Home** operating mode icon shown in the TNC bar changes if the code number 123 is enabled. The icon additionally shows an unlocked padlock to illustrate that machine parameters can be accessed.
- The **CFGREAD** function allows you to read the values from machine parameters. Use the table view to determine the parameters required for **CFGREAD**.
Further information: Programming and Testing User's Manual
- The machine manufacturer defines which machine parameters are saved user-specifically by the control when user administration is active. These machine parameters can be changed at any time without, for example, having to restart the control.
Further information: "User administration", Page 549
- The machine manufacturer offers further applications for machine parameters.
- If later customization of the machine configuration by the machine manufacturer is intended, the machine operator might incur additional costs.

23.26 Configuring the control's user interface

Application

Each user can create and activate configurations in which the control's user interface is customized.

Related topics

- Workspaces

Further information: "Workspaces", Page 87

- Control interface

Further information: "Areas of the control's user interface", Page 83

Description of function

To navigate to this function:

Settings ► Configurations ► Configurations

A configuration contains all adaptations to the control's user interface that do not influence the control's actual functions.

- Settings for the TNC bar
- Arrangement of workspaces
- Font size
- Favorites

The **Configurations** area contains the following functions:

Function	Meaning
Active Configuration	Activate a configuration from a selection menu Further information: "The Desktop menu workspace", Page 102
Default configuration	Use the Reset button to apply the settings of the OEM configuration to the active configuration.
Save as OEM Configuration	The machine manufacturer can use the Save button to overwrite the OEM configuration .
Save current settings	With the Save button, you can save the current version of the active configuration.
Restore last configuration	With the Reset button, you can discard any customizations and revert to the saved version of the active configuration.
Enable autosave	If the toggle switch is active, the control automatically saves changes to the active configuration. If the toggle switch is not active, you can save the active configuration with the Save current settings function. If, for example, more than one person is working on the same configuration, the configuration will not be overwritten unintentionally.

The control displays the following information about all available configurations in a table:

Column	Meaning
Configuration Name	Name of the configuration
Selectable	If this toggle switch is active, you can select the configuration in the Active configuration selection menu.
Exportable	If this toggle switch is active, you can export the configuration. Further information: "Exporting and importing configurations", Page 548
Edit	This column contains two buttons, for renaming and deleting the configuration.

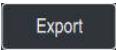
Press the **Add** button to create a new configuration.

23.26.1 Exporting and importing configurations

To export configurations:



- ▶ Select the **Settings** application
- ▶ Select **Configurations**
- > The control opens the **Configurations** area.
- ▶ Activate the **Exportable** toggle switch for the desired configuration, if necessary



- ▶ Select **Export**
- > The control opens the **Save as** window.
- ▶ Select the target file
- ▶ Enter a file name



- ▶ Select **Create**
- > The control saves the configuration file.

To import configurations:



- ▶ Select **Import**
- > The control opens the **Import configurations** window.
- ▶ Select file



- ▶ Select **Import**
- > If importing a configuration would overwrite a file with the same name, the control displays a prompt.
- ▶ Select the procedure:
 - **Overwrite:** The control overwrites the original configuration.
 - **Keep:** The control does not import the configuration.
 - **Cancel:** The control cancels the import process.

Notes

- Delete only inactive configurations. If you delete an active configuration, the control first activates a default configuration. This can lead to delays.
- The **Overwrite** function permanently replaces existing configurations.

24

User administration

24.1 Fundamentals

Application

User administration enables you to create and administrate different users with different access rights to various functions of the control. You can assign roles to the various users that reflect their respective tasks, such as machine operator or setup technician.

User administration is inactive in the control's factory default setting. This status is called **Legacy-Mode**.

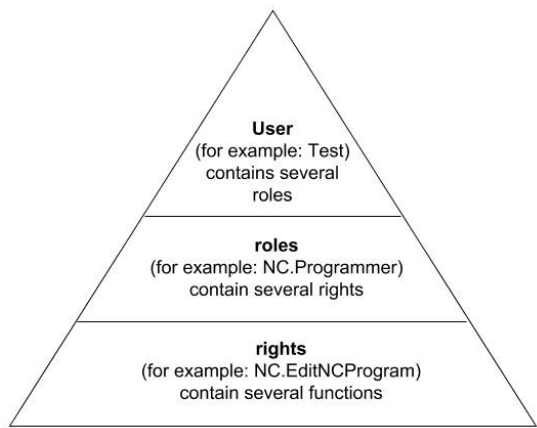
Description of function

User administration supports you in the following fields of security, based on the requirements of the IEC 62443 series of standards:

- Application security
- Network security
- Platform security

The user administration differentiates between the following terms:

- User
Further information: "Users", Page 550
- Roles
Further information: "Roles", Page 552
- Rights
Further information: "Rights", Page 552



Users

The user administration offers the following types of users:

- Function users pre-defined by HEIDENHAIN
- Function users pre-defined by the machine manufacturer
- Self-defined users

Depending on the task assigned, you can use one of the pre-defined function users or you have to create a new user.

Further information: "Creating a new user", Page 556

If you deactivate user administration, the control saves all configured users. Thus they will be available again when user administration is reactivated.

If you want to delete the configured users upon deactivation, you need to set this explicitly when deactivating user administration.

Further information: "Deactivating user administration", Page 557

HEIDENHAIN function users

HEIDENHAIN function users are pre-defined users that are automatically created upon activation of user administration. Function users cannot be changed.

HEIDENHAIN provides four different function users in the control's factory default setting.

- **useradmin**

The **useradmin** function user is automatically created upon activation of user administration. The **useradmin** function user allows you to configure and edit user administration.

- **sys**

The **sys** function user allows you to access the **SYS:** drive of the control. This function user is reserved for use by HEIDENHAIN service personnel.

- **user**

In **legacy mode**, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect. The logged-on user of the type **user** cannot be changed in **legacy mode**.

- **oem**

The **oem** function user is intended for the machine manufacturer. The **oem** function user allows you to access the **PLC:** drive of the control.

The useradmin function user

The **useradmin** user is comparable to the local administrator of a Windows system.

The **useradmin** account provides the following functions:

- Creating databases
- Assigning the password data
- Activating the LDAP database
- Exporting LDAP server configuration files
- Importing LDAP server configuration files
- Emergency access if the user database was destroyed
- Retroactive change of the database connection
- Deactivating user administration

Function users pre-defined by the machine manufacturer

Your machine manufacturer can define up to 32 function users, such as for machine maintenance or for setting up and operating external systems.

Function users defined by the machine manufacturer can also be used as a substitute for code numbers. You can use the function users' passwords to enable their additional rights temporarily.

Further information: "The Active user window", Page 559

The machine manufacturer's function users can already be active in **legacy mode** and replace code numbers.

Roles

HEIDENHAIN combines several rights for individual task areas to roles. Different pre-defined roles that you can use to assign rights to your users are available. The tables below describe the individual rights of the different roles.

Further information: "List of roles", Page 603

Advantages of classification in roles:

- Simplified administration
- Different rights are compatible between different software versions of the control and different machine manufacturers.

User administration offers roles for the following tasks:

- **Operating system roles:** access to functions of the operating system and interfaces
- **NC operator roles:** access to functions for programming, setting up and running NC programs
- **Machine tool builder (PLC) roles:** access to functions for configuring and checking the control

Every user should have at least one role from the operating system area and at least one role from the programming area.

HEIDENHAIN recommends permitting more than one person to access an account with the HEROS.Admin role. This ensures that necessary changes to user administration can also be made in the administrator's absence.

Local or remote registration

You can enable a role either for local login or for remote login. With local login, the user directly logs on to the control at the control's screen. A remote login (DNC) is a connection via SSH.

Further information: "SSH-secured DNC connection", Page 572

If a role is only enabled for local login, "Local." is added to the role name (e.g., Local.HEROS.Admin instead of HEROS.Admin).

If a role is only enabled for remote login, "Remote." is added to the role name (e.g., Remote.HEROS.Admin instead of HEROS.Admin).

You can therefore also make the rights of a user dependent on the access used to operate the control.

Rights

The user administration is based on the Unix rights management. Access to the control is controlled by means of rights.

Rights gather various functions of the control (e.g., editing the tool table).

User administration offers rights for the following tasks:

- HEROS rights
- NC rights
- PLC rights (machine manufacturer)

If more than one role is assigned to a user, he will be granted all rights contained in these roles.



Ensure that every user is assigned all access rights he needs. The access rights result from the tasks a user performs on the control.

The access rights of HEIDENHAIN function users are already pre-defined in the control's factory default setting.

Further information: "List of rights", Page 606

Password settings

If you use an LDAP database, users with the HEROS.Admin role can define password requirements. For this, the control provides the **Password settings** tab.

Further information: "Saving user data", Page 560

The following parameters are available:

Password lifetime

- **Validity period of password:**

Here, you can indicate how long the password can be used.

- **Warning before expiration:**

From the defined time, a warning will be issued that the password will soon expire.

Password quality

- **Minimum password length:**

Here, you can indicate the minimum password length.

- **Minimal number of character classes (upper/lower, digits, special):**

Here, you can indicate the minimum number of different character classes required in the password.

- **Maximum number of repeated characters:**

Here, you can indicate the maximum number of identical successive characters in the password.

- **Maximum length of character sequences:**

Here, you can indicate the maximum length of the character sequences to be used in the password (e.g., 123).

- **Dictionary check (number of matching characters):**

Here, you can enable a check whether the password contains known words and specify the allowed number of meaningful characters.

- **Minimum number of characters changed compared to previous password:**

Here, you can specify how many characters in the new password must be different from the previous one.

You define the values for each parameter on a scale.

For reasons of security, passwords should comply with the following criteria:

- Eight characters minimum
- Letters, numbers, and special characters
- Avoid using whole words or a sequence of characters (e.g., Anna or 123)



If you want to use special characters, pay attention to the keyboard layout. HEROS assumes a US keyboard, the NC software assumes a HEIDENHAIN keyboard. External keyboards can be freely configured.

Additional directories

HOME: drive

When user administration is active, a private **HOME:** directory, to which you can save your private programs and files, is available to every user.

The **HOME:** directory can be viewed by the respectively logged-in users as well as users with the HEROS.Admin role.

public directory

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

The **public** directory can be accessed by any user.

In the **public** directory you can, for example, make files available to other users.

24.1.1 Configuring user administration

User administration needs to be configured before you can use it.

Perform the following steps for configuration:

- 1 Open the **User administration** window
- 2 Activating user administration
- 3 Defining the password for the **useradmin** function user
- 4 Setting up a database
- 5 Creating a new user



- You can exit the **User administration** window after each configuration step.
- If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

Open the User administration window

To open the **User administration** window:



- ▶ Select the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **UserAdmin**
- ▶ The control opens the **User administration** window in the **Settings** tab.

Further information: "The User administration window",
Page 558

Activating user administration

To activate user administration:

- ▶ Select **User administration active**
- ▶ The control shows the message **Enter password for user 'useradmin'**.
- ▶ Retain or reactivate the active status of the **Anonymize users in log data** function



- The purpose of the **Anonymize users in log data** function is data privacy; this function is active by default. While this function is active, user data in all log files of the control will be anonymized.
- If you exit the **User administration** window directly after having activated user administration, the control will prompt you for a restart once.
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.

Defining the password for the useradmin function user

If you are activating user administration for the first time, you must define a password for the **useradmin** function user.

Further information: "Users", Page 550

To define a password for the **useradmin** function user:

- ▶ Select **Password for useradmin**
- ▶ The control opens the **Password for user 'useradmin'** pop-up window.
- ▶ Enter the password for the **useradmin** function user



Please observe the recommendations for passwords.

Further information: "Password settings", Page 553

- ▶ Repeat the password
- ▶ Select **Set new password**
- ▶ The control shows the message **Settings were changed**.

Setting up a database

To set up a database:

- ▶ Select the database for saving your user data (e.g., **Local LDAP database**)
- ▶ Select **Configuration**
- ▶ The control opens a window for configuring the corresponding database.
- ▶ Follow the instructions from the control in the window
- ▶ Select **OK**



The following options are available for saving your user data:

- **Local LDAP database**
- **LDAP on remote computer**
- **Connection to Windows domain**

Parallel operation of Windows users and users from an LDAP database is possible.

Further information: "Saving user data", Page 560

Creating a new user

To create a new user:

- ▶ Select the **User administration** tab
- ▶ Select the **Editing on** checkbox
- > The control opens the **Enter password of LDAP database** window.
- ▶ Enter password
- ▶ Select **OK**
- ▶ Select **Create new user**
- > The control adds a new user to the **User list**.
- ▶ Change the name as needed
- ▶ Edit a password as needed
- ▶ Define a profile image as needed
- ▶ Enter a description as needed
- ▶ Select **Add role**
- > The control opens the **Add role** window.
- ▶ Select a role
- ▶ Select **Add**



You can also add roles using the **Add external login** and **Add local login** buttons.

Further information: "Roles", Page 552

- ▶ Select **Close**
- > The control closes the **Add role** window.
- ▶ Select **Apply**
- > The control adopts the changes.
- ▶ Select **Exit**
- > The control opens the **System reboot required** window.
- ▶ Select **Yes**
- > The control restarts.



The user must change the password when logging in for the first time.

24.1.2 Deactivating user administration

User administration can be deactivated only by the following function users:

- **useradmin**
- **OEM**
- **SYS**

Further information: "Users", Page 550

To deactivate user administration:

- ▶ Log in as a function user
- ▶ Open the **User administration** window
- ▶ Select **User administration inactive**
- ▶ If desired, check **Delete existing user databases** to delete all configured users and user-specific directories
- ▶ Select **Apply**
- ▶ Select **Exit**
- > The control opens the **System reboot required** window.
- ▶ Select **Yes**
- > The control restarts.

Notes

NOTICE

Caution: Unwanted data transfer is possible!

If you deactivate the **Anonymize users in log data** function, the system will show personalized user data in all control log files.

If servicing becomes necessary or if the log files need to be transmitted for another reason, the contracting party will be able to view this user data. In this case, it is your responsibility to ensure that all required data protection provisions have been made at your company.

- ▶ Retain or reactivate the active status of the **Anonymize users in log data** function

- Some user administration areas are configured by the machine manufacturer. Refer to your machine manual.
- HEIDENHAIN recommends activating user administration as part of an IT safety concept.
- If both user administration and a screensaver are active, then the current user's password must be entered to unlock the screen.

Further information: "HEROS menu", Page 579

- If you used **Remote Desktop Manager** to establish private connections before user administration was activated, these connections are no longer available after the activation of user administration. Save your private connections before activating user administration.

Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522

24.2 The User administration window

Application

In the **User administration** window you can activate and deactivate user administration, as well as define its settings.

Related topics

- The **Active user** window
Further information: "The Active user window", Page 559

Requirement

- If user administration is active, the HEROS.Admin role
Further information: "List of roles", Page 603

Description of function

To navigate to this function:

Settings ► Operating System ► UserAdmin

The **User administration** window contains the following tabs:

Tab	Meaning
Settings	Configure user administration Further information: "Configuring user administration", Page 554
User administration	Create or remove users, change rights, add profile images Further information: "Creating a new user", Page 556
Password settings	Define password requirements Further information: "Password settings", Page 553

24.3 The Active user window

Application

In the **Active user** window, the control displays information about the logged on user, such as assigned rights. You can also manage other user settings, such as keys for SSH-secured DNC connections or smartcards for logon, and change the password.

Related topics

- SSH-secured DNC connections
Further information: "SSH-secured DNC connection", Page 572
- Logon with smartcards
Further information: "Logon with smartcards", Page 568
- Available roles and rights
Further information: "User administration roles and rights", Page 603

Description of function

To navigate to this function:

Settings ► Operating System ► Current User

When you open the **Active user** window, by default the window shows the **Base rights** tab. On this tab the control displays information about the user and all assigned roles and rights.

The **Base rights** tab contains the following buttons:

Button	Meaning
Add rights	On the Added rights tab, enable rights for another user or function user until the next logoff
Open user administration	Open the User administration window Further information: "The User administration window", Page 558
SSH keys and certificates	Manage keys and certificates for client connections Further information: "SSH-secured DNC connection", Page 572 Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505
Create token	Manage smartcards for logon with a card reader Further information: "Logon with smartcards", Page 568
Delete token	
Close	Close the Active user window

On the **Change password** tab you can check your password against the current requirements or set a new password.

Further information: "Password settings", Page 553

Note

In legacy mode, the **user** function user is automatically logged on to the system during control startup. When user administration is active, the **user** function user has no effect.

Further information: "Users", Page 550

24.4 Saving user data

24.4.1 Overview

The following options are available for saving your user data:

- **Local LDAP database**

Further information: "Local LDAP database", Page 560

- **LDAP on remote computer**

Further information: "LDAP database on a remote computer", Page 561

- **Connection to Windows domain**

Further information: "Connection to Windows domain", Page 562



Parallel operation of Windows users and users from an LDAP database is possible.

24.4.2 Local LDAP database

Application

With the **Local LDAP database** setting the control saves the user data locally. That way you can activate user administration even on machines without a network connection.

Related topics

- Using an LDAP database on multiple controls

Further information: "LDAP database on a remote computer", Page 561

- Connecting a Windows domain with user administration

Further information: "Connection to Windows domain", Page 562

Requirements

- User administration is active

Further information: "Activating user administration", Page 555

- **useradmin** user is logged on

Further information: "Users", Page 550

Description of function

A local LDAP database offers the following options:

- Using user administration on one single control
- Setting up a central LDAP server for more than one control
- Exporting an LDAP server configuration file if the exported database is to be used by more than one control

Setting up a Local LDAP database

To set up a **Local LDAP database**:

- ▶ Open the **User administration** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Configuration**
- > The control opens the **Configure local LDAP database** window.
- ▶ Enter the **Name of the LDAP domain**:
- ▶ Enter the password
- ▶ Repeat the password
- ▶ Select **OK**
- > The control closes the **Configure local LDAP database** window.

Notes

- Before you can start editing the user administration, the control prompts you to enter the password of your local LDAP database.
Passwords must not be trivial and must be known only to the administrators.
- If the host name or domain name of the control changes, you need to reconfigure the local LDAP databases.

24.4.3 LDAP database on a remote computer

Application

With the **LDAP on remote computer** function you can transmit the configuration of a local LDAP database between controls and computers. That way you can use the same users on multiple controls.

Related topics

- Configuring an LDAP database on a control
Further information: "Local LDAP database", Page 560
- Connecting a Windows domain with user administration
Further information: "Connection to Windows domain", Page 562

Requirements

- User administration is active
Further information: "Activating user administration", Page 555
- **useradmin** user is logged on
Further information: "Users", Page 550
- LDAP database has been set up in the company network
- Server configuration file of an existing LDAP database is stored on the control or a PC in the network
If the configuration file is stored on a PC, the PC must be running and accessible through the network.
Further information: "Providing a server configuration file", Page 562

Description of function

The **useradmin** function user can export the server configuration file of an LDAP database.

Providing a server configuration file

To provide a server configuration file:

- ▶ Open the **User administration** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **Local LDAP database**
- ▶ Select **Export server configuration**
- > The control opens the **Export LDAP configuration file window**.
- ▶ Enter the name for the server configuration file into the name field
- ▶ Save the file to the desired folder
- > The control exports the server configuration file.

Setting up LDAP on remote computer

To set up **LDAP on remote computer**:

- ▶ Open the **User administration** window
- ▶ Select **LDAP user database**
- > The control enables the dimmed area for editing the LDAP user database.
- ▶ Select **LDAP on remote computer**
- ▶ Select **Import server configuration**
- > The control opens the **Import LDAP configuration file window**.
- ▶ Select the existing configuration file
- ▶ Select **Open**
- ▶ Select **Apply**
- > The control imports the configuration file.

24.4.4 Connection to Windows domain

Application

With the **Connection to Windows domain** function, you can connect the data of a domain controller with the control's user administration.

Ask your IT administrator to configure the connection to the Windows domain.

Related topics

- Configuring an LDAP database on a control
Further information: "Local LDAP database", Page 560
- Using an LDAP database on multiple controls
Further information: "LDAP database on a remote computer", Page 561

Requirements

- User administration is active
Further information: "Activating user administration", Page 555
- **useradmin** user is logged on
Further information: "Users", Page 550
- Windows domain controller present in the network
- Domain controller accessible in the network
- Organizational unit for HEROS roles known
- Function user is defined in the organization
- User name and password of the function user are known

Description of function

Your IT administrator sets up a function user for connecting to the Windows domain.

Further information: "Joining a Windows domain", Page 566

Buttons

The **Connection to Windows domain** area provides the following buttons:

Button	Meaning
Configuration	The control opens the Configure Windows domain with function user window. Further information: "The Configure Windows domain with function user window", Page 564
Find domain	The control selects a Windows domain.
Export the Windows config.	Once you have connected the control to the Windows domain, you can export the configurations for other controls. Further information: "Exporting and importing a Windows configuration file", Page 566
Import the Windows config.	Using a present configuration, you can connect the control easily and quickly to the Windows domain. Further information: "Exporting and importing a Windows configuration file", Page 566
Check missing role definitions	The control checks whether all of the required roles have been created in the Windows domain.
Add role definition	If any roles required in the Windows domain are missing, you can add the missing roles. Further information: "Groups of the domain", Page 565

The Configure Windows domain with function user window

After the domain search, you can customize the Windows domain information or specify new information in the **Configure Windows domain with function user** window.

Your IT administrator will provide the required information.

The **Configure Windows domain with function user** window provides the following settings:

Setting	Meaning
Domain name:	Server name of the Windows domain Is populated by domain search
Key Distribution Center (KDC):	KDC address Is populated by domain search
Alternative admin server:	Deviating server name where the passwords are managed
Map SIDs to Unix UIDs	Map the Windows user SIDs (Security IDs) in Active Directory to the matching Unix UIDs on the control
Use LDAPs	Transfer data using secure LDAPs LDAPs encrypt user data and passwords. You can select a certificate or disable certificate validation.
Group for login authorization:	Define a special group of Windows users to whom you want to restrict the connection to this control
Organizational unit for HEROS roles:	Modify the organizational unit in which the HEROS role names are stored Specify the configuration of your domain.
Prefix for HEROS role names:	Change the prefix in order to manage users from different workshops, for example. Each prefix given to a HEROS role name can be changed (e.g., HEROS hall 1 and HEROS hall 2) Is populated by domain search
Separator for HEROS role names:	Modify the separator within the HEROS role names
Function user:	User name and password of the Active Directory function user
Organizational unit for function user:	Organizational unit of the function user
Advanced configuration of domain section	Only for IT administrators

The function user's user name must not contain blanks. The name and organizational unit form the complete path (Distinguished Name, DN) in the Active Directory.

Groups of the domain

If not all of the required roles have been created in the domain as groups, the control issues a warning.

If the control issues a warning, proceed in one of the two following ways:

- Use the **Add role definition** function to enter a role directly in the domain
- Use the **Export role definition** function to export the roles to an *.ldif file

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

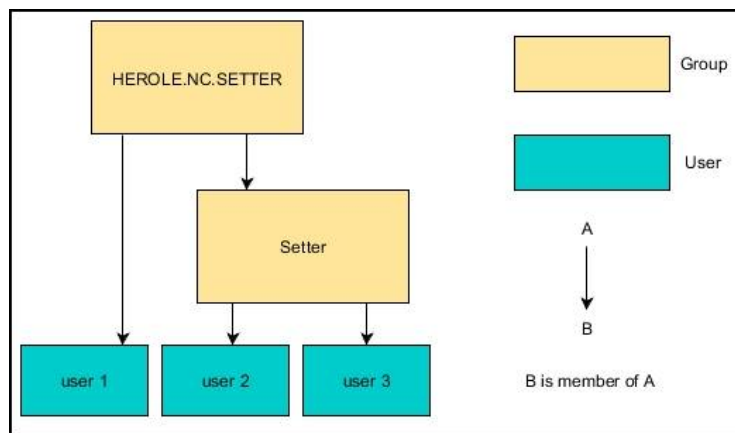
- Automatically when entering the Windows domain by specifying a user with administrator rights
- By importing an import file in .ldif format to the Windows server

The Windows administrator must add the users manually to the roles (security groups) on the domain controller.

Two suggestions describing how the groups can be structured by the Windows administrator are given by below.

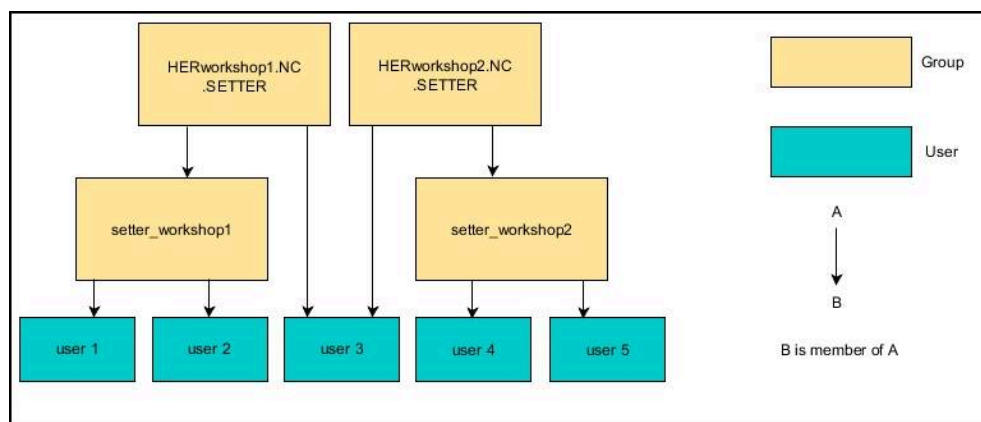
Example 1

The user is a direct or indirect member of the respective group:



Example 2

Users from various sectors (workshops) are members of groups with different prefixes:



Joining a Windows domain

To join a Windows domain:

- ▶ Open the **User administration** window
- ▶ Select **Connection to Windows domain**
- ▶ Select **Find domain**
- > The control selects a domain.
- ▶ Select **Configuration**
- ▶ Check the data for **Domain name:** and **Key Distribution Center (KDC):**
- ▶ Enter **Organizational unit for HEROS roles:**
- ▶ Enter the user name and password of the function user
- ▶ Press **OK**
- ▶ Select **Apply**
- > The control connects to the Windows domain found.
- > The control checks whether all of the required roles have been created in the domain as groups.

Exporting and importing a Windows configuration file

If you have connected the control to the Windows domain, you can export the required configurations for other controls.

To export the Windows configuration file:

- ▶ Open the **User administration** window
- ▶ Select **Connect to Windows domain**
- ▶ Select **Export the Windows config.**
- > The control opens the **Export the Windows domain configuration** window.
- ▶ Select the directory for the file
- ▶ Enter the name for the file
- ▶ Select the **Export the function user's password?** check box, if required
- ▶ Select **Export**
- > The control saves the Windows configuration as a BIN file.

To import the Windows configuration file of another control:

- ▶ Open the **User administration** window
- ▶ Select **Connect to Windows domain**
- ▶ Select **Import the Windows config.**
- > The control opens the **Import the Windows domain configuration** window.
- ▶ Select the existing configuration file
- ▶ Select the **Import the function user's password?** check box, if required
- ▶ Select **Import**
- > The control adopts the configurations for the Windows domain.

24.5 Autologin with user administration

Application

If the **Autologin** function is enabled, during startup the control automatically logs on a selected user without the need to enter a password.

As opposed to the **legacy mode**, this enables you to restrict a user's rights without entering a password.

Related topics

- User login
Further information: "Logging on with user administration", Page 568
- Configuring user administration
Further information: "Configuring user administration", Page 554

Requirements

- User administration has been configured
- The user for **Autologin** has been defined

Description of function

With the **Enable autologin** check box in the **User administration** window, you can define a user for autologin.

Further information: "The User administration window", Page 558

The control then automatically logs this user on and displays the user interface according to the defined rights.

For further authorizations, the control still requires an authentication to be entered.

Further information: "Window for requesting additional rights", Page 571

24.6 Logging on with user administration

Application

The control displays a dialog window for user logon. Within the dialog the user can log on with a password or a smartcard.

Related topics

- Automatic user logon

Further information: "Autologin with user administration", Page 567

Requirements

- User administration has been configured
- For logon with smartcards:
 - Euchner EKS card reader
 - Smartcard assigned to a user

Further information: "Assigning a smartcard to a user", Page 569

Description of function

The control displays the Login dialog in the following cases:

- After the **User logout** function
- After the **Switch users** function
- After the **Lock display** function
- Immediately after control startup if user administration is active and **Autologin** is not enabled

When user administration is active, the control provides these functions in the **Start/Login** application and in the HEROS menu.

Further information: "HEROS menu", Page 579

The logon dialog gives you the following options:

- Users who logged in at least once
- **Other** user

Logon with smartcards

You can save a user's logon data on a smartcard and then log the user on with a card reader, without needing to enter a password. You can define whether a PIN is necessary for logon.

The card reader is attached over a USB port. You assign the smartcard to a reader as a token.

Further information: "Assigning a smartcard to a user", Page 569

The smartcard also has additional memory space, where the machine manufacturer can store his own user-specific data.

24.6.1 Logging on a user with password

To logon a user the first time:

- ▶ Select **Other** in the login dialog
- > The control enlarges the user icon you selected.
- ▶ Enter the user name
- ▶ Enter the user's password



The control shows in the Login dialog whether CAPS LOCK is active.

- > The control opens a window with the message **Password expired. Change the password now.**
- ▶ Enter the current password
- ▶ Enter a new password
- ▶ Repeat the new password
- > The control uses the new user to log you in.
- > The control displays this user in the dialog during the next logon procedure.

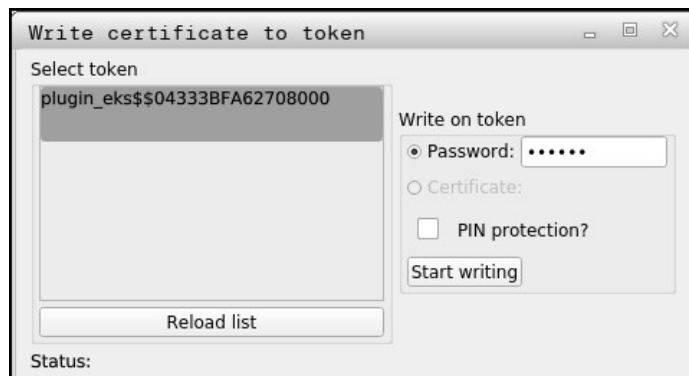
24.6.2 Assigning a smartcard to a user

To assign a smartcard to a user:

- ▶ Insert a blank smartcard in the card reader
- ▶ Logon the desired smartcard user in user administration



- ▶ Select the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- ▶ Select **Create token**
- > The control opens the **Write certificate to token** window.
- > The control displays the smartcard in the **Select token** area.
- ▶ Select the smartcard as the token to be written
- ▶ Activate the **PIN protection?** check box, if desired
- ▶ Enter user password (and PIN, if desired)
- ▶ Select **Start writing**
- > The control saves the user's logon data on the smartcard.



Notes

- You must restart the control in order for it to detect a card reader.
- You can overwrite smartcards that already contain information.
- If you change a user's password, you must reassign the smartcard.

24.7 Window for requesting additional rights

Application

If you do not have the rights required for a specific **HEROS menu** item, the control opens the window for requesting additional rights.

In this window, you can temporarily obtain more rights by adding another user's rights.

Related topics

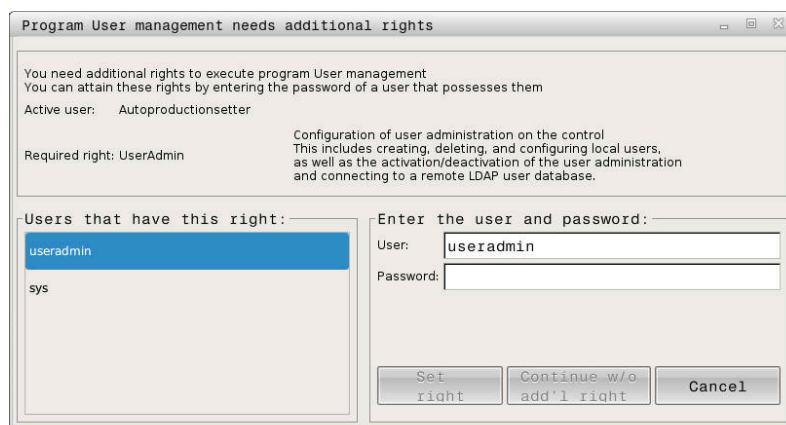
- Temporarily granting additional rights in the **Active user** window

Further information: "The Active user window", Page 559

Description of function

In the **Users that have this right:** field, the control lists all existing users that have the right to use this function.

You must enter the password in order to enable user rights.



Window for requesting additional rights

To attain the rights of users that are not shown, enter their user data. The control will then recognize those users that are contained in the user database.

Notes

- If **Connection to Windows domain** is used, only users that were recently logged on are shown in the selection menu.
- You can't use this window to change user administration settings. The user with the HEROS.Admin role must be logged on in order to do so.

24.8 SSH-secured DNC connection

Application

If user administration is active, external applications also need to authenticate a user so that the suitable rights can be assigned.

For DNC connections using the RPC or LSV2 protocol, the connection is routed through an SSH tunnel. This method assigns the remote user to a user set up on the control, granting the remote user this user's rights.

Related topics

- Forbidding non-secure connections
Further information: "Firewall", Page 528
- Roles for remote logon
Further information: "Roles", Page 552

Requirements

- TCP/IP network
- The remote computer acts as SSH client
- The control acts as SSH server
- Key pair consisting of
 - Private key
 - Public key

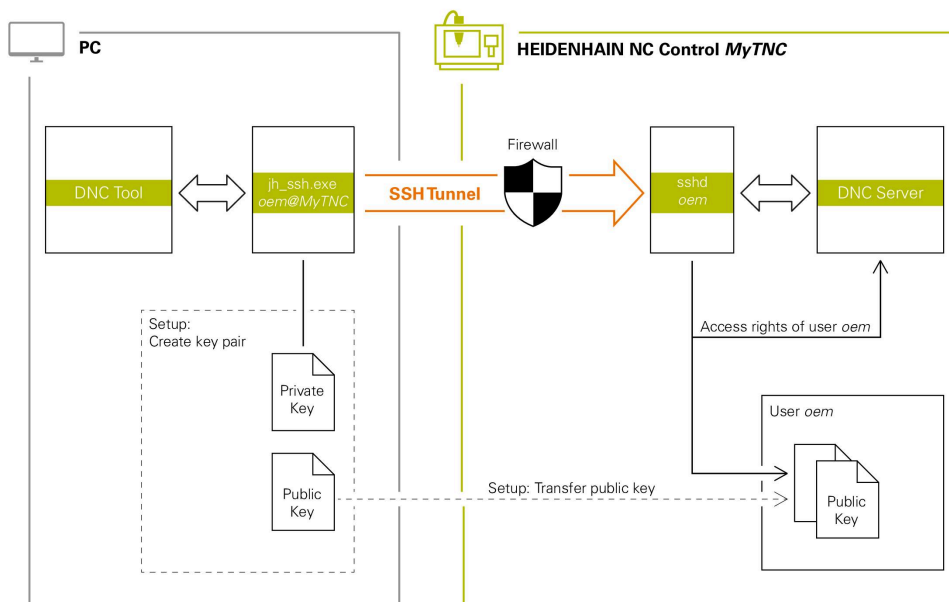
Description of function

Concept of transmission through an SSH tunnel

An SSH connection is always set up between an SSH client and an SSH server.

A key pair is used to protect the connection. This key pair is generated on the client. The key pair consists of a private key and a public key. The private key remains with the client. During setup, the public key is transferred to the server and assigned to a certain user.

The client tries to connect to the server using the pre-defined user name. The server can use the public key to verify that the requester of the connection holds the associated private key. If yes, the server accepts the SSH connection and assigns it to the user that has been used for the login. Communication can then be "tunneled" through this SSH connection.



Use in external applications

The PC tools available from HEIDENHAIN, such as TNCremo with version **v3.3** or higher, provide all functions for setting up, establishing, and managing secure connections through an SSH tunnel.

When the connection is set up, the required key pair is generated in TNCremo and the public key is transferred to the control.

This also applies to applications that are using the HEIDENHAIN DNC component from RemoTools SDK for communication. There is no need to adapt existing customer applications.



In order to expand the connection configuration using the associated **CreateConnections** tool, you need to update to **HEIDENHAIN DNC v1.7.1**. A modification of the application source code is not required.

24.8.1 Setting up SSH-secured DNC connections

To set up an SSH-secured DNC connection for the logged-on user:



- ▶ Select the **Settings** application
- ▶ Select **Network/Remote Access**
- ▶ Select **DNC**
- ▶ Activate the **Setup permitted** toggle switch
- ▶ Use **TNCremo** to set up the secure connection (TCP secure).



For details, refer to the integrated help system of TNCremo.

- > TNCremo transmits the public key to the control.



In order to ensure maximum security, deactivate the **Allow password authentication** function after the public key has been stored.

- ▶ Deactivate the **Setup permitted** toggle switch

24.8.2 Removing a secure connection

If you delete a private key from the control, that user no longer has the possibility of a secure connection.

To delete a key:



- ▶ Select the **Settings** application
- ▶ Select **Operating System**
- ▶ Double-tap or double-click **Current User**
- > The control opens the **Active user** window.
- ▶ Select **Certificate and keys**
- ▶ Select the key to be deleted
- ▶ Select **Delete SSH key**
- > The control deletes the selected key.

Notes

- The encryption used with the SSH tunnel protects the communication from attackers.
- For OPC UA connections, a stored user certificate is used for authentication.
Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505
- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections.
If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- Once the connection configurations have been set up, they can be shared among all HEIDENHAIN PC tools for establishing a connection.
- You can also transfer a public key to the control by using a USB device or network drive.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

25

**HEROS operating
system**

25.1 Fundamentals

HEROS is the fundamental basis for all NC controls from HEIDENHAIN. The HEROS operating system is based on Linux, and was adapted for the purposes of NC controls.

The TNC7 basic features the version HEROS 5.

25.2 HEROS menu

Application

In the HEROS menu the control shows information about the operating system. You can change settings or use HEROS functions.

By default you open the HEROS menu through the taskbar at the bottom edge of the screen.

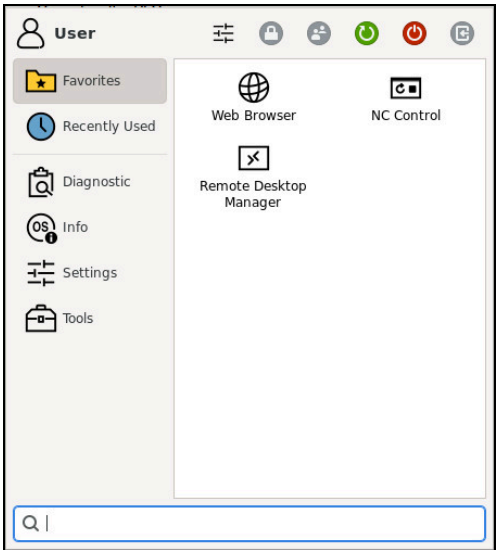
Related topics

- Opening HEROS functions through the **Settings** application
Further information: "The Settings application", Page 469

Description of function

You open the HEROS menu with the green DIADUR icon in the task bar or with the **DIADUR** key.

Further information: "Taskbar", Page 582



Standard view of the HEROS menu

The HEROS menu contains the following functions:

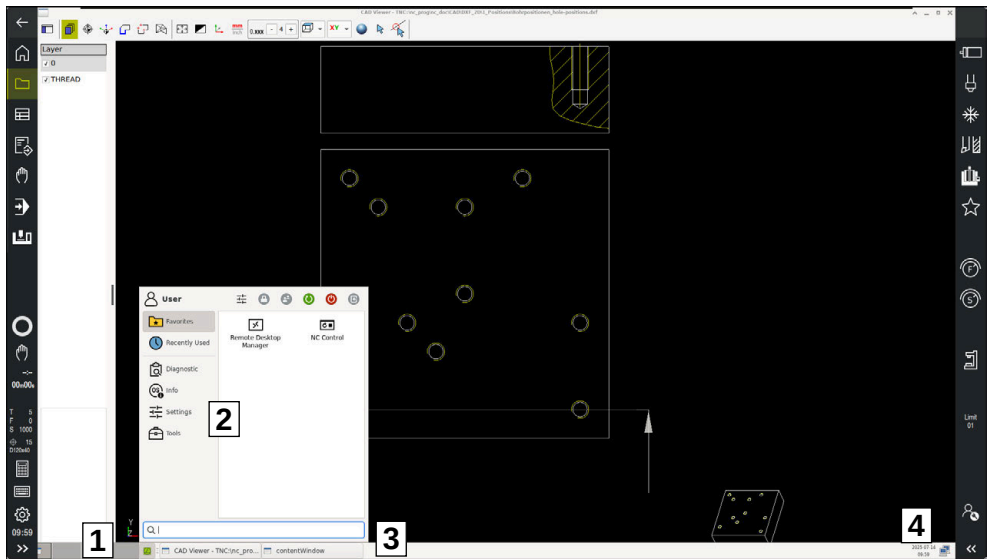
Area	Function
Header	■ User name Further information: "The Active user window", Page 559
	■ User-specific settings
	■ Lock display Only if user administration is active
	■ Switch users Only if user administration is active
	■ Restart
	■ Shut down
	■ Log out Only if user administration is active
	Further information: "User administration", Page 549

Area	Function
Navigation	■ Favorites ■ Recently used
Diagnostic	■ GSmartControl: Available only to authorized specialists ■ HeLogging: Define settings for internal diagnostic files ■ ITC VNC: Display the screen contents of the additional operating station (ITC) ■ perf2: Check processor load and process load ■ Portscan: Test active connections Further information: "Portscan", Page 533 ■ Portscan OEM: Available only to authorized specialists ■ Terminal: Enter and execute console commands ■ TNCdiag: Evaluates status and diagnostic information of HEIDENHAIN components with a focus on the drives and presents it graphically Further information: "TNCdiag", Page 537 ■ TNCscope: Available only to authorized specialists
Settings	■ Adjust screen brightness: Adjust screen brightness ■ Screensaver: Screensaver ■ Current User Further information: "The Active user window", Page 559 ■ Date/Time Further information: "The Adjust system time window", Page 488 ■ Firewall Further information: "Firewall", Page 528 ■ Language/Keyboards Further information: "Conversational language of the control", Page 489 ■ Network Further information: "Ethernet interface", Page 496 ■ OEM Function Users Further information: "Users", Page 550 ■ OPC UA NC Server Connection Assistant Further information: "The OPC UA connection assistant function (#56-61 / #3-02-1*)", Page 511 ■ OPC UA NC Server License Further information: "The OPC UA license settings function (#56-61 / #3-02-1*)", Page 512 ■ PKI Admin: Manage certificates for the control, such as for OPC UA NC Server Further information: "OPC UA NC Server (#56-61 / #3-02-1*)", Page 505 ■ Printer Further information: "Printers", Page 516 ■ Reset Touchscreen Calibration ■ Screenshot Config In the Screenshot settings window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).

Area	Function
	■ SELinux Further information: "SELinux security software", Page 491 ■ Shares Further information: "Network drives on the control", Page 492 ■ Touchscreen Calibration ■ Touchscreen Configuration ■ UserAdmin Further information: "The User administration window", Page 558 ■ VNC Further information: "The VNC menu item", Page 519 ■ WindowManagerConfig: Settings for the Window Manager Further information: "Window Manager", Page 583
Info	■ About HeROS: Open information about the operating system of the control ■ About Xfce: Open information on the Window manager
Tools	■ Switch-off: Shut-down or restart ■ Screenshot: Create screenshots ■ File Manager: Available only to authorized specialists ■ Document Viewer: Display and print files (e.g., PDF files) ■ Geeqie: Open, manage, and print graphics ■ Gnumeric: Open, edit, and print tables ■ Hostkey: Show a unique ASCII image to identify the control ■ IDS Camera Manager: Manage cameras connected to the control ■ keypad horizontal: Open virtual keyboard ■ keypad vertical: Open virtual keyboard ■ Leafpad: Open and edit text files ■ Meld: Compare files ■ Mozilla Firefox: Start the browser ■ NC Control: Start or stop the NC software independently of the operating system ■ NC/PLC Backup Further information: "Backup and restore", Page 534 ■ NC/PLC Restore Further information: "Backup and restore", Page 534 ■ Real VNC Viewer: Connect to a remote device via a VNC server Available only to network specialists ■ Remote Desktop Manager Further information: "The Remote Desktop Manager window (#133 / #3-01-1)", Page 522 ■ Ristretto Image Viewer: Open graphics ■ Secure Remote Access Further information: "Secure Remote Access", Page 590

Area	Function
	■ Combine fixtures Further information: "Combining fixtures in the New Fixture window", Page 252 ■ Virtual keyboard ■ Touchscreen Cleaning ■ Web Browser: Start the browser ■ Xarchiver: Extract or compress directories
Search	Full-text search of individual functions

Taskbar



CAD Viewer opened in the third desktop with taskbar shown and active HEROS menu

The taskbar consists of the following areas:

- 1 Workspaces
- 2 HEROS menu
 - Further information:** "Description of function", Page 579
- 3 Opened applications, e.g.:
 - Control interface
 - **CAD Viewer**
 - Window of HEROS functions

You can move the opened applications into any other workspaces.
- 4 Widgets
 - Calendar
 - Status of the firewall
 - Further information:** "Firewall", Page 528
 - Network status
 - Further information:** "Ethernet interface", Page 496
 - Notifications
 - Shut down or restart the operating system

Window Manager

With the Window Manager, you manage functions of the HEROS operating system as well as windows opened in the third desktop, such as **CAD Viewer**.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces.

The following functions are possible with the window manager:

- Display a taskbar for switching between various applications (user interfaces)
- Manage an additional desktop, on which special applications from your machine manufacturer can run
- Control the focus between NC software applications and those of the machine manufacturer
- You can change the size and position of pop-up windows. It is also possible to close, minimize and restore pop-up windows

If a window is opened in the third desktop, the control displays the **Window Manager** icon in the information bar. You can switch between the open applications by selecting the icon.

You can minimize the control's user interface by pulling down from the information bar. The TNC bar and the OEM bar remain visible.

Further information: "Areas of the control's user interface", Page 83

Notes

- If a window is opened in the third desktop, the control displays an icon in the information bar.

Further information: "Areas of the control's user interface", Page 83

- The machine manufacturer determines the scope of function and behavior of the window manager.
- The control shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.

25.3 Serial data transfer

Application

The TNC7 basic automatically uses the LSV2 transmission protocol for serial data transfer. All parameters of the LSV2 protocol are invariably fixed except for the baud rate in the machine parameter **baudRateLsv2** (no. 106606).

Description of function

The machine parameter **RS232** (no. 106700) allows you to define another transmission type (interface). The settings described below are effective only for the respective newly defined interface.

Further information: "Machine parameters", Page 540

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate) Input: BAUD_110, BAUD_150, BAUD_300, BAUD_600, BAUD_1200, BAUD_2400, BAUD_4800, BAUD_9600, BAUD_19200, BAUD_38400, BAUD_57600, BAUD_115200
protocol (no. 106702)	Data transfer protocol ■ STANDARD: Standard data transfer, line-by-line ■ BLOCKWISE: Packet-based data transfer ■ RAW_DATA: Transfer without protocol (purely character-by-character) Input: STANDARD, BLOCKWISE, RAW_DATA
dataBits (no. 106703)	Data bits in each transferred character Input: 7 Bit, 8 Bit
parity (no. 106704)	Parity bit used to check for transmission errors ■ NONE: No parity, no error detection ■ EVEN: Even parity, error if the number of bits set is odd ■ ODD: Odd parity, error if the number of bits set is even Input: NONE, EVEN, ODD
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize to each transmitted character during serial data transmission. Input: 1 Stop-Bit, 2 Stop-Bits
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking. ■ NONE: No data-flow check ■ RTS_CTS: Hardware handshaking, transmission stop is active through RTS ■ XON_XOFF: Software handshaking, transmission stop is active through DC3 Input: NONE, RTS_CTS, XON_XOFF
fileSystem (no. 106707)	File system for the serial interface ■ EXT: Minimum file system for printers or non-HEIDENHAIN transmission software ■ FE1: Communication with TNCserver or an external floppy disk unit If you require no special file system, this machine parameter is not needed. Input: EXT, FE1

Machine parameters	Setting
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. ■ TRUE: The BCC does not correspond to any control character ■ FALSE: Function not active Input: TRUE, FALSE
rtsLow (no. 106709)	This optional parameter determines the level of the RTS line in the idle state. ■ TRUE: Level is LOW in idle state ■ FALSE: Level is HIGH in idle state Input: TRUE, FALSE
noEotAfterEtx (no. 106710)	This optional parameter sets whether an EOT character (End of Transmission) is to be transmitted after receiving an ETX character (End of Text). ■ TRUE: The EOT character is not sent ■ FALSE: The EOT character is sent Input: TRUE, FALSE

Example

In order to use the TNCserver PC software for data transfer, define the following settings in the machine parameter **RS232** (no. 106700):

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

TNCserver is part of the TNCremo software for PCs.

Further information: "PC software for data transfer", Page 586

25.4 PC software for data transfer

Application

HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Related topics

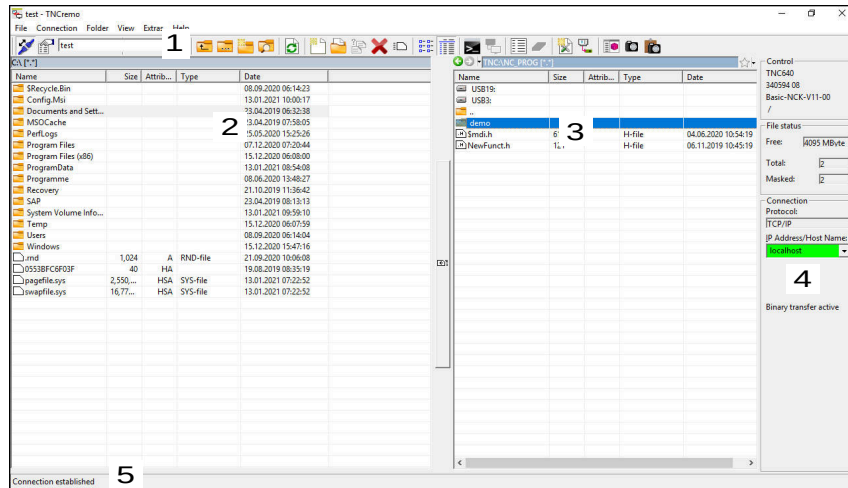
- Comparison of the transmission duration of different protocols
Further information: "Example: Transmission duration of different transmission types", Page 593

Requirements

- PC operating system:
 - Windows 10
 - Windows 11
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- A network connection to the control

Description of function

The TNCremo data transfer software provides the following areas:



- 1 Toolbar
This area provides the most important TNCremo functions.
- 2 File list of the PC
In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).
- 3 File list of control
In this area, TNCremo displays all of the folders and files of the connected drive of the control.
- 4 Status display
In the status display, TNCremo shows information about the current connection.
- 5 Connection status
The connection status indicates whether a connection is currently active.



For more information, refer to the integrated help system of TNCremo. You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- You can download the current version of the TNCremo software from the **HEIDENHAIN website** for free.

25.5 File transfer with SFTP (SSH File Transfer Protocol)

Application

SFTP (SSH File Transfer Protocol) provides a secure way to connect client applications to the control and to transfer files at high speed from a PC to the control. The connection is routed via an SSH tunnel.

Related topics

- User administration
Further information: "User administration", Page 549
- Principle of the SSH connection
Further information: "Concept of transmission through an SSH tunnel", Page 573
- Firewall settings
Further information: "Firewall", Page 528
- Comparison of the transmission duration of different protocols
Further information: "Example: Transmission duration of different transmission types", Page 593

Requirements

- PC software TNCremo with version 3.3 or higher is installed
Further information: "PC software for data transfer", Page 586
- SSH service is permitted in the firewall of the control
Further information: "Firewall", Page 528

Description of function

SFTP is a secure transmission protocol supported by various operating systems for client applications.

To set up the connection, you need a key pair consisting of a public and a private key. You transfer the public key to the control and assign it to a user through the user administration. The private key is required by the client application to set up a connection to the control.

HEIDENHAIN recommends using the CreateConnections application to generate the key pair. CreateConnections is installed together with the PC software TNCremo with version 3.3 and higher. CreateConnections lets you transfer the public key directly to the control and assign it to a user.

You can also use other software to generate the key pair.

25.5.1 Setting up an SFTP connection with CreateConnections

For an SFTP connection using CreateConnections, the following are required:

- Connection with secure protocol, such as **TCP/IP Secure**
- User name and password of the desired user are known



When you transfer the public key to the control, you must enter the user's password twice.

If user administration is inactive, the user **user** is logged in. The password for the user **user** is **user**.

To set up an SFTP connection:



- ▶ Select the **Settings** application
- ▶ Select **Network/Remote Access**
- ▶ Select **DNC**
- ▶ Activate the **Setup permitted** toggle switch
- ▶ Create a key pair with CreateConnections and transfer it to the control



For more information, refer to the integrated help system of TNCremo.

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

- ▶ Deactivate the **Setup permitted** toggle switch
- ▶ Transfer the private key to the client application
- ▶ Connect the client application to the control



Please refer to the manual of the client application.

Notes

- When user administration is active, you can set up only secure network connections via SSH or OPC UA (#56-61 / #3-02-1*). If non-secure network connections exist, you must set them up again as secure connections. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections.
- During the connection, the rights of the user to whom the used key is assigned are active. The directories and files displayed, as well as the access options, vary depending on the permissions.
- You can also transfer a public key to the control by using a USB device or network drive. In this case, you do not need to activate the **Allow password authentication** check box.
- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.

25.6 Secure Remote Access

Application

Secure Remote Access (SRA) allows you to set up an encrypted connection between a PC and your control via the Internet. SRA allows the control to be displayed and operated on a PC, such as for service trainings or remote maintenance.

Related topics

- VNC settings
Further information: "The VNC menu item", Page 519

Requirements

- Existing Internet connection
Further information: "Network configuration with Advanced Network Configuration", Page 596
- The following settings in the **VNC settings** window:
 - **Enable RemoteAccess and IPC** check box is active
 - In the **Enabling other VNC** area, the **Inquire** or **Permitted** check box is active**Further information:** "The VNC menu item", Page 519
- PC with paid RemoteAccess software including the extension **Secure Remote Access**

HEIDENHAIN website



For more information, refer to the integrated help system of RemoteAccess.
 You can open the context-sensitive help function of the RemoteAccess software by pressing the **F1** key.

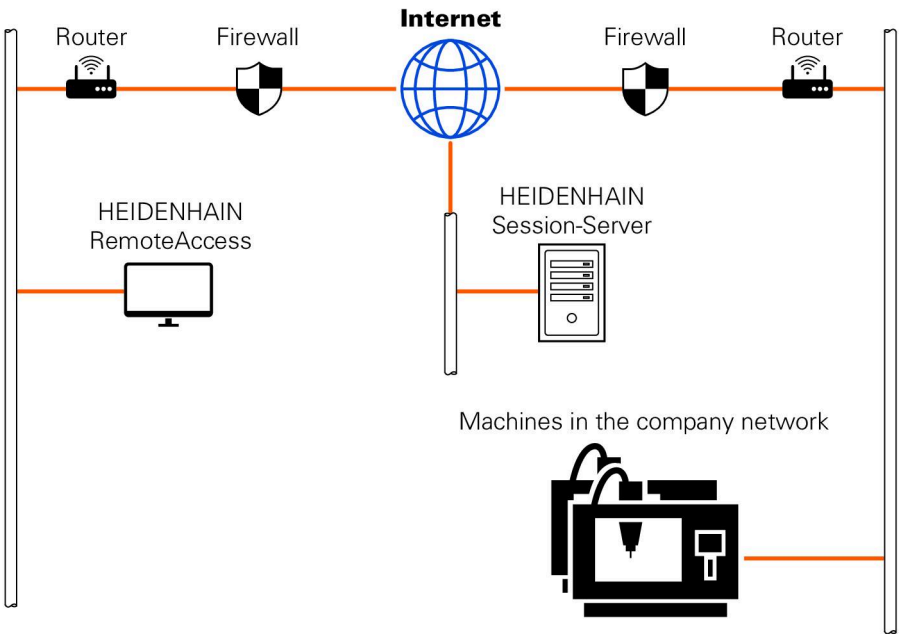
Description of function

To navigate to this function:

Tools ▶ Secure Remote Access

The PC provides a ten-digit session ID for you to enter in the **HEIDENHAIN Secure Remote Access** window.

SRA enables connection via a VPN server.



In the **Extended** area, the control shows the progress of the connection setup.
The **HEIDENHAIN Secure Remote Access** window provides the following buttons:

Button	Function
Mount	The control starts the connection with the entered session ID.
Update	The control manually searches for updates for SRA. The control automatically searches for available updates when you open the HEIDENHAIN Secure Remote Access window. If an update is available, you can install it. The control restarts during the update.
Config.	The control opens the Network settings window. Only for network specialists
Show log	The control opens the log files of the SRA.

Notes

If, in the **VNC settings** window, you set the **Enabling other VNC** setting to **Inquire**, you can permit or deny any connection.

25.7 Data backup

Application

If you create or modify files on the control, then you should back up these files periodically.

Related topics

- File management

Further information: Programming and Testing User's Manual

Description of function

With the functions **NC/PLC Backup** and **NC/PLC Restore** you can create back-up files for specific directories or even an entire drive, and restore them as needed. You should store these backup files on an external storage medium.

Further information: "Backup and restore", Page 534

You have the following options for transferring files from the control:

- TNCremo

With TNCremo you can transfer files from the control to a computer.

Further information: "PC software for data transfer", Page 586

- External drive

You can transfer files from the control directly to an external drive.

Further information: "Network drives on the control", Page 492

- External data carriers

You can back-up files to external data carriers or use external data carriers to transfer the files.

Further information: Programming and Testing User's Manual

Notes

- You should back-up all machine-specific data, such as the PLC program or machine parameters. Consult your machine manufacturer about this.
- You must transmit files with the extensions PDF, XLS, ZIP, BMP, GIF, JPG and PNG in binary format from the PC to the control's hard disk.
- Backing up all files of the internal memory can take several hours. If required, perform the backup during a time when you don't need the machine.
- Periodically delete files that are no longer required. This ensures that the control has enough memory available for system files, such as the tool table.
- HEIDENHAIN recommends having the hard disk inspected after three to five years. After this time, and depending on the operating conditions (e.g., vibration loads), you must expect increased failure rates.

25.7.1 Example: Transmission duration of different transmission types

The following table shows the measurement results regarding how long the different services take to transmit an NC program with a size of 1 GB to and from the control.

A CAD laptop and a TNC7 with an MC 356 main computer were used for the test.

Service	Duration of transmission from the control	Duration of transmission to the control	Further information
TNCremo v3	Approx. 3 min 30 s	Approx. 14 min 30 s	Page 586
TNCremo v4	Approx. 0 min 12 s	Approx. 0 min 12 s	
HEIDENHAIN DNC	Approx. 0 min 12 s	Approx. 0 min 12 s	Page 513
OPC UA NC Server	Approx. 0 min 27 s	Approx. 0 min 50 s	Page 505
SFTP	Approx. 0 min 09 s	Approx. 0 min 10 s	Page 588
SMB	Approx. 0 min 12 s	Approx. 0 min 09 s	Page 496

The duration of transmission can increase due to factors, such as:

- Number of simultaneously transmitted files
- Network utilization
- Age of the hardware being used

25.8 Opening files with additional software

Application

The control provides various tools for opening and editing standard file types.

Related topics

- File types
Further information: Programming and Testing User's Manual
- Opening image, PDF, and HTML files in the **Document** workspace
Further information: Programming and Testing User's Manual
- Opening text files in the **Text editor** workspace
Further information: Programming and Testing User's Manual
- Opening 3D models in **CAD Viewer**
Further information: "CAD Viewer ", Page 267

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XLSX (XLS)	Gnumeric
CSV	
INI	Leafpad
A	
TXT	
CFG	
CFG	Combine fixtures or KinematicsDesign Further information: "Combining fixtures in the New Fixture window", Page 252
CFT	ToolHolderWizard Further information: "Customizing tool carrier templates with ToolHolderWizard", Page 188
HTM/HTML	Web Browser or Mozilla Firefox
	 For networks and the Internet, the machine manufacturer or network administrator must guarantee that the control is protected against viruses and malware (e.g., by a firewall).
ZIP	Xarchiver
BMP	Ristretto Image Viewer or Geeqie
GIF	
JPG/JPEG	 Ristretto can only open graphics files. Geeqie can also edit and print graphics.
PNG	

File type	Tool
OGG	Parole



With Parole you can open the file types OGA, OGG, OGV and OGX. The Fuendo Codec Pack (available for payment) is needed only for other formats, such as MP4 files.

If you double-tap or double-click a file in the file manager, the control automatically starts the file in the correct workspace or tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

25.8.1 Opening tools

To open a tool:

- ▶ Select the HEIDENHAIN icon in the taskbar
- > The control opens the HEROS menu.
- ▶ Select **Tools**
- ▶ Select the tool (e.g. **Leafpad**)
- > The control opens the tool in its own workspace.

Notes

- You can also open several tools from the **Desktop menu** workspace.
- Use the **ALT+TAB** key combination to switch between open workspaces.
- More information on how to use the various tools is provided within the respective tool under Help.
- After starting, the **web browser** checks at regular intervals whether updates are available.

If you want to update the **web browser**, then you must deactivate the SELinux security software during this time and establish a connection to the Internet. Reactivate SELinux after the update!

Further information: "SELinux security software", Page 491

25.9 Network configuration with Advanced Network Configuration

Application

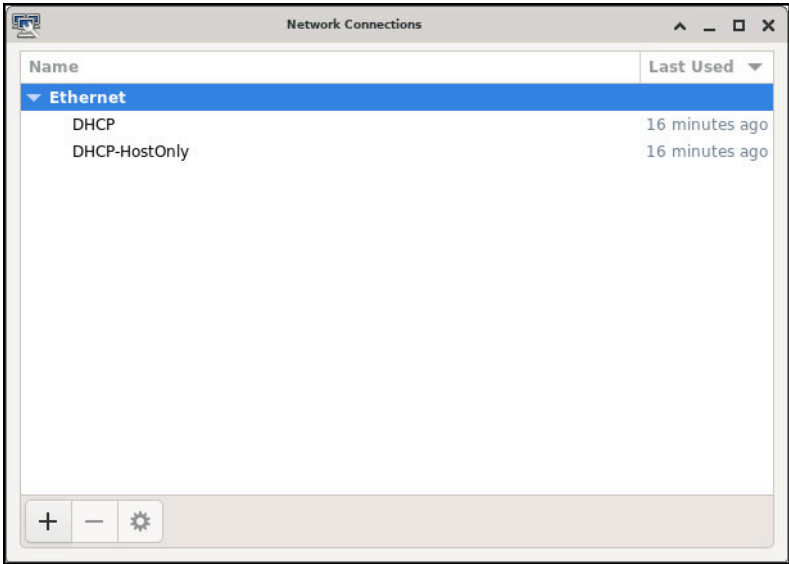
Use **Advanced Network Configuration** to edit or remove profiles for the network connection.

Related topics

- Network settings
Further information: "The Editing network connection window", Page 597

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network Connections** window.



The **Network Connections** window

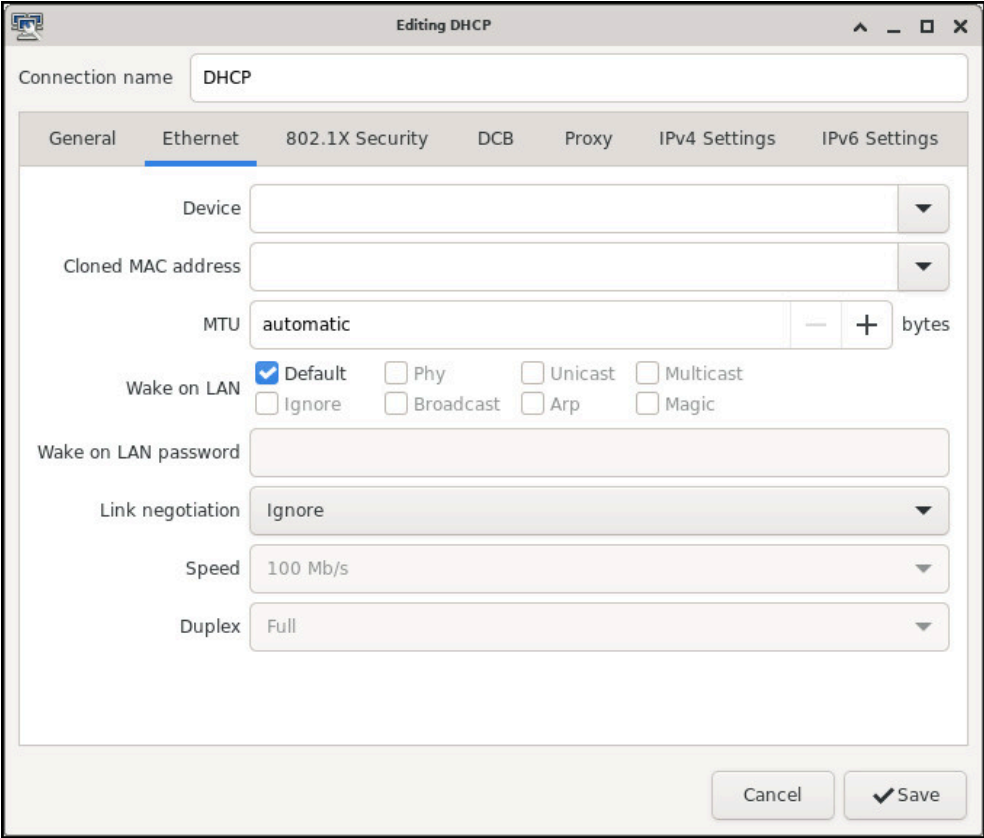
Icons in the Network Connections window

The following icons are shown in the **Network Connections** window:

Icon	Function
+	Add network connection
-	Remove network connection
⚙	Edit network connection The control opens the Editing network connection window. Further information: "The Editing network connection window", Page 597

25.9.1 The Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.



The **Editing network connection** window

The General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect automatically	If you are using several profiles, you can define an order of priority for the connection here. The control connects the network with the highest priority first. Input: -999...999
All users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to VPN when using this connection	Currently no function
Bonded connections:	Currently no function

The Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
Service:	Here you can select the Ethernet interface. If you do not select an Ethernet interface, this profile can be used for any Ethernet interface. Selection by means of a selection window
Cloned MAC address:	Currently no function
MTU:	Here you can define the maximum package size in bytes. Input: Automatic, 1...10000
Private key password:	Currently no function
Wake-on-LAN password	Currently no function
Link negotiation	Here you have to configure the settings for the Ethernet connection: ■ Ignore Retain the configurations already existing on the device. ■ Automatic The speed and duplex settings are configured automatically for the connection. ■ Manual Configure the speed and duplex settings for the connection manually. Selection by means of a selection window
Speed	Here you have to select the speed settings: ■ 10 Mb/s ■ 100 Mb/s ■ 1 Gb/s ■ 10 Gb/s Only if Link negotiation has been selected Manual Selection by means of a selection window
Full duplex	Here you have to select the duplex setting: ■ Half ■ Full Only if Link negotiation has been selected Manual Selection by means of a selection window

The 802.1X Security tab

Currently no function

The DCB tab

Currently no function

The Proxy tab

Currently no function

The IPv4 Settings tab

The **IPv4 Settings** tab contains the following settings:

Setting	Meaning
Method:	Here you have to select a network connection method: ■ Automatic (DHCP) If the network uses a DHCP server for IP address assignment ■ Automatic (DHCP) addresses only If the network uses a DHCP server for IP address assignment, but you are assigning the DNS server manually ■ Manual Assign the IP address manually ■ Link-Local Only Currently no function ■ Shared to other computers Currently no function ■ Disabled Deactivate IPv4 for this connection
Automatic, addresses only	Here you can add static IP addresses that will be set up in addition to the IP addresses that are assigned automatically. Only with Method: Manual
Additional DNS servers:	Here you can add the IP addresses of DNS servers that are used to resolve computer names. Separate multiple IP addresses by commas. Only with Method: Manual and Automatic (DHCP) addresses only
Additional search domains:	Here you can add domains used by computer names. Separate multiple domains by commas. Only with Method: Manual
DHCP client ID:	Currently no function
Require IPv4 addressing for this connection to complete	Currently no function

The IPv6 Settings tab

Currently no function

26

Overviews

26.1 Pin layout and cables for data interfaces

26.1.1 V.24/RS-232-C interface for HEIDENHAIN devices



The interface complies with the requirements of EN 50178 for Secure separation from the power grid.

Control		25-pin: VB 274545-xx			9-pin: VB 366964-xx		
Male	Assignment	Male	Color	Female	Female	Color	Female
1	Do not assign	1	White/Brown	1	1	Red	1
2	RXD	3	Yellow	2	2	Yellow	3
3	TXD	2	Green	3	3	White	2
4	DTR	20	Brown	8	4	Brown	6
5	Signal GND	7	Red	7	5	Black	5
6	DSR	6		6	6	Violet	4
7	RTS	4	Gray	5	7	Gray	8
8	CTR	5	Pink	4	8	White/Green	7
9	Do not assign	8	Violet	20	9	Green	9
Housing	External shield	Housing	External shield	Housing	Housing	External shield	Housing

26.1.2 Ethernet interface RJ45 socket

Maximum cable length:

- 100 m unshielded
- 400 m shielded

Pin	Signal
1	TX+
2	TX-
3	RX+
4	Vacant
5	Vacant
6	RX-
7	Vacant
8	Vacant

26.2 User administration roles and rights

26.2.1 List of roles



The following contents can change in the following software versions of the control:

- HEROS role names
- Unix groups
- Basic ID number

Further information: "Roles", Page 552

Operating system roles:

Role	Privileges		
	HEROS role name	UNIX group	Basic ID number
HEROS.RestrictedUser	Role for a user with minimum rights on the operating system.		
	■ HEROS.MountShares	■ mnt	■ 335
	■ HEROS.Printer	■ lp	■ 9
	■ HEROS.VMSharedFolders	■ vboxsf	■ 1000
HEROS.NormalUser	Role for a normal user with limited rights on the operating system.		
	This role grants the rights of the RestrictedUser role, as well as the following rights:		
	■ HEROS.SetShares	■ mntcfg	■ 334
	■ HEROS.ControlFunctions	■ ctrlfct	■ 340
HEROS.Admin	■ HEROS.MountUSBDevices	■ mntusb	■ 345
	The configuration of the network and the configuration of the user administration are some of the rights granted by this role.		
	This role grants the rights of the NormalUser role, as well as the following rights:		
	■ HEROS.BackupUsers	■ userbck	■ 337
	■ HEROS.PrinterAdmin	■ lpadmin	■ 16
	■ HEROS.ReadLogs	■ logread	■ 342
	■ HEROS.SWUpdate	■ swupdate	■ 341
	■ HEROS.SetNetwork	■ netadmin	■ 336
	■ HEROS.SetTimezone	■ tz	■ 333
	■ HEROS.UserAdmin	■ useradmin	■ 339
	■ HEROS.BackupMachine	■ backup	■ 338

Role	Privileges																							
	HEROS role name	UNIX group	Basic ID number																					
HEROS.LegacyUser	Legacy-User is the default role if user administration is inactive. HEIDEN-HAIN recommends assigning other roles to users when user administration is active. This role grants the rights of the NormalUser role, as well as the following rights: <table><tr><td>■ HEROS.BackupUsers</td><td>■ userbck</td><td>■ 337</td></tr><tr><td>■ HEROS.PrinterAdmin</td><td>■ lpadmin</td><td>■ 16</td></tr><tr><td>■ HEROS.ReadLogs</td><td>■ logread</td><td>■ 342</td></tr><tr><td>■ HEROS.SWUpdate</td><td>■ swupdate</td><td>■ 341</td></tr><tr><td>■ HEROS.SetNetwork</td><td>■ netadmin</td><td>■ 336</td></tr><tr><td>■ HEROS.SetTimezone</td><td>■ tz</td><td>■ 333</td></tr><tr><td>■ HEROS.UserAdmin</td><td>■ useradmin</td><td>■ 339</td></tr></table>			■ HEROS.BackupUsers	■ userbck	■ 337	■ HEROS.PrinterAdmin	■ lpadmin	■ 16	■ HEROS.ReadLogs	■ logread	■ 342	■ HEROS.SWUpdate	■ swupdate	■ 341	■ HEROS.SetNetwork	■ netadmin	■ 336	■ HEROS.SetTimezone	■ tz	■ 333	■ HEROS.UserAdmin	■ useradmin	■ 339
■ HEROS.BackupUsers	■ userbck	■ 337																						
■ HEROS.PrinterAdmin	■ lpadmin	■ 16																						
■ HEROS.ReadLogs	■ logread	■ 342																						
■ HEROS.SWUpdate	■ swupdate	■ 341																						
■ HEROS.SetNetwork	■ netadmin	■ 336																						
■ HEROS.SetTimezone	■ tz	■ 333																						
■ HEROS.UserAdmin	■ useradmin	■ 339																						
HEROS.LegacyUserNoCtrlfct	This role determines the rights for remote log-in when user administration is disabled (e.g., via SSH). The control assigns this role automatically. This role grants the rights of the LegacyUser role, with the exception of the following right: <table><tr><td>■ HEROS.ControlFunctions</td><td>■ ctrlfct</td><td>■ 340</td></tr></table>			■ HEROS.ControlFunctions	■ ctrlfct	■ 340																		
■ HEROS.ControlFunctions	■ ctrlfct	■ 340																						

NC operator roles:

Role	Privileges		
	HEROS role name	UNIX group	Basic ID number
NC.Operator	This role allows you to run NC programs.		
	■ NC.OPModeProgramRun	■ NCOpPgmRun	■ 302
	■ NC.OpModeSingleStep	■ NCOpSin- glesStep	■ 303
NC.Programmer	This role grants the rights of NC programming.		
	This role grants the rights of the Operator role, as well as the following rights:		
	■ NC.EditNCProgram	■ 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	■ NCAp- proveFsAxis	■ 319
	■ NC.EditPocketTable		■ 307
	■ NC.SetupDrive	■ NCEdPocket	■ 315
	■ NC.SetupProgramRun	■ NCSetupDrv ■ NCSe- tupPgRun	■ 303

Role	Privileges		
	HEROS role name	UNIX group	Basic ID number
NC.AutoProductionSetter	This role allows you to execute all NC functions, including programming a scheduled NC program start.		
	This role grants the rights of the Setter role, as well as the following rights:		
	■ NC.ScheduleProgramRun	■ NCSchedulePgRun	■ 304
NC.LegacyUser	With the LegacyUser role, the control's behavior regarding NC programming is identical to that of older software versions without user administration. User administration remains active. The LegacyUser has the same rights as the AutoProductionSetter. Exception: The Legacy-User does not have the NC.ApproveFsAxis right.		
NC.AdvancedEdit	This role allows you to use special functions of the NC and table editors.		
	■ Special functions for the programming of variables and for editing the table header		
	Replacement for code number 555343		
	■ NC.EditNCProgramAdv	■ NCEdit-NCPgmAdv	■ 327
	■ NC.EditTableAdv	■ NCEdit-TableAdv	■ 328
NC.RemoteOperator	This role allows you to start NC programs from an external application.		
	■ NC.RemoteProgramRun	■ NCRemotePgmRun	■ 329

Machine manufacturer (PLC) roles:

Role	Privileges		
	HEROS role name	UNIX group	Basic ID number
PLC.ConfigureUser	This roles grants the rights on code number 123 .		
	■ NC.ConfigUserAdv	■ 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 manufacturer can adapt the PLC roles.

When the **Machine manufacturer (PLC) roles:** are adapted by the machine manufacturer, the following contents may change:

- The names of the roles
- The number of roles
- The functionality of the roles

26.2.2 List of rights

The table below lists all of the individual rights.

Further information: "Rights", Page 552

Rights:

HEROS role name	Description
HEROS.Printer	Data output to network printers
HEROS.PrinterAdmin	Configuration of network printers
HEROS.ReadLogs	Currently no function
NC.OPModeManual	Operation of the machine in the Manual operating mode
NC.OPModeMDi	Working in the MDI application
NC.OpModeProgramRun	Running NC programs in Full Sequence mode of the Program Run operating mode
NC.OpModeSingleStep	Running NC programs in Single Block mode of the Program Run operating mode
NC.SetupProgramRun	Probing in the Setup application Using the AFC and ACC functions
NC.ScheduleProgramRun	Programming a scheduled NC program start
NC.EditNCProgram	Editing NC programs
NC.EditToolTable	Editing the tool table
NC.EditPocketTable	Editing the pocket table
NC.EditPresetTable	Editing the preset table
NC.EditPalletTable	Editing pallet tables
NC.SetupDrive	Adjustment of drives by the end user
NC.ApproveFsAxis	Confirming test position of safe axes Resetting the test position of the axes
NC.EditNCProgramAdv	Additional NC functions
NC.EditTableAdv	Additional table programming functions (e.g., editing of the table head)
HEROS.SetTimezone	Setting the date and time, time zone and time synchronization via NTP and the HEROS menu
HEROS.SetShares	Configuration of public network drives mounted on the control
HEROS.MountShares	Connecting and disconnecting network shares with the control
HEROS.MountUSB-Devices	Access to USB devices (e.g., USB flash drive, external hard disk) via the file manager
HEROS.SetNetwork	Configuration of network and relevant settings for data security
HEROS.BackupUsers	Data backup on the control—for all users configured on the control
HEROS.BackupMachine	Backup and restoring data of the entire machine configuration

HEROS role name	Description
HEROS.UserAdmin	Configuration of user administration on the control This includes creating, deleting, and configuring local users
HEROS.ControlFunctions	Control function of the operating system ■ Auxiliary functions, such as starting and stopping NC software ■ Telemaintenance ■ Advanced diagnostic functions, such as log data
HEROS.SWUpdate	Installation of software updates for the control
HEROS.VMShared-Folders	Access to shared folders of a virtual machine Only relevant when running a programming station within a virtual machine
NC.RemoteProgram-Run	Defining the NC program start and override values from an external application (e.g., via the DNC interface) Read-access and write-access to the counter by means of FUNCTION COUNT via OPC UA NC Server (#56-61 / #3-02-1*)
NC.ConfigUserAdv	Configuration access to the contents that have been enabled through code number 123
NC.DataAccessServiceRead	Read-only access to the PLC: drive during servicing
NC.OpcUaOEMConfiguredDataRead	Read-access through OPC UA NC Server (#56-61 / #3-02-1*) to data defined by the machine manufacturer
NC.OpcUaOEMConfiguredData	Read-access and write-access through OPC UA NC Server (#56-61 / #3-02-1*) to data defined by the machine manufacturer
NC.OpcUaPwAuth	Logon to OPC UA NC Server (#56-61 / #3-02-1*) with your user name and password
NC.OpcUaPwAuthOnlyMachineNet	For connection via the eth1 network interface: Logon to OPC UA NC Server (#56-61 / #3-02-1*) with your user name and password

26.3 Overview of signals for Process Tracking Interface (PTI) (#3-04-1*)



The control always outputs signals in the metric system with Process Tracking Interface (PTI), regardless of the unit of the NC program.

You can output the following signals with PTI:

Signal designator	Description
a act	Actual axis acceleration value [m/s ²] or [°/s ²], calculated via the position encoder. Actual spindle acceleration value [rpm/sec], calculated via the speed encoder.
a nom	Nominal axis acceleration value [m/s ²] or [°/s ²]. Nominal spindle acceleration value [rpm/sec]
v act	Actual axis feed rate [mm/min] or [°/min], calculated via the position encoder. Actual spindle speed value [rpm], calculated via the speed encoder.
v nom	Nominal value of the axis feed rate [mm/min] or [°/min]. Axis feed rate calculated from the difference between the nominal position values. The following error is not included
Feed rate F	Contouring feed rate [mm/min] or [°/min]
Block no.	Block number of the NC program for triggering
s act	Actual position [mm] or [°] with respect to the machine coordinate system M-CS
s nom	Nominal position according to the nominal position value filter [mm] or [°] with respect to the machine coordinate system M-CS
s diff	Following error of the position controller [mm] or [°]
PosDiff	Difference between position and speed encoder [mm] or [°]
j act	Actual jerk value [m/s ³], for rotary axes in [°/s ³]. Calculated via the position encoder.
j nom	Nominal jerk value [m/s ³], for rotary axes in [°/s ³].
GantryDiff	Difference between synchronous axes before and during coupling [mm]
Analog	Analog axis/spindle: Analog voltage = Nominal velocity value [V]
SyncPosDiff	Position difference during spindle synchronism (slave spindle) [°]
AFC Diag.	Diagnosis information for AFC (Adaptive Feed Control, option 45). The following parameters are available: ■ Spindle utilization ■ Override factor ■ Cutting condition ■ Position ■ Status ■ Spindle util. accord. to CC ■ Load for breakage monitoring

Signal designator	Description
	■ Upper overload limit ■ Lower overload limit ■ Idle load of spindle The sampling interval should match the control loop cycle
Thread quality	Thread groove deviates from programmed contour
actTcpWcsX	Current TCP position in X in the workpiece coordinate system
actTcpWcsY	Current TCP position in Y in the workpiece coordinate system
actTcpWcsZ	Current TCP position in Z in the workpiece coordinate system
actTOriwcsZ	Current tool orientation, Euler angle Z
actTOriWcsX'	Current tool orientation, Euler angle X'
actTOriWcsZ''	Current tool orientation, Euler angle Z''
PLCPrePgm	The PLC operands (B, W, D, M, I, O, T, C, IB, IW, ID, OB, OW, OD, s) are recorded before the PLC program run. This means that the values of the operands are read at the beginning of the PLC program cycle. For types B, W and D the contents are recorded, and for the other types the logical state of the operands is recorded. For s, a symbolic value can be transferred
PLCPostPgm	The PLC operands (B, W, D, M, I, O, T, C, IB, IW, ID, OB, OW, OD, s) are recorded after the PLC program run. This means that the values of the operands are read at the end of the PLC program cycle. For types B, W and D the contents are recorded, and for the other types the logical state of the operands is recorded. For s, a symbolic value can be transferred
Comp	Diagnosis information on non-linear and linear axis-error compensations as well as the effective temperature compensation [mm or °]
kinemComp	Compensation value of KinematicsComp and the kinematic temperature compensation [mm]
tempComp	Pending compensation value of the lag-tracked PLC temperature compensation (PLC Module 9231) [mm or °].
backlashComp	Active compensation of reversal error [mm]
v act vctrl	Actual speed value on the output side, calculated from the motor encoder. Unit of rotary axis: [°/min] Unit of the linear axis: [mm/min]
v nom vctrl	Nominal speed value on the output side at position controller output. Unit of rotary axis: [°/min] Unit of the linear axis: [mm/min]
Iq int	Integral-action component of nominal current value [A]; CC 61xx/CC 3xx: effective value
Iq nom	Torque-defining nominal current value [A]; CC 61xx/CC 3xx: effective value
I2t mot	Current value of the I2t monitoring of the motor [%]
I2t inv	Current value of the I2t monitoring of the power module [%]
util-mot	Utilization of a motor with respect to the nominal point [%]

Signal designator	Description
util-DriveTrain	Utilization of the drive train in percent (power module + motor) [%]
J total	Estimated mass moment of inertia [$\text{kg} \cdot \text{m}^2$] (for rotary axes) from Load Adaptive Control (LAC—software option 143), if active
m total	Estimated mass [kg] (for linear axes) from Load Adaptive Control (LAC), if active
I act	Momentary actual current of the motor [A]
I d act	Momentary actual magnetizing current of the motor [A]
P mot elec	Electric power consumption of the motor [W]
P UV	Actual power released by the supply module [W]
U nom	Nominal voltage [V]
P mot mech	Mechanical power [W]
M act	Actual torque value [Nm]
P ZK	DC-link power of the power supply module [W]
U DC-link	DC-link voltage [V]
n act	Momentary motor speed [rpm]
n nom	Nominal value of motor speed [rpm]
Brake	Brake is released
s diff Ctrl	Servo lag of the position controller [mm] or [°]
U DC-link (UVR)	UVR DC-link voltage [V]
I DC-link	DC-link current [A]
I L1	Current of phase conductor L1 [A]
I L2	Current of phase conductor L2 [A]
I L3	Current of phase conductor L3 [A]
I leak	Leakage current [A]
P in	Supply unit power consumption [W]
U AC	Emulated line voltage [V]
f AC	Emulated mains frequency [Hz]
I out 24V	Current of the 24 V supply bar

Index

3

3D basic rotation.....	211
3D mesh.....	288
3D-ROT menu.....	218
3D tool model.....	190

A

About the product.....	61
About the User's Manual.....	47
ACC.....	266
Accessories.....	81
Active Chatter Control (ACC).....	266
Active Directory.....	562
Export configuration.....	566
Function user.....	566
Adaptive Feed Control (AFC).....	256
Additional documentation.....	50
Additional software.....	594
Additional status display.....	129
Adjustment of analog voltage offset.....	487
AFC.....	256
Basic settings.....	423
Programming.....	259
Teach-in cut.....	262
AFC settings.....	263
Angle encoder.....	167
Application	
Configuration editor.....	541
Functional safety.....	464
Help.....	55
Manual operation.....	160
MDI.....	301
Move to reference position....	155
MP for setters.....	540
MP for Users.....	540
Pocket table.....	406
Preset table.....	415
Retract.....	366
Settings.....	469
Setup.....	305
Start/Login.....	85
Tool management.....	184
Automatic program start.....	361
Axes	
Moving.....	162
Referencing.....	155
Axis angle.....	216
Axis designation.....	166
Axis display.....	122
Axis key.....	162

B

Backup.....	534
Basic coordinate system.....	200
Basic rotation.....	211

Basic transformation.....	417
B-CS.....	200
Block scan.....	352
Multi-level.....	356
Pallet table.....	358
Point table.....	357
Returning to the contour.....	360
Single-level.....	355

C

CAD file.....	267
CAD Import.....	280
Contour, saving.....	281
Position, saving.....	282
CAD Viewer.....	267
Manipulating.....	274
Calibrating.....	321
Length.....	323
Radius.....	324
Cartesian coordinate system....	197
Certificate.....	503
CFG file.....	244
Chatter Control.....	266
Code number.....	474
Collision monitoring.....	224
Activating.....	228
Fixtures.....	231
Compensation table	
Program run.....	364
Component monitoring	
Status.....	133
Conditional stop.....	450
Configuration editor.....	541
List.....	541
Table.....	544
Connecting cable.....	602
Connection	
Network.....	496
Network drive.....	492
Connection assistant.....	511
Contact.....	59
Context-sensitive help.....	57
Control	
Powering off.....	157
Switch-on.....	152
Control's user interface.....	83
Control-in-operation symbol.....	346
Control user interface	
User-defined.....	547
Conversational language.....	489
Coordinate system.....	196
Basics.....	197
Coordinate origin.....	197
CR2.....	174
CreateConnections.....	589
CSV file.....	387
Current user.....	559

D

Data backup.....	534, 592
Comparison of duration.....	593
Database ID.....	177
Data interface.....	584
Comparison.....	593
OPC UA.....	505
pin layout.....	602
Data transfer	
Software.....	586
Data transmission	
Comparison of duration.....	593
Date and time.....	488
Datum table	
Program run.....	364
DCM.....	224
Activating.....	228
Fixtures.....	231
Directory	
public.....	554
Display handwheel	
Incremental jog positioning...	442
Display unit.....	77
DNC.....	513
Port.....	515
Secure connection.....	572
Drive	
HOME.....	554
Dynamic Collision Monitoring	
DCM.....	224

E

Embedded Workspace.....	458
Encoder.....	167
Error message.....	297
Error window.....	297
Ethernet interface.....	496 , 602
Configuration.....	596
Settings.....	497
Transmission duration.....	593
Export version.....	68
Extended Workspace.....	460
External access.....	513
Port.....	515

F

Feed control.....	256
Feed rate limit.....	345
File	
Backing up.....	592
Tools.....	594
Firewall.....	528
First steps.....	105
Program run.....	116
Setup.....	113
Tool.....	109
Fixture	
Measuring.....	234

Fixture, setting up.....	234	Defining.....	250	Mini dashboard.....	127
Fixture monitoring		Integrated product aid		Model Aided Setup.....	326
CFG file.....	232	TNCguide.....	54	Using.....	332
Combined.....	252	Interface		MOD menu.....	469
Integrating.....	234	Ethernet.....	496	Overview.....	470
M3D file.....	233	OPC UA.....	505	Move to reference position.....	155
STL file.....	232	User-defined.....	547	Moving	
Fixtures.....	231	J		Axis key.....	162
CFG file.....	244	Jog increment.....	163	Incremental jog.....	163
Combining.....	252	Handwheel.....	442	N	
Form		Handwheel without display....	434	NC fundamentals.....	166
For tables.....	384	K		Network.....	496
Functional safety (FS).....	461	Keyboard.....	77	Configuration.....	596
Functional safety (FS) operating		Formula.....	296	Settings.....	497
modes.....	463	NC functions.....	295	Transmission duration.....	593
G		Text.....	296	Network configuration.....	596
General status display.....	121	Virtual.....	294	DCB.....	598
Gestures.....	91	Keys.....	91	Ethernet.....	598
GUI.....	83	Kinematics.....	475	General.....	597
H		KinematicsDesign.....	244	IPv4 Settings.....	599
Handwheel.....	429	L		IPv6 Settings.....	599
Activating.....	433	Language.....	489	Proxy.....	599
Contents of display.....	436	Changing.....	490	Security.....	598
Incremental jog positioning....	442	License settings.....	512	Network drive.....	492
Operating elements.....	432	Licensing terms.....	75	Connecting.....	492
Overview.....	430	Linear encoder.....	167	Network setting	
Soft key.....	437	L-shaped stylus.....	322, 322	Ping.....	501
Wireless handwheel.....	439	M		Routing.....	501
With display.....	435	M92 datum M92-ZP.....	168	SMB share.....	501
Without display.....	432	Machine		Network settings	
Handwheel mode.....	160	Powering off.....	157	DHCP Server.....	500
Handwheel with display.....	435	Switch-on.....	152	Interface.....	500
Contents of display.....	436	Machine axes, moving.....	162	Status.....	499
Operating elements.....	435	Machine coordinate system.....	198	New table, creating.....	374
Soft key.....	437	Machine datum.....	168	Notes, types of.....	51
Hardware.....	77	Machine information.....	478	O	
HEROS.....	577	Machine parameters.....	540	OC 310.....	447
HEROS function		Editing.....	540	Offset.....	417
Overview.....	579	Table.....	544	OPC UA NC Server.....	505
Settings Application.....	469	Machine parameters for users...	540	Connection assistant.....	511
HEROS menu.....	579	Machine settings.....	475	License settings.....	512
HEROS tool.....	594	Machine times.....	483	Restart.....	510
HOME.....	554	Machining time.....	146	Operating elements.....	91
Host computer operation.....	513	Main menu.....	102	Operating mode	
Host Key Fingerprint.....	514	Manual axis.....	361	Machine.....	85
I		Manual operation.....	160	Manual.....	85
Icons, miscellaneous.....	99	Manual tilting, activating.....	218	Overview.....	85
I-CS.....	206	MAS.....	326	Program Run.....	340
Incremental jog positioning.....	163	Maximum feed rate.....	345	RDP.....	458
Handwheel.....	442	M-CS.....	198	Start.....	85
Handwheel without display....	434	MDI.....	301	Tables.....	370
Indexed tool.....	178	Message.....	297	Operating system.....	577
Input coordinate system.....	206	Message menu.....	297	Override controller.....	447
Insertion point		Mid-program startup.....	352	Conditional stop.....	450
				Displaying breakpoints.....	453
				Overview of touch probes.....	484

P			
Pallet table			
Block scan.....	358		
Parameter list.....	150		
Parameters			
Tool table.....	393		
Touch probe table.....	402		
Pin layout			
data interface.....	602		
PKI admin.....	503		
Place of operation.....	64		
Pocket table.....	406		
Port DNC.....	515		
Portscan.....	533		
Position display.....	122		
Mode.....	148		
Status overview.....	128		
Position encoder.....	167		
Positioning with Manual Data			
Input.....	301		
Powering off.....	157		
Powering on and off.....	151		
Preset.....	209		
Activating.....	213		
Inches.....	421		
Scratching.....	210		
Setting.....	212		
Preset management.....	209		
Preset table.....	415		
Columns.....	416		
Inches.....	421		
Write-protection.....	418		
Printer.....	516, 516		
Process Tracking Interface			
(PTI).....	608		
Program call			
Structure.....	350		
Program Run.....	340		
Automatic program start.....	361		
Block scan.....	352		
Canceling.....	346		
Compensation table.....	364		
Contextual reference.....	347		
Datum table.....	364		
Manual traverse.....	350		
Navigation path.....	348		
Retract.....	366		
Returning to the contour.....	360		
Program run time.....	146		
Proper and intended operation.....	62		
PTI.....	608		
Public directory.....	554		
Q			
Q parameter list.....	150		
Q parameters			
Show.....	150		
R			
RDP.....	458		
Reference point.....	168		
Reference system.....	196		
Basic coordinate system.....	200		
Input coordinate system.....	206		
Machine coordinate system..	198		
Tool coordinate system.....	207		
Working plane coordinate			
system.....	204		
Workpiece coordinate system.....	202		
Remaining run time.....	146		
Remote Desktop Manager.....	522		
External computer, shutting			
down.....	522		
VNC.....	523		
Windows Terminal Service...	523		
Remote maintenance.....	590		
Remote Service.....	590		
Restarting.....	157		
Restore.....	534		
Retract.....	366		
Returning to the contour.....	360		
Run time			
Machine information.....	483		
Program run.....	146		
S			
Safety precaution.....	65		
Content.....	51		
Scratching			
Measuring the tool.....	334		
Preset.....	210		
Secure connection.....	572		
Fingerprint.....	514		
Transmission duration.....	593		
Secure Remote Access.....	590		
Security software SELinux.....	491		
Selection function			
Structure.....	350		
SELinux.....	491		
Service file.....	297		
Creating.....	299		
Settings.....	469		
Network.....	497		
VNC.....	519		
Settings application			
Overview.....	470		
Setting up a vise.....	241		
Setting up fixtures			
Sequence.....	240		
Vise.....	241		
SFTP.....	588		
SIK menu.....	480		
Simulation status.....	145		
Software number.....	68		
Software option.....	69		
Enabling.....	482		
Settings.....	480		
Spatial angles.....	216		
SRA.....	590		
SSH connection.....	572		
SSH File Transfer Protocol.....	588		
Startup process.....	152		
Status display.....	119		
Additional status display.....	129		
Axis.....	122		
Overview.....	120		
Position.....	122		
Position display.....	148		
Simulation.....	145		
technology.....	123		
TNC bar.....	127		
Status overview.....	127		
Control-in-operation symbol..	128		
Remaining run time.....	146		
Step index.....	178		
STL file			
Optimizing.....	288		
Switch-on.....	152		
System time.....	488		
T			
Table			
Creating.....	374		
Exporting and importing.....	387		
Filter.....	378		
In the configuration editor.....	544		
Preset table.....	415		
Tool tables.....	392		
Workspace.....	376		
Table contents, exporting.....	387		
Table contents, importing.....	387		
Target group.....	48		
Taskbar.....	582		
TCP.....	172		
T-CS.....	207		
Tilting			
Manual.....	214		
Spatial angle or axis angles..	216		
Time.....	488		
Time zone.....	488		
TIP.....	172		
TLP.....	173		
TNCdiag.....	537		
TNCguide.....	55		
TNCremo.....	586		
Tool.....	169		
Database ID.....	177		
Definition.....	184		
Exporting and importing.....	387		
Measuring.....	334		
Overview.....	170		
Preset.....	171		
Table.....	392		

Touch probe.....	401	566	
Tool carrier management.....	186	Logging on.....	568
Tool carrier reference point.....	171	Overview of roles and rights..	603
Tool center point TCP.....	172	Rights.....	552
Tool change position.....	168	Roles.....	552
Tool coordinate system.....	207	Secure connection.....	574
Tool data.....	175	Settings.....	558
Tool ID number.....	176	Users.....	550
Tooling list.....	413	Windows domain.....	562
Tool location point TLP.....	173	User Aids.....	293
Tool management.....	184	User interface of the control.....	83
Tool model.....	190	User's Manuals.....	49
Tool name.....	176		
Tool parameters.....	175	V	
Tool radius 2 center CR2.....	174	Virtual keyboard.....	294
Tool rotation point TRP.....	173	VNC.....	519
Tool table.....	392		
Inches.....	405	W	
Input options.....	392	W-CS.....	202
Parameters.....	393	Window Manager.....	583
Tool tip TIP.....	172	Windows domain.....	562
Tool types.....	182	Export configuration.....	566
Tool usage file.....	409	Function user.....	566
Tool usage test.....	193	Joining.....	566
Touch gestures.....	91	Wireless handwheel.....	439
CAD Viewer.....	274	Configuring.....	444
Touch probe		Working plane.....	166
Calibrating.....	321	Working plane, tilting	
Length, calibrating.....	323	Fundamentals.....	214
Radius, calibrating.....	324	Head rotary axis.....	215
Setting up a fixture.....	234	Manually.....	214
Workpiece, setting up.....	326	Table rotary axis.....	215
Touch probe cycle		Working plane coordinate	
Manual.....	305	system.....	204
Touch probe data.....	402	Workpiece, setting up.....	326
Touch Probe Function.....	305	Workpiece coordinate system...	202
Overview.....	307	Workpiece datum.....	168
Workpiece, setting up.....	326	Workpiece preset.....	168, 209
Touch probe monitoring.....	336	Workspace	
Touch probe table.....	401	Form for tables.....	384
Parameters.....	402	Keyboard.....	294
Touchscreen.....	77	List.....	541
Traverse		Main menu.....	102
Handwheel.....	429	Overview.....	88
Traverse limit.....	475	Overview of FS.....	464
TRP.....	173	Positions.....	121
T usage order.....	411	Probing Function.....	305
		RDP.....	458
		Simulation status.....	145
		Startup.....	152
		Status.....	129
		Table in the Tables operating	
		mode.....	376
		WPL-CS.....	204
		Write-protection, preset table....	418
		Write protection for preset table	
		Activating.....	419
		Removing.....	420

U

Unit of measure.....	475
UserAdmin.....	558
User administration.....	550
Activating.....	554
Autologin.....	567
Current user.....	559
Database.....	560
Domain.....	560
Export Windows configuration....	

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104
service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101
service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103
service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102
service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com

The Information Site for
HEIDENHAIN Controls

Klartext App

Klartext on your
mobile device

Google
Play Store

Apple
App Store



Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.



For more details on touch probes and vision systems:

www.heidenhain.com/products/touch-probes-and-vision-systems

