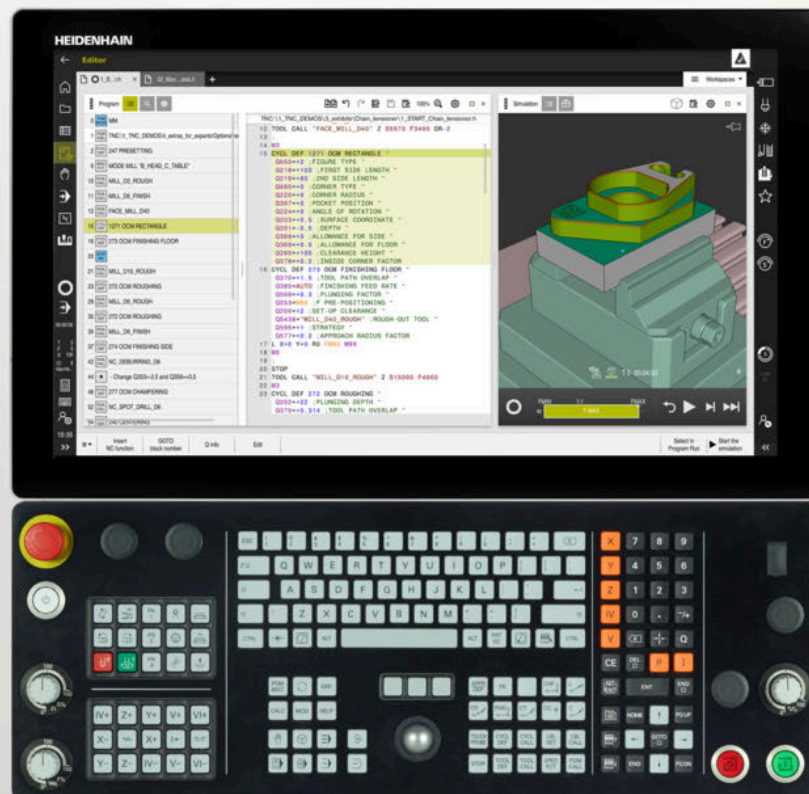




HEIDENHAIN



TNC7

User's Manual
Setup and Program Run

NC Software
817620-16
817621-16
817625-16

English (en)
01/2022

Contents

1	About the User's Manual.....	31
2	About the Product.....	41
3	First Steps.....	81
4	Status Displays.....	91
5	Powering On and Off.....	121
6	Manual Operation.....	129
7	NC Fundamentals.....	135
8	Tools.....	139
9	Coordinate Transformation.....	177
10	Collision Monitoring.....	203
11	Control Functions.....	229
12	Monitoring.....	251
13	Opening CAD Files with the CAD-Viewer.....	275
14	User Aids.....	297
15	Touch Probe Functions in the Manual Operating Mode.....	305
16	Application MDI.....	329
17	Program Run.....	333
18	Tables.....	357
19	Electronic Handwheel.....	415
20	Touch Probes.....	429
21	Embedded Workspace and Extended Workspace.....	433
22	Integrated Functional Safety (FS).....	437
23	Settings Application.....	445
24	HEROS Operating System.....	497
25	Overviews.....	513

1	About the User's Manual.....	31
1.1	Target group: Users.....	32
1.2	Available user documentation.....	33
1.3	Types of notes used.....	34
1.4	Notes on using NC programs.....	35
1.5	User's Manual as integrated product help: TNCguide.....	36
1.5.1	Search in TNCguide.....	39
1.5.2	Copying NC examples to clipboard.....	39
1.6	Contact to the editorial staff.....	39

2	About the Product.....	41
2.1	The TNC7.....	42
2.1.1	Proper and intended operation.....	42
2.1.2	Intended place of operation.....	43
2.2	Safety precautions.....	44
2.3	Software.....	48
2.3.1	Software options.....	48
2.3.2	Feature Content Level.....	55
2.3.3	Information on licensing and use.....	55
2.4	Hardware.....	56
2.4.1	Monitor.....	56
2.4.2	Keyboard unit.....	58
2.4.3	Hardware enhancements.....	61
2.5	Areas of the control's user interface.....	63
2.6	Overview of operating modes.....	64
2.7	Workspaces.....	66
2.7.1	Operating elements within the workspaces.....	66
2.7.2	Symbols within the workspaces.....	67
2.7.3	Overview of workspaces.....	67
2.8	Operating elements.....	70
2.8.1	Common gestures for the touchscreen.....	70
2.8.2	Operating elements of the keyboard unit.....	70
2.8.3	Icons on the control's user interface.....	76
2.8.4	Desktop menu workspace.....	78

3	First Steps.....	81
3.1	Chapter overview.....	82
3.2	Switching the machine and the control on.....	82
3.3	Configuring a tool.....	83
3.3.1	Selecting the Tables operating mode.....	83
3.3.2	Configuring the control's user interface.....	84
3.3.3	Preparing and measuring tools.....	84
3.3.4	Editing within tool management.....	85
3.3.5	Editing the pocket table.....	86
3.4	Setting up a workpiece.....	87
3.4.1	Selecting an operating mode.....	87
3.4.2	Clamping the workpiece.....	87
3.4.3	Workpiece presetting with a touch probe.....	87
3.5	Machining a workpiece.....	89
3.5.1	Selecting an operating mode.....	89
3.5.2	Opening an NC program.....	90
3.5.3	Starting an NC program.....	90
3.6	Switching the machine off.....	90

4	Status Displays.....	91
4.1	Application.....	92
4.2	Positions workspace.....	93
4.3	Status overview on the control bar.....	99
4.4	Status workspace.....	101
4.5	Simulation status workspace.....	116
4.6	Display of the program run time.....	117
4.7	Position displays.....	118
4.7.1	Switching the position display mode.....	119
4.8	Defining the contents of the QPARA tab.....	120

5	Powering On and Off.....	121
5.1	Powering on.....	122
5.1.1	Powering the machine and the control on.....	123
5.2	Referencing workspace.....	124
5.2.1	Axis reference run.....	125
5.3	Powering off.....	126
5.3.1	Shutting down the control and powering-off the machine.....	127

6	Manual Operation.....	129
6.1	Manual operation application.....	130
6.2	Moving the machine axes.....	132
6.2.1	Using axis keys to move the axes.....	132
6.2.2	Incremental jog positioning of axes.....	133

7	NC Fundamentals.....	135
7.1	NC fundamentals.....	136
7.1.1	Programmable axes.....	136
7.1.2	Designation of the axes on milling machines.....	136
7.1.3	Position encoders and reference marks.....	137
7.1.4	Presets in the machine.....	137

8	Tools.....	139
8.1	Fundamentals.....	140
8.2	Presets on the tool.....	140
8.2.1	Tool carrier reference point.....	141
8.2.2	Tool tip TIP.....	142
8.2.3	Tool center point (TCP, tool center point).....	143
8.2.4	Tool location point (TLP, tool location point).....	143
8.2.5	Tool rotation point (TRP, tool rotation point).....	144
8.2.6	Tool radius 2 center (CR2, center R2).....	144
8.3	Tool data.....	145
8.3.1	Tool ID number.....	145
8.3.2	Tool name.....	145
8.3.3	Indexed tool.....	146
8.3.4	Tool types.....	150
8.3.5	Tool data for the tool types.....	152
8.4	Tool management.....	165
8.4.1	Importing and exporting tool data.....	166
8.5	Tool carrier management.....	169
8.5.1	Parameterizing tool carrier templates.....	171
8.5.2	Assigning a tool carrier.....	171
8.6	Tool usage test.....	172
8.6.1	Performing the tool usage test.....	175

9	Coordinate Transformation.....	177
9.1	Reference systems.....	178
9.1.1	Overview.....	178
9.1.2	Basics of coordinate systems.....	179
9.1.3	Machine coordinate system M-CS.....	179
9.1.4	Basic coordinate system B-CS.....	182
9.1.5	Workpiece coordinate system W-CS.....	184
9.1.6	Working plane coordinate system WPL-CS.....	186
9.1.7	Input coordinate system I-CS.....	188
9.1.8	Tool coordinate system T-CS.....	190
9.2	Preset management.....	193
9.2.1	Setting a preset manually.....	196
9.2.2	Activating a preset manually.....	197
9.3	Tilting the working plane (option 8).....	198
9.3.1	Fundamentals.....	198
9.3.2	3-D rotation window (option 8).....	199

10 Collision Monitoring.....	203
10.1 Dynamic Collision Monitoring (DCM, option 40).....	204
10.1.1 Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes.....	208
10.1.2 Activating the graphic display of the collision objects.....	209
10.2 Fixture monitoring (option 40).....	210
10.2.1 Fundamentals.....	210
10.2.2 Integrating the fixtures into collision monitoring (option 140).....	213
10.2.3 Editing CFG files with KinematicsDesign.....	221

11 Control Functions.....	229
11.1 Adaptive Feed Control (AFC, option 45).....	230
11.1.1 Fundamentals.....	230
11.1.2 Activating and deactivating AFC.....	232
11.1.3 AFC teach-in cut.....	235
11.1.4 Monitoring tool wear and tool load.....	236
11.2 Active Chatter Control (ACC, option 145).....	237
11.3 Global Program Settings (GPS, option 44).....	238
11.3.1 Fundamentals.....	238
11.3.2 Function Additive offset (M-CS).....	241
11.3.3 Function Additive basic rotat. (W-CS).....	242
11.3.4 Function Shift (W-CS).....	243
11.3.5 Function Mirroring (W-CS).....	244
11.3.6 Function Shift (mW-CS).....	245
11.3.7 Function Rotation (WPL-CS).....	246
11.3.8 Function Handwheel superimp.....	247
11.3.9 Function Feed rate factor.....	250

12 Monitoring.....	251
12.1 Process Monitoring (option 168).....	252
12.1.1 Fundamentals.....	252
12.1.2 Process Monitoring workspace (option 168).....	253

13 Opening CAD Files with the CAD-Viewer.....	275
13.1 Fundamentals.....	276
13.2 Workpiece preset in the CAD model.....	281
13.2.1 Set the workpiece preset or workpiece datum and align the coordinate system.....	283
13.3 Workpiece datum in the CAD model.....	284
13.4 Applying contours and positions to NC programs with CAD Import (option 42).....	286
13.4.1 Selecting and saving a contour.....	289
13.4.2 Select positions.....	290
13.5 Generating STL files with 3D mesh (option 152).....	292
13.5.1 Positioning the 3D model for rear-face machining.....	295

14 User Aids.....	297
14.1 Virtual keyboard of the control bar.....	298
14.1.1 Opening and closing the virtual keyboard.....	301
14.2 Message menu on the information bar.....	301
14.2.1 Creating a service file.....	304

15 Touch Probe Functions in the Manual Operating Mode.....	305
15.1 Fundamentals.....	306
15.1.1 Setting a preset in a linear axis.....	313
15.1.2 Determining the circle center point of a stud using the automatic probing method.....	315
15.1.3 Determining and compensating the rotation of a workpiece.....	316
15.1.4 Using touch probe functions with mechanical probes or dial gages.....	317
15.2 Calibrating the workpiece touch probe.....	319
15.2.1 Calibrating the length of the workpiece touch probe.....	321
15.2.2 Calibrating the radius of the workpiece touch probe.....	322
15.2.3 3D calibration of workpiece touch probe (option 92).....	323
15.3 Suppressing touch probe monitoring.....	324
15.3.1 Deactivating touch probe monitoring.....	324
15.4 Comparison of offset and 3D basic rotation.....	326

16 Application MDI.....	329
--------------------------------	------------

17 Program Run.....	333
17.1 Program Run operating mode.....	334
17.1.1 Fundamentals.....	334
17.1.2 Manual traverse during an interruption.....	341
17.1.3 Block scan for mid-program startup.....	342
17.1.4 Returning to the contour.....	349
17.2 Compensation during program run.....	351
17.2.1 Opening tables from within Program Run operating mode.....	352
17.3 Retract application.....	352

18 Tables.....	357
18.1 Tables operating mode.....	358
18.1.1 Editing the contents of tables.....	359
18.2 Table workspace.....	359
18.3 Form workspace for tables.....	363
18.4 Tool Tables.....	367
18.4.1 Overview.....	368
18.4.2 Tool table tool.t.....	368
18.4.3 Turning tool table toolturn.trn (option 50).....	377
18.4.4 Grinding tool table toolgrind.grd (option 156).....	382
18.4.5 Dressing tool table tooldress.drs (option 156).....	389
18.4.6 Touch probe table tchprobe.tp.....	392
18.4.7 Creating a tool table in inches.....	395
18.5 Pocket table tool_p.tch.....	396
18.6 Tool usage file.....	398
18.7 T usage order (option 93).....	400
18.8 Tooling list (option 93).....	402
18.9 Preset table.....	403
18.9.1 Activating write protection.....	406
18.9.2 Removing write protection.....	406
18.9.3 Creating a preset table in inches.....	407
18.10 Tables for AFC (option 45).....	408
18.10.1 Basic AFC settings in AFC.tab.....	408
18.10.2 AFC.DEP settings file for teach-in cuts.....	411
18.10.3 Log file AFC2.DEP.....	412
18.10.4 Editing tables for AFC.....	414

19 Electronic Handwheel.....	415
19.1 Fundamentals.....	416
19.1.1 Entering spindle speed S.....	421
19.1.2 Entering the feed rate F.....	421
19.1.3 Entering miscellaneous functions M.....	421
19.1.4 Creating a positioning block.....	422
19.1.5 Incremental jog positioning.....	422
19.2 HR 550FS wireless handwheel.....	423
19.3 Configuration of wireless handwheel window.....	425
19.3.1 Assigning a handwheel to a handwheel holder.....	427
19.3.2 Selecting the transmission power.....	427
19.3.3 Setting the radio channel.....	428
19.3.4 Reactivating the handwheel.....	428

20 Touch Probes.....	429
20.1 Setting up touch probes.....	430

21	Embedded Workspace and Extended Workspace.....	433
21.1	Embedded Workspace (option 133).....	434
21.2	Extended Workspace.....	436

22 Integrated Functional Safety (FS).....	437
22.1 Checking axis positions manually.....	443

23 Settings Application.....	445
23.1 Overview.....	446
23.2 Code numbers.....	448
23.3 Machine settings menu item.....	448
23.4 General information menu item.....	450
23.5 SIK menu item.....	451
23.5.1 Viewing of software options.....	452
23.6 Machine times menu item.....	452
23.7 Adjust system time window.....	453
23.8 Conversational language of the control.....	453
23.8.1 Changing the language.....	454
23.9 SELinux security software.....	455
23.10 Network drives on the control.....	456
23.11 Ethernet interface.....	459
23.11.1 Network settings window.....	461
23.12 OPC UA NC Server (options 56 to 61).....	465
23.12.1 Fundamentals.....	465
23.12.2 OPC UA menu item (options 56 to 61).....	468
23.12.3 OPC UA connection wizard function (options 56 to 61).....	468
23.12.4 OPC UA license settings function (options 56 to 61).....	469
23.13 DNC menu item.....	470
23.14 Printers.....	471
23.14.1 Creating a printer.....	474
23.15 VNC menu item.....	474
23.16 Remote Desktop Manager window (option 133).....	478
23.16.1 Configuring an external computer for Windows Terminal Service (RemoteFX).....	482
23.16.2 Establishing and starting a connection.....	483
23.16.3 Exporting and importing connections.....	483
23.17 Firewall.....	484
23.18 Portscan.....	487
23.19 Remote servicing.....	488
23.19.1 Installing a session certificate.....	489

23.20 Backup and restore.....	489
23.20.1 Backing up data.....	490
23.20.2 Restoring data.....	491
23.21 TNCdiag.....	491
23.22 Machine parameters.....	492

24 HEROS Operating System.....	497
24.1 Fundamentals.....	498
24.2 HEROS menu.....	498
24.3 Serial data transfer.....	502
24.4 PC software for data transfer.....	504
24.5 Data backup.....	506
24.6 Opening files with additional software.....	506
24.6.1 Opening tools.....	507
24.7 Network configuration with Advanced Network Configuration.....	508
24.7.1 Editing network connection window.....	509

25	Overviews.....	513
25.1	Pin layout and cables for data interfaces.....	514
25.1.1	V.24/RS-232-C interface for HEIDENHAIN devices.....	514
25.1.2	Ethernet interface RJ45 socket.....	514
25.2	Machine parameters.....	514
25.2.1	List of user parameters.....	515
25.3	Keycaps for keyboard units and machine operating panels.....	525

1

**About the
User's Manual**

1.1 Target group: Users

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

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

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

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



HEIDENHAIN offers separate information products for other target groups:

- Leaflets and overview of the product program 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.

1.2 Available user documentation

User's Manual

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

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

- As a printed version, sub-divided into the modules below:
 - The **Setting up and running** User's Manual contains all information needed for setting up the machine and for running NC programs.
ID: 1358774-xx
 - The **Programming and testing** User's Manual contains all information needed for creating and testing NC programs. Touch probe and machining cycles are not included.
ID for Klartext programming: 1358773-xx
 - The **machining cycles** User's Manual contains all functions of the machining cycles.
ID: 1358775-xx
 - The **Measuring cycles for Workpieces and Tools** User's Manual contains all functions of the touch probe cycles.
ID: 1358777-xx

- As PDF files, sub-divided according to the printed versions or as complete PDF, containing all modules

TNCguide

- As HTML file for use as an integrated product help **TNCguide** directly on the control

TNCguide

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

Further information: "Proper and intended operation", Page 42

Further information products for users

The following information products are available to you as users:

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

1.3 Types of notes used

Safety precautions

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

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

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

Sequence of information in precautionary statements

All precautionary statements contain the following four sections:

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

Informational notes

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

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



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



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



The book symbol represents a **cross reference** to external documentation, e.g. the documentation of your machine manufacturer or other supplier.

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

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

tnc-userdoc@heidenhain.de

1.4 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.

1.5 User's Manual as integrated product help: TNCguide

Application

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

Further information: "Available user documentation", Page 33

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

Further information: "Proper and intended operation", Page 42

Requirement

In the factory default setting, the control offers the integrated product help **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** e.g. 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 **TNCguide** product help can be selected within the **Help** application or in the **Help** workspace.

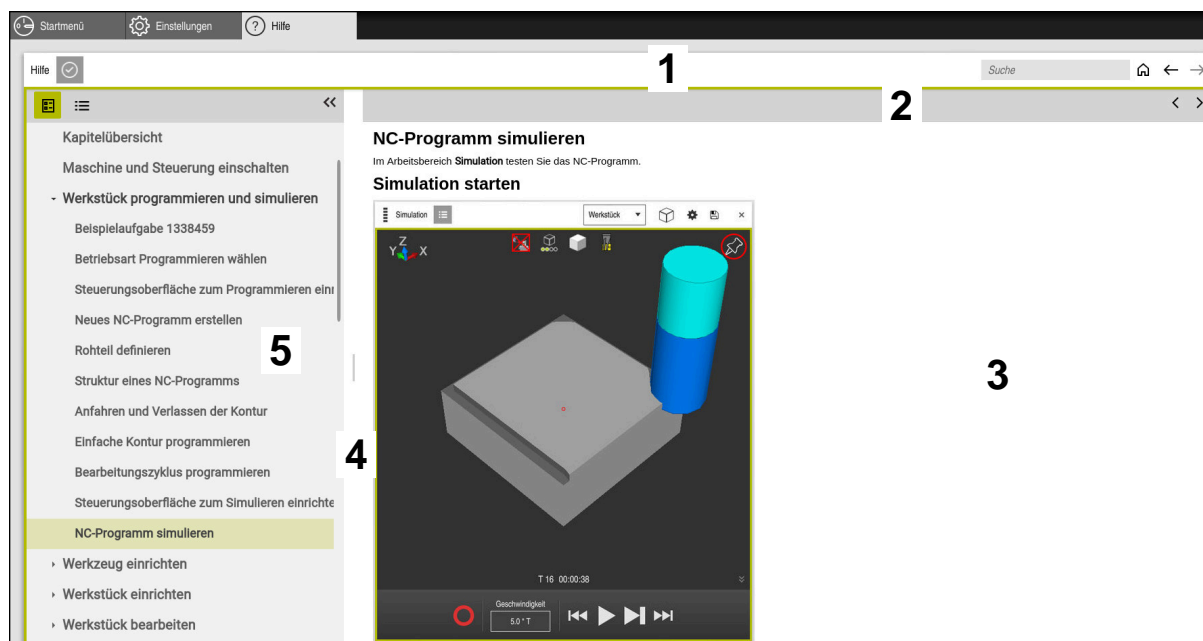
Further information: "Help application", Page 37

Further information: User's Manual for Programming and Testing

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

Further information: "Symbols", Page 38

Help application








Help application with open TNCguide

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








- 1 Title bar in the **Help** application
Further information: "Symbols in the Help application", Page 38
- 2 Title bar of the integrated **TNCguide** product help
Further information: "Symbols in the integrated TNCguide product help ", Page 38
- 3 Content column of **TNCguide**
- 4 Separator between the columns of **TNCguide**
 Adjust the column width by means of the separator.
- 5 Navigation column of **TNCguide**

Symbols

Symbols in the Help application

Symbol	Function
	<p>Show start page</p> <p>The start page displays all available documentation. Select the desired documentation, using a navigation tiles, e.g. TNCguide.</p> <p>If only one piece of documentation is available, the control opens the content directly.</p> <p>When a documentation is open, you can use the search function.</p>
	Displaying tutorials
	Navigating between the last opened contents
	
	<p>Displaying or hiding search results</p> <p>Further information: "Search in TNCguide", Page 39</p>

Symbols in the integrated TNCguide product help

Symbol	Function
	<p>Display documentation structure</p> <p>The structure consists of the content headings.</p> <p>The structure serves for main navigation within the documentation.</p>
	<p>Display documentation index</p> <p>The index consists of important keywords.</p> <p>The index serves as an alternative navigation within the documentation.</p>
	Display previous or next page within the documentation
	
	Display or hide the navigation
	
	<p>Copy NC examples to clipboard</p> <p>Further information: "Copying NC examples to clipboard", Page 39</p>

1.5.1 Search in TNCguide

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

Use the search function as follows:

- ▶ Enter a character string



The entry field is located in the title bar, to the left of the Home symbol that you use for navigating to the start page.

The search starts automatically after you have entered e.g. a letter.

If you wish to delete the entry, use the X symbol within the entry field.

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

1.5.2 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
- ▶ **Notes on using NC programs** Expand
- ▶ **Notes on using NC programs** Read and pay attention to
Further information: "Notes on using NC programs", Page 35



- ▶ 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 contents according to the **Notes on using NC programs**
- ▶ Use the Simulation mode to test the NC program
Further information: User's Manual for Programming and Testing

1.6 Contact to the editorial staff

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

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

tnc-userdoc@heidenhain.de

2

About the Product

2.1 The TNC7

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

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

Such an extension provides e.g. the chance to perform turning and grinding in addition to milling and drilling processes.

Further information: User's Manual for Programming and Testing

The ease of use increases e.g. when using touch probes, handwheels or a 3D mouse.

Further information: "Hardware", Page 56

Definitions

Abbreviation	Definition
TNC	TNC is derived from the acronym CNC (computerized numerical control). The T (tip or touch) represents the possibility to enter 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.

2.1.1 Proper and intended operation

The information about the proper and intended operation supports you as user in the safe handling of 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 into consideration the customer feedback. This results e.g. in function adaptations of the controls and safety precautions in the information products.



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

Further information: "Contact to the editorial staff", Page 39

2.1.2 Intended place of operation

In accordance with DIN EN 50370-1 standard referring to 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.

2.2 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

DANGER

Caution: hazard to the user!

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

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

⚠ WARNING**Caution: hazard to the user!**

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

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

NOTICE**Danger of collision!**

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

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

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!**

With NC software 81762x-16, the TNC7 does not support ISO programming. There is a risk of collision during execution due to the missing support.

- ▶ Use Klartext NC programs exclusively.

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 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 **TOOL CALL 0**. Danger of collision during subsequent tool positioning movements!

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

NOTICE**Caution: Significant property damage!**

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

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

NOTICE**Danger of collision!**

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

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

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

The **DELETE** function permanently deletes the file. The control does not perform an automatic backup of the file prior to deletion (e.g., there is no recycle bin). Files are thereby irreversibly deleted.

- ▶ Regularly back up important data to external drives

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

Never remove a connected USB device during data transfer—data can be damaged or deleted!

- ▶ Use the USB port for data transfer and backup only; do not use it for editing and executing NC programs
- ▶ Use the soft key to remove a USB device when data the transfer is complete

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

The control must be shut down so that running processes can be concluded and data can be saved. Immediate switch-off of the control by turning off the main switch can lead to data loss no matter what state the control was in!

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

NOTICE**Danger of collision!**

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

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

2.3 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 48

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
817620-16	TNC7
817621-16	TNC7 E
817625-16	TNC7 programming station



Refer to your machine manual.

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

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

Definition

Abbreviation	Definition
E	The suffix E indicates the export version of the control. In this version, the software option 9 Advanced Function Set 2 is restricted to 4-axis interpolation.

2.3.1 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 452

Overview and definitions

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



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



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

Further information: "Hardware", Page 56

Software option	Definition and application
Additional Axis (options 0 to 7)	Additional control loop A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required e.g. for detachable and motor-driven tilting tables.
Advanced Function Set 1 (option 8)	Advanced functions (set 1) On machines with rotary axes this software option enables the machining of multiple workpiece sides in a single setup. The software option includes the following functions: <ul style="list-style-type: none"> ■ Tilting the working plane, e.g. with PLANE SPATIAL Further information: User's Manual for Programming and Testing ■ Programming of contours on the unrolled surface of a cylinder, e.g. by using Cycle 27 CYLINDER SURFACE Further information: User's Manual for Machining Cycles ■ Programming the rotary axis feed rate in mm/min with M116 Further information: User's Manual for Programming and Testing ■ 3-axis circular interpolation with a tilted working plane The advanced functions (set 1) reduce the setup effort and increase the workpiece accuracy.
Advanced Function Set 2 (option 9)	Advanced functions (set 2) On machines with rotary axes this software option enables the simultaneous 5-axis machining of workpieces. The software option includes the following functions: <ul style="list-style-type: none"> ■ TCPM (tool center point management): Automatic tracking of linear axes during the rotary axis positioning Further information: User's Manual for Programming and Testing ■ Running of NC programs with vectors, including optional 3D tool compensation Further information: User's Manual for Programming and Testing ■ Manual moving of axes in the active tool coordinate system T-CS ■ Linear interpolation in more than 4 axes (max. 4 axes in case of an export version) The advanced functions (set 2) can be used to produce free-form surfaces.

Software option	Definition and application
HEIDENHAIN DNC (option 18)	<p>HEIDENHAIN DNC</p> <p>This software option enables external Windows applications to access data of the control via the TCP/IP protocol.</p> <p>Potential fields of application are e.g.:</p> <ul style="list-style-type: none"> ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data <p>HEIDENHAIN DNC is required in conjunction with external Windows applications.</p>
Dynamic Collision Monitoring (option 40)	<p>Dynamic Collision Monitoring (DCM)</p> <p>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.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Automatic interruption of the program run whenever a collision is imminent. ■ Warnings in case of manual axis movements ■ Collision monitoring in Test Run mode <p>With DCM you can prevent collisions and thus avoid additional costs resulting from material damage or a machine downtime.</p> <p>Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204</p>
CAD Import (option 42)	<p>CAD Import</p> <p>This software option is used to select positions and contours from CAD files and to transfer them into an NC program.</p> <p>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.</p> <p>"Applying contours and positions to NC programs with CAD Import (option 42)"</p>
Global PGM Settings (option 44)	<p>Global Program Settings GPS</p> <p>This software option can be used for superimposed coordinate transformations and handwheel movements during program run without changing the NC program.</p> <p>With GPS you can adapt externally created NC programs to the machine and increase flexibility during program run.</p> <p>Further information: "Globale Programmeinstellungen GPS", Page</p>
Adaptive Feed Control (option 45)	<p>Adaptive Feed Control AFC</p> <p>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.</p> <p>With AFC you can shorten the machining time without adapting the NC program, while preventing machine damage from overload at the same time.</p> <p>Further information: "Adaptive Feed Control (AFC, option 45)", Page 230</p>

Software option	Definition and application
KinematicsOpt (option 48)	<p>KinematicsOpt</p> <p>This software option uses automatic probing processes to check and optimize the active kinematics.</p> <p>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 temperature-induced deviations through repeated measurements and corrections.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p>
Turning (option 50)	<p>Mill-turning</p> <p>This software option offers a comprehensive milling-specific function package for milling machines with rotary tables.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Turning-specific tools ■ Turning-specific cycles and contour elements such as undercuts ■ Automatic tool radius compensation <p>Mill-turning enables mill-turning machining operations on only one machine, thus reducing e.g. the setup work effort considerably.</p> <p>Further information: User's Manual for Programming and Testing</p>
KinematicsComp (option 52)	<p>KinematicsComp</p> <p>This software option uses automatic probing processes to check and optimize the active kinematics.</p> <p>With KinematicsComp the control can correct position and component errors in three dimensions. This means it can compensate the errors of rotary and linear axes in three dimensions. Compared to KinematicsOpt (option 48), the corrections are even far more comprehensive.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p>
OPC UA NC Server 1 to 6 (options 56 to 61)	<p>OPC UA NC Server</p> <p>The software options offer the OPC UA standardized interface for the external access to data and functions of the control.</p> <p>Potential fields of application are e.g.:</p> <ul style="list-style-type: none"> ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data <p>Each software option enables one client connection each. Several parallel connections require the use of multiple OPC UA NC servers.</p> <p>Further information: "OPC UA NC Server (options 56 to 61)", Page 465</p>
4 Additional Axes (option 77)	<p>4 additional control loops</p> <p>see "Additional Axis (options 0 to 7)"</p>
8 Additional Axes (option 78)	<p>8 additional control loops</p> <p>see "Additional Axis (options 0 to 7)"</p>

Software option	Definition and application
3D-ToolComp (option 92)	<p>3D-ToolComp only in connection with Advanced Function Set 2 (option 9)</p> <p>With this software option, shape deviations on ball cutters and workpiece probes can be automatically compensated using a correction value table. 3D-ToolComp enables increasing the workpiece accuracy in conjunction with free-form surfaces, for example.</p> <p>Further information: User's Manual for Programming and Testing</p>
Extended Tool Management (option 93)	<p>Extended tool management</p> <p>This software option extends tool management by the two tables Tooling list and T usage order.</p> <p>The tables show the following contents:</p> <ul style="list-style-type: none"> ■ The Tooling list shows the tool requirements of the NC program or pallet to be run Further information: "Tooling list (option 93)", Page 402 ■ The T usage order shows the tool order of the NC program or pallet to be run Further information: "T usage order (option 93)", Page 400 <p>The extended tool management enables you to detect the tool requirements in time and thus prevent interruptions during program run.</p>
Advanced Spindle Interpolation (option 96)	<p>Interpolating Spindle</p> <p>This software option enables interpolation turning, as the control couples the tool spindle with the linear axes.</p> <p>The software option includes the following cycles:</p> <ul style="list-style-type: none"> ■ Cycle 291 COUPLG.TURNG.INTERP. for simple turning machining operations without contour subprograms ■ Cycle 292 CONTOUR.TURNG.INTRP. for finishing rotationally symmetrical contours <p>The interpolating spindle enables you to execute a turning operation also on machines without rotary table.</p> <p>Further information: User's Manual for Machining Cycles</p>
Spindle Synchronism (option 131)	<p>Spindle synchronism</p> <p>This software option synchronizes two or more spindles and thus enables e.g. the manufacture gears by hobbing.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Spindle synchronism for special machining operation, e.g. polygonal turning ■ Cycle 880 GEAR HOBGING only in conjunction with mill-turning (option 50) <p>Further information: User's Manual for Machining Cycles</p>
Remote Desktop Manager (option 133)	<p>Remote Desktop Manager</p> <p>This software option is used to display and operate externally linked computer units.</p> <p>With Remote Desktop Manager you reduce the distances covered between several workplaces and as a result increase the efficiency.</p> <p>Further information: "Remote Desktop Manager window (option 133)", Page 478</p>

Software option	Definition and application
Dynamic Collision Monitoring v2 (option 140)	<p>Dynamic Collision Monitoring (DCM) version 2</p> <p>This software option includes the functions of software option 40 (Dynamic Collision Monitoring, DCM).</p> <p>In addition, this software option can be used for the collision monitoring of workpiece fixtures.</p> <p>Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 213</p>
Cross Talk Compensation (option 141)	<p>Compensation of axis couplings CTC</p> <p>Using this software option, the machine manufacturer can e.g. compensate acceleration-induced deviations at the tool and thus increase accuracy and dynamic performance.</p>
Position Adaptive Control (option 142)	<p>Position adaptive control PAC</p> <p>Using this software option, the machine manufacturer can e.g. compensate position-induced deviations at the tool and thus increase accuracy and dynamic performance.</p>
Load Adaptive Control (option 143)	<p>Load adaptive control LAC</p> <p>Using this software option, the machine manufacturer can e.g. compensate load-induced deviations at the tool and thus increase accuracy and dynamic performance.</p>
Motion Adaptive Control (option 144)	<p>Motion adaptive control MAC</p> <p>Using this software option, the machine manufacturer can e.g. change speed-dependent machine settings and thus increase the dynamic performance.</p>
Active Chatter Control (option 145)	<p>Active chatter control ACC</p> <p>With this software option the chatter tendency of a machine used for heavy machining can be reduced.</p> <p>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 %.</p> <p>Further information: "Active Chatter Control (ACC, option 145)", Page 237</p>
Machine Vibration Control (option 146)	<p>Vibration damping for machines MVC</p> <p>Damping of machine oscillations for improving the workpiece surface quality through the following functions:</p> <ul style="list-style-type: none"> ■ AVD Active Vibration Damping ■ FSC Frequency Shaping Control
CAD Model Optimizer (option 152)	<p>Optimization of CAD models</p> <p>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.</p> <p>Further information: "Generating STL files with 3D mesh (option 152)", Page 292</p>

Software option	Definition and application
Batch Process Manager (option 154)	<p>Batch Process Manager BPM</p> <p>This software option makes it easy to plan and execute multiple production jobs.</p> <p>If pallet management and extended tool management (option 93) are extended or combined, BPM offers the following additional data, for example:</p> <ul style="list-style-type: none"> ■ Machining time ■ Availability of necessary tools ■ Manual interventions to be made ■ Program test results of assigned NC programs <p>Further information: User's Manual for Programming and Testing</p>
Component Monitoring (option 155)	<p>Component monitoring</p> <p>This software option enables the automatic monitoring of machine components configured by the machine manufacturer.</p> <p>Component monitoring assists the control in preventing machine damage due to overload by way of hazard warnings and error messages.</p>
Grinding (option 156)	<p>Jig grinding</p> <p>This software option offers a comprehensive grinding-specific function package for milling machines.</p> <p>The software option includes the following functions:</p> <ul style="list-style-type: none"> ■ Grinding-specific tools including dressing tools ■ Cycles for reciprocating stroke and dressing <p>Jig-turning enables complete machining operations on just one machine, thus reducing e.g. the setup work effort considerably.</p> <p>Further information: User's Manual for Programming and Testing</p>
Gear Cutting (option 157)	<p>Gear manufacturing</p> <p>This software option enables the manufacturing of cylindrical gears or helical gears of any angle.</p> <p>The software option includes the following cycles:</p> <ul style="list-style-type: none"> ■ Cycle 285 DEFINE GEAR to define the gear geometry ■ Cycle 286 GEAR HOBGING ■ Cycle 287 GEAR SKIVING <p>Gear manufacturing expands the scope of functionality of milling machines with rotary tables even without mill-turning (option 50).</p> <p>Further information: User's Manual for Machining Cycles</p>
Turning v2 (option 158)	<p>Mill-turning version 2</p> <p>This software option includes all functions of Mill-Turning (software option 50). In addition, this software option offers the following advanced turning functions:</p> <ul style="list-style-type: none"> ■ Cycle 882 SIMULTANEOUS ROUGHING FOR TURNING ■ Cycle 883 TURNING SIMULTANEOUS FINISHING <p>The advanced turning functions not only enable you to manufacture undercut workpieces but also to use a larger area of the indexable insert during the machining operation.</p> <p>Further information: User's Manual for Machining Cycles</p>

Software option	Definition and application
Optimized Contour Milling (option 167)	<p>Optimized contour machining (OCM)</p> <p>This software option enables the trochoidal milling of closed or open pockets and islands of any shape. During trochoidal milling, the full cutting edge is used under constant cutting conditions.</p> <p>The software option includes the following cycles:</p> <ul style="list-style-type: none"> ■ Cycle 271 OCM CONTOUR DATA ■ Cycle 272 OCM ROUGHING ■ Cycle 273 OCM FINISHING FLOOR and cycle 274 OCM FINISHING SIDE ■ Cycle 277 OCM CHAMFERING ■ In addition, the control offers OCM FIGURES for frequently needed contours <p>With OCM you can shorten the machining time while reducing tool wear at the same time.</p> <p>Further information: User's Manual for Machining Cycles</p>
Process Monitoring (option 168)	<p>Process monitoring</p> <p>Reference-based monitoring of the machining process</p> <p>The control uses this software option to monitor defined machining sections during program run. The control compares changes in conjunction with the tool spindle or the tool with the values of a reference machining operation.</p> <p>Further information: "Arbeitsbereich Prozessüberwachung (Option #168)", Page</p>

2.3.2 Feature Content Level

New functions or functional enhancements of the control software may be protected by software options or by means of the Feature Content Levels.

On purchasing a new control you will receive the highest level of the **FCL** possible with the installed software. A subsequent software update e.g. within the frame of a service requirement, does not automatically increase the **FCL** version.



As yet, no functions are protected by Feature Content Level. If functions are protected in future, the User's Manual will indicate the marking **FCL n**. The **n** shows the required number of the **FCL** version.

2.3.3 Information on licensing and use

Open-source software

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

To get to the licensing terms on the control:



▶ Select the **Home** operating mode

▶ Select the **Settings** application

▶ Select the **Operating system** tab



▶ Double-tap or double-click **About HeROS**

➤ The control opens the **HEROS Licence Viewer** window.

OPC UA

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

OPC UA NC Server (options 56 to 61) and HEIDENHAIN DNC (option 18) can be used to influence the behavior of the control. Before using these interfaces for productive purposes, system tests must be performed to exclude the occurrence of any malfunctions or performance failures of the control. The manufacturer of the software product that uses these communication interfaces is responsible for performing these tests.

Further information: "OPC UA NC Server (options 56 to 61)", Page 465

2.4 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 48

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

2.4.1 Monitor



BF 360

The TNC7 is delivered with a 24-inch touchscreen.

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

Further information: "Common gestures for the touchscreen", Page 70

Further information: "Operating elements of the keyboard unit", Page 70

Operation and cleaning



Avoiding electrostatic discharge when operating touchscreens

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

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

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



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

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

- Glass cleaner
- Foaming screen cleaners
- Mild detergents



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

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

Further information: "Settings Application", Page 445



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

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners

2.4.2 Keyboard unit



TE 360 with standard potentiometer layout



TE 360 with alternative potentiometer layout



TE 361

The TNC7 is delivered with various keyboard units.

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

Further information: "Common gestures for the touchscreen", Page 70

Further information: "Operating elements of the keyboard unit", Page 70



Refer to your machine manual.

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

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

Cleaning



Use operating gloves to prevent the device from becoming dirty.

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



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

Switch the control off before cleaning the keyboard unit.



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

- Aggressive solvents
- Abrasives
- Compressed air
- Steam blasters



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

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

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



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

- ▶ Apply a small amount of an isopropyl alcohol cleaner to a lint-free and clean cloth



Please observe the information for the cleaner.

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

Exchanging keycaps

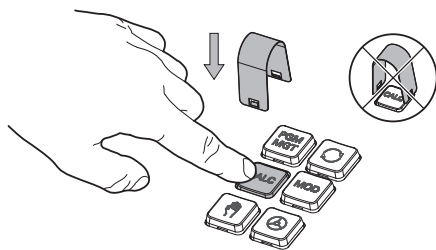
If you need replacements for the keycaps of the keyboard unit, contact HEIDENHAIN or the machine manufacturer.

Further information: "Keycaps for keyboard units and machine operating panels", Page 525



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

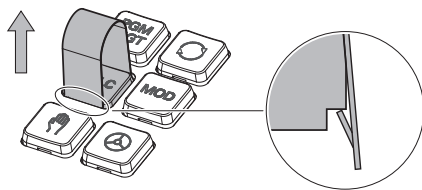
To exchange the keycaps:



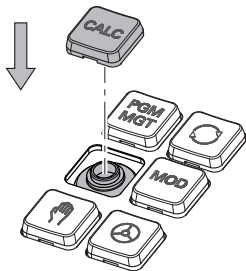
- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage



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



- Pull off the keycap



- Place the keycap onto the seal and push it down



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

- Verify proper seating and correct functioning

2.4.3 Hardware enhancements

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


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



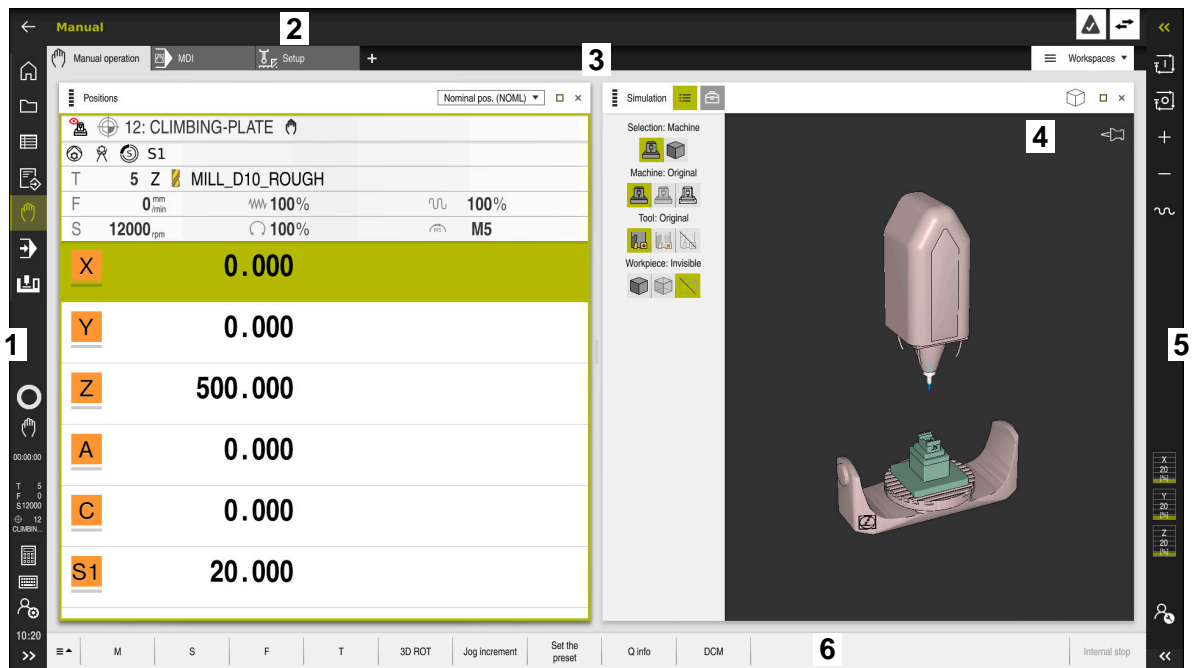
Keep in mind that particular hardware enhancements require additional software options.

Further information: "Software options", Page 48

Hardware enhancements	Definition and application
Electronic handwheels	<p>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:</p> <ul style="list-style-type: none"> ■ Portable or installed in the machine operating panel ■ With or without display ■ With or without functional safety <p>Electronic handwheels, for example, greatly simplify workpiece setup.</p> <p>Further information: "Electronic Handwheel", Page 415</p>
Workpiece touch probes	<p>The control uses this enhancement for automatic and precise detection of workpiece positions and misalignments. The workpiece touch probes have the following differing features:</p> <ul style="list-style-type: none"> ■ With radio or infrared transmission ■ With or without cable <p>Workpiece touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions during program run.</p> <p>Further information: "Touch Probe Functions in the Manual Operating Mode", Page 305</p>
Tool touch probes	<p>The control uses this enhancement for automatic and precise calibration of tools directly in the machine. Tool touch probes have the following differing features:</p> <ul style="list-style-type: none"> ■ Contact-free or tactile measurement ■ With radio or infrared transmission ■ With or without cable <p>Tool touch probes, for example, are useful for quick workpiece setup and for automatic correction of dimensions and breakage control during program run.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p>
Vision systems	<p>Use this enhancement to inspect the tools used. With the VT 121 vision system, you can visually inspect the cutting edges during program run without removing the tool. The vision systems help to avoid damage during program run, thus preventing unnecessary costs.</p>

Hardware enhancements	Definition and application
Additional operating stations	<p>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:</p> <ul style="list-style-type: none"> ■ 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 750 and ITC 860 are additional screens that increase the display area of the main screen, so that you can view multiple applications at the same time. <div data-bbox="576 678 1463 779" style="border: 1px solid black; padding: 5px;"> <p> By adding a keyboard unit, the ITC 750 and ITC 860 can be used as full-fledged additional operating stations.</p> </div> <p>The additional operating stations increase operator comfort, especially on large machining centers.</p>
Industrial PC	<p>You use this enhancement to install and run Windows-based applications. With Remote Desktop Manager (option 133), you can display the applications on the control's screen.</p> <p>Further information: "Remote Desktop Manager window (option 133)", Page 478</p> <p>The industrial PC is a secure and powerful alternative to external PCs.</p>

2.5 Areas of the control's user interface



The control's user interface in the **Manual operation** application

The control's user interface shows the following areas:

- 1 TNC bar
 - Back
Use this function to go backwards in the application history since booting the control.
 - Operating modes
Further information: "Overview of operating modes", Page 64
 - Status overview
Further information: "Status overview on the control bar", Page 99
 - Calculator
Further information: User's Manual for Programming and Testing
 - Screen keyboard
Further information: "Virtual keyboard of the control bar", Page 298
 - Settings
In the settings you can choose among several pre-defined views for the control's user interface.
 - Date and time
- 2 Information bar
 - Active operating mode
 - Message menu
Further information: "Message menu on the information bar", Page 301
 - Symbols

- 3 Application bar
 - Tabs of opened applications
 - Selection menu for workspaces

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

Further information: "Workspaces", Page 66
- 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.

2.6 Overview of operating modes

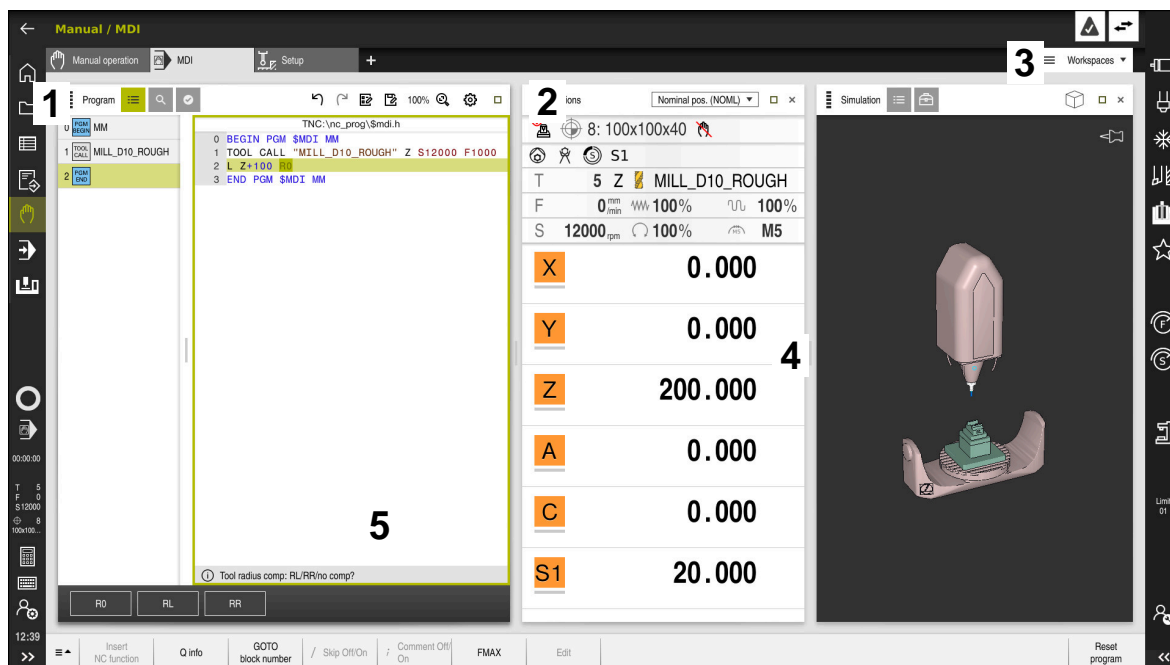
The control provides the following operating modes:

Symbols	Operating modes	Further information
	The Home operating mode contains the following applications:	
	<ul style="list-style-type: none"> ■ Start/Login application <p>During the startup process, the control is in the Start/Login application.</p> 	
	<ul style="list-style-type: none"> ■ Settings application 	Page 445
	<ul style="list-style-type: none"> ■ Help application ■ Applications for machine parameters 	Page 492
	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 358
	In the Editor operating mode you can do the following: <ul style="list-style-type: none"> ■ 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:	
	<ul style="list-style-type: none"> ■ Manual operation application 	Page 130
	<ul style="list-style-type: none"> ■ MDI application 	Page 329
	<ul style="list-style-type: none"> ■ Setup application 	Page 305
	<ul style="list-style-type: none"> ■ Move to ref. point application 	Page 124

Symbols	Operating modes	Further information
	In the Program Run operating mode you produce workpieces by having the control execute NC programs either one block at a time or in full sequence. You also execute pallet tables in this operating mode.	Page 334
	In the Retract application you can move the tool away from the workpiece, for example after a power failure.	Page 352
	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 433
	In the Machine operating mode the machine manufacturer defines his own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	

2.7 Workspaces

2.7.1 Operating elements within the workspaces






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

The control displays the following operating elements:

- 1 Gripper
Use the gripper in the title bar to change positions of the workspaces. You can also align two workspaces vertically above each other.
- 2 Title bar
In the title bar the control shows the title of the workspace, and different symbols or settings, depending on the workspace.
- 3 Selection menu for workspaces
Use the selection menu for workspaces in the application bar to open individual workspaces. The available workspaces depend on the active application.
- 4 Separator
You use the separator between two workspaces to change the scaling of the workspaces.
- 5 Action bar
In the action bar the control shows selection possibilities for the current dialog; for example, an NC function.

2.7.2 Symbols within the workspaces

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

Icon	Function
	Maximize workspace
	Reduce workspace
	Close workspace

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

2.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function In the Probing function workspace you set presets on the workpiece and determine and compensate for workpiece misalignment and rotations. You can also calibrate the touch probe, measure tools, and set up fixtures.	Page 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 The Open File workspace allows e. g. selecting and creating files.	See the User's Manual for Programming and Testing
Form for tables In the Form workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.	Page 363
Form for pallets In the Form workspace, the control shows the contents of the pallet table for the selected row.	See the User's Manual for Programming and Testing
Retract In the Retract workspace you disengage the tool after a power interruption.	Page 352
GS (option 44) In the GS workspace you define selected transformations and settings without modifying the NC program.	Page 238
Desktop menu In the Desktop menu workspace the control displays selected control and HEROS functions.	Page 78

Workspace	Further information
Help In the Help workspace the control displays a help graphic for the current syntax element of an NC function or the TNCguide integrated product help.	See the User's Manual for Programming and Testing
Contour In the Contour workspace you use lines and arcs to draw a 2D sketch and then generate a Klartext contour from it. You can also import program sections with contours from an NC program to the Contour workspace for graphical editing.	See the User's Manual for Programming and Testing
List In the List workspace the control shows the machine parameter structure; you might be able to edit some of the parameters.	Page 493
Positions In the Positions workspace the control displays information about the status of various functions of the control and about current axis positions.	Page 93
Program The control displays the NC program in the Program workspace.	See the User's Manual for Programming and Testing
RDP (option 133) If the machine manufacturer has defined an embedded workspace, you can see and operate the screen of an external computer on the control. The machine manufacturer can change the name of the workspace. Refer to your machine manual.	Page 433
Quick selection In the Quick selection workspace you open an existing table or create a file, e. g. an NC program.	See the User's Manual for Programming and Testing
Simulation In the Simulation workspace the control shows the simulated or actual movements, depending on the operating mode.	See the User's Manual for Programming and Testing
Simulation status In the Simulation status workspace the control shows data based on the simulation of the NC program.	Page 116
Start/Login In the Start/Login workspace the control shows the steps that are performed while booting.	Page 82
Status In the Status workspace the control shows the status and values of individual functions.	Page 101
Table In the Table workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.	Page 359










Workspace	Further information
<p>Table for machine parameters</p> <p>In the Table workspace the control shows the machine parameters; you might be able to edit some of them.</p>	Page 493
<p>Keyboard</p> <p>In the Keyboard workspace you can enter NC functions, letters and numbers, and also navigate.</p>	Page 298
<p>Overview</p> <p>In the Overview workspace the control displays the information on the status of individual functional safety (FS) aspects.</p>	Page 440
<p>Monitoring</p> <p>In the Process Monitoring workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.</p>	Page 253

2.8 Operating elements

2.8.1 Common gestures for the touchscreen

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

You can use the following gestures:

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

2.8.2 Operating elements of the keyboard unit

Application

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









Further information: "Common gestures for the touchscreen", Page 70

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







Description of function

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

Keycaps for alphabetic keyboard

Key	Function
  	Enter texts (e.g., file names)
SHIFT + 	Uppercase Q If an NC program is open, in the Editor operating mode for entering a Q parameter formula; in the Manual operating mode for opening the Q parameter list window Further information: User's Manual for Programming and Testing
	Close windows and context menus
	Create screenshot
	Left DIADUR key Open the HEROS menu
	In Klartext programming open a context menu

Keycaps for operating aids

Key	Function
	Open the Open File workspace in the Editor and Program Run operating modes Further information: User's Manual for Programming and Testing
	Select the button at the extreme right margin
	Open and close the message menu Further information: "Message menu on the information bar", Page 301
	Open and close the calculator Further information: User's Manual for Programming and Testing
	Open the Settings application Further information: "Settings Application", Page 445
	Open the online help Further information: "User's Manual as integrated product help: TNCguide", Page 36

Operating modes



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

Key	Function
	Opening the Manual operation application in the Manual operating mode Further information: "Manual operation application", Page 130
	Activating and deactivating the electronic handwheel in the Manual operating mode Further information: "Electronic Handwheel", Page 415
	Opening the Tool Management tab in the Tables operating mode Further information: "Tool management ", Page 165
	Opening the MDI application in the Manual operating mode Further information: "Application MDI", Page 329
	Opening the Program Run operating mode in Single Block mode Further information: "Program Run operating mode", Page 334
	Opening the Program Run operating mode Further information: "Program Run operating mode", Page 334
	Opening the Editor operating mode Further information: User's Manual for Programming and Testing
	While the NC program is running, opening of the Simulation workspace in the Editor operating mode Further information: User's Manual for Programming and Testing

Keycaps for NC dialog



















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











Key	Function
	In the Insert NC function window, open the Path contour folder in order to select an approach or departure function
	Open the Contour 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: User's Manual for Measuring Cycles for Workpieces and Tools
	In the Insert NC function window, open the 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 in order 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 data in the NC program
	In the Insert NC function window, open the Special functions folder (e.g., for later programming of a workpiece blank)
	In the Insert NC function window, open the Selection folder (e.g., to call an external NC program)

Further information: User's Manual for Programming and Testing

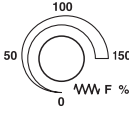
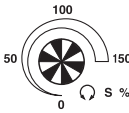
Keycaps for axis input and value input

Key	Function
 ... 	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
	In the Editor operating mode, within the Insert NC function window, open the FN folder
	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	Function
 	Position the cursor
	<ul style="list-style-type: none"> Position the cursor by using the block number of an NC block Open the selection menu while editing
	Jump to first line of an NC program or first column of a table
	Jump to last line of an NC program or last column of a table
	Go one page up in an NC program or table
	Go one page down in an NC program or table
	Mark the active application in order to navigate between applications
 	Navigate between areas of an application

Potentiometers





















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









2.8.3 Icons on the control's user interface

Overview of icons not specific to any operating mode

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

Icons that are specific to individual workspaces are described there.

Icon or shortcut	Function
	Back
	Select the Home operating mode
	Select the Files operating mode
	Select the Tables operating mode
	Select the Editor operating mode
	Select the Manual operating mode
	Select the Program Run operating mode
	Select the Machine operating mode
	Open and close the calculator
	Open and close the virtual keyboard
	Open and close the settings
	<ul style="list-style-type: none"> ■ White: Expand control bar or machine manufacturer bar ■ Green: Collapse control bar or machine manufacturer bar or go back ■ Gray: Confirm message
	Add
	Open file
	Close
	Maximize workspace
	Reduce workspace
	<ul style="list-style-type: none"> ■ Black: Add to favorites ■ Yellow: Remove from favorites
	Save
CTRL+S	
	Save as

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

2.8.4 Desktop menu workspace

Application

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

Description of function

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

- **Control**

In this area you can open operating modes or applications.

Further information: "Overview of operating modes", Page 64

Further information: "Overview of workspaces", Page 67

- **Tools**

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

Further information: "HEROS Operating System", Page 497

- **Help**

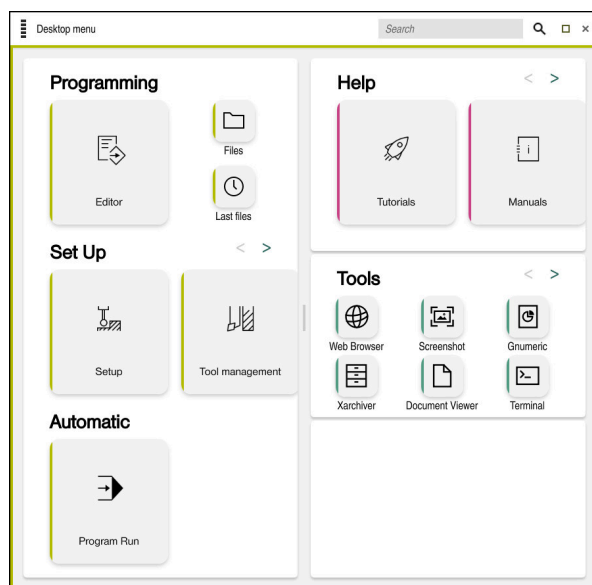
In this area you can open training videos or TNCguide.

- **Favorites**

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

Further information: "Adding and removing favorites", Page 79

In the title bar you can perform a full-text search for any strings.



Desktop menu workspace

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

Showing or hiding an area

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

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



Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

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

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



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

Removing favorites

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

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



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

3

First Steps

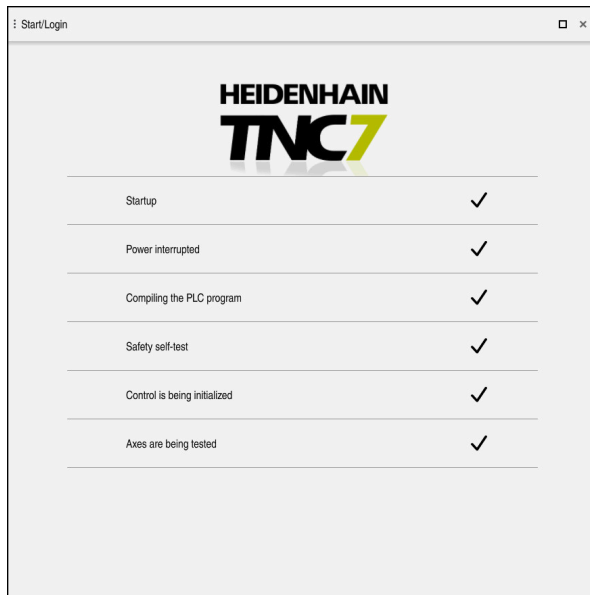
3.1 Chapter overview

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

The chapter covers the following topics:

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

3.2 Switching the machine and the control on



Start/Login workspace

DANGER

Caution: hazard to the user!

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

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

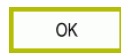


Refer to your machine manual.

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

To switch the machine on:

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



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

Further information: "Referencing workspace", Page 124



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

Further information: "Manual operation application", Page 130

More detailed information

- Switching on and off
- Position encoders

Further information: "Position encoders and reference marks", Page 137

- Axis reference run

3.3 Configuring a tool

3.3.1 Selecting the Tables operating mode

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

To select the **Tables** operating mode:



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

More detailed information

- Operating mode: **Tables**

Further information: "Tables operating mode", Page 358

3.3.2 Configuring the control's user interface

The screenshot displays the 'Form' workspace interface. It features a sidebar on the left with expandable sections: 'basic geometry data', 'correction data', 'others', and 'tool icon'. The main area contains several data entry fields and sections: 'tool life' with parameters like RT, LAST_USE, TIME1, TIME2, CUR_TIME, OVRTIME, and TL; 'additional geometry data' with KINEMATIC, LIFTOFF, LCUTS, ANGLE, LU, RN, and CUT; and 'tool identification' with fields for T, NAME, and TYP. At the bottom, there are status indicators for 'Min' and 'Max' values.

Form workspace in the **Tables** operating mode

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



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

To open the **Form** workspace:

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

More detailed information

- **Form** workspace
Further information: "Form workspace for tables", Page 363
- Workspace: **Table**
Further information: "Table workspace", Page 359

3.3.3 Preparing and measuring tools

To prepare tools:

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

3.3.4 Editing within tool management

Table Filter: all tools > all tool types 100%

T	P	NAME	TYP
0		NULLWERKZEUG	MILL_R
1	1.1	MILL_D2_ROUGH	MILL_R
2	1.2	MILL_D4_ROUGH	MILL_R
3	1.3	MILL_D6_ROUGH	MILL_R
4	1.4	MILL_D8_ROUGH	MILL_R
5	1.5	MILL_D10_ROUGH	MILL_R
6	0.0	MILL_D12_ROUGH	MILL_R
7	1.7	MILL_D14_ROUGH	MILL_R
8	1.8	MILL_D16_ROUGH	MILL_R
9	1.9	MILL_D18_ROUGH	MILL_R
10	1.10	MILL_D20_ROUGH	MILL_R
11	1.11	MILL_D22_ROUGH	MILL_R
12	1.12	MILL_D24_ROUGH	MILL_R
13	1.13	MILL_D26_ROUGH	MILL_R
14	1.14	MILL_D28_ROUGH	MILL_R

Tool name? Min: Max:

Tool management application in the **Table** workspace

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

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

To enter the tool data in tool management:

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

More detailed information

- Operating mode: **Tables**
Further information: "Tables operating mode", Page 358
- Workspace: **Form**
Further information: "Form workspace for tables", Page 363
- Tool management
Further information: "Tool management ", Page 165
- Tool types
Further information: "Tool types", Page 150

3.3.5 Editing the pocket table



Refer to your machine manual!

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

Table Filter: main magazine

TNC:\table\tool_p.tch

P	T	NAME	...	ST	F
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	MILL_D10_ROUGH			
1.6	6	MILL_D12_ROUGH			
1.7	7	MILL_D14_ROUGH			
1.8	8	MILL_D16_ROUGH		R	
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 name? Text width 32

Pocket table application in the **Table** workspace

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

There are various ways of accessing the pocket table:

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

To enter the data in the pocket table:

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



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

More detailed information

- Pocket table

Further information: "Pocket table tool_p.tch", Page 396

3.4 Setting up a workpiece

3.4.1 Selecting an operating mode

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

To select the **Manual** operating mode:



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

More detailed information

- Operating mode: **Manual**

Further information: "Overview of operating modes", Page 64

3.4.2 Clamping the workpiece

Mount the workpiece with a fixture on the machine table.

3.4.3 Workpiece presetting with a touch probe

Inserting a workpiece touch probe

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

To insert a workpiece touch probe:



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



Setting a workpiece preset

To set a workpiece preset at a corner:

- ▶ Select the **Setup** application



- ▶ Select **Intersection point (P)**

- > The control opens the probing cycle.

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



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



- ▶ Press the **NC Start** key

- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.

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



- ▶ Press the **NC Start** key

- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.

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



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



- ▶ Press the **NC Start** key

- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.

- ▶ Manually position the touch probe near the second touch point of the second workpiece edge



- ▶ Press the **NC Start** key

- > The control moves the touch probe in the probing direction to the workpiece edge and then back to the starting point.

- > The control then displays the coordinates of the determined corner point in the **Measuring result** area.

- ▶ Select **Compensate the active preset**

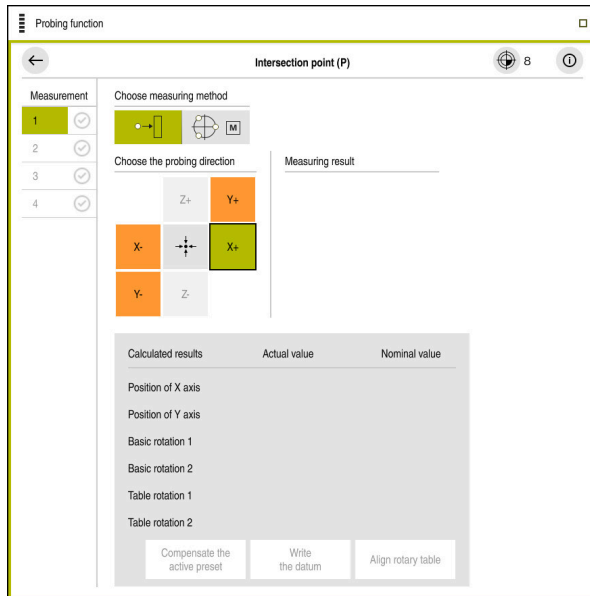
- > The control applies the calculated results to the workpiece preset.

Compensate the active preset



- ▶ Select **Exit probing**

- > The control closes the probing cycle.



Probing function workspace with an open manual probing function

More detailed information

- Workspace: **Probing function**
Further information: "Touch Probe Functions in the Manual Operating Mode", Page 305
- Reference points in the machine
Further information: "Presets in the machine", Page 137
- Tool change in the **Manual operation** application
Further information: "Manual operation application", Page 130

3.5 Machining a workpiece

3.5.1 Selecting an operating mode

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

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

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

More detailed information

- Operating mode: **Program Run**
Further information: "Program Run operating mode", Page 334

3.5.2 Opening an NC program

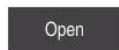
To open an NC program:



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



- ▶ Select an NC program



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

More detailed information

- Workspace: **Open File**

Further information: User's Manual for Programming and Testing

3.5.3 Starting an NC program

To start an NC program:



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

3.6 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 no matter what state the control was in!

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

To shut down the control:



- ▶ Select the **Home** operating mode



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



- ▶ Select **Shut down**
- The control shuts down.
- Once shutdown has concluded, the control displays the text **Now you can switch off.**

4

Status Displays

4.1 Application

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

The control offer the following status displays:

- General status display and position display in the **Positions** workspace
Further information: "Positions workspace", Page 93
- Status overview on the control bar
Further information: "Status overview on the control bar", Page 99
- Additional status displays for specific areas in the **Status** workspace
Further information: "Status workspace", Page 101
- Additional status displays in the **Editor** operating mode in the **Simulation status** workspace, based on the machining status of the simulated workpiece
Further information: "Simulation status workspace", Page 116

4.2 Positions workspace

Application

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

Description of function

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

Positions workspace with general status display

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

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

Further information: "Overview of operating modes", Page 64

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

- Symbols of active and inactive functions, such as Dynamic Collision Monitoring (DCM, option 40)
- Active tool
- Technology values
- Settings of the spindle and feed-rate potentiometers
- Active miscellaneous functions for the spindle
- Axis values and statuses, such as "Axis not referenced"








Further information: "Test status of the axes", Page 442

Axis display and position display




Refer to your machine manual.











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












Icon	Meaning
ACTL	Position display mode (e.g., actual or nominal coordinates of the current tool position) You can select the mode in the title bar of the workspace. Further information: "Position displays", Page 118
	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 98
?	The axis is not referenced.
	The axis is not in safe mode. Further information: "Checking axis positions manually", Page 443
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.
	Feed rate when stopped Further information: "Functional safety (FS) in the Positions workspace", Page 439
	Spindle when stopped Further information: "Functional safety (FS) in the Positions workspace", Page 439





Presets and technology values

Symbol	Meaning
	<p>Number of the active workpiece preset</p> <p>The number corresponds to the active line number of the preset table.</p> <p>Further information: "Preset management", Page 193</p>
T	<p>In the T area, the control shows the following information:</p> <ul style="list-style-type: none"> ■ Number of the active tool ■ Tool axis of the active tool ■ Symbol of the defined tool type ■ Name of the active tool
F	<p>In the F area, the control shows the following information:</p> <ul style="list-style-type: none"> ■ Active feed rate in mm/min <p>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.</p> <ul style="list-style-type: none"> ■ Setting of the rapid-traverse potentiometer in percent ■ Setting of the feed-rate potentiometer in percent <p>Further information: "Potentiometers", Page 75</p>
S	<p>In the S area, the control shows the following information:</p> <ul style="list-style-type: none"> ■ Active shaft speed in rpm <p>If you have programmed a cutting speed instead of a rotational speed, the control automatically converts this value to a rotational speed.</p> <ul style="list-style-type: none"> ■ Setting of the spindle potentiometer in percent ■ Active miscellaneous function for the spindle

Active functions

Icon	Meaning
	The Manual traverse function is active.
	The Manual traverse function is not active. Further information: "Program Run operating mode", Page 334
	RL tool radius compensation is active. Further information: User's Manual for Programming and Testing
	RR tool radius compensation is active. Further information: User's Manual for Programming and Testing These icons are transparent while the Block scan function of the control is active. Further information: "Block scan for mid-program startup", Page 342
	R+ tool radius compensation is active. Further information: User's Manual for Programming and Testing
	R- tool radius compensation is active. Further information: User's Manual for Programming and Testing These icons are transparent while the Block scan function of the control is active. Further information: "Block scan for mid-program startup", Page 342
	3D tool compensation is active. Further information: User's Manual for Programming and Testing This icon is transparent while the Block scan function of the control is active. Further information: "Block scan for mid-program startup", Page 342
	A basic rotation is defined in the active preset. Further information: "Basic rotation and 3D basic rotation", Page 195
	The basic rotation will be taken into account while moving the axes. Further information: "Basic rotation setting", Page 202
	A 3D basic rotation is defined in the active preset. Further information: "Basic rotation and 3D basic rotation", Page 195

Icon	Meaning
	<p>The tilted working plane will be taken into account while moving the axes.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>Further information: "3D ROT setting", Page 201</p>
	<p>The Tool axis function is active.</p> <p>Further information: "Tool axis setting", Page 202</p>
	<p>Either the TRANS MIRROR function or Cycle 8 MIRRORING is active. The axes programmed in the function or cycle are mirrored and moved.</p> <p>Further information: User's Manual for Machining Cycles</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>The pulsing spindle speed function S-PULSE is active.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>The PARAXCOMP DISPLAY function is active.</p>
	<p>The PARAXCOMP MOVE function is active.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>The PARAXMODE function is active.</p> <p>This icon might be superimposed on the icons for PARAXCOMP DISPLAY and PARAXCOMP MOVE.</p> <p>Further information: User's Manual for Programming and Testing</p>
TCPM	<p>Either the M128 function or TCPM FUNCTION (option 9) is active.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>The turning mode FUNCTION MODE TURN (option 50) is active.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>The grinding mode FUNCTION MODE GRIND (option 156) is active.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>Dressing mode (option 156) is active.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>Dynamic Collision Monitoring (DCM, option 40) is active.</p>

Icon	Meaning
	Dynamic Collision Monitoring (DCM, option 40) is not active. Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204
AFC 	Adaptive Feed Control (AFC, option 45) is active in teach-in cut mode.
AFC	Adaptive Feed Control (AFC, option 45) is active in closed-loop mode. Further information: "Adaptive Feed Control (AFC, option 45)", Page 230
ACC	Active Chatter Control (ACC, option 145) is active. Further information: "Active Chatter Control (ACC, option 145)", Page 237
	Global Program Settings (GPS, option 44) are active. Further information: "Global Program Settings (GPS, option 44)", Page 238
	Process Monitoring (option 168) is active. Further information: "Process Monitoring (option 168)", Page 252



In the optional machine parameter **iconPrioList** (no. 100813) you can change the sequence in which the control displays these symbols. The symbol for Dynamic Collision Monitoring (DCM, option 40) is always visible and cannot be configured.

Definition

Auxiliary axes

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

4.3 Status overview on the control bar

Application

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

Description of function

General information

	Positions (NOML.)	
	X	-118.632
	Y	-141.428
	Z	0.000
	A	0.000
	C	0.000
	m	0.000
	S1	328.015

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

- **Control-in-operation:** current machining status
Further information: "Definition", Page 100
- Symbol of the application used for machining
- Program run time



The control displays the same value for the program run time as on the **PGM** tab of the **Status** workspace.

Further information: "Display of the program run time", Page 117

- Active tool
- Active feed rate
- Current spindle speed
- Active workpiece preset

Position display

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

Further information: "Positions workspace", Page 93

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.

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 338

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

4.4 Status workspace

Application

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

Description of function

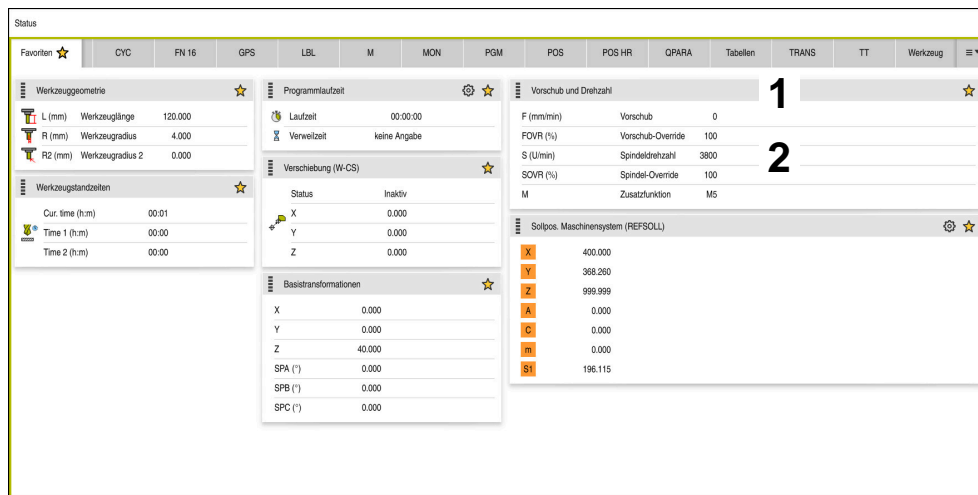
You can open the **Status** workspace in the following operating modes:

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

Further information: "Overview of operating modes", Page 64

Favorites tab

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



Favorites tab

- 1 Area
- 2 Contents

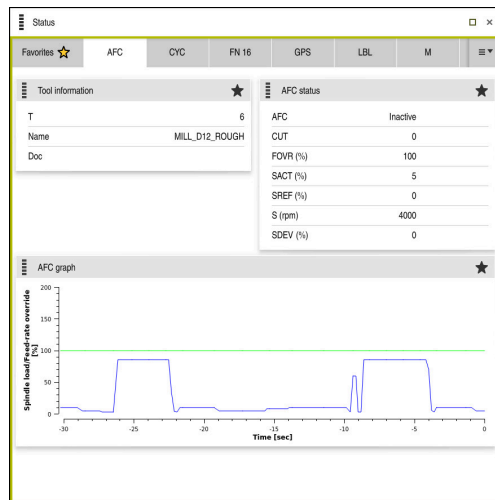
Every area in the status display has a **Favorites** icon. If you select the icon, the control adds that area to the **Favorites** tab.

Further information: "Icons on the control's user interface", Page 76

AFC tab (option 45)

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

Further information: "Adaptive Feed Control (AFC, option 45)", Page 230



AFC tab

Area	Contents
Tool information	<ul style="list-style-type: none"> ■ T Tool number ■ Name Tool name ■ Doc Comment about the tool from the tool management
AFC status	<ul style="list-style-type: none"> ■ AFC If AFC is being used to control the feed rate, then Control is displayed in this area. If the control is not controlling the feed rate, then Inactive is displayed in this area. ■ CUT Counts the quantity of cuts that have been performed with FUNCTION AFC CUT BEGIN, starting from zero. ■ FOVR (%) Active factor of the feed-rate potentiometer in percent ■ SACT (%) Current spindle load in percent ■ SREF (%) Reference load of the spindle in percent Define the reference load of the spindle in the syntax element LOAD of the FUNCTION AFC CUT BEGIN function. Further information: "NC functions for AFC (option 45)", Page 232 ■ S (rpm) Spindle shaft speed in rpm ■ SDEV (%) Current deviation of the speed in percent

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

CYC tab

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

Area	Contents
Active cycle definition	When you use the CYCLE DEF function to define a cycle, the control shows the cycle number in this area.
Cycle 32 TOLERANCE	<ul style="list-style-type: none"> ■ Status Shows whether Cycle 32 TOLERANCE is active or inactive ■ Values of Cycle 32 TOLERANCE ■ Values from the machine manufacturer for path and angle tolerance, such as predefined machine-specific roughing or finishing filters ■ Values of Cycle 32 TOLERANCE that are restricted by Dynamic Collision Monitoring (DCM, option 40)



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

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

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

FN16 tab

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

Further information: User's Manual for Programming and Testing

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

GPS tab (option 44)

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

Further information: "Global Program Settings (GPS, option 44)", Page 238

Area	Contents
Additive offset (M-CS)	<ul style="list-style-type: none"> ■ Status The Status shows whether a function is active or inactive. A function can be active even if its values are zero. ■ A (°) Additive offset (M-CS) in the A axis The Additive offset (M-CS) function is also available for the other rotary axes B (°) and C (°).
Additive basic rotat. (W-CS)	<ul style="list-style-type: none"> ■ Status ■ (°) The Additive basic rotat. (W-CS) function is in effect in the workpiece coordinate system W-CS. Entries are in degrees. Further information: "Workpiece coordinate system W-CS", Page 184
Shift (W-CS)	<ul style="list-style-type: none"> ■ Status ■ X Shift (W-CS) in the X axis The Shift (W-CS) function is also available for the other linear axes Y and Z.
Mirroring (W-CS)	<ul style="list-style-type: none"> ■ Status ■ X Mirroring (W-CS) in the X axis The Mirroring (W-CS) function is also available for the other linear axes Y and Z, as well as for the rotary axes available in the respective machine kinematics.
Rotation (WPL-CS)	<ul style="list-style-type: none"> ■ Status ■ (°) Rotation (WPL-CS) in degrees The Rotation (WPL-CS) function is in effect in the working plane coordinate system WPL-CS. Entries are in degrees. Further information: "Working plane coordinate system WPL-CS", Page 186
Shift (mW-CS)	<ul style="list-style-type: none"> ■ Status ■ X Shift (mW-CS) in the X axis The Shift (mW-CS) function is also available for the other linear axes Y and Z, as well as for the rotary axes available in the respective machine kinematics.

Area	Contents
Handwheel superimp.	<ul style="list-style-type: none"> ■ Status ■ Coordinate system This area contains the selected coordinate system for Handwheel superimp., such as the machine coordinate system M-CS. ■ X ■ Y ■ Z ■ A (°) ■ B (°) ■ C (°) ■ VT
Feed rate factor	<p>If the Feed rate factor function is active, the control displays the defined percentage in this field.</p> <p>If the Feed rate factor function is not active, the control displays 100.00 % in this field.</p>

LBL tab

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


Further information: User's Manual for Programming and Testing

Area	Contents
Subprograms	<ul style="list-style-type: none"> ■ Blk. no. Block number of the call ■ LBL no./Name Called label
Repetitions	<ul style="list-style-type: none"> ■ Blk. no. ■ LBL no./Name ■ Program-section repeat Number of repetitions still to be performed, e.g. 4/5

M tab

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

Further information: User's Manual for Programming and Testing

Area	Contents
Active M functions	<ul style="list-style-type: none"> ■ Function Active miscellaneous functions, such as M3 ■ Description Descriptive text about the respective miscellaneous function. <div style="border: 1px solid black; padding: 10px; margin-top: 10px;">  Refer to your machine manual. Only the machine manufacturer can create a descriptive text for machine-specific miscellaneous functions. </div>

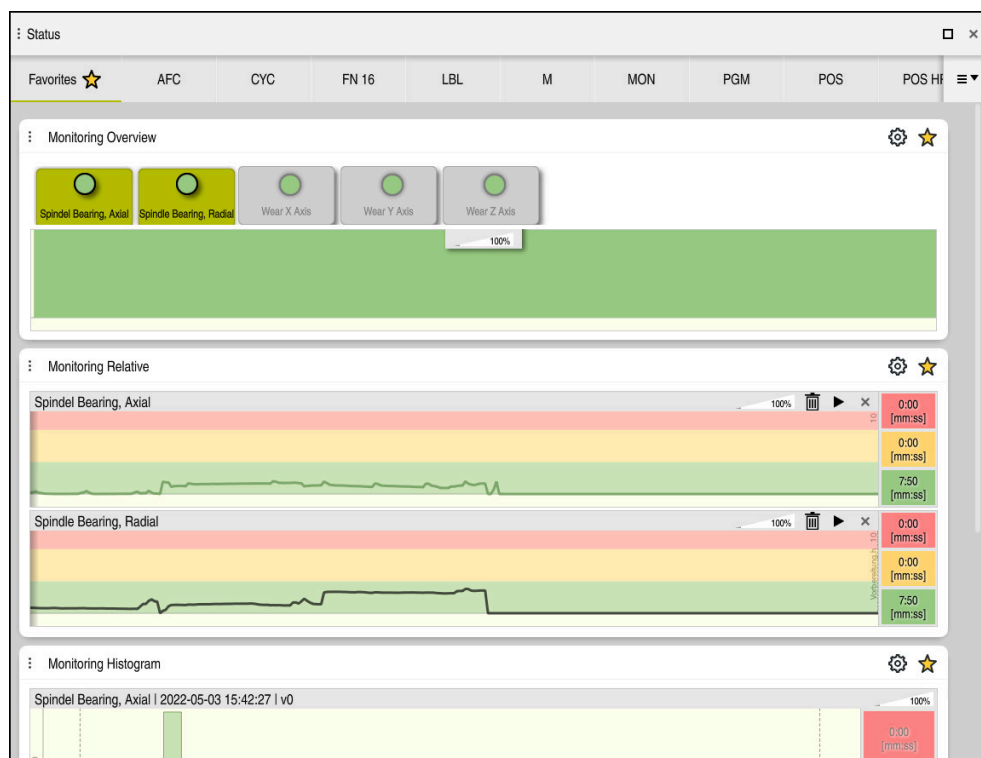
MON tab (option 155)

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



Refer to your machine manual.

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



MON tab with configured spindle speed monitoring

Area	Contents
Monitoring Overview	The control displays the machine components defined for monitoring. By selecting a component, you hide or show whether it is being monitored.
Monitoring Relative	<p>The control displays the monitoring information for the components being shown in the Monitoring Overview area.</p> <ul style="list-style-type: none"> ■ Green: component works under conditions defined as safe ■ Yellow: component works under warning zone conditions ■ Red: component is overloaded <p>In the Display settings window you can select which component the control shows.</p>
Monitoring Histogram	The control shows a graphical evaluation of previous monitoring sessions.

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

PGM tab

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

Area	Contents
Parts counter	<ul style="list-style-type: none"> ■ Quantity Actual value and nominal value of the parts counter defined with the FUNCTION COUNT function Further information: User's Manual for Programming and Testing
Program run time	<ul style="list-style-type: none"> ■ Runtime Run time of the NC program in hh:mm:ss ■ Dwell time Countdown of the waiting time in seconds from the following functions: <ul style="list-style-type: none"> ■ 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 117
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

POS tab


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

Area	Contents
Position display, e.g., Actual reference position (RFACTL)	<p>In this area the control shows the current position of all axes that are present.</p> <p>You can choose between the following views in the position display:</p> <ul style="list-style-type: none"> ■ Nominal pos. (NOML) ■ Actual pos. (ACT) ■ Nominal reference position (RFNOML) ■ Actual reference position (RFACTL) ■ Servo lag (LAG) ■ Handwheel superimposed (M118) <p>Further information: "Position displays", Page 118</p>
Feed and Speed	<ul style="list-style-type: none"> ■ Active Feed in mm/min ■ Active Feed-rate override in % ■ Active Rapid-traverse override in % ■ Active Spindle speed in rpm ■ Active Spindle override in % ■ Active Miscellaneous function in reference to the spindle, such as M3
Orientation of the working plane	<p>Spatial angles or axis angles for the active working plane</p> <p>Further information: User's Manual for Programming and Testing</p> <p>If axis angles are active, the control displays in this area only the values of the physically present axes.</p> <p>The defined values are in the window 3-D rotation</p> <p>Further information: "3D ROT setting", Page 201</p>
OEM transformation	<p>The machine manufacturer can define an OEM transformation for special turning kinematics.</p> <p>Further information: "Definitions", Page 115</p>
Basic transformations	<p>In this area the control shows the values of the active workpiece preset and active transformations in linear and rotary axes, such as a transformation in the X axis with the function TRANS DATUM.</p> <p>Further information: "Preset management", Page 193</p>
Special turning transformations	<p>Transformations relevant for turning operations (option 50), such as a defined precession angle from the following sources:</p> <ul style="list-style-type: none"> ■ Defined by the machine manufacturer ■ Cycle 800 ADJUST XZ SYSTEM ■ Cycle 801 RESET ROTARY COORDINATE SYSTEM ■ Cycle 880 GEAR HOBBING
Active traverse ranges	<p>Active traverse range, such as Limit 1 for traverse range 1</p> <p>Traverse ranges are machine-specific. If no traverse range is active, then Traverse range not defined is displayed in this area.</p>

Area	Contents
Active kinemat.	Name of the active machine kinematics

POS HR tab

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

Area	Contents
Coordinate system	<ul style="list-style-type: none"> ■ Machine (M-CS) With M118 the handwheel superimpositioning is always in effect in the machine coordinate system M-CS. Further information: User's Manual for Programming and Testing <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> With the Global Program Settings (GPS, option 44) the coordinate system can be chosen. Further information: "Global Program Settings (GPS, option 44)", Page 238</p> </div>
Handwheel superimp.	<ul style="list-style-type: none"> ■ Max. val. Maximum value of the individual axes programmed in M118 or in the GS workspace ■ Actl.val. Current superimpositioning

QPARA tab

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

Further information: User's Manual for Programming and Testing

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

Further information: "Defining the contents of the QPARA tab", Page 120

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

Tables tab

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

Area	Contents
Active tables	<p>In this area the control shows the path for the following active tables:</p> <ul style="list-style-type: none">■ Tool table■ Turning tool table■ Preset table■ Datum table■ Pocket table■ Touch probe table■ Grinding tool table■ Dressing tool table

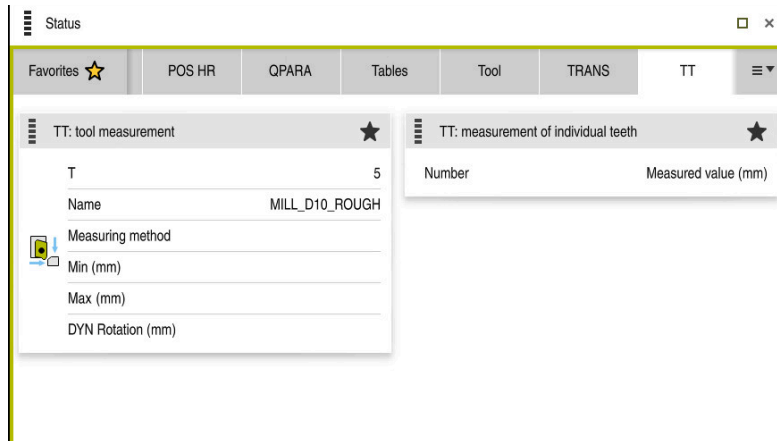
TRANS tab

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


Area	Contents
Active datum	<ul style="list-style-type: none"> ■ Path of the selected datum table ■ Line number of the selected datum table ■ Doc Contents of the DOC column of the datum table
Active datum shift	Datum shift that was defined with the TRANS DATUM function Further information: User's Manual for Programming and Testing
Mirrored axes	Axes mirrored with either the TRANS MIRROR function or Cycle 8 MIRRORING Further information: User's Manual for Programming and Testing Further information: User's Manual for Machining Cycles
Active angle of rotation	Rotation angle defined with either the TRANS ROTATION function or Cycle 10 ROTATION Further information: User's Manual for Programming and Testing Further information: User's Manual for Machining Cycles
Orientation of the working plane	Spatial angles for the active working plane Further information: User's Manual for Programming and Testing
Center coordinates	Center of scaling that was defined with Cycle 26 AXIS-SPECIFIC SCALING Further information: User's Manual for Machining Cycles
Active scaling factors	Scaling factors that were defined for the individual axes with the TRANS SCALE function, Cycle 11 SCALING FACTOR or Cycle 26 AXIS-SPECIFIC SCALING Further information: User's Manual for Programming and Testing Further information: User's Manual for Machining Cycles

TT tab

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



TT tab with values from the measurement of tool cutting edges of a milling cutter

Area	Contents
TT Tool to be measured	<ul style="list-style-type: none"> ■ T Tool number ■ Name Tool name ■ Measure what? Measurement method chosen for tool measurement, such as Length ■ Min (mm) When measuring milling cutters, in this area the control shows the smallest measured value of a cutting edge. When measuring turning tools (option 50), in this area the control shows the smallest measured tipping angle. The value of the angle can also be negative. Further information: "Definitions", Page 115 ■ Max (mm) When measuring milling cutters, in this area the control shows the greatest measured value of a cutting edge. When measuring turning tools, in this area the control shows the greatest measured tipping angle. The value of the angle can also be negative. ■ DYN Rotation (mm) When measuring milling cutters with a rotating spindle, the control shows values in this area. When measuring turning tools, the value DYN ROTATION describes the tolerance for the tipping angle. If the tolerance for the tipping angle is exceeded during calibration, the control marks the affected value in the MIN or MAX fields with an *. <div style="border: 1px solid black; padding: 10px; margin-top: 10px;"> <p> 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.</p> </div>
TT Tool cutting edges	<p>Number</p> <p>List of the measurements performed and the measured values of the individual cutting edges</p>

Tool tab

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

Further information: "Tool types", Page 150

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

Area	Contents
Tool information	<ul style="list-style-type: none"> ■ T Tool number ■ Name Tool name ■ Doc Note on the tool
Tool geometry	<ul style="list-style-type: none"> ■ L Tool length ■ R Tool radius ■ R2 Corner radius of the tool
Tool allowances	<ul style="list-style-type: none"> ■ DL Delta value for the tool length ■ DR Delta value for the tool radius ■ DR2 Delta value for the corner radius of the tool
Tool ages	<ul style="list-style-type: none"> ■ Cur. time (h:m) Time in hours and minutes the tool has been engaged ■ Time 1 (h:m) Service life of the tool ■ Time 2 (h:m) Maximum service life at tool call
Replacement tool	<ul style="list-style-type: none"> ■ RT Tool number of the replacement tool ■ Name Tool name of the replacement tool
Tool type	<ul style="list-style-type: none"> ■ Tool Axis Tool axis programmed in the tool call (e.g., Z) ■ Type Tool type of the active tool (e.g., DRILL)

Other contents for turning tools (option 50)

Area	Contents
Tool geometry	<ul style="list-style-type: none"> ■ ZL (mm) Tool length in Z direction ■ XL (mm) Tool length in X direction ■ RS (mm) Cutter radius ■ YL (mm) Tool length in Y direction
Tool allowances	<ul style="list-style-type: none"> ■ DZL (mm) Delta value in Z direction ■ DXL (mm) Delta value in X direction ■ DRS (mm) Delta value for the cutter radius ■ DCW (mm) Delta value for the width of the recessing tool
Tool type	<ul style="list-style-type: none"> ■ Tool Axis ■ TO Tool orientation ■ Type Tool type (e.g. TURN)

Definitions**OEM transformations for special turning kinematics**

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

Tipping angle

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

Angle of misalignment

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

4.5 Simulation status workspace

Application

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

Description of function

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

- **Favorites**
Further information: "Favorites tab", Page 101
- **CYC**
Further information: "CYC tab", Page 103
- **FN16**
Further information: "FN16 tab", Page 103
- **LBL**
Further information: "LBL tab", Page 105
- **M**
Further information: "M tab", Page 105
- **PGM**
Further information: "PGM tab", Page 107
- **POS**
Further information: "POS tab", Page 108
- **QPARA**
Further information: "QPARA tab", Page 109
- **Tables**
Further information: "Tables tab", Page 110
- **TRANS**
Further information: "TRANS tab", Page 111
- **TT**
Further information: "TT tab", Page 112
- **Tool**
Further information: "Tool tab", Page 114

4.6 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**. In the **Status** workspace the control takes traverse movements and dwell times into account. The control does not take dwell times into account in the **Simulation status** workspace.

Description of function

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

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

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

Further information: "PGM tab", Page 107

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

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

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

4.7 Position displays

Application

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


Description of function



The control has position displays in the following areas:

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

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

The control offers the following modes for the position display:

Mode	Meaning
Nominal pos. (NOML)	<p>This mode shows the value of the currently calculated target position in the input coordinate system I-CS.</p> <p>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.</p> <div style="border: 1px solid black; padding: 5px; margin-top: 10px;"> <p> The Nominal pos. (NOML) and Actual pos. (ACT) modes differ solely with regard to the servo lag.</p> </div>
Actual pos. (ACT)	<p>This mode shows the currently measured tool position in the input coordinate system I-CS.</p> <p>The actual position is the measured position of the axes, as determined by encoders at the time of comparison.</p>

Mode	Meaning
Nominal reference position (RFNOML)	<p>This mode shows the calculated target position in the machine coordinate system M-CS.</p> <div>  <p>The Nominal reference position (RFNOML) and Actual reference position (RFACTL) modes differ solely with regard to the servo lag.</p> </div>
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)	<p>This mode shows the values that you move using the M118 miscellaneous function.</p> <p>Further information: User's Manual for Programming and Testing</p> <div>  <p>Refer to your machine manual.</p> <p>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.</p> </div>

4.7.1 Switching the position display mode

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

- Select the **POS** tab



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

Notes

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

Further information: "Axis display and position display", Page 94

4.8 Defining the contents of the QPARA tab

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

Further information: "QPARA tab", Page 109

To define the contents of the **QPARA** tab, proceed as follows:



- ▶ Select the **QPARA** tab
- ▶ Select the **Settings** in the desired area, such as QL parameters
- > The control opens the **Parameter list** window.
- ▶ Enter numbers, such as **1,3,200-208**
- ▶ Press **OK**
- > The control displays the values of the defined variables.



- Use commas to separate single variables and connect sequential variables with a hyphen.
- The control always shows eight decimal places on the **QPARA** tab. For example, the control shows the result of **Q1 = COS 89.999** as 0.00001745. Very large and very small values are shown in exponential notation. The control shows the result of **Q1 = COS 89.999 * 0.001** as +1.74532925e-08, with e-08 corresponding to the factor of 10^{-8} .
- For variable texts in QS parameters the control shows the first 30 characters, i.e. the contents might be truncated.

5

**Powering On and
Off**

5.1 Powering on

Application

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



Refer to your machine manual.

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

Related topics

- Absolute and incremental position encoders

Further information: "Position encoders and reference marks", Page 137

Description of function

DANGER

Caution: hazard to the user!

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

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

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

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

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

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

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

- Absolute position encoders

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

- Incremental position encoders

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

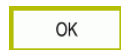
Further information: "Referencing workspace", Page 124

Further information: "Manual operation application", Page 130

5.1.1 Powering the machine and the control on

To switch the machine on:

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



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

Further information: "Referencing workspace", Page 124



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

Further information: "Manual operation application", Page 130

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 tool builder

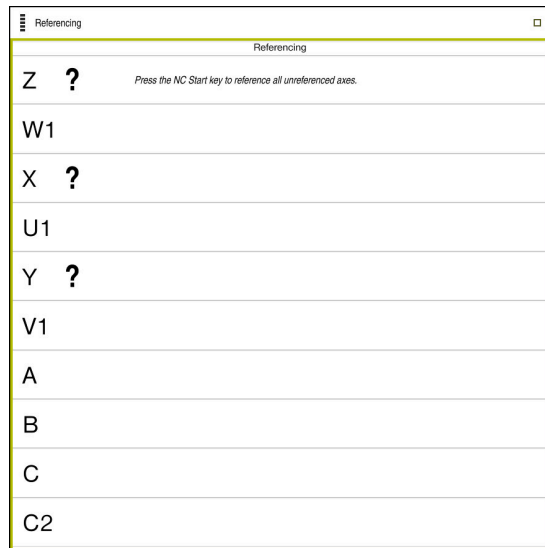
5.2 Referencing workspace

Application

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

Description of function

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



Referencing workspace with axes to be referenced

The control displays a question mark behind all axes that need to be referenced. Once all axes have been referenced, the control closes the **Move to ref. point** application and switches to the **Manual operation** application.

5.2.1 Axis reference run

To reference the axes in the prescribed sequence:



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

To reference the axes in any sequence:



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

Notes

NOTICE

Danger of collision!

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

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

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

Notes about traversing reference points in a tilted working plane

If the **Tilt working plane** (option 8) function was active before the control was shut down, then the control will automatically reactivate this function after restarting. This means that movements via the axis keys take place in the tilted working plane.

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

Further information: User's Manual for Programming and Testing

5.3 Powering off

Application

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

Description of function

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

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

5.3.1 Shutting down the control and powering-off the machine

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**
- > The control shuts down.
- > Once shutdown has concluded, the control displays the text **Now you can switch off.**
- ▶ Switch off the main 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 no matter what state the control was in!

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

- Different machines have different power-off procedures.
Refer to your machine manual.
- Applications on the control can delay the shutdown, such as a connection with **Remote Desktop Manager** (option 133)
"Remote Desktop Manager window (option 133)"

6

Manual Operation

6.1 Manual operation application

Application

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

Related topics

- Moving the machine axes
Further information: "Moving the machine axes", Page 132
- Incremental jog positioning of machine axes
Further information: "Incremental jog positioning of axes", Page 133

Description of function

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

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

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

Button	Meaning
Handwheel	The control displays this switch if a handwheel is configured for the control. If the handwheel is active, the operating mode's icon in the sidebar changes. Further information: "Electronic Handwheel", Page 415
M	Define a miscellaneous function M or use the selection window to choose one and activate it with the NC start key. Further information: User's Manual for Programming and Testing
S	Define the spindle speed S activate it with the NC start key, and also switch on the spindle. Further information: User's Manual for Programming and Testing
F	Define the feed rate F and activate it with the OK button. Further information: User's Manual for Programming and Testing
T	Define a tool T or use the selection window to choose one and insert it with the NC start key. Further information: User's Manual for Programming and Testing
Jog increment	Define the jog increment Further information: "Incremental jog positioning of axes", Page 133
Set the preset	Enter and set a preset Further information: "Preset management", Page 193
3D ROT	The control opens a window for the 3D rotation settings (option 8). Further information: User's Manual for Programming and Testing
Q info	The control opens the Q parameter list window, where you can see and edit the current values and descriptions of the variables. Further information: User's Manual for Programming and Testing
DCM	The control opens the Collision monitoring (DCM) window, where you can activate or deactivate Dynamic Collision Monitoring (DCM, option 40). Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 208

Note

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

6.2 Moving the machine axes

Application

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

Further information: "Touch Probe Functions in the Manual Operating Mode", Page 305

Related topics

- Programming traverse movements
Further information: User's Manual for Programming and Testing
- Executing traverse movements in the **MDI** application
Further information: "Application MDI", Page 329

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 415

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

Further information: "Status Displays", Page 91

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

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

Further information: "Status overview on the control bar", Page 99

6.2.1 Using axis keys to move the axes

To move an axis manually with the axis keys:



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



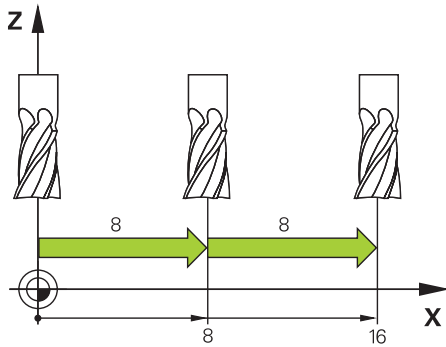
- ▶ Select an application, e.g. **Manual operation**
- ▶ Press the axis key of the desired axis
- ▶ The control moves the axis as long as you press the key.



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.

6.2.2 Incremental jog positioning of axes

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



To position an axis incrementally:



- ▶ Select the **Manual** operating mode

Jog increment

- ▶ Select the **Manual operation** application
- ▶ Select **Jog increment**
 - The control opens the **Positions** workspace, if necessary, and shows the **Jog increment** area.

X+

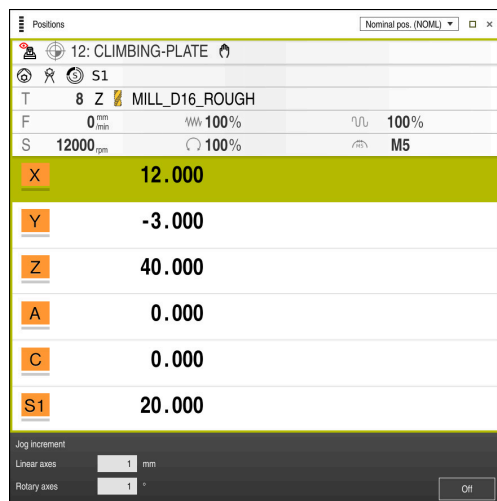
- ▶ Enter the jog increment for linear axes and rotary axes
- ▶ Press the axis key of the desired axis
 - The control positions the axis in the selected direction by the defined jog increment.

Jog increment On

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



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



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

Note

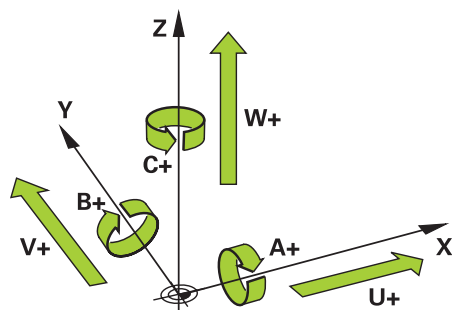
When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.

7

NC Fundamentals

7.1 NC fundamentals

7.1.1 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.

7.1.2 Designation of the axes on milling machines

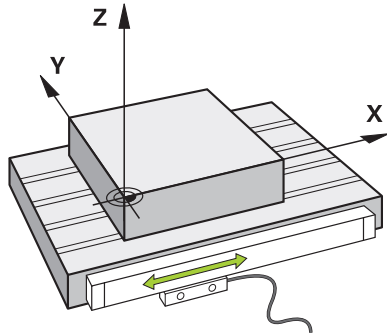
The axes **X**, **Y** and **Z** on your milling machine are designated as the main axis (1st axis), secondary axis (2nd axis) and tool axis. The main axis and secondary axis define the working plane.

The axes are associated as follows:

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

7.1.3 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 178

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

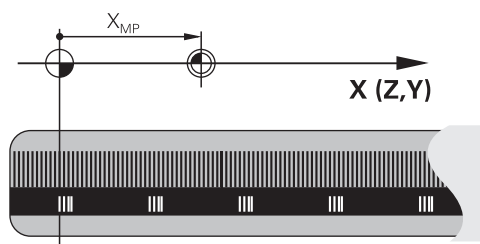
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

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 125








7.1.4 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 140

Icon	Preset
	<p>Machine datum</p> <p>The machine datum is a fixed point defined in the machine configuration by the machine manufacturer.</p> <p>The machine datum is the origin of the machine coordinate system M-CS.</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>If you program M91 in an NC block, the defined values are referenced to the machine datum.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>M92 datum M92-ZP (zero point)</p> <p>The M92 datum is a fixed point defined relative to the machine datum by the machine manufacturer in the machine configuration.</p> <p>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.</p> <p>Further information: User's Manual for Programming and Testing</p>
	<p>Tool change position</p> <p>The tool change position is a fixed point defined relative to the machine datum by the machine manufacturer in the tool-change macro.</p>
	<p>Reference point</p> <p>The reference point is a fixed point for initializing position encoders.</p> <p>Further information: "Position encoders and reference marks", Page 137</p> <p>If the machine has incremental position encoders, the axes must traverse the reference point after booting.</p> <p>Further information: "Axis reference run", Page 125</p>
	<p>Workpiece preset</p> <p>With the workpiece preset you define the origin of the workpiece coordinate system W-CS.</p> <p>Further information: "Workpiece coordinate system W-CS", Page 184</p> <p>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.</p> <p>Further information: "Preset management", Page 193</p> <p>Further information: User's Manual for Programming and Testing</p> <p>If no transformations are defined, the entries in the NC program refer to the workpiece preset.</p>
	<p>Workpiece datum</p> <p>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.</p> <p>If you tilt the working plane (option 8), the workpiece datum is the point around which the workpiece is rotated.</p>

8

Tools

8.1 Fundamentals

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

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

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

Further information: "Tool carrier reference point", Page 141

- Further tool data are needed to completely define the tool. Take these tool data e. g. from the manufacturer's tool catalog.

Further information: "Tool data for the tool types", Page 152

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

Further information: "Tool management ", Page 165

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

Further information: "Tool carrier management", Page 169

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

Further information: User's Manual for Programming and Testing

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

Further information: User's Manual for Programming and Testing

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

Further information: "Tool usage test", Page 172

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

Further information: User's Manual for Programming and Testing

8.2 Presets on the tool

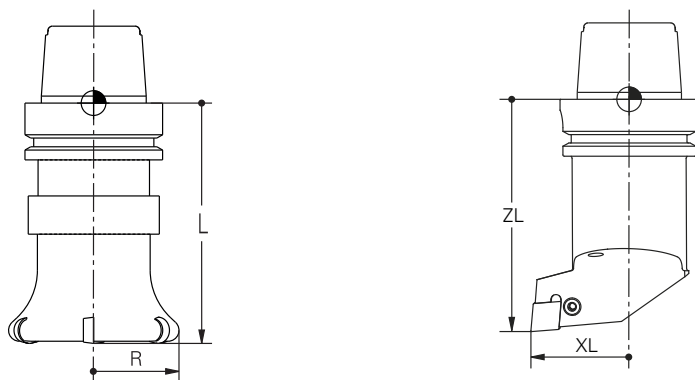
The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

Further information: "Presets in the machine", Page 137

8.2.1 Tool carrier reference point

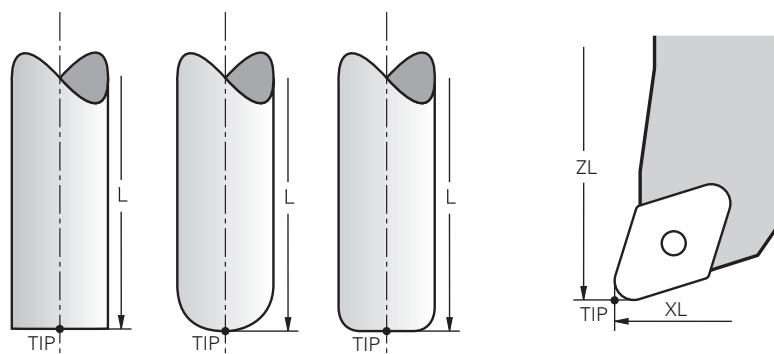


The tool carrier reference point is a fixed point defined by the machine manufacturer. The tool carrier reference point is usually located on the spindle nose.

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

Further information: "Tool management ", Page 165

8.2.2 Tool tip TIP



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

Further information: "Tool coordinate system T-CS", Page 190

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

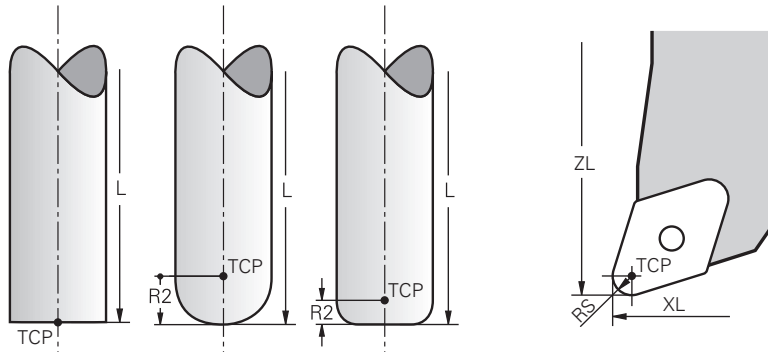
You define the tool tip with the following columns of the tool management relative to the tool carrier reference point:

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

Further information: "Tool data for the tool types", Page 152

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

8.2.3 Tool center point (TCP, tool center point)



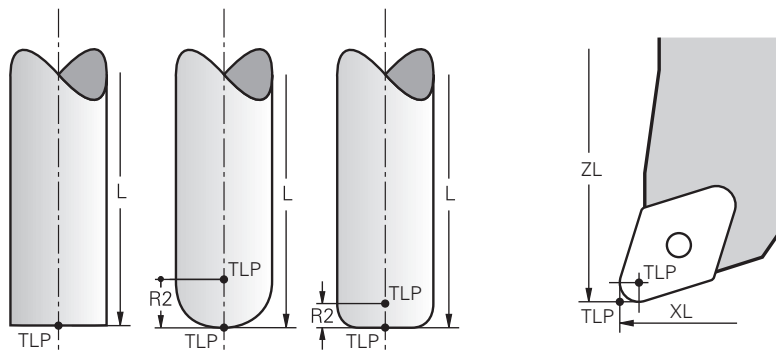
The tool center point is the center of the tool radius **R**. If a second tool radius (**R2**) is defined, the tool center point is offset from the tool tip by this value.

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

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

Further information: "Tool data for the tool types", Page 152

8.2.4 Tool location point (TLP, tool location point)

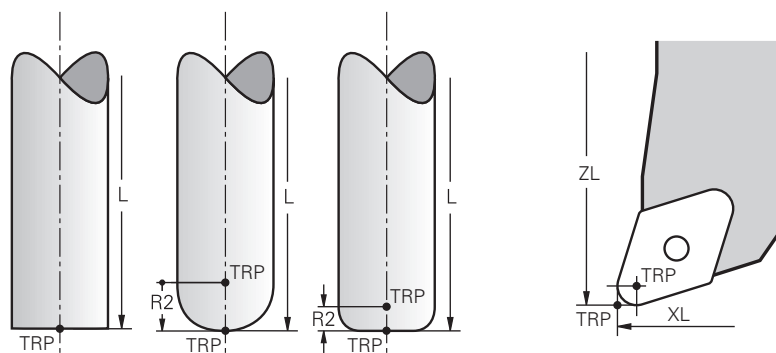


The control positions the tool on the tool location point. By default, the tool location point is at the tool tip.

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

Further information: User's Manual for Programming and Testing

8.2.5 Tool rotation point (TRP, tool rotation point)



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

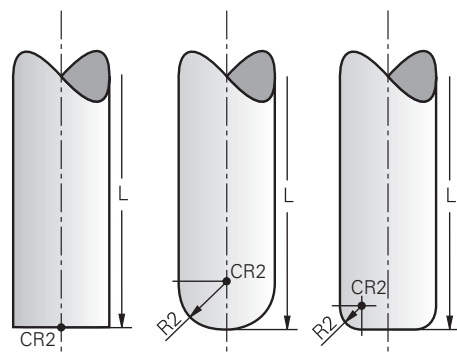
When selecting **MOVE** in **PLANE** functions, the syntax element **DIST** is used to define the relative position between the workpiece and the tool. The control shifts the tool rotation point from the tool tip by this value. When **DIST** is not defined, the control keeps the tool tip constant.

Further information: User's Manual for Programming and Testing

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

Further information: User's Manual for Programming and Testing

8.2.6 Tool radius 2 center (CR2, center R2)



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

Further information: User's Manual for Programming and Testing

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

8.3 Tool data

8.3.1 Tool ID number

Application

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

Further information: "Tool management ", Page 165

Description of function

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

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

Further information: User's Manual for Programming and Testing

8.3.2 Tool name

Application

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

Description of function

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

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

A tool name may contain up to 32 characters.

Permitted characters

You can use the following characters for the tool name:

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

When entering lowercase letters, the control will substitute them by uppercase letters upon saving.

Note

- Assign unique tool names!

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

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



Refer to your machine manual.

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

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

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

8.3.3 Indexed tool

Application

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

Description of function

Tools with multiple lengths and radii cannot be defined in one line of the tool management table. Additional table lines are required, specifying the full definitions of the indexed tools. 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 141

Further information: "Creating an indexed tool", Page 147

Examples of an application of indexed tools:

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

Creating an indexed tool

To create an indexed tool:



- ▶ Select the **Tables** operating mode



- ▶ Select **Tool management**

- ▶ Activate **Edit**

- The control enables tool management for editing.



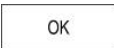
- ▶ Select **Insert tool**

- The control opens the **Insert tool** pop-up window.

- ▶ Define the tool type

- ▶ Define the tool number of the main tool, e. g. **T5**

- ▶ Press **OK**



- The control adds table line **5**.

- ▶ Define all necessary tool data including the maximum tool length.

Further information: "Tool data for the tool types", Page 152



- ▶ Select **Insert tool**

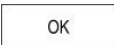
- The control opens the **Insert tool** pop-up window.

- ▶ Define the tool type

- ▶ Define the tool number of the indexed tool, e. g. **T5.1**



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



- ▶ Press **OK**

- The control adds table line **5.1**.

- ▶ Define all required tool data

Further information: "Tool data for the tool types", Page 152



The control does not adopt any main tool data!
The lengths of the indexed tools approach the tool carrier preset as the index rises, starting from the maximum tool length.

Further information: "Tool carrier reference point", Page 141

Notes

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

Further information: "Tool table tool.t", Page 368

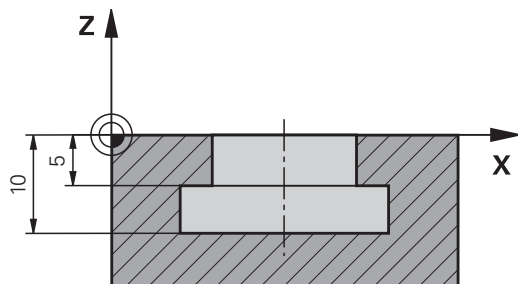
- Index numbers do not need to be sequential. It is possible e. g. to create the tools **T5**, **T5.1** and **T5.3**.

- Up to nine indexed tools can be added to each main tool.

When defining a replacement tool **RT**, this applies to the respective table line exclusively. When an indexed tool is worn and consequently blocked, this also does not apply to all other indices. This ensures e. g. that the main tool can still be used.

Further information: User's Manual for Programming and Testing

Example of T-slot milling cutter



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

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

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



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

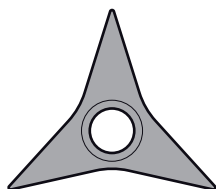
The T-slot is programmed in two machining steps:

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

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

Example FreeTurn tool







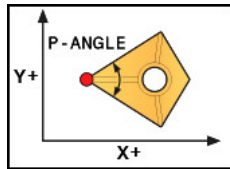



You need the following tool data for a FreeTurn tool:



FreeTurn tool with three finishing teeth



Integrating information about the point angles **P-ANGLE** and the tool length **ZL**, e.g. **FT1_35-35-35_100**, into the tool name is recommended.

Icon and parameter	Meaning	Intended use
 ZL	Tool length 1	<p>The tool length ZL equals the total tool length, relating to the tool carrier preset.</p> <p>Further information: "Presets on the tool", Page 140</p>
 XL	Tool length 2	<p>The tool length XL equals the difference between the spindle center and the tool tip of the tooth. XL must always be defined as a negative value with FreeTurn tools.</p> <p>Further information: "Presets on the tool", Page 140</p>
 YL	Tool length 3	The tool length YL is always 0 with FreeTurn tools.
 RS	Cutting radius	You can take the radius RS from the tool catalog.
 TYPE	Lathe tool type	<p>You select between a rough-turning tool (ROUGH) and finishing tool (FINISH).</p> <p>Further information: "Subgroups of technology-specific tool types", Page 151</p>
 TO	Tool orientation	<p>The tool orientation TO is always 18 with FreeTurn tools.</p> 
 ORI	Angle of orientation	<p>The angle of orientation ORI defines the offset of the single teeth with respect to one another. If the first tooth has the value 0, define the second tooth of symmetrical tools at 120 and the third tooth at 240.</p>
 P-ANGLE	Point angle	You can get the point angle P-ANGLE from the tool catalog.
 CUTLENGTH	Cutting-edge length	You can get the tooth length CUTLENGTH from the tool catalog.
	Toolcarrier kinematics	<p>Using the optional tool-carrier kinematics, the control can monitor e.g. the tool for collisions. Assign the same kinematics to each single tooth.</p>

8.3.4 Tool types

Application

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

Related topics


















- Editing the tool data in the tool management







Further information: "Tool management ", Page 165

Description of function

A number is additionally assigned to each tool type.

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

Icon	Tool type	Number
	Milling cutter (MILL)	0
	Rough cutter (MILL_R)	9
	Finishing cutter (MILL_F)	10
	Ball-nose cutter (BALL)	22
	Toroid cutter (TORUS)	23
	Drill (DRILL)	1
	Tap (TAP)	2
	NC center drill (CENT)	4
	Turning tool (TURN) Further information: "Types within the turning tools", Page 151	29
	Touch probe (TCHP)	21
	Reamer (REAM)	3
	Countersink (CSINK)	5
	Piloted counterbore (TSINK)	6
	Boring tool (Bor)	7
	Back boring tool (BCKBOR)	8
	Thread miller (GF)	1
	Thread miller with chamfer (GSF)	16

Icon	Tool type	Number
	Thread mill with single thread (EP)	17
	Thread mill with indexable insert (WSP)	18
	Thread drilling/milling cutter (BGF)	19
	Circular thread mill (ZBGF)	20
	Grinding wheel (GRIND) Further information: "Types within the grinding tools", Page 152	30
	Dressing tool (DRESS) Further information: "Types within the dressing tools", Page 152	31

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







Further information: "Tool management ", Page 165

Subgroups of technology-specific tool types

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







Types within the turning tools

Select between the types below within the turning tools:

Icon	Tool type	Number
	Rough-turning tool (ROUGH)	11
	Finish-turning tool (FINISH)	12
	Thread-turning tool (THREAD)	14
	Recessing tool (RECESS)	15
	Button tool (BUTTON)	21
	Recess-turning tool (RECTURN)	26






Types within the grinding tools

Select between the types below within the grinding tools:

Icon	Tool type	Number
	Grinding pin (GRIND_M)	1
	Special grinding pin (GRIND_MS)	2
	Cup wheel (GRIND_MT)	3
	Straight wheel (GRIND_S) Currently no function	26
	Slant wheel (GRIND_A) Currently no function	27
	Facing wheel (GRIND_P) Currently no function	28

Types within the dressing tools

Select between the types below within the dressing tools:

Icon	Tool type	Number
	Profile dresser (DIAMOND)	101
	Horn-type dresser (HORNED) Currently no function	102
	Dressing spindle (SPINDLE)	103
	Dressing plate (PLATE)	110
	Dressing roll (ROLL)	120

8.3.5 Tool data for the tool types

Application

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

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

Related topics

- Editing the tool data in the tool management
Further information: "Tool management ", Page 165
- Tool types
Further information: "Tool types", Page 150

Description of function

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

- You can measure your tools in the machine (e. g., with a tool touch probe) or externally with a tool presetter.

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

- Take further tool information from the manufacturer's tool catalog, e. g. the material or the number of teeth.

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

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

- Simulation

Further information: User's Manual for Programming and Testing

- Machining or touch probe cycles

Further information: User's Manual for Machining Cycles












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




- Dynamic Collision Monitoring (DCM, option 40)

Further information: "Dynamic Collision Monitoring (DCM, option 40)",
Page 204

Tool data for milling and drilling tools

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

Icon and parameter	Meaning	Intended use
 L	Length	Required for all milling and drilling tool types
 R	Radius	Required for all milling and drilling tool types
 R2	Radius 2	Required for the drilling and milling tool types below: <ul style="list-style-type: none"> ■ Ball-nose cutter ■ Toroid cutter
 DL	Delta value of length	Optional The control describes this parameter in connection with touch probe cycles.
 DR	Delta value of radius	Optional The control describes this parameter in connection with touch probe cycles.
 DR2	Delta value of radius 2	Optional The control describes this parameter in connection with touch probe cycles.
 LCUTS	Tooth length	Recommended
 RCUTS	Tooth width	Recommended
 LU	Useful length	Recommended
 RN	Neck radius	Recommended
 ANGLE	Plunge angle	Recommended for the drilling and milling tool types below: <ul style="list-style-type: none"> ■ Milling tool ■ ROUGHING MILL ■ Finishing cutter ■ Ball-nose cutter ■ Toroid cutter












Icon and parameter	Meaning	Intended use
 PITCH	Thread pitch	Recommended for the drilling and milling tool types below: <ul style="list-style-type: none"> ■ Tapping tools ■ Thread mill ■ Thread miller with chamfer ■ Thread mill with single thread ■ Thread mill w/ indexable insert ■ Thread drilling/milling cutter ■ Circular thread mill
 T-ANGLE	Point angle	Recommended for the drilling and milling tool types below: <ul style="list-style-type: none"> ■ Drill ■ NC center drill ■ Countersink
 NMAX	Maximum spindle speed	Optional












- All tool types listed in the **TYPE** column are milling and drilling tools except for:
 - **Touch probe**
 - **Turning tool**
 - **Grinding wheel**
 - **Dressing tool****Further information:** "Tool types", Page 150
- The parameters are described in the tool table.
 Further information: "Tool table tool.t", Page 368

Tool data for turning tools (option 50)


The control offers the following parameters for turning tools:











Icon and parameter	Meaning	Intended use
 ZL	Tool length 1	Required for all turning tool types
 XL	Tool length 2	Required for all turning tool types
 YL	Tool length 3	Required for all turning tool types
 RS	Cutting radius	Required for the turning tool types below: <ul style="list-style-type: none"> ■ Roughing tool ■ Finish-turning tool ■ Button tool ■ Recessing tool ■ Recess-turning tool
 TYPE	Lathe tool type	Required for all turning tool types
 TO	Tool orientation	Required for all turning tool types
 DZL	Delta value of tool length 1	Optional The control describes this value in connection with touch probe cycles.
 DXL	Delta value of tool length 2	Optional The control describes this value in connection with touch probe cycles.
 DYL	Delta value of tool length 3	Optional The control describes this value in connection with touch probe cycles.
 DRS	Delta value of cutter radius	Optional The control describes this value in connection with touch probe cycles.
 DCW	Delta value of cutter width	Optional The control describes this value in connection with touch probe cycles.













Icon and parameter	Meaning	Intended use
	Angle of orientation	Required for all turning tool types
ORI		
 T-ANGLE	Tool angle	Required for the turning tool types below: <ul style="list-style-type: none"> ■ Roughing tool ■ Finish-turning tool ■ Button tool ■ Threading tool
 P-ANGLE	Point angle	Required for the turning tool types below: <ul style="list-style-type: none"> ■ Roughing tool ■ Finish-turning tool ■ Button tool ■ Threading tool
	Cutting-edge length	Recommended
 CUTLENGTH		
	Tooth width	Required for the turning tool types below: <ul style="list-style-type: none"> ■ Recessing tool ■ Recess-turning tool
 CUTWIDTH		Recommended for the other turning tool types
 SPB-INSERT	Angular offset	Required for all turning tool types
<div>  <ul style="list-style-type: none"> ■ The TYPE column of the Turning tool tool type as well as the associated technology-specific tool types in the TYPE column define turning tools. Further information: "Tool types", Page 150 Further information: "Types within the turning tools", Page 151 ■ The parameters are described in the turning tool table. Further information: "Turning tool table toolturn.trn (option 50)", Page 377 </div>		

Tool data for grinding tools (option 156)

The control offers the following parameters for grinding tools:

Icon and parameter	Meaning	Intended use
 TYPE	Grinding tool type	Required for all grinding tool types
 R-OVR	Radius	Required for all grinding tool types After initial dressing, this value can only be read.
 L-OVR	Overhang	Required for the grinding tool types below: <ul style="list-style-type: none"> ■ Special grinding pin ■ Cup wheel After initial dressing, this value can only be read.
 LO	Overall length	Required for the grinding tool types below: <ul style="list-style-type: none"> ■ Grinding pin ■ Special grinding pin After initial dressing, this value can only be read.
 LI	Length to the inner edge	Required for the Special grinding pin grinding tool type After initial dressing, this value can only be read.
 B	Width	Required for the grinding tool types below: <ul style="list-style-type: none"> ■ Grinding pin ■ Cup wheel After initial dressing, this value can only be read.
 G	Depth of grinding tool	Required for the Cup wheel grinding tool type After initial dressing, this value can only be read.
ALPHA	Slant angle	Required for the Special grinding pin grinding tool type Unchangeable default value for the grinding tool types below: <ul style="list-style-type: none"> ■ Grinding pin 0° ■ Cup wheel 90°
GAMMA	Corner angle	Required for the grinding tool types below: <ul style="list-style-type: none"> ■ Special grinding pin ■ Cup wheel Unchangeable default value for the Grinding pin 90° tool type

Icon and parameter	Meaning	Intended use
 RV	Radius at the edge for L-OVR	Optional for the grinding tool types below: <ul style="list-style-type: none"> ■ Grinding pin ■ Special grinding pin
 RV1	Radius at the edge for LO	Optional for the grinding tool types below: <ul style="list-style-type: none"> ■ Grinding pin ■ Special grinding pin
 RV2	Radius at the edge for LI	Optional for the Special grinding pin grinding tool type
HW	Wheel has a relief cut	Required for the Cup wheel grinding tool type Optional for the remaining grinding tool types
 HWI	Angle for a relief cut on the inner edge	Required for the Cup wheel grinding tool type Optional for the remaining grinding tool types
 HWA	Angle for a relief cut on the outer edge	Required for the Cup wheel grinding tool type Optional for the remaining grinding tool types
INIT_D_OK	Initial dressing	Required for all grinding tool types The control activates the checkbox after initial dressing. The checkbox can be cleared, which will require another initial dressing process.
 dR-OVR	Delta value of radius	This value can be changed only by cycles.
 dL-OVR	Delta value of overhang	This value can be changed only by cycles.
 dLO	Delta value of total length	This value can be changed only by cycles.
 dLI	Delta value of length up to the inner edge	This value can be changed only by cycles.
 DRESS-N-D	Default value of diameter dressing counter	Optional

Icon and parameter	Meaning	Intended use
	Default value of outer edge dressing counter	Optional
DRESS-N-A		
	Default value of inner edge dressing counter	Optional
DRESS-N-I		
	Diameter dressing counter	The control increments this value.
DRESS-N-D-ACT		
	Outer edge dressing counter	The control increments this value.
DRESS-N-A-ACT		
	Inner edge dressing counter	The control increments this value.
DRESS-N-I-ACT		
	Radius of the tool shank	Optional
R_SHAFT		
	Min. permissible radius	Optional
R_MIN		
	Min. permissible width	Optional
B_MIN		
	Maximum permissible cutting speed	Optional
V_MAX		
	Retraction amount at the diameter	Required for all grinding tool types
AD		
	Retraction amount at the outer edge	Required for all grinding tool types
AA		
	Retraction amount at the inner edge	Required for all grinding tool types
AI		



- The **TYPE** column of the **Grinding wheel** tool type as well as the associated technology-specific tool types in the **TYPE** column define grinding tools.

Further information: "Tool types", Page 150











Further information: "Types within the grinding tools", Page 152

- The parameters are described in the grinding tool table.

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 382

Tool data for dressing tools (option 156)

The control offers the following parameters for dressing tools:

Icon and parameter	Meaning	Intended use
 ZL	Tool length 1	Required for dressing tool types
 XL	Tool length 2	Required for all dressing tool types
 YL	Tool length 3	Required for all dressing tool types
 RS	Cutting radius	Required for the dressing tool types below: <ul style="list-style-type: none"> ■ Profile dresser ■ Dressing spindle
CUTWIDTH	Width of tooth	Required for the dressing tool types below: <ul style="list-style-type: none"> ■ Dressing plate ■ Dressing roll
 TYPE	Dressing tool type	Required for all dressing tool types
 TO	Tool orientation	Required for all dressing tool types
 DZL	Delta value of tool length 1	Optional
 DXL	Delta value of tool length 2	Optional
 DYL	Delta value of tool length 3	Optional
 DRS	Delta value of cutter radius	Optional
N-DRESS	Tool speed	Required for the dressing tool types below: <ul style="list-style-type: none"> ■ Dressing spindle ■ Dressing roll



- The **TYPE** column of the **Dressing tool** tool type as well as the associated technology-specific tool types in the **TYPE** column define dressing tools.

Further information: "Tool types", Page 150













Further information: "Types within the dressing tools", Page 152



- The parameters are described in the dressing tool table.

Further information: "Dressing tool table tooldress.drs (option 156)", Page 389

Tool data for touch probes

The control offers the following parameters for touch probes:

Icon and parameter	Meaning	Intended use
 L	Length	Required
 R	Radius	Required
TP_NO	Number in the touch probe table	Required
 TYPE	Type of touch probe	Required
 F	Probing feed rate	Required
 FMAX	Rapid traverse in probing cycle	Optional
 F_PREPOS	Pre-positioning at rapid traverse	Required
 TRACK	Orienting the touch probe in each probing process	Required
 REACTION	Trigger NCSTOP or EMERGSTOP in case of collision	Required
 SET_UP	Set-up clearance	Recommended
 DIST	Maximum measuring range	Recommended
 CAL_OF1	Center offset in the main axis	Required if TRACK = ON The control describes this value in connection with the calibration cycle.
 CAL_OF2	Center offset in the secondary axis	Required if TRACK = ON The control describes this value in connection with the calibration cycle.

Icon and parameter	Meaning	Intended use
 CAL_ANG	Spindle angle during calibration	Required if TRACK = OFF
<div>  <ul style="list-style-type: none"> The TYPE column of the Touch probe tool type as well as the touch probe model in the TYPE column define touch probes. Further information: "Tool types", Page 150 The parameters are described in the touch probe table. Further information: "Touch probe table tchprobe.tp", Page 392 </div>		

8.4 Tool management

Application

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

The tool management allows adding tools, editing tool data and deleting tools.

Related topics

- Creating new tools
Further information: "Configuring a tool", Page 83
- Table workspace
Further information: "Table workspace", Page 359
- Form workspace
Further information: "Form workspace for tables", Page 363

Description of function

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

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

- Tool table **tool.t**
Further information: "Tool table tool.t", Page 368
- Turning tool table **toolturn.trn** (option 50)
Further information: "Turning tool table toolturn.trn (option 50)", Page 377
- Grinding tool table **toolgrind.grd** (option 156)
Further information: "Grinding tool table toolgrind.grd (option 156)", Page 382
- Dressing tool table **tooldress.drs** (option 156)
Further information: "Dressing tool table tooldress.drs (option 156)", Page 389
- Touch probe table **tchprobe.tp**
Further information: "Touch probe table tchprobe.tp", Page 392

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

Further information: "Pocket table tool_p.tch", Page 396

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

Further information: "Tool data", Page 145

Notes

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

8.4.1 Importing and exporting tool data

Application

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

Description of function

The control transmits tool data as a CSV file.

Further information: User's Manual for Programming and Testing

The tool data transfer file is structured as follows:

- The first line contains the tool table column names that are transferred.
- The other lines contain the tool data to be transferred. The order of the data must match the order of the column names in line 1. Decimal numbers are separated by a point.

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

Please note the following regarding the transfer file:

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

Importing tool data

To import tool data:



- ▶ Select the **Tables** operating mode



- ▶ Select **Tool management**

- ▶ Activate **Edit**

- ▶ The control enables tool management for editing.



- ▶ Select **Import**

- ▶ The control opens a selection window.

- ▶ Select the desired file



- ▶ Select **Import**

- ▶ The control adds the tool data to the tool management.

- ▶ If required, the control opens the **Confirm import** window, e. g. in case of identical tool numbers.

- ▶ Selecting the procedure:

- **Append:** the control adds the tool data into new lines at the end of the table.
- **Overwrite:** the control overwrites the initial tool data with the tool data from the transfer file.
- **Cancel:** the control cancels the import process.

NOTICE

Caution: Data may be lost!

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

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

Exporting tool data

To export tool data:



- ▶ Select the **Tables** operating mode



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

Further information: User's Manual for Programming and Testing

- ▶ Select **Mark row**
- ▶ Mark further tools if required
- ▶ Select **Export**
- The control opens the **Save as** window.
- ▶ Select a path

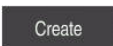


By default, the control saves the transfer file under **TNC:\table**.

- ▶ Enter the file name
- ▶ Select the file type



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



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

Notes

NOTICE

Caution: Possible material damage!

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

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

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

The TNC7 can both import and export both transfer files.

8.5 Tool carrier management

Application

Tool carrier management allows parameterizing and assigning tool carriers.

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

Related topics

- **Simulation** workspace

Further information: User's Manual for Programming and Testing

- Dynamic Collision Monitoring (DCM, option 40)

Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204

Description of function

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

- Save the tool carrier or tool carrier templates
- Parameterize the tool carrier templates

Further information: "Parameterizing tool carrier templates", Page 171

- Assign a tool carrier

Further information: "Assigning a tool carrier", Page 171



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

Tool carriers in STL format must meet the following requirements:

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

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

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

Further information: "Options for fixture files", Page 211

Tool-carrier templates

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

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



They can be downloaded through the following link:

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

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

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

Further information: "Parameterizing tool carrier templates", Page 171

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

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

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

8.5.1 Parameterizing tool carrier templates

To parameterize a tool carrier template:



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



8.5.2 Assigning a tool carrier

To assign a tool carrier to a tool:



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



- ▶ In the **Spec. functions** panel, select the **KINEMATIC** parameter
- > The control displays the available tool carriers in the **Tool-carrier kinematics** window.
- ▶ Select the desired tool carrier
- ▶ Select **OK**
- > The control assigns the tool carrier to the tool.



- The tool carrier will only be taken into account after the next tool call.
 - Parameterized tool carriers can consist of several subfiles. If the subfiles are incomplete, the control will display an error message. Only use fully parameterized tool carriers and error-free STL or M3D files!
For tool carriers, the same requirements with respect to STL and M3D files apply as for fixtures.
- Further information:** "Fixture monitoring (option 40)", Page 210

Notes

- In the simulation, the tool carriers can be checked for collisions with the workpiece.
Further information: User's Manual for Programming and Testing
- On 3-axis machines with rectangular angle heads, tool carriers of angle heads are advantageous in connection with the tool axes **X** and **Y** because the control takes the dimensions of the angle heads into account.
 HEIDENHAIN recommends machining with tool axis **Z**. Using Advanced Functions Set 1 (software option 8), you can tilt the working plane to the angle of exchangeable angle heads and continue working with tool axis **Z**.
- The control monitors the tool carriers by means of Dynamic Collision Monitoring (DCM, option 40). This enables the tool carriers to protect against collisions with fixtures or machine components.
Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204
- A grinding tool that is to be dressed must not contain any tool carrier kinematics (option 156).

8.6 Tool usage test

Application

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

Related topics

- Contents of the tool usage file
Further information: "Tool usage file", Page 398
- Tool usage test in Batch Process Manager (option 154)
Further information: User's Manual for Programming and Testing

Requirement

- A tool usage file is needed for performing a tool usage test
 The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
Further information: "Tool usage file", Page 398
- The **Generate tool-usage file** function setting is set to either **once** or **always**
Further information: "Channel settings", Page 449
- Use the same tool table for the simulation as for the program run
Further information: User's Manual for Programming and Testing

Description of function

Creating the tool usage file

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

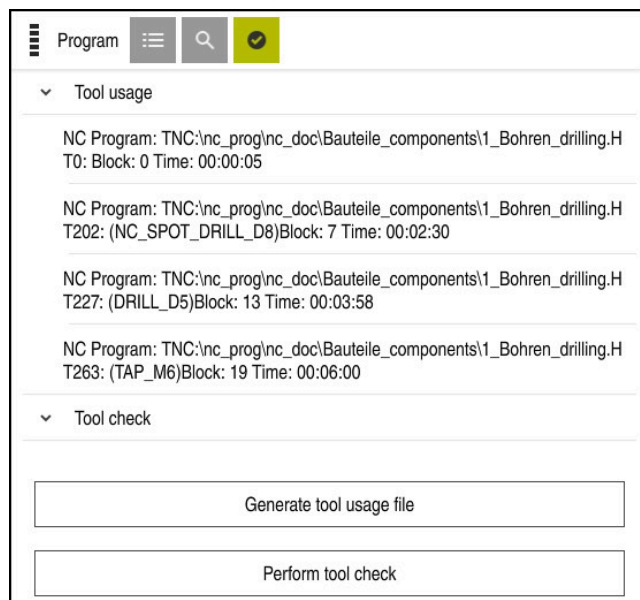
When setting the **Generate tool-usage file** setting to **once** or **always**, the control will generate a tool usage file in the following cases:

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

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

Further information: "Tool usage file", Page 398

Tool check column in the Program workspace



Tool check column in the **Program** workspace

The control displays the **Tool usage** and **Tool check** areas in the **Tool check** column of the **Program** workspace.

Further information: User's Manual for Programming and Testing

Tool usage area

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

Further information: "Creating the tool usage file", Page 173

Further information: "Tool usage file", Page 398

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

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

Tool check area

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

Further information: "Performing the tool usage test", Page 175

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

- The tool is defined in the tool management

Further information: "Tool management ", Page 165

- The tool is defined in the pocket table

Further information: "Pocket table tool_p.tch", Page 396

- The tool has sufficient remaining tool life

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

Further information: "Tool table tool.t", Page 368

Further information: "Tool usage file", Page 398

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

- **OK:** All tools are available and have sufficient remaining tool life
- **No suitable tool:** The tool is not defined in the tool management
In this case, check if the correct tool is selected in the tool call. Otherwise, create the tool in the tool management.
- **External tool:** The tool is defined in the tool management, but not in the pocket table
If your machine is equipped with a magazine, position the missing tool in the magazine.
- **Insufficient remaining tool life:** The tool is blocked or does not have sufficient remaining tool life
Change the tool or use a replacement tool.

Further information: User's Manual for Programming and Testing

8.6.1 Performing the tool usage test

Use the tool usage test as follows:



- ▶ Select the **Home** operating mode



- ▶ Select the **Settings** application



- ▶ Select the **Machine settings** group



- ▶ Select the **Machine settings** menu item

- ▶ In the **Channel settings** area, select Generate tool usage file **Once** for the simulation.

Further information: "Channel settings", Page 449



- ▶ Press **Apply**



- ▶ Select the **Editor** operating mode



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



- ▶ Select **Open**
- The control opens the NC program in a new tab.



- ▶ Select the **Tool check** column
- The control opens the **Tool check** column.
- ▶ Select **Generate tool usage file**
- The control generates a tool usage file and displays the tools used in the **Tool usage** area.

Further information: "Tool usage file", Page 398

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

Notes

- If you selected **never** in the **Generate tool-usage file** function, the **Generate tool usage file** button in the **Tool check** column is grayed-out.
Further information: "Channel settings", Page 449
- The **Simulation settings** window allows selecting when the control generates a tool usage file for the simulation.
Further information: User's Manual for Programming and Testing
- The control creates dependency files (***.dep**); for example, the tool-usage file in order to perform a tool usage test.
Further information: "Tool usage file", Page 398
- The control displays the order of tool calls of the NC program that is active in the program run in the **T usage order** table (option 93).
Further information: "T usage order (option 93)", Page 400
- The control shows an overview of all tool calls of the NC program that are active in the program run in the **Tooling list** table (option 93).
Further information: "Tooling list (option 93)", Page 402
- Function **FN 18: SYSREAD ID975 NR1** allows querying the tool usage test for an NC program.
- Function **FN 18: SYSREAD ID975 NR2 IDX** allows querying the tool usage test for a pallet table. The pallet table row is defined by **IDX**.
- The machine manufacturer uses the machine parameter **autoCheckPrg** (no. 129801) to define whether the control automatically generates a tool usage file upon selecting an NC program.
- The machine manufacturer uses the machine parameter **autoCheckPal** (no. 129802) to define whether the control automatically generates a tool usage file upon selecting a pallet table.
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files with the *.dep extension in the file manager. Even if the control displays no dependency files, it still generates a tool usage file.

9

Coordinate Transformation

9.1 Reference systems

9.1.1 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 179
B-CS	Basic coordinate system basic coordinate system	Page 182
W-CS	Workpiece coordinate system workpiece coordinate system	Page 184
WPL-CS	Working plane coordinate system working plane coordinate system	Page 186
I-CS	Input coordinate system input coordinate system	Page 188
T-CS	Tool coordinate system tool coordinate system	Page 190

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.

9.1.2 Basics of coordinate systems

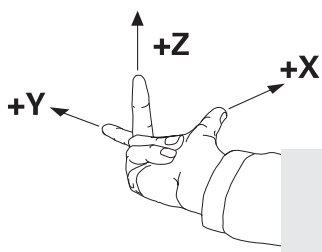
Types of coordinate systems

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.

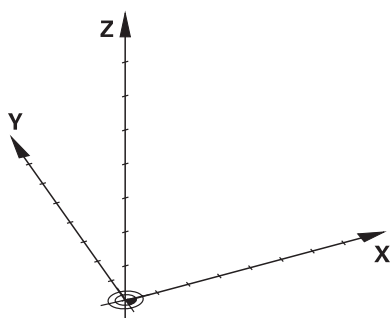
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

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**.



9.1.3 Machine coordinate system M-CS

Application

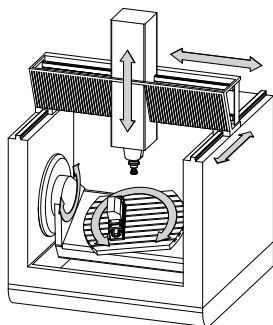
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

Description of function

Properties of M-CS machine coordinate system

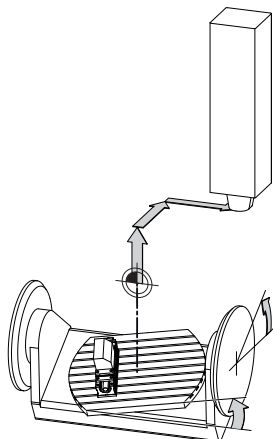
The machine coordinate system **M-CS** corresponds to the kinematics description and therefore to the actual mechanical design of the machine tool. The physical axes of a machine tool are not necessarily always exactly perpendicular to each other, and therefore do not represent a Cartesian coordinate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

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



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

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



Position of the machine datum in the machine

Transformations in the machine coordinate system M-CS

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

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

Further information: "Preset table", Page 403



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

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

Further information: "Global Program Settings (GPS, option 44)", Page 238



The machine manufacturer can also define further transformations.

Further information: "Note", Page 181

Position display

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

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

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

Programming coordinate entry in machine coordinate system M-CS

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

Further information: User's Manual for Programming and Testing

Note

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

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

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

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

Example

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

Traverse movement without M91

```
11 L IY+10
```

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

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

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

Traverse movement with M91

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

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

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

9.1.4 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**.

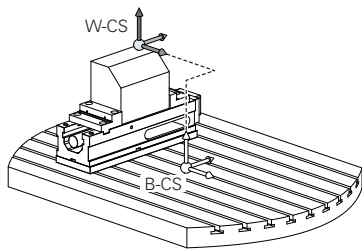
Transformations in the basic coordinate system B-CS

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 193



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

The machine manufacturer can also define further transformations.

Further information: "Note", Page 183

Note

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

NOTICE

Danger of collision!

The control may feature an additional pallet preset table, depending on the machine. Values that the machine manufacturer defined in the pallet preset table take effect before values that you defined in the preset table. Since the values of the pallet preset table are neither visible nor editable, there is a risk of collision during any movement!

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

9.1.5 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.

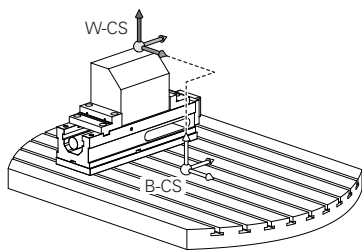
Description of function

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 193



Transformations in the workpiece coordinate system (W-CS)

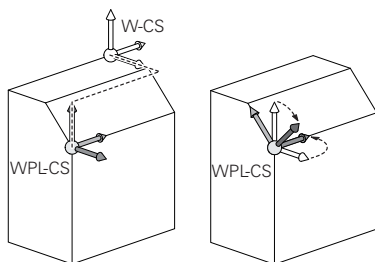
HEIDENHAIN recommends using the following transformations in the workpiece coordinate system **W-CS**:

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



The control also offers Cycle **19 WORKING PLANE** for tilting the working plane.

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



NOTICE**Danger of collision!**

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

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



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

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

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

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

In the **GS** workspace (option 44) you can define additional transformations in the workpiece coordinate system **W-CS**:

- **Additive basic rotat. (W-CS)**
The effects of this function are added to a basic rotation or a 3D basic rotation from the preset table or the pallet preset table. This function is the first transformation that is possible in the **W-CS**.
- **Shift (W-CS)**
This function is in effect in addition to a datum shift defined in the NC program with the **TRANS DATUM** function and before the working plane is tilted.
- **Mirroring (W-CS)**
The function is effective in addition to a mirror image (function **TRANS MIRROR** or Cycle **8 MIRRORING**) defined in the NC program and before tilting the working plane.
- **Shift (mW-CS)**
This function is in effect in the modified workpiece coordinate system. This function is in effect after the **Shift (W-CS)** and **Mirroring (W-CS)** functions and before the working plane is tilted.

Further information: "Globale Programmeinstellungen GPS", Page

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.

Further information: "Input coordinate system I-CS", Page 188

- 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 186

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

9.1.6 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 188

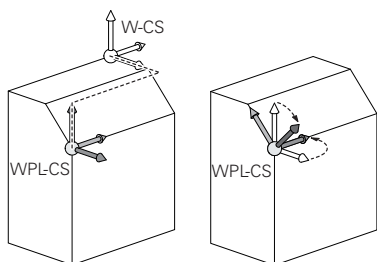
Description of function

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 184

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

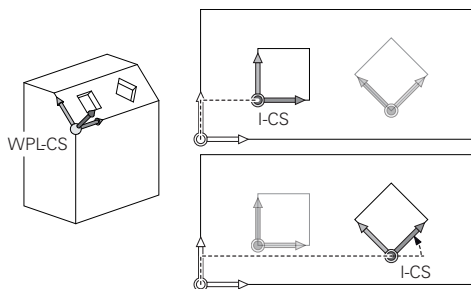


Transformations in the working plane coordinate system WPL-CS

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

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

With these transformations you modify the position and orientation of the input coordinate system **I-CS**.



NOTICE

Danger of collision!

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

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

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

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

Further information: "Globale Programmeinstellungen GPS", Page

Additional transformations with mill-turning (option 50)

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

- Precession angle with the following cycles:
 - Cycle **800 ADJUST XZ SYSTEM**
 - Cycle **801 RESET ROTARY COORDINATE SYSTEM**
 - Cycle **880 GEAR HOBBING**
- OEM transformations defined by machine manufacturers for special turning kinematics



Machine manufacturers can also define an OEM transformation and a precession angle without software option 50.

An OEM transformation takes effect before the precession angle.

If an OEM transformation or a precession angle is defined, the control shows the values on the **POS** tab of the **Status** workspace. These transformations are also in effect in milling mode!

Further information: "POS tab", Page 108

Additional transformation with Gear Cutting (option 157)

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

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



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

Notes

- The programmed values in the NC program refer to the input coordinate system **I-CS**. If you do not program any transformations in the NC program, then the origin and position of the workpiece coordinate system **W-CS**, the working plane coordinate system **WPL-CS**, and the **I-CS** are identical.
Further information: "Input coordinate system I-CS", Page 188
- During pure 3-axis machining, the workpiece coordinate system **W-CS** and the working plane coordinate system **WPL-CS** are identical. In this case, all transformations influence the input coordinate system **I-CS**.
- The result of transformations built upon each other depends on the programming sequence.
- As a **PLANE** function (option 8), **PLANE RELATIV** has an effect in the workpiece coordinate system **W-CS** and orients the working plane coordinate system **WPL-CS**. The values of additive tilting always relate to the current **WPL-CS**.

9.1.7 Input coordinate system I-CS

Application

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

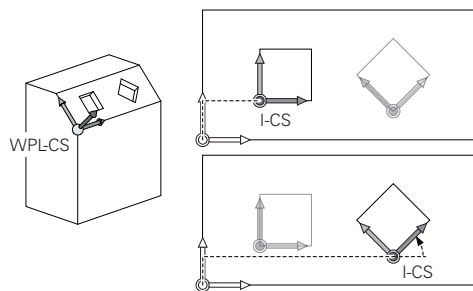
Description of function

Properties of the input coordinate system I-CS

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

Further information: "Working plane coordinate system WPL-CS", Page 186

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 190

You can define the following positioning blocks:

- Paraxial positioning blocks
- Path functions with Cartesian or polar coordinates
- Straight lines **LN** with Cartesian coordinates and surface normal vectors (option 9)
- Cycles

11 X+48 R+	; Paraxial positioning block
-------------------	------------------------------

11 L X+48 Y+102 Z-1.5 R0	; Path function L
---------------------------------	--------------------------

11 LN X+48 Y+102 Z-1.5 NX-0.04658107 NY0.00045007 NZ0.8848844 R0	; Straight line LN with Cartesian coordinates and surface normal vector
---	--

Position display

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

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

Notes

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

Further information: "Working plane coordinate system WPL-CS", Page 186

9.1.8 Tool coordinate system T-CS

Application

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

Description of function

Properties of the tool coordinate system T-CS

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

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

Further information: "Presets in the machine", Page 137

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

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

Further information: "Tool carrier reference point", Page 141

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

Further information: "Input coordinate system I-CS", Page 188

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

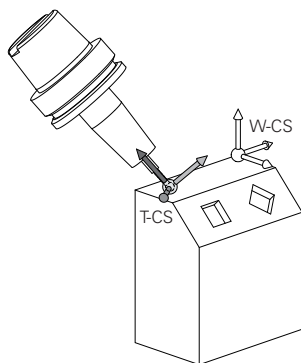
Further information: User's Manual for Programming and Testing

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

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

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

Further information: User's Manual for Programming and Testing



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

Further information: User's Manual for Programming and Testing

Further information: User's Manual for Programming and Testing

11 L X+10 Y+45 A+10 C+5 R0 M128

; Straight line with miscellaneous function **M128** and axis angles

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

11 FUNCTION TCPM F TCP AXIS SPAT
PATHCTRL AXIS

; **FUNCTION TCPM** with spatial angles

12 L A+0 B+45 C+0 R0 F2500

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

; Straight line **LN** with surface normal vector and tool orientation

Transformations in the tool coordinate system T-CS

The following tool compensations have an effect in the tool coordinate system **T-CS**:

- Compensation values from the tool management
Further information: User's Manual for Programming and Testing
- Compensation values from the tool call
Further information: User's Manual for Programming and Testing
- Values of the compensation tables ***.tco**
Further information: User's Manual for Programming and Testing
- Values of **FUNCTION TURNDATA CORR T-CS** (option 50)
Further information: User's Manual for Programming and Testing
- 3D tool compensation with surface normal vectors (option 9)
Further information: User's Manual for Programming and Testing
- 3D tool radius compensation depending on the tool's contact angle using compensation-value tables (option 92)
Further information: User's Manual for Programming and Testing

Position display

The display of the virtual tool axis **VT** refers to the tool coordinate system **T-CS**.

The control shows the values of **VT** in the **GS workspace** (option 44) and on the **GS** tab of the **Status** workspace.

Further information: "Global Program Settings (GPS, option 44)", Page 238

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

Further information: "Contents of an electronic handwheel display", Page 418

9.2 Preset management

Application

The preset management allows setting and activating single presets. The presets to be saved may include e. g. the position and the misalignment of a workpiece in the preset table. The active row of the preset table serves as the workpiece preset in the NC program and as the origin of workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 137

Use the preset management in the following cases:

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

Related topics

- Contents of preset table, write protection

Further information: "Preset table", Page 403

Description of function

Setting presets

Presets can be set in the following ways:

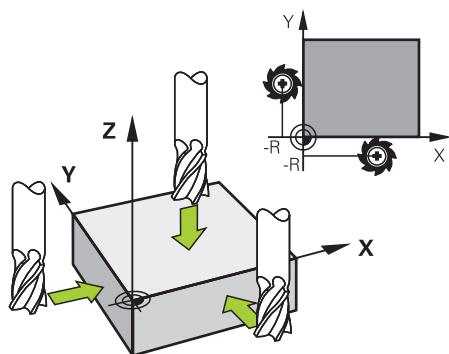
- Setting axis positions manually
Further information: "Setting a preset manually", Page 196
- Touch probe cycles in the **Setup** application
Further information: "Touch Probe Functions in the Manual Operating Mode", Page 305
- Touch probe cycles in the NC program
Further information: User's Manual for Measuring Cycles for Workpieces and Tools

If you try to write a value in a write-protected preset table row, the control cancels this process with an error message. Write-protection for this row must be rescinded first.

Further information: "Removing write protection", Page 406

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

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

Further information: "Setting a preset manually", Page 196

Activating presets

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

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

Presets can be activated in the following ways:

- Activating manually in the **Tables** operating mode
Further information: "Activating a preset manually", Page 197
- Cycle **247 PRESETTING**
Further information: User's Manual for Machining Cycles
- **PRESET SELECT** function
Further information: User's Manual for Programming and Testing

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

- Datum shift with the **TRANS DATUM** function
 - Mirror image with the **TRANS MIRROR** function or Cycle **8 MIRRORING**
 - Rotation with the **TRANS ROTATION** function or Cycle **10 ROTATION**
 - Scaling with the **TRANS SCALE** function or Cycle **11 SCALING FACTOR**
 - Axis-specific scaling with Cycle **26 AXIS-SPECIFIC SCALING**
- Tilting the working plane by using **PLANE** functions or Cycle **19 WORKING PLANE** will not be reset by the control.

Basic rotation and 3D basic rotation

The **SPA**, **SPB** and **SPC** columns define a spatial angle for orienting the workpiece coordinate system **W-CS**. This spatial angle defines the basic rotation or 3D basic rotation of the preset.

Further information: "Workpiece coordinate system W-CS", Page 184

When a rotation around the tool axis is defined, the preset contains a basic rotation, e. g. **SPC** for tool axis **Z**. If one of the remaining columns is defined, the preset contains a 3D basic rotation. If the workpiece preset contains a basic rotation or 3D basic rotation, the control takes these values into account when executing an NC program.

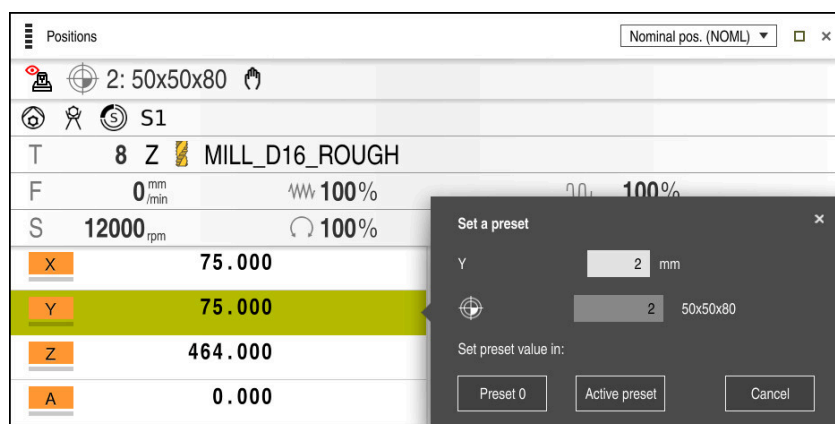
The **3D ROT** button (option 8) allows defining that the control takes a basic rotation or 3D basic rotation into account in the **Manual operation** application as well.

Further information: User's Manual for Programming and Testing

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

Further information: User's Manual for Programming and Testing

9.2.1 Setting a preset manually



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

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

To set a preset manually in an axis:



- ▶ Select the **Manual operation** application in the **Manual** operating mode
- ▶ Open the **Positions** workspace
- ▶ Traverse the tool to the desired position, e. g. scratching
- ▶ Select the row of the desired axis
- ▶ The control opens the **Set a preset** window.
- ▶ Enter the value of the current axis position, relating to the new preset, e. g. **0**
- ▶ The control activates the **Preset 0** and **Active preset** buttons as options.
- ▶ 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.

Active preset



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

9.2.2 Activating a preset manually

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

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

To activate a preset manually:



- ▶ Select the **Tables** operating mode

- ▶ Select the **Presets** application

- ▶ Select the desired row

- ▶ Select **Activate the preset**

- > The control activates the preset.

- > The control shows the number of the active preset in the **Positions** workspace and in the status overview.

Activate
the preset

Further information: "Description of function", Page 93

Further information: "Status overview on the control bar", Page 99

Notes

- The machine manufacturer uses the optional **initial** machine parameter (no. 105603) to define a default value for each column of a new row.
- In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). If the rotary axes are positioned so that they differ from the definition in the **3-D rotation** window, the control cancels this process with an error message by default.

Further information: User's Manual for Programming and Testing

In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.

- When scratching a workpiece with the radius of a milling cutter, the radius value must be taken into account in the preset.
- Even if the current preset contains a basic rotation or a 3D basic rotation, the **PLANE RESET** function will position the rotary axes at 0° in the **MDI** application.

Further information: "Application MDI", Page 329

- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

Further information: User's Manual for Programming and Testing

9.3 Tilting the working plane (option 8)

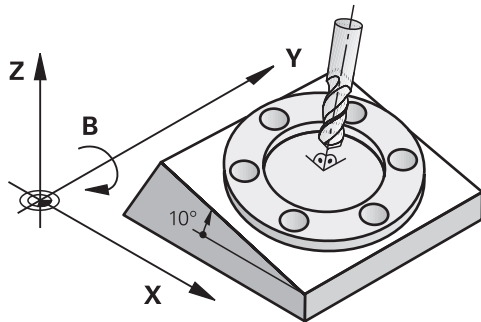
9.3.1 Fundamentals

Machines with rotary axes allow machining e. g. 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 186



There are three functions available for tilting the working plane:

- Manual tilting with the **3-D rotation** window in the **Manual operation** application

Further information: "3-D rotation window (option 8)", Page 199

- Tilting under program control with the **PLANE** functions in the NC program

Further information: User's Manual for Programming and Testing

- Tilting under program control with Cycle **19 WORKING PLANE**

Further information: User's Manual for Machining Cycles

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 182

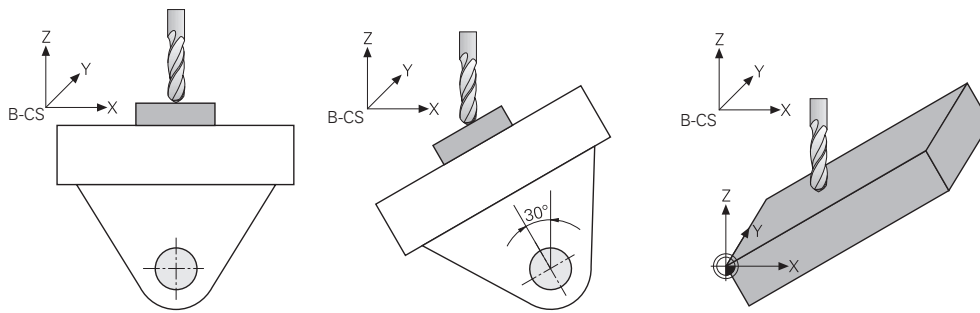
When tilting the working plane, the control moves the machine axes according to the kinematics.

Please observe the aspects below regarding the machine kinematics:

■ Machine with table rotary axes

With this kinematics, 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**.

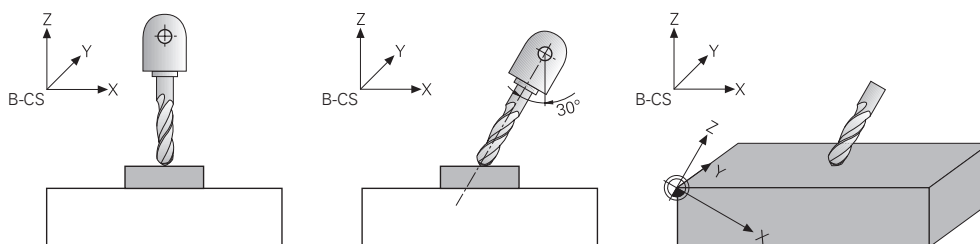
Further information: "Working plane coordinate system WPL-CS", Page 186



■ Machine with head rotary axes

With this kinematics, 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 186



9.3.2 3-D rotation window (option 8)

Application

The **3-D rotation** window allows activating and deactivating tilting of the working plane for the **Manual** and **Program Run** operating modes. This allows restoring the tilted working plane and retracting the tool, e. g. after program cancellation in the **Manual operation** application.

Related topics

■ Tilting the working plane in the NC program

Further information: User's Manual for Programming and Testing

■ Reference systems of the control

Further information: "Reference systems", Page 178

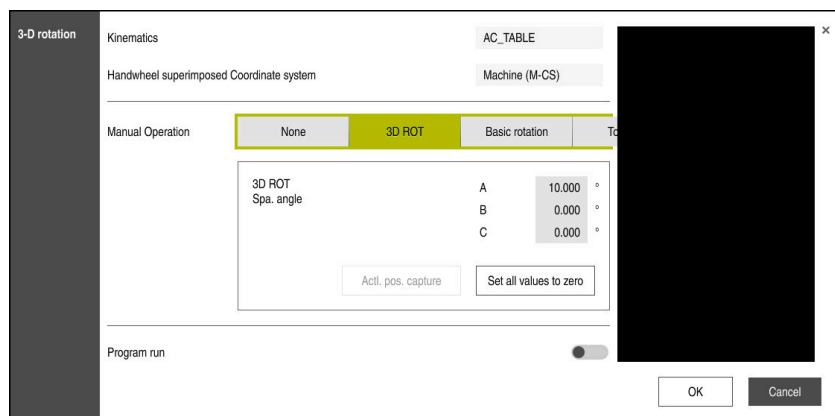
Requirements

- Machine with rotary axes
- Kinematics description
To calculate the tilting angles, the control requires a kinematics description prepared by the machine manufacturer.
- Advanced Functions Set 1 (software option 8)
- Function enabled by the machine manufacturer
In the machine parameter **rotateWorkPlane** (no. 201201), the machine manufacturer defines whether tilting the working plane is allowed on the machine.
- Tool with tool axis **Z**

Description of function

The **3-D rotation** window can be opened with the **3D ROT** button in the **Manual operation** application.

Further information: "Manual operation application", Page 130



3-D rotation window

The **3-D rotation** window contains the following information:

Area	Contents
Kinematics	Name of the active machine kinematics
Handwheel superimpositioning coordinate system	Coordinate system in which handwheel superimpositioning is effective Further information: "Reference systems", Page 178 Further information: "Function Handwheel superimp.", Page 247 Further information: User's Manual for Programming and Testing

Area	Contents
Manual Operation	<p>Status of tilting function in the Manual operating mode:</p> <ul style="list-style-type: none"> ■ None The control will not take the rotary axes positions that are not equal to 0 into account. Traverses are effective in the W-CS workpiece coordinate system. Further information: "Workpiece coordinate system W-CS", Page 184 ■ 3D ROT The control takes the positions of rotary axes and columns SPA, SPB and SPC of the preset table into account. The traverses are effective in the WPL-CS working plane coordinate system. Further information: "3D ROT setting", Page 201 ■ 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 are effective in the W-CS workpiece coordinate system. Further information: "Basic rotation setting", Page 202 ■ Tool axis This is relevant only for head rotary axes. The traverses are effective in the T-CS tool coordinate system. Further information: "Tool axis setting", Page 202
Program run	<p>When activating the Tilt working plane function for the Program run operating mode, the entered angle of rotation applies from the first NC block of the NC program to be run.</p> <p>If you use Cycle 19 WORKING PLANE or the PLANE function in the NC program, the angular values defined there become effective. The control will reset the entered angular values to 0.</p>

Confirm the settings with **OK**.

When a tilting function is active in the **3-D rotation** window, the control displays the corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 93

3D ROT setting

When selecting the **3D ROT** setting, all axes move in the tilted working plane. The traverses are effective in the **WPL-CS** working plane coordinate system.

Further information: "Working plane coordinate system WPL-CS", Page 186

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

The control displays the angles that are currently effective in the entry field of the **Manual Operation** area. The spatial angle can also be edited.

The **3D ROT** entry field in the **Manual Operation** area displays the currently active angles. 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.



When editing values in the **3D ROT** entry field, the rotary axes must be positioned afterwards, e. g. in the **MDI** application.

Basic rotation setting

When selecting the **Basic rotation** setting, the axes move while taking a basic rotation or 3D basic rotation into account.

Further information: "Basic rotation and 3D basic rotation", Page 195

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

Further information: "Workpiece coordinate system W-CS", Page 184

If the active workpiece preset contains a basic rotation or 3D basic rotation, the control additionally displays the corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 93

The **3D ROT** entry field has no function in this setting.

Tool axis setting

When selecting the **Tool axis** setting, traversing is possible in both the positive and negative direction of the tool axis. The control locks all other axes. This setting makes sense only for machines with head rotary axes.

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

Further information: "Tool coordinate system T-CS", Page 190

This setting is used e. g. in the cases below:

- When retracting the tool in the direction of the tool axis during an interruption of a 5-axis machining program.
- When traversing with the axis keys or the handwheel with a pre-positioned tool.

The **3D ROT** entry field has no function in this setting.

Notes

- The control uses the **COORD ROT** transformation type in the following situations:
 - if a **PLANE** function was previously executed with **COORD ROT**
 - after **PLANE RESET**
 - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- The control uses the **TABLE ROT** transformation type in the following situations:
 - if a **PLANE** function was previously executed with **TABLE ROT**
 - with corresponding configuration of the machine parameter **CfgRot-WorkPlane** (no. 201200) by the machine manufacturer
- When setting a preset, the positions of the rotary axes must match the tilting situation in the **3-D rotation** window (option 8). If the rotary axes are positioned so that they differ from the definition in the **3-D rotation** window, the control cancels this process with an error message by default.
In the optional machine parameter **chkTiltingAxes** (no. 204601) the machine manufacturer defines the control reaction.
- A tilted working plane will remain active even after a control restart.
Further information: "Referencing workspace", Page 124
- PLC positionings defined by the machine manufacturer are not allowed when the working plane is tilted.

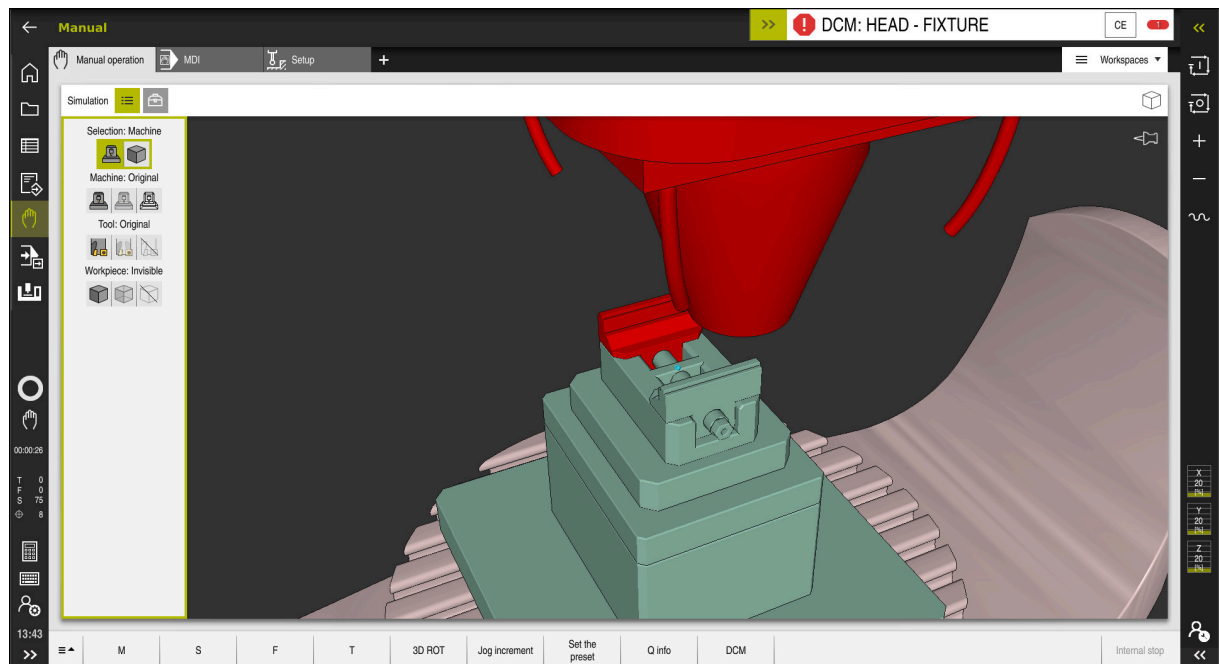
10

Collision Monitoring

10.1 Dynamic Collision Monitoring (DCM, option 40)

Application

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



Dynamic Collision Monitoring (DCM) including collision warning

Requirements

- Dynamic Collision Monitoring (DCM, software option 40)
- Control prepared by the machine manufacturer
The machine manufacturer must define a kinematics model of the machine, insertion point for fixtures and the safety distance between collision objects.
Further information: "Fixture monitoring (option 40)", Page 210
- Tools with a positive radius **R** and length **L**.
Further information: "Tool table tool.t", Page 368
- The values in the tool management equal the actual tool dimensions
Further information: "Tool management ", Page 165

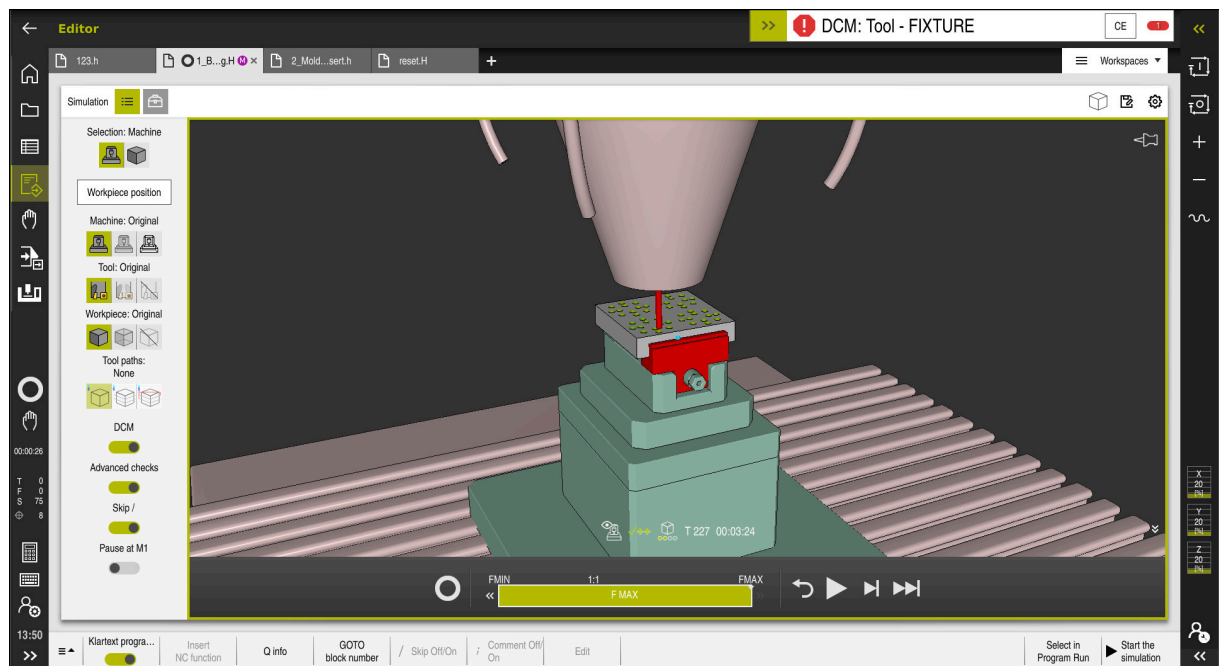
Description of function



Refer to your machine manual.

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

The machine manufacturer can define machine components and minimum distances that are to be monitored by the control during all machine movements. If two collision objects come closer to each other than a defined minimum distance, the control generates an error message and terminates the movement.



Dynamic Collision Monitoring (DCM) in the **Simulation** workspace

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- ▶ Make sure to activate DCM whenever possible
- ▶ Make sure to always re-activate DCM immediately after a temporary deactivation
- ▶ Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

The control displays the collision objects graphically in the following operating modes:

- **Editor** operating mode
- **Manual** operating mode
- **Program Run** operating mode

The control also monitors the tools, as defined in tool management, for collision.

NOTICE**Danger of collision!**

Even if Dynamic Collision Monitoring (DCM) is active, the control does not automatically monitor the workpiece for collisions, neither with the tool nor with other machine components. There is a risk of collision during machining!

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

Further information: User's Manual for Programming and Testing

Dynamic Collision Monitoring (DCM) in the Manual and Program Run operating modes

Dynamic Collision Monitoring (DCM) is activated separately for the **Manual** and **Program Run** operating modes, using the **DCM** button.

Further information: "Activating Dynamic Collision Monitoring (DCM) for the Manual and Program Run operating modes", Page 208

In the **Manual** and **Program Run** operating modes, the control stops the movement if two collision objects approach each other by less than a minimum distance. In this case, the control displays an error message naming the two objects causing collision.



Refer to your machine manual.

The machine manufacturer can define the minimum distance between two collision-monitored objects.

Before the collision warning, the control dynamically reduces the feed rate of movements. This ensures that the axes stop in good time before a collision occurs. When the collision warning is triggered, the control displays the colliding objects in red in the **Simulation** workspace.



When a collision warning has been issued, machine movements via the axis direction keys or the handwheel are only possible if they increase the distance between the collision objects.

With active collision monitoring and a simultaneous collision warning, no movements are permitted that reduce the distance or leave it unchanged.

Dynamic Collision Monitoring (DCM) in the Editor operating mode

Dynamic Collision Monitoring (DCM) is activated for simulation in the **Simulation** workspace.

Further information: User's Manual for Programming and Testing

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

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



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

Further information: User's Manual for Programming and Testing

To obtain a simulation result that is similar to the program run, the following aspects must match:

- Workpiece preset
- Basic rotation
- Offsets of each axis
- Tilting condition
- Active kinematic model

The active workpiece preset for the simulation must be selected. The active workpiece preset from the preset table can be adopted into the simulation.

Further information: User's Manual for Programming and Testing

In a simulation, the following aspects may differ from the actual machine or may not be available at all:

- The simulated tool change position may differ from the tool change position in the machine.
- Changes in the kinematics may have a delayed effect in the simulation.
- PLC positioning movements are not displayed in the simulation.
- Global program settings (GPS, option 44) are not available
- Handwheel superimposition is not available
- Editing of job lists is not available
- Traverse range limits from the **Settings** application are not available.

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

NOTICE

Danger of collision!

If Dynamic Collision Monitoring (DCM) is inactive, the control will not perform any automatic collision checking. This means that movements that might cause collisions will not be prevented. There is a danger of collision during all movements!

- ▶ Make sure to activate DCM whenever possible
- ▶ Make sure to always re-activate DCM immediately after a temporary deactivation
- ▶ Carefully test your NC program or program section in the **Single Block** mode while DCM is deactivated

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



- ▶ Select the **Manual** operating mode



- ▶ Select the **Manual** application
- ▶ Select **DCM**
- > The control opens the **Dyna. Coll. Monitoring (DCM)** window.
- ▶ Activate DCM in the desired operating modes, using the switches

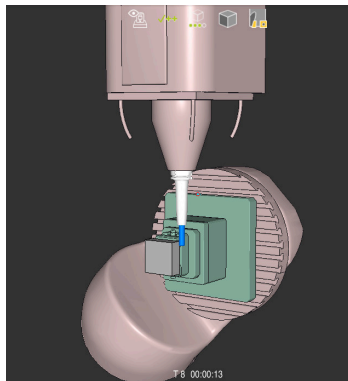


- ▶ Press **OK**
- > The control activates DCM in the selected operating modes.



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.

10.1.2 Activating the graphic display of the collision objects



Simulation in the **Machine** mode

To activate the graphic display of the collision objects:



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

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



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

Changing the representation

To change the graphic display of the collision objects:

- ▶ Activate the graphic display of the collision objects



- ▶ Select the **Visualization options** column



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

Notes

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

10.2 Fixture monitoring (option 40)

10.2.1 Fundamentals

Application

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

Related topics

- Dynamic Collision Monitoring (DCM, option 40)
Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204
- Integrating an STL file as workpiece blank
Further information: User's Manual for Programming and Testing

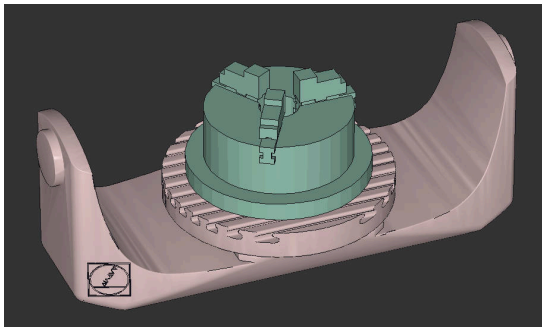
Requirements

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

Description of function

To use fixture monitoring, the steps below are needed:

- Creating a fixture or loading into the control
Further information: "Options for fixture files", Page 211
- Fixture placement
 - **Set up fixtures** function in the **Setup** application (option 140)
Further information: "Integrating the fixtures into collision monitoring (option 140)", Page 213
 - Manual fixture placement
- When changing fixtures, load or remove the fixture in the NC program
Further information: User's Manual for Programming and Testing



Three-jaw chuck loaded as fixture

Options for fixture files

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

You can use the **3D mesh** function (option 152) to create STL files from other file types and adapt STL files to the requirements of your control.

Further information: "Generating STL files with 3D mesh (option 152)", Page 292

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

Fixtures from STL files

STL files allow you to map both individual components and entire assemblies as an immobile fixture. The STL format is useful, in particular, for integrated fixture systems and recurring setups.

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

With CAD Model Optimizer (software option 152), you can adapt STL files that do not meet the requirements and then use them as fixtures.

Further information: "Generating STL files with 3D mesh (option 152)", Page 292

Fixtures from M3D files

M3D is a file type designed by HEIDENHAIN. The paid M3D Converter software from HEIDENHAIN allows you to create M3D files from STL or STEP files.

In order to use an M3D file as a fixture, you need to use the M3D Converter software to create and check the file.

Fixtures from CFG files

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

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

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

Further information: "Editing CFG files with KinematicsDesign", Page 221

Notes

NOTICE

Danger of collision!

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

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

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

By storing multiple fixtures, you can choose the appropriate fixture for your machining operation without needing to configure it.

- Example files for setups used in everyday manufacturing are provided in the NC database of the Klartext Portal:

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

10.2.2 Integrating the fixtures into collision monitoring (option 140)

Application

The **Simulation** function determines the position of a 3D model in the **Set up fixtures** workspace, matching the real fixture in the machine envelope. Once the fixture has been set-up, the control considers it in Dynamic Collision Monitoring (DCM).

Related topics

- **Simulation** workspace
Further information: User's Manual for Programming and Testing
- Dynamic Collision Monitoring (DCM)
Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204
- Fixture monitoring
Further information: "Fixture monitoring (option 40)", Page 210

Requirements

- Dynamic Collision Monitoring (DCM version 2, software option 140)
- Workpiece touch probe
- Permitted fixture file matching the real fixture
Further information: "Options for fixture files", Page 211

Description of function

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

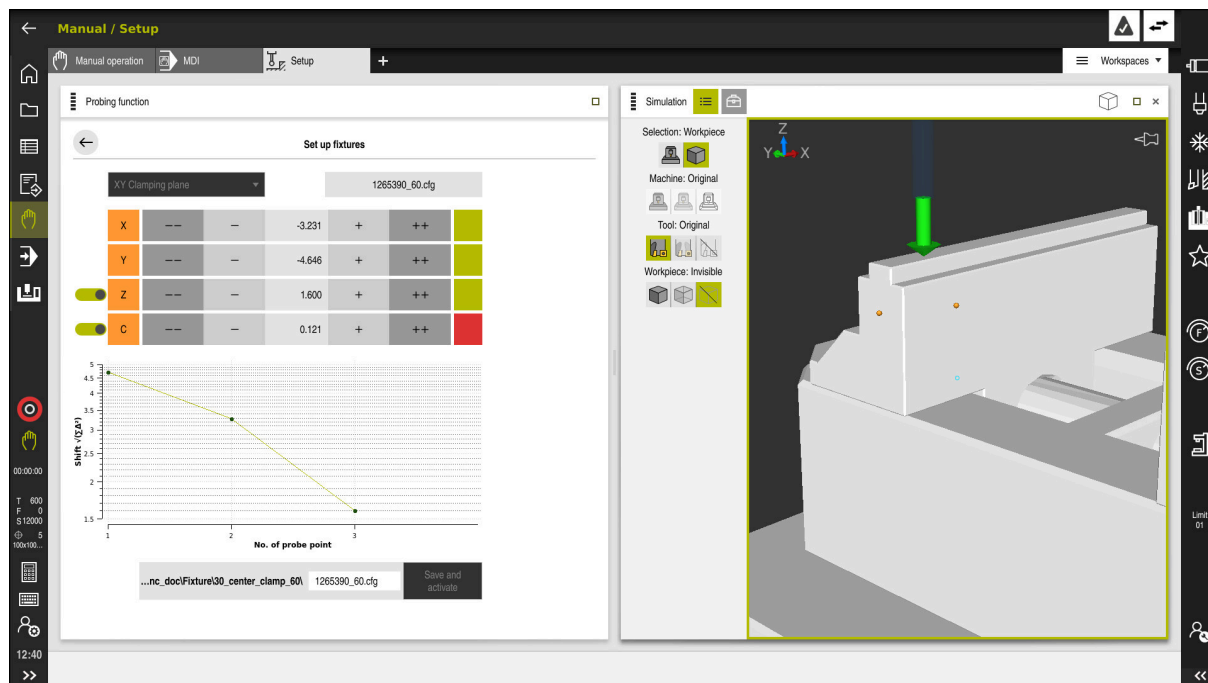
The **Set up fixtures** function determines the fixture position, using various probing processes. First one point on the fixture is probed in every linear axis. The position of the fixture is defined in this way. After probing one point in all linear axes, further points can be integrated in order to improve positioning accuracy. After defining the position in one axis, the control changes the status of that axis from red to green.

The revision diagram shows by which value the 3D model is shifted on the real fixture resulting from the individual probing processes.

Further information: "Revision diagram", Page 217

Extensions of the Simulation workspace

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












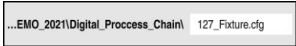
Set up fixtures function with open **Simulation** workspace

When the **Set up fixtures** function is active, the **Simulation** workspace shows the content below:

- Current position of fixture as viewed by the control
 - Probed points on the fixture
 - Possible direction of probing by means of an arrow:
 - No arrow
Probing is not possible. The workpiece touch probe is too distant from the fixture or the workpiece touch probe is positioned within the fixture, as seen by the control.
 - Red arrow
Probing in the direction of the arrow is not possible. The angle between the workpiece touch probe and the fixture is not correct.
- i** 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. Probing will presumably not improve the set-up process, e. g. the probing point in the displayed axis direction is too close to an already probed point.
 - Green arrow
Probing in the direction of the arrow is possible. The angle between the workpiece touch probe and the fixture is correct and the distance is right.

Icons and buttons

The **Set up fixtures** function contains the following icons and buttons:

Icon or button	Function
XY Clamping plane	<p>This selection menu defines the plane in which the fixture is in contact with the machine.</p> <p>The control offers the following planes:</p> <ul style="list-style-type: none"> ■ XY clamping plane ■ XZ clamping plane ■ YZ clamping plane <div>  Depending on the selected clamping plane, the control displays the axes in question. In the XY Clamping plane, the control displays e. g. the axes X, Y, Z and C. </div>
 127_Fixture.cfg	Name of fixture file
	<p>Shifts the position of the virtual fixture by 10 mm or 10° in the negative axis direction</p> <div>  Shifts the fixture in mm in a linear axis and in degrees in a rotary axis. </div>
	Shifts the position of the virtual fixture by 1 mm or 1° in the negative axis direction
 0.000	Enter the position of the virtual fixture directly
	Shifts the position of the virtual fixture by 1 mm or 1° in the positive axis direction
	Shifts the position of the virtual fixture by 10 mm or 10° in the positive axis direction
	<p>Status of axis</p> <p>The control displays the following colors:</p> <ul style="list-style-type: none"> ■ Gray The axis is hidden in this set-up process and is not considered. ■ White At the beginning of the set-up process, i. e. before probing points are calculated, the control displays the status of all axes in white. ■ Red The position of the fixture in this axis is not unambiguously defined. ■ Yellow The position of the fixture in this axis already contains information. The information is not meaningful yet. ■ Green The position of the fixture in this axis is unambiguously defined.
 ...EMO_2021\Digital_Process_Chain\ 127_Fixture.cfg	<p>Path of fixture file</p> <p>The control automatically saves the fixture file in the initial folder. The fixture file name can be edited.</p>

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



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

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

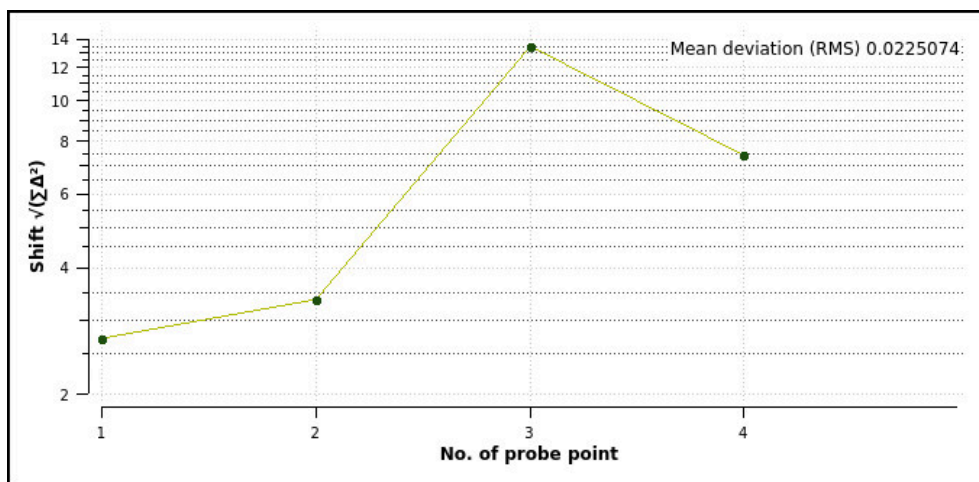
Revision diagram

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

The revision diagram shows the curve of revisions performed during the setting-up process. The set-up process is successfully completed when only revisions in the accuracy range of e. g. 0.05 mm appear.

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



Revision diagram in the **Set up fixtures** function

The revision diagram of the **Set up fixtures** function shows the following information:

- **Mean deviation (QMW)**

This area shows the average distance of the measured probing points from the 3D model in mm.

- **Shift**

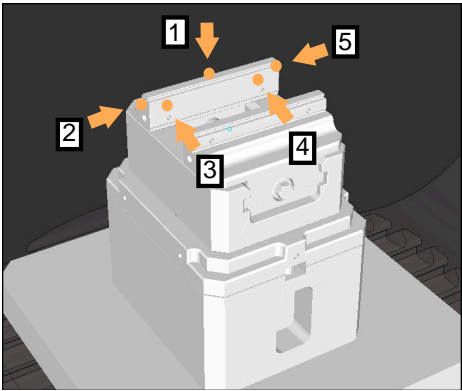
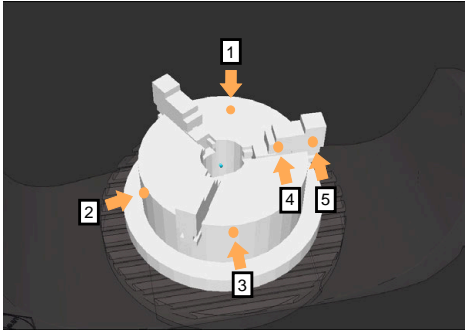
This axis shows the course of the revised model position by means of the added probing points. The individual values show by which value the 3D model was shifted by the respective probing process.

- **No. of probe point**

This axis shows the numbers of the individual probing points.

Example of sequence of fixture probing points

The following probing points can e. g. be set for different fixtures:

Chucking equipment/fixtures	Possible sequence
	<p>The following probing points can be set when measuring a vice:</p> <ol style="list-style-type: none"> 1 Touching the fixed vice jaw in Z- 2 Touching the fixed vice jaw in X+ 3 Touching the fixed vice jaw in Y+ 4 Touching the second value in Y+ for rotation 5 To improve accuracy, touching the check point in X-
	<p>The following probing points can be set when measuring a three-point chuck:</p> <ol style="list-style-type: none"> 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

Probing points for a vice with a fixed vice jaw

Probing points with a three-jaw chuck

Measuring the fixed-jaw vice



The desired 3D model must meet the requirements of the control.

Further information: "Options for fixture files", Page 211

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

- ▶ Affix a real vice in the working space



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



This step makes the subsequent steps easier.



Open

++

- ▶ Select the **Setup** application
- ▶ Select **Set up fixtures**
- ▶ The control opens the **Set up fixtures** menu.
- ▶ Select a 3D model matching the real vice
- ▶ Select **Open**
- ▶ The control opens the selected 3D model in the simulation.
- ▶ Pre-position the 3D model by using the buttons for the individual axes within the virtual working space



For pre-positioning the vice, use the workpiece touch probe as a point of reference.

At this point in time, the control does not know the precise position of the fixture, but of the workpiece touch probe. Pre-positioning the 3D model in accordance with the position of the workpiece touch probe and by using e. g. the table's T-slots produces values close to the position of the real vice.

Even after recording the first measuring points, the shifting functions are still available for correcting the fixture position manually.

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



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

Do not touch directly near edges, chamfers and roundings.



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



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

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



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

Save and
activate

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

Notes

NOTICE

Danger of collision!

To touch the exact position of the fixture on the machine, 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 221

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

Further information: "Editing CFG files with KinematicsDesign", Page 221

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

Further information: User's Manual for Programming and Testing

10.2.3 Editing CFG files with KinematicsDesign

Application

KinematicsDesign allows editing CFG files in the control. In this process, **KinematicsDesign** displays the fixtures graphically and thus supports troubleshooting and removal of errors. Several fixtures can be joined in order to take complex clamping situations into account in Dynamic Collision Monitoring (DCM).

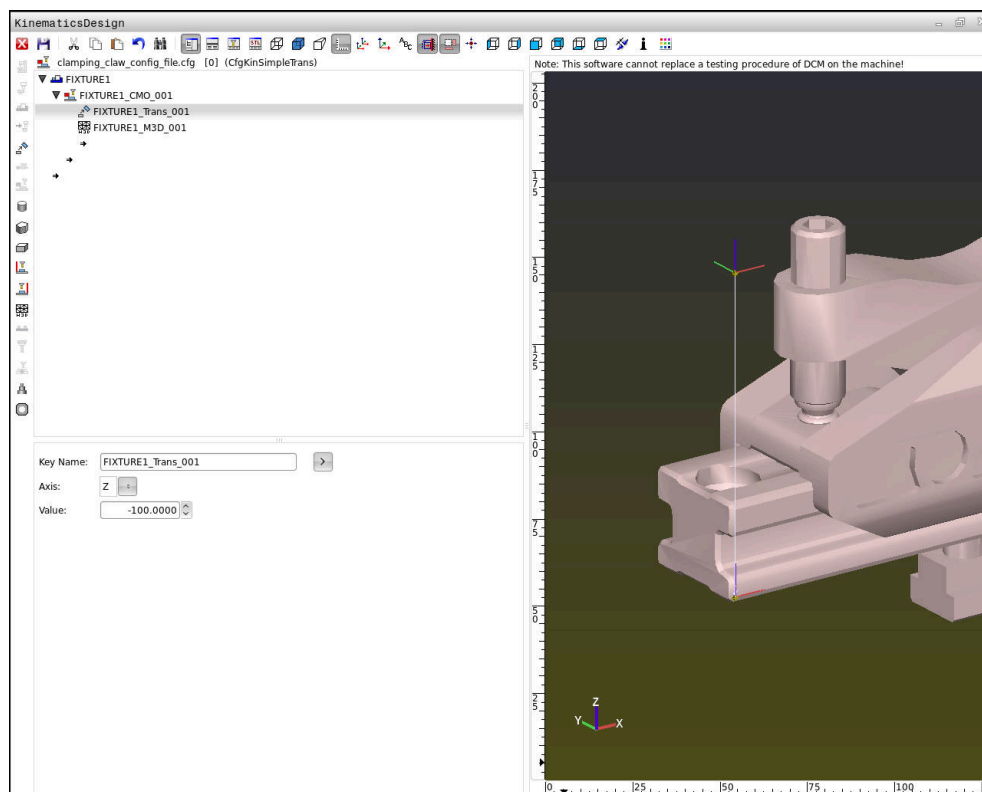
Description of function

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

KinematicsDesign offers the following functions:

- Editing of fixtures with graphic support
- Feedback in case of incorrect entries
- Integration of transformations
- Addition of new elements
 - 3D model (M3D or STL files)
 - Cylinder
 - Prism
 - Cuboid
 - Truncated cone
 - Hole

You can integrate both STL files and M3D files into CFG files more than once.




Syntax in CFG files

The following syntax elements are used within the various CFG functions:

Function	Description
<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 if a collision occurs (optional input)
<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	Description
<code>CfgCMOMesh3D(key:="Fixture_body", filename:="1.STL",name:=" ")</code>	Definition of fixture component <div>  You can also enter an absolute path for the defined fixture component (e.g., TNC:\nc_prog\1.STL) </div>
<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
<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 .

Function	Description
<code>CfgKinFixModel (key:="Fix_Model", kinObjects:=["fixture"])</code>	Fixture designation CfgKinFixModel contains one or more CfgCMO elements.

Geometric shapes

You can add simple geometric objects to your collision object either directly in the CFG file or by using **KinematicsDesign**.

All integrated geometric shapes are subelements of the higher-order **CfgCMO**, in which they are listed as **primitives**.

The following geometric objects are available:

Function	Description
<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 fixture**
- **KinematicsDesign** creates a new fixture entry within the CFG file.
- ▶ Enter a **keyname** for the fixture (e.g., **clamping jaw**)
- ▶ Confirm your input
- **KinematicsDesign** loads the input.



- ▶ Move cursor down one level




- ▶ Select **Insert collision object**
- ▶ Confirm your input
- **KinematicsDesign** creates a new collision object.

Defining geometric shapes

KinematicsDesign allows you to define various geometric shapes. You can construct simple fixtures by combining several geometric shapes.

To define a geometric shape:


- ▶ Create a fixture entry with a collision object
- ⇒
- ▶ Select the cursor key beneath the collision object
- 
- ▶ Select the desired geometric shape (e.g., a cuboid)
 - ▶ Define the position of the cuboid (e.g., **X = 0, Y = 0, Z = 0**)
 - ▶ Define the dimensions of the cuboid (e.g., **X = 100, Y = 100, Z = 100**)
 - ▶ Confirm your input
 - The control displays the defined cuboid in the graphic.

Integrating 3D models

The integrated 3D models must meet the requirements of the control.

Further information: "Options for fixture files", Page 211


To integrate a 3D model as a fixture:

- ▶ Create a fixture entry with a collision object
- ⇒
- ▶ Select the cursor key beneath the collision object
- 
- ▶ Select **Insert 3D model**
 - The control opens the **Open file** window.
 - ▶ Select the desired STL or M3D file
 - ▶ Press **OK**
 - The control integrates the selected file and displays the file in the graphic window.

Fixture placement

You can place the integrated fixture at any position (e.g., for correcting the orientation of an external 3D model). For this purpose, insert transformations for all axes you wish to use.

To position a fixture with **KinematicsDesign**:

- ▶ Define the fixture
- ⇒
- ▶ Select the cursor key beneath the element to be positioned
- 
- ▶ Select **Insert transformation**
 - ▶ Enter a **key name** for the transformation (e.g., **Z shift**)
 - ▶ Select the **axis** for the transformation (e.g., **Z**)
 - ▶ Select the **value** for the transformation (e.g., **100**)
 - ▶ Confirm your input
 - **KinematicsDesign** inserts the transformation.
 - **KinematicsDesign** depicts the transformation in the graphic.

Note

As an alternative to using **KinematicsDesign**, you can also create fixture files directly from the CAM system or by using the appropriate code in a text editor.

Example

The example below describes the syntax of a CFG file for a vise with two movable jaws.

Files used

Various STL files are used to describe the vise. Since the jaws of the vise are dimensionally identical, they are defined using the same STL file.

Code	Explanation
<code>CfgCMOMesh3D (key:="Fixture_body", filename:="vice_47155.STL", name:=" ")</code>	Body of the vise
<code>CfgCMOMesh3D (key:="vice_jaw_1", filename:="vice_jaw_47155.STL", name:=" ")</code>	First jaw of the vise
<code>CfgCMOMesh3D (key:="vice_jaw_2", filename:="vice_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_vice_jaw", dir:=Z, val:=60) CfgKinSimpleTrans (key:="TRANS_C_180", dir:=C, val:=180) CfgKinSimpleTrans (key:="TRANS_SPC", dir:=C, val:=0) CfgKinSimpleTrans (key:="TRANS_SPB", dir:=B, val:=0) CfgKinSimpleTrans (key:="TRANS_SPA", dir:=A, val:=0)</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 components following 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>CfgCMO (key:="FIXTURE", primitives:= ["TRANS_X", "TRANS_Y", "TRANS_Z", "TRANS_SPC", "TRANS_SPB", "TRANS_SPA", "Fixture_body", "TRANS_Z_vice_jaw", "TRANS_opening_width_2", "vice_jaw_1", "TRANS_opening_width", "TRANS_C_180", "vice_jaw_2"], active:=TRUE, name:=" ")</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

Control Functions

11.1 Adaptive Feed Control (AFC, option 45)

11.1.1 Fundamentals

Application

Adaptive Feed Control (AFC) saves time when processing NC programs and reduces wear on the machine. The control regulates the contouring feed rate during program run depending on the spindle power. In addition, the control responds to overloading of the spindle.

Related topics

- Tables related to AFC

Further information: "Tables for AFC (option 45)", Page 408

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Approved 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 408
- Define settings for AFC for each tool in the tool management
Further information: "Tool table tool.t", Page 368
- Define AFC in the NC program
Further information: "NC functions for AFC (option 45)", Page 232
- Define AFC in the **Program Run** operating mode with the **AFC** switch.
Further information: "AFC switch in Program Run operating mode", Page 234
- Prior to automatic control, determine the reference spindle power with a teach-in cut
Further information: "AFC teach-in cut", Page 235

If AFC is active in the teach-in cut or in control mode, the control displays an icon in the **Positions** workspace.

Further information: "Positions workspace", Page 93

Detailed information about the function is provided by the control on the **AFC** tab of the **Status** workspace.

Further information: "AFC tab (option 45)", Page 102

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 236

- Protection of the machine's mechanical elements

Timely feed rate reduction and shutdown responses help to avoid machine overload.

Tables related to AFC

The control offers the following tables in conjunction with AFC:

- **AFC.tab**

In the **AFC.TAB** table, you can enter the feed rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

Further information: "Basic AFC settings in AFC.tab", Page 408

- ***.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**. **<name>** is 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 411

- ***.H.AFC2.DEP**

During a teach-in cut, the control stores information for each machining step in the **<name>.H.AFC2.DEP** file. The **<name>** corresponds to the name of the NC program for which you perform 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 412

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

Further information: "Editing tables for AFC", Page 414

Notes

NOTICE

Caution: Danger to the tool and workpiece!

As soon as Adaptive Feed Control (AFC) is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate, e.g. due to wear, before it was deactivated, the control accelerates the feed rate up to the programmed value. This behavior applies regardless of how the function is deactivated. This feed acceleration may result in damage to the tool and/or the workpiece!

- ▶ If the feed rate falling below the **FMIN** value is imminent, stop the machining operation without deactivating AFC
 - ▶ Define the overload reaction 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 reaction.
 - If, with the reference spindle load, the minimum feed factor is fallen below
The control executes the shutdown response from the **OVLD** column of the **AFC.tab** table.
Further information: "Basic AFC settings in AFC.tab", Page 408
 - If the programmed feed rate falls below the 30 % threshold
The control executes an NC stop.
 - Adaptive feed control is not intended for tools with diameters less than 5 mm. If the rated power consumption of the spindle is very high, the limit diameter of the tool may be larger.
 - Do not work with adaptive feed control in operations in which the feed rate and spindle speed must be adapted to each other, such as tapping.
 - In NC blocks containing **FMAX**, the adaptive feed control is **not active**.
 - With the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control displays dependency files in the file management.

11.1.2 Activating and deactivating AFC

NC functions for AFC (option 45)

Application

Adaptive Feed Control (AFC) is activated and deactivated from the NC program.

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table
Further information: "Basic AFC settings in AFC.tab", Page 408
- Desired control setting defined for all tools
Further information: "Tool table tool.t", Page 368
- **AFC** switch active
Further information: "AFC switch in Program Run operating mode", Page 234

Description of function

The control provides several functions that enable you to start and stop AFC:

- **FUNCTION AFC CTRL:** The **AFC CTRL** function activates feedback control mode starting with this NC block, even if the learning phase has not been completed yet.
- **FUNCTION AFC CUT BEGIN TIME1 DIST2 LOAD3:** The control starts a sequence of cuts with active **AFC**. The changeover from the teach-in cut to feedback control mode begins as soon as the reference power has been determined in the teach-in phase, or once one of the **TIME**, **DIST** or **LOAD** conditions has been met.
- **FUNCTION AFC CUT END:** The **AFC CUT END** function deactivates the AFC control.

Input

FUNCTION AFC CTRL

11 FUNCTION AFC CTRL	; Start AFC in control mode
-----------------------------	-----------------------------

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CTRL	Syntax initiator for the start of control mode

FUNCTION AFC CUT

11 FUNCTION AFC CUT BEGIN TIME10 DIST20 LOAD80	; Start AFC machining step, limit the duration of the teach-in phase
---	--

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION AFC CUT	Syntax initiator for an AFC machining step
BEGIN or END	Start or end machining step
TIME	End teach-in phase after the defined time in seconds Optional syntax element Only if BEGIN has been selected
DIST	End teach-in phase after the defined distance in mm Optional syntax element Only if BEGIN has been selected
LOAD	Enter the reference load of the spindle directly, max. 100% Optional syntax element Only if BEGIN has been selected

Notes

NOTICE

Caution: Danger to the tool and workpiece!

If you activate the **FUNCTION MODE TURN** machining mode, the control will clear the current **OVLD** values. This means that you need to program the machining mode before the tool call! If the programming sequence is not correct, no tool monitoring will take place, which might result in damage to the tool or workpiece!

- ▶ Program the **FUNCTION MODE TURN** machining mode before the tool call

- The **TIME**, **DIST** and **LOAD** defaults are modally effective. They can be reset by entering **0**.
- Execute the function **AFC CUT BEGIN** only after the starting rotational speed has been reached. If this is not the case, then the control issues an error message, and the AFC cut is not started.
- You can define a feedback-control reference power with the **AFC LOAD** tool table column and the **LOAD** input in the NC program. You can activate the **AFC LOAD** value via the tool call and the **LOAD** value with the **FUNCTION AFC CUT BEGIN** function.

If you program both values, the control will use the value programmed in the NC program!

AFC switch in Program Run operating mode

Application

The **AFC** switch allows you to activate or deactivate Adaptive Feed Control (AFC) in the **Program Run** operating mode.

Related topics

- Activating AFC in the NC program
Further information: "NC functions for AFC (option 45)", Page 232

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Approved 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** switch must be activated for the NC functions to have an effect for AFC.

If you do not specifically deactivate AFC using the switch, AFC remains active. The control remembers the setting of the switch even if the control is restarted.

If the **AFC** switch is active, the control displays an icon in the **Positions** workspace. In addition to the current setting of the feed rate potentiometer, the control shows the controlled feed value as a percentage (%).

Further information: "Positions workspace", Page 93

Notes

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

As soon as the AFC function is deactivated, the control immediately switches back to the programmed machining feed rate. If AFC decreased the feed rate (e.g. due to wear) before it was deactivated, the control accelerates the feed rate up to the programmed value. This applies regardless of how the function is deactivated (e.g. feed rate potentiometer). This acceleration may result in damages to the tool or the workpiece!

- ▶ If it is imminent that the feed rate falls below the **FMIN** value, stop the machining operation (instead of deactivating the **AFC** function)
 - ▶ Define the overload reaction for cases in which the feed rate falls below the **FMIN** value
- If Adaptive Feed Control is active in **Control** mode, the control internally sets the spindle override to 100 %. This means that you can no longer change the spindle speed.
 - If Adaptive Feed Control is active in **Control** mode, the control takes over the value from the feed rate override function.
 - Increasing the feed rate override has no influence on the control.
 - If you reduce the feed override with the potentiometer by more than 10% in relation to the position at the start of the program, the control switches AFC off.
You can reactivate control with the **AFC** 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.

11.1.3 AFC teach-in cut**Application**

With the teach-in cut, the control determines the reference power of the spindle for the machining step. Based on the reference power, the control adjusts the feed rate in control mode.

If you have already determined the reference power for a machining operation, you can specify the value for the machining operation. For this, the control provides the **AFC-LOAD** column in the tool management and the **LOAD** syntax element in the **FUNCTION AFC CUT BEGIN** function. In this case, the control no longer performs a teach-in cut, but uses the specified value immediately for control.

Related topics

- Enter the known reference power in the **AFC-LOAD** column in the tool management
Further information: "Tool table tool.t", Page 368
- Define the known reference power in the **FUNCTION AFC CUT BEGIN** function
Further information: "NC functions for AFC (option 45)", Page 232

Requirements

- Adaptive Feed Control (AFC, software option 45)
- Control settings defined in the **AFC.tab** table
Further information: "Basic AFC settings in AFC.tab", Page 408
- Desired control setting defined for all tools
Further information: "Tool table tool.t", Page 368
- Desired NC program selected in the **Program Run** operating mode
- **AFC** switch active
Further information: "AFC switch in Program Run operating mode", Page 234

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 411

When you are performing a teach-in cut, the control shows the spindle reference power determined until this time in a pop-up window.

When the control has determined the control reference power, it ends the teach-in cut and switches to control mode.

Notes

- When you record a teach-in cut, the control internally sets the spindle override to 100%. Then you can no longer change the spindle speed.
- During the teach-in cut, you can influence the measured reference load by using the feed rate override to make any changes to the contouring feed rate.
- You can repeat a teach-in cut as often as desired. Manually change the status from **ST** back to **L**. If the programmed feed rate value is far too high and forces you to sharply decrease the feed rate override during the machining step, you will have to repeat the teach-in cut.
- If the determined reference load is greater than 2 %, the control changes the status from teach-in (**L**) to controlling (**C**). Adaptive feed control is not possible for smaller values.
- In **FUNCTION MODE TURN** machining mode, the minimum reference load is 5 %. Even if the control determines lower values, it will still use this minimum reference load. Thus, the overload limits (indicated as percentage values) are based on a minimum reference load of at least 5 %.

11.1.4 Monitoring tool wear and tool load

Application

With Adaptive Feed Control (AFC), you can monitor the tool for wear and breakage. The **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management can be used for this.

Related topics

- **AFC-OVLD1** and **AFC-OVLD2** columns in the tool management
Further information: "Tool table tool.t", Page 368

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 408

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 shutdown 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 shutdown response, the control always executes a machining stop and locks the momentary tool.

In turning mode, the control can check for tool wear and tool breakage.

A tool breakage leads to a sudden load decrease. If you want the control to monitor the load decrease, too, enter the value 1 in the **SENS** column.

Further information: "Basic AFC settings in AFC.tab", Page 408

11.2 Active Chatter Control (ACC, option 145)**Application**

Chatter marks can be caused during heavy-duty machining, in particular. **ACC** reduces chattering, thereby reducing wear on the tool and machine. In addition, **ACC** increases metal removal rates.

Related topics

- ACC column in the tool table

Further information: "Tool table tool.t", Page 368

Requirements

- Active Chatter Control (ACC, software option 145)
- Control adapted by the machine manufacturer
- **ACC** column in the tool management defined with **Y**
- Number of tool cutting edges defined in the **CUT** column

Description of function

Strong forces come into play during roughing (power milling). Depending on the tool spindle speed, the resonances in the machine tool and the chip volume (metal-removal rate during milling), the machine can sometimes begin to **chatter**. This chattering places heavy strain on the machine, and causes ugly marks on the workpiece surface. The tool, too, is subject to heavy and irregular wear from chattering. In extreme cases it can result in tool breakage.

In order to reduce a machine's tendency to chatter, HEIDENHAIN offers an effective control function known as Active Chatter Control (**ACC**). The use of this control function is particularly advantageous during heavy machining. ACC makes substantially higher metal removal rates possible. Depending on the type of machine, the metal-removal rate can often be increased by more than 25 %. You reduce the mechanical load on the machine and increase the life of your tools at the same time.

ACC was developed especially for roughing and heavy machining and is particularly effective in this area. You need to conduct appropriate tests to see whether ACC will also be advantageous on your machine and with your tool.

ACC is activated and deactivated using the **ACC** switch in the **Program Run** operating mode or the **MDI** application.

Further information: "Program Run operating mode", Page 334

Further information: "Application MDI", Page 329

If ACC is active, the control shows a corresponding icon in the **Positions** workspace.

Further information: "Positions workspace", Page 93

Notes

- ACC reduces or prevents vibrations in the range of 20 Hz to 150 Hz. If ACC does not appear to have an effect, the vibrations may be outside of this range.
- With Machine Vibration Control (MVC, software option 146), you can also positively influence the result.

11.3 Global Program Settings (GPS, option 44)

11.3.1 Fundamentals

Application

The Global Program Settings (GPS) allow you to define selected transformations and settings without changing the NC program. All of the settings apply globally and are superimposed on the relevant active NC program.

Related topics

- Coordinate transformations in the NC program
 - Further information:** User's Manual for Programming and Testing
 - Further information:** User's Manual for Machining Cycles
- **GPS** tab in the **Status** workspace.
 - Further information:** "GPS tab (option 44)", Page 104
- Reference systems of the control
 - Further information:** "Reference systems", Page 178

Requirements

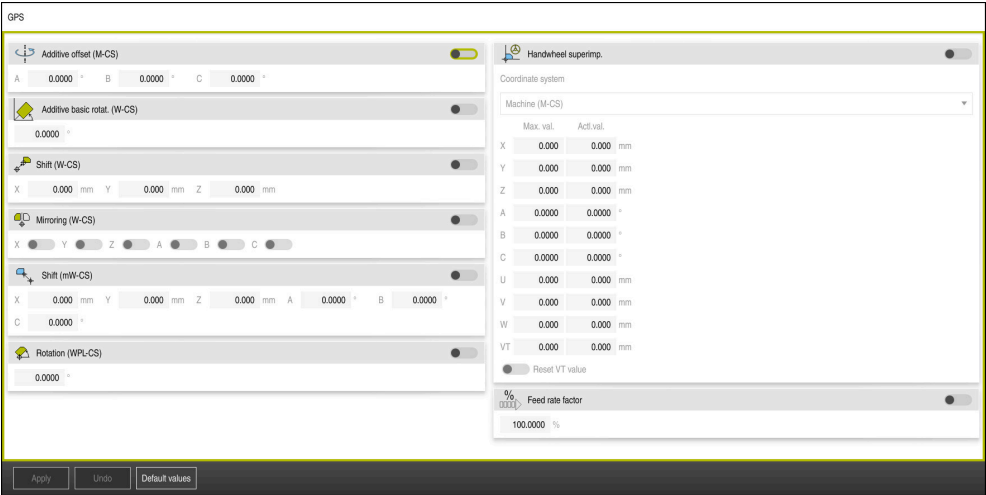
- Approval by the machine manufacturer
The machine manufacturer uses the optional machine parameter **CfgGlobalSettings** (no. 128700) to define which GPS functions are available on the control.
- Global Program Settings (GPS, software option 44)

Description of function

The values of the Global Program Settings are defined and activated in the **GS** workspace.

The **GS** workspace is available in the **Program Run** operating mode and in the **MDI** application of the **Manual** operating mode.

The transformations of the **GS** workspace are effective in all operating modes and are persistent across reboots of the control.



GS workspace with active functions

The functions of GPS are activated using switches.

The control marks the sequence in which the transformations are effective with green digits.

The control shows the active settings of GPS on the **GPS** tab of the **Status** workspace.

Further information: "GPS tab (option 44)", Page 104

Before executing an NC program with active GPS in the **Program Run** operating mode, you must confirm use of the GPS functions in a pop-up window.

Buttons

The control provides the following buttons in the **GS** workspace:

Button	Description
Apply	Save changes in the GS workspace
Undo	Reset unsaved changes in the GS workspace
Default values	Set the Feed rate factor function to 100%, reset all other functions to zero

Overview of Global Program Settings (GPS)

The Global Program Settings (GPS) include the following functions:

Function	Description
Additive offset (M-CS)	Shift of the zero position of an axis in the machine coordinate system M-CS Further information: "Function Additive offset (M-CS)", Page 241
Additive basic rotat. (W-CS)	Additional rotation based on basic rotation or 3D basic rotation in the workpiece coordinate system W-CS . Further information: "Function Additive basic rotat. (W-CS)", Page 242
Shift (W-CS)	Shift of workpiece preset in a single axis in the workpiece coordinate system W-CS Further information: "Function Shift (W-CS)", Page 243
Mirroring (W-CS)	Mirroring of individual axes in the workpiece coordinate system W-CS Further information: "Function Mirroring (W-CS)", Page 244
Shift (mW-CS)	Additional shift of a workpiece datum already shifted in the modified workpiece coordinate system (mW-CS). Further information: "Function Shift (mW-CS)", Page 245
Rotation (WPL-CS)	Rotation around the active tool axis in the working plane coordinate system WPL-CS Further information: "Function Rotation (WPL-CS)", Page 246
Handwheel superimposition	Superimposed movement of NC program positions with the electronic handwheel Further information: "Function Handwheel superimp.", Page 247
Feed rate factor	Manipulation of the active feed rate Further information: "Function Feed rate factor", Page 250

Define and activate Global Program Settings (GPS)

To define and activate the Global Program Settings (GPS):



- ▶ Select an operating mode, e.g. **Program run**
- ▶ Open the **GS** workspace
- ▶ Activate the switch for the required function, e.g. **Additive offset (M-CS)**
- The control activates the selected function.
- ▶ Enter a value in the desired field, e.g. **A=10.0°**
- ▶ Press **Apply**
- The control accepts the entered values.

Apply



If you select an NC program for program run, you must confirm the Global Program Settings (GPS).

Resetting Global Program Settings (GPS)

To reset the Global Program Settings (GPS):



- ▶ Select an operating mode, e.g. **Program Run**

- ▶ Open the **GS** workspace

Default values

- ▶ Select **Default values**



Provided that you have not selected the **Apply** button, you can restore the values with the **Undo** function.

- > The control sets the values of all Global Program Settings (GPS) to zero except for the feed factor.
- > The control sets the feed factor to 100%.

Apply

- ▶ Press **Apply**
- > The control saves the values that have been reset.

Notes

- The control grays out any axes that are not active on your machine.
- Value inputs are defined in the selected unit of measurement for the position display (mm or inch), e.g. shift values and values of **Handwheel superimp.** Angles are always entered in degrees.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.

11.3.2 Function Additive offset (M-CS)

Application

With the **Additive offset (M-CS)** function, you can shift the zero position of a machine axis in the machine coordinate system **M-CS**. You can use this function, for example, on large machines, to compensate an axis when using axis angles.

Description of function

The control adds the value to the active axis-specific offset from the preset table.

Further information: "Preset table", Page 403

If you activate a value in the **Additive offset (M-CS)** function, the zero position of the affected axis changes in the position display of the **Positions** workspace. The control assumes a different zero position of the axes.

Further information: "Positions workspace", Page 93

Application example

The travel range of a machine with AC fork head is increased using the **Additive offset (M-CS)** function. An eccentric tool chuck is used and the zero position of the C axis is shifted by 180°.

Initial situation:

- Machine kinematics with AC fork head
- Use of an eccentric tool chuck

The tool is clamped in an eccentric tool chuck outside the center of rotation of the C axis.
- The machine parameter **presetToAlignAxis** (no. 300203) for the C axis is set to **FALSE**

To increase the traversing distance:

- ▶ Open the **GS** workspace
- ▶ Activate the **Additive offset (M-CS)** switch
- ▶ Enter **C 180°**

Apply

- ▶ Press **Apply**
- ▶ Program a positioning movement with **L C+0** in the desired NC program
- ▶ Select an NC program
- ▶ The control considers the 180° rotation for all C axis positioning movements as well as the changed tool position.
- ▶ The position of the C axis does not affect the position of the workpiece preset.

Notes

NOTICE

Danger of collision!

The setting of the machine parameter **presetToAlignAxis** (no. 300203) determines whether a preset is taken into account in a rotary axis offset. There is a risk of collision during the subsequent machining operation!

- ▶ Test the behavior at the machine
- ▶ If necessary, set a new preset after the offsets have been activated (mandatory for rotary axes on a table)

With the machine parameter **presetToAlignAxis** (no. 300203), the machine manufacturer defines for each axis what effect an offset of a rotary axis has on the preset:

- **True** (default): Use offset for aligning the workpiece
- **False**: Use offset for inclined-tool machining

11.3.3 Function Additive basic rotat. (W-CS)

Application

The function **Additive basic rotat. (W-CS)** facilitates e.g. better utilization of the working space. For example, you can rotate an NC program by 90° so that the X and Y directions are inverted during execution.

Description of function

The function **Additive basic rotat. (W-CS)** takes effect in addition to the basic rotation or 3D basic rotation from the preset table. The values of the preset table do not change in this respect.

Further information: "Preset table", Page 403

The function **Additive basic rotat. (W-CS)** has no effect on the position display.

Application example

You rotate the CAM output of an NC program by 90° and compensate for the rotation using the function **Additive basic rotat. (W-CS)**.

Initial situation:

- Available CAM output for gantry-type milling machine with a large range of traverse of the Y axis
- The available machining center has the necessary traversing range only in the X axis
- The workpiece blank is clamped with a 90° rotation (long side along the X axis)
- The NC program must be rotated by 90° (algebraic sign depends on the preset position)

To rotate the CAM output:

- ▶ Open the **GS** workspace
- ▶ Activate the switch for **Additive basic rotat. (W-CS)**
- ▶ Enter **90°**



- ▶ Press **Apply**
- ▶ Select NC program
- ▶ The control considers the 90° rotation for all axis positioning movements.

11.3.4 Function Shift (W-CS)

Application

You can use the **Shift (W-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult, for example.

Description of function

The **Shift (W-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 184

The **Shift (W-CS)** function affects the position display. The control shifts the display by the active value.

Further information: "Position displays", Page 118

Application example

The surface of a workpiece to be reworked is determined using the handwheel and the offset is compensated for using the **Shift (W-CS)** function.

Initial situation:

- Reworking of a free-form surface is required
- Workpiece clamped
- Basic rotation and workpiece preset measured in the working plane
- Z coordinate must be defined with the handwheel due to the presence of a free-form surface

To shift the workpiece surface of a workpiece to be reworked:

- ▶ Open the **GS** workspace
- ▶ Activate the **Handwheel superimp.** switch
- ▶ Determine the workpiece surface by scratching, using the handwheel
- ▶ Activate the **Shift (W-CS)** switch
- ▶ Transfer the determined value to the corresponding axis of the **Shift (W-CS)** function, e.g. **Z**

Apply

- ▶ Press **Apply**
- ▶ Start the NC program
- ▶ Activate **Handwheel superimp.** with the **Workpiece (WPL-CS)** coordinate system
- ▶ Determine the workpiece surface by scratching, using the handwheel for fine adjustment
- ▶ Select NC program
- The control takes the **Shift (W-CS)** into account.
- The control uses the current values from **Handwheel superimp.** in the **Workpiece (WPL-CS)** coordinate system.

11.3.5 Function Mirroring (W-CS)

Application

You can use the **Mirroring (W-CS)** function to execute mirror-inverted execution of an NC program without having to modify the NC program.

Description of function

The **Mirroring (W-CS)** function acts on an axis-by-axis basis. The value is additive to mirroring defined in the NC program before tilting the working plane with Cycle **8 MIRRORING** or the **TRANS MIRROR** function.

Further information: User's Manual for Machining Cycles

Further information: User's Manual for Programming and Testing

The **Mirroring (W-CS)** function has no effect on the position display in the **Positions** workspace.

Further information: "Position displays", Page 118

Application example

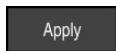
An NC program can be edited with the **Mirroring (W-CS)** function mirror-inverted.

Initial situation:

- Available CAM output for right mirror cap
- NC program set to the center of the ball-nose cutter and **FUNCTION TCPM** function with spatial angles output
- The workpiece datum is centered on the workpiece blank
- Mirroring required in the X axis to produce the left mirror cap

To mirror the CAM output of an NC program:

- ▶ Open the **GS** workspace
- ▶ Activate the **Mirroring (W-CS)** switch
- ▶ Activate the **X** switch



- ▶ Press **Apply**
- ▶ Run the NC program
- ▶ The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.

Notes

- If you use **PLANE** functions or the **FUNCTION TCPM** function with spatial angles, the rotary axes are mirrored accordingly along with the mirrored main axes. This always creates the same constellation, regardless of whether the rotary axes were marked in the **GS** workspace.
- With **PLANE AXIAL**, the mirroring of rotary axes is irrelevant.
- With the **FUNCTION TCPM** function with axis angles, you must activate all axes to be mirrored individually in the **GS** workspace.

11.3.6 Function Shift (mW-CS)

Application

You can use the **Shift (mW-CS)** function to compensate for an offset relative to the workpiece preset for a reworking operation where probing is difficult in the modified workpiece coordinate system **mW-CS**, for example.

Description of function

The **Shift (mW-CS)** function acts on an axis-by-axis basis. The value is added to an existing shift in the **W-CS** workpiece coordinate system.

Further information: "Workpiece coordinate system W-CS", Page 184

The **Shift (mW-CS)** function affects the position display. The control shifts the display by the active value.

"Position displays"

A modified workpiece coordinate system **mW-CS** is present with active **Shift (W-CS)** or active **Mirroring (W-CS)**. Without these preceding coordinate transformations, the **Shift (mW-CS)** option would be effective directly in the workpiece coordinate system (**W-CS**) and would thus be identical to **Shift (W-CS)**.

Application example

You mirror the CAM output of an NC program. After mirroring, you shift the workpiece datum in the mirrored coordinate system to produce the counterpart to a mirror cap.

Initial situation:

- Available CAM output for right mirror cap
- The workpiece datum is located in the left front corner of the workpiece blank.
- NC program set to the center of the ball-nose cutter and **Function TCPM** function with spatial angles output
- The left mirror cap is to be machined

To shift the datum in the mirrored coordinate system:

- ▶ Open the **GS** workspace
- ▶ Activate the **Mirroring (W-CS)** switch
- ▶ Activate the **X** switch
- ▶ Activate the **Shift (mW-CS)** switch
- ▶ Enter the value for shifting the workpiece datum in the mirrored coordinate system

Apply

- ▶ Press **Apply**
- ▶ Run the NC program
- ▶ The control takes the **Mirroring (W-CS)** value for the X axis and the required rotary axes into account.
- ▶ The control takes the modified position of the workpiece datum into account.

Notes

NOTICE

Danger of collision!

The machine parameter **presetToAlignAxis** (no. 300203) defines whether the **Shift (mW-CS)** of a rotary axis is taken into account. There is a risk of collision during the subsequent machining operation!

- ▶ Test the behavior at the machine

The **Shift (mW-CS)** function of the rotary axes to be taken into account is defined by the machine manufacturer in the machine parameter **presetToAlignAxis** (no. 300203) separately for each axis:

- **True** (default): Use offset for aligning the workpiece
- **False**: Use offset for inclined-tool machining

11.3.7 Function Rotation (WPL-CS)

Application

With the **Rotation (WPL-CS)** function, you can, for example, compensate for the misalignment of a workpiece in the already swiveled working plane coordinate system **WPL-CS** without modifying the NC program.

Description of function

The **Rotation (WPL-CS)** function is effective in the tilted working plane coordinate system **WPL-CS**. The value is additive to rotation in the NC program with Cycle **10 ROTATION** or the function **TRANS ROTATION**.

Further information: User's Manual for Machining Cycles

Further information: User's Manual for Programming and Testing

The **Rotation (WPL-CS)** function has no effect on the position display.

11.3.8 Function Handwheel superimp.

Application

With the **Handwheel superimp.** function, you can traverse the axes with the superimposed handwheel during program run. You select the coordinate system in which the **Handwheel superimp.** function is effective.

Related topics

- Handwheel superimpositioning with **M118**

Further information: User's Manual for Programming and Testing

Description of function

In the **Max. val.** column, you define the maximum traversing distance for the respective axis. You can move the input value either positively or negatively. The maximum path is therefore twice as large as the input value.

In the **Actl.val.** column, the control displays the path traversed using the handwheel for each axis.

The **Actl.val.** column can also be edited manually. If you enter a value greater than the **Max. val.**, you cannot activate the value. The control marks an incorrect value in red. The control displays a warning message and prevents the form from being closed.

If the **Actl.val.** column contains a value when you activate the function, the control will use the menu for returning to move to the new position.

Further information: "Returning to the contour", Page 349

The **Handwheel superimp.** function affects the position display in the **Positions** workspace. The control shows the values offset by the handwheel in the position display.

Further information: "Positions workspace", Page 93

The control displays the values of the two methods for **Handwheel superimp.** on the **POS HR** tab of the additional status display.

On the **POS HR** tab of the **Status** workspace, the control shows whether the **Max. val.** is defined using the **M118** function or the Global Program Settings (GPS).

Further information: "POS HR tab", Page 109

Virtual tool axis VT

The virtual tool axis **VT** is needed for machining operations with inclined tools, e.g. for manufacturing oblique holes without using a tilted working plane.

Handwheel superimp. can also be executed in the active tool axis direction. The **VT** always corresponds to the direction of the active tool axis. On machines with head rotation axes, this direction may not correspond to the basic coordinate system **B-CS**. You activate the function with the **VT** line.

Further information: "Notes concerning different machine kinematics", Page 199

By default, values traversed with the handwheel in the **VT** remain active even after a tool change. If you activate the **Reset VT value** switch, the control resets the actual value of the **VT** when a tool is changed.

The control displays the values of the virtual tool axis **VT** on the **POS HR** tab of the **Status** workspace.

Further information: "POS HR tab", Page 109

For the control to display values, you must define a value greater than 0 in the **VT** function for **Handwheel superimp.**

Notes

NOTICE**Danger of collision!**

The coordinate system chosen in the selection menu also takes effect on **Handwheel superimp.** with **M118**, even if the Global Program Settings function (GPS) is not active. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- ▶ Before exiting the form, always make sure to select the **Machine (M-CS)** coordinate system
- ▶ Test the behavior at the machine

NOTICE**Danger of collision!**

When both methods for **Handwheel superimp.** with **M118** and with the Global Program Settings GPS are effective at the same time, the definitions influence each other, depending on their sequence of activation. There is a risk of collision during the execution of **Handwheel superimp.** and the subsequent machining operations!

- ▶ Use only one method for **Handwheel superimp.**
- ▶ Preferably use the **Handwheel superimp.** option of the **Global Program Settings** function
- ▶ Test the behavior at the machine

HEIDENHAIN does not recommend using both methods for **Handwheel superimp.** at the same time. If **M118** cannot be removed from the NC program, you should at least activate **Handwheel superimp.** from GPS prior to selecting the program. This ensures that the control uses the GPS function rather than **M118**.

- If neither the NC program nor the Global Program Settings were used to activate coordinate system transformations, **Handwheel superimp.** is effective in the same manner in all coordinate systems.
- If you want to use **Handwheel superimp.** while machining with active Dynamic Collision Monitoring (DCM), then the control must be in a stopped or interrupted state. Alternatively, you can also deactivate DCM.
Further information: "Dynamic Collision Monitoring (DCM, option 40)", Page 204
- **Handwheel superimp.** in virtual axis direction **VT** requires neither a **PLANE** function nor the **FUNCTION TCPM** function.
- Use the machine parameter **axisDisplay** (no. 100810) to define whether the control also shows the virtual axis **VT** in the position display of the **Positions** workspace.
Further information: "Positions workspace", Page 93

11.3.9 Function Feed rate factor

Application

You can use the **Feed rate factor** function to influence the effective feed rates on the machine, e.g. to adjust the feed rates of a CAM program. This will prevent the CAM program from being re-output using the postprocessor. When doing so, you change all feed rates as a percentage without making any changes in the NC program.

Related topics

- Feed rate limit **F MAX**

The **Feed rate factor** function has no influence on the feed rate limit with **F MAX**.

Further information: "Feed rate limit F MAX", Page 337

Description of function

All feed rates are changed as a percentage. You define a percentage value from 1% to 1000%.

The **Feed rate factor** function acts on the programmed feed rate and the feed rate potentiometer, but not on rapid traverse **FMAX**.

The control shows the current feed rate in field **F** of the **Positions** workspace. If the **Feed rate factor** function is active, the feed rate is shown with the defined values taken into account.

Further information: "Presets and technology values", Page 95

12

Monitoring

12.1 Process Monitoring (option 168)

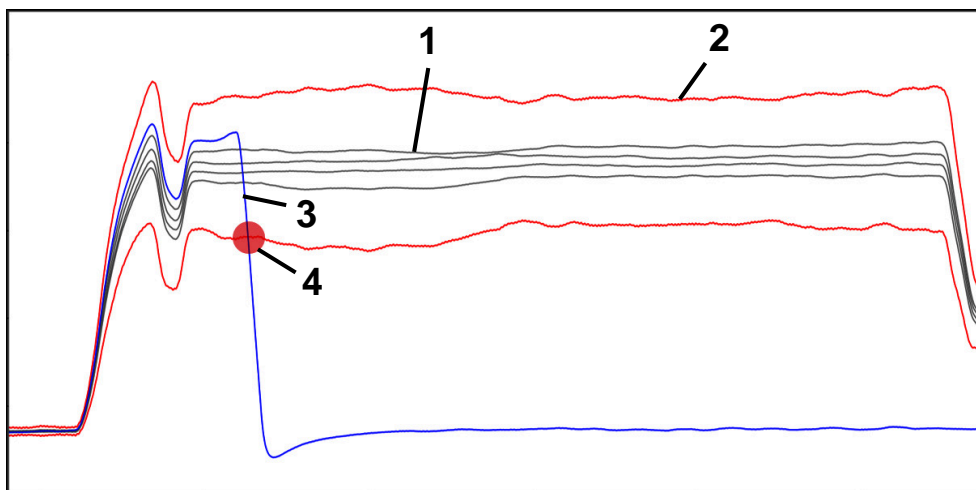
12.1.1 Fundamentals

The control uses process monitoring to detect disturbances in the machining process, e.g.:

- Tool breakage
- Incorrect or missing workpiece pre-machining
- Changed position or size of the workpiece blank
- Wrong material, e.g. aluminum instead of steel

Process monitoring allows you to monitor the machining process during program run using monitoring tasks. The monitoring task compares the signal curve of the current execution of an NC program with one or more reference machining operations. The monitoring task uses these reference machining operations to determine an upper and lower limit. If the current machining operation is outside the limits for a defined hold time, the monitoring task responds with a defined reaction. If, for example, the spindle current drops due to a tool breakage, the monitoring task stops the NC program.

Further information: "Interrupting, stopping or canceling program run", Page 338



Drop in spindle current due to tool breakage

- | | | |
|---|---|--|
| 1 | — | References |
| 2 | — | Limits consisting of tunnel width and, if necessary, expansion |
| 3 | — | Current machining operation |
| 4 | ● | Process fault, e.g. due to tool breakage |

If you are using process monitoring, the following steps are required:

- Defining monitoring sections in the NC program
Further information: User's Manual for Programming and Testing
- Slowly running-in the NC program in Single Block mode before activating process monitoring
Further information: "Program Run", Page 333
- Activating process monitoring
Further information: "Monitoring options column", Page 269
- If necessary, configuring settings for the monitoring tasks
 - Selecting a strategy template
Further information: "Strategy template", Page 259
 - Adding or removing monitoring tasks
Further information: "Icons", Page 255
 - Defining settings and responses within the monitoring tasks
Further information: "Monitoring tasks", Page 261
 - Displaying monitoring task in the simulation as a process heat map
Further information: "Monitoring options column within a monitoring section", Page 270
Further information: User's Manual for Programming and Testing
- Running the NC program in Full Sequence
Further information: "Program Run", Page 333
- Selecting the required references as necessary for the monitoring tasks
Further information: "Monitoring tasks", Page 261
Further information: "Records of monitoring sections", Page 272

Related topics

- **Component monitoring** (option 155) with **MONITORING HEATMAP**
Further information: User's Manual for Programming and Testing

12.1.2 Process Monitoring workspace (option 168)

Application

In the **Process Monitoring** workspace the control visualizes the machining process during program run. You can activate various monitoring tasks that are relevant to the process. If necessary, you can adapt the monitoring tasks.

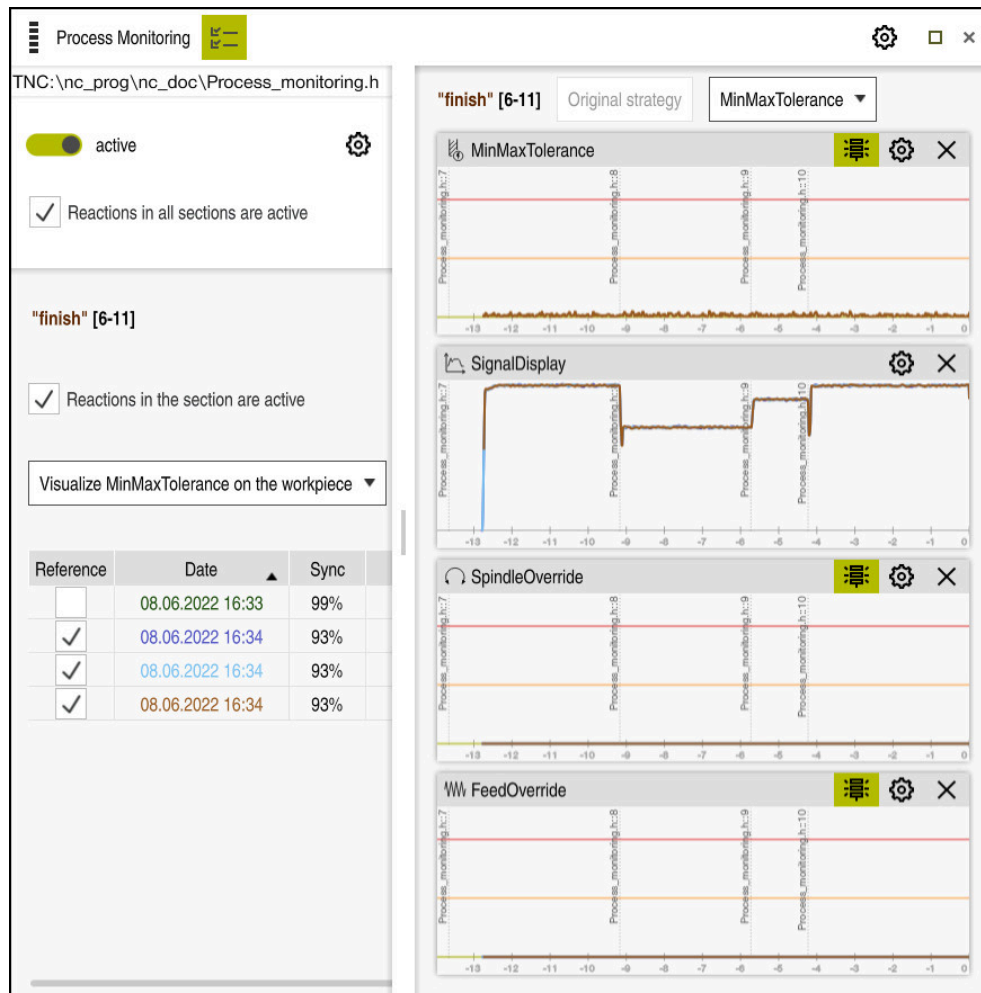
Further information: "Monitoring tasks", Page 261

Requirements

- Process Monitoring (software option 168)
- Monitoring sections defined with **MONITORING SECTION**
Further information: User's Manual for Programming and Testing
- Possible without limitation in the **FUNCTION MODE MILL** machining mode
 The monitoring tasks **FeedOverride** and **SpindleOverride** are functional in the **FUNCTION MODE TURN** machining mode (option 50).
Further information: User's Manual for Programming and Testing

Functionality

The **Process Monitoring** workspace provides information and settings for monitoring the machining process.








Process Monitoring workspace

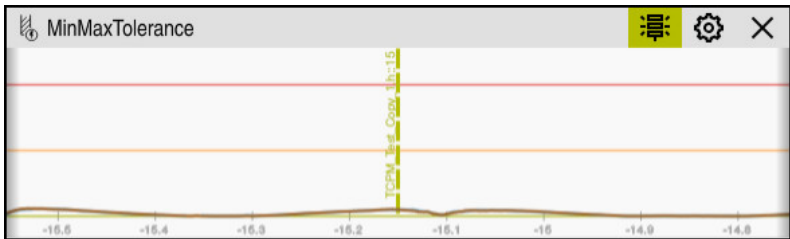
Depending on the cursor position in the NC program, the control provides the following areas:

- **Global area**
The control shows information about the active NC program.
Further information: "Global area", Page 256
- **Strategy range**
The control shows the monitoring tasks and the graphs of the recordings. You can configure settings for the monitoring tasks.
Further information: "Strategy range", Page 258
- **Monitoring options** column in the global range
The control displays information on the recordings that relate to all monitoring sections of the NC program.
Further information: "Monitoring options column in the global range", Page 270
- **Monitoring options** column within a monitoring section
The control displays information on the recordings that relate only to the currently selected monitoring section.
Further information: "Monitoring options column within a monitoring section", Page 270

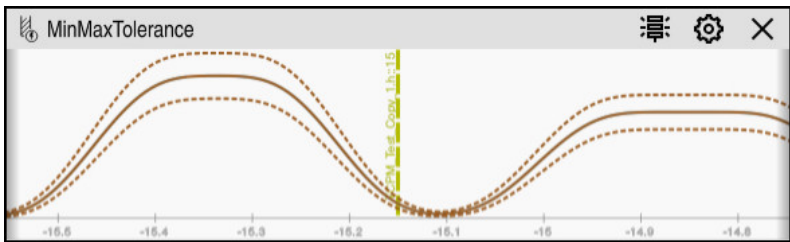
Icons

The following icons are shown in the **Process Monitoring** workspace:

Icon	Meaning
	Show or hide the Monitoring options column Further information: "Monitoring options column", Page 269
	Remove monitoring task Further information: "Monitoring tasks", Page 261
	Add monitoring task Further information: "Monitoring tasks", Page 261
	Open settings You can open the following settings: <ul style="list-style-type: none"> ■ Process Monitoring workspace setting Further information: "Settings for the Process Monitoring workspace", Page 268 ■ Setting in the Settings for NC program window of the Monitoring options column Further information: "Settings for NC program window", Page 274 ■ Monitoring task setting Further information: "Monitoring tasks", Page 261
	Show or hide warning and error limits If you show the warning and error limits, the control shows the monitored signal in relation to the defined limits. The control shows the following warning and error limits: <ul style="list-style-type: none"> ■ Green line If the current machining operation is at the bottom line, the current machining operation corresponds to the reference. ■ Orange line This line shows the warning limit. If the current machining operation exceeds the middle line, the current machining operation deviates by half the set limit of the reference. ■ Red line This line shows the error limit. If the current machining operation exceeds the upper line for a defined hold time, the monitoring task triggers a defined response, e.g. NC stop. If you hide the warning and error limits, the control shows an absolute display of the monitored signal. The dashed lines represent the upper and lower error limits, i.e. the tunnel width.



Warning and error limits displayed: The control shows the signal in relation to the defined limits



Warning and error limits hidden: The solid line represents the signal and the dashed lines represent the tunnel width determined at the time

Global area

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the global range.

Type	Description	Program
1	1 sections in 1 (sub)programs	
1	NC program has been altered compatibly	




Global area in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the global range:

- 1 **Monitoring options** icon
Further information: "Monitoring options column", Page 269
- 2 **Settings** icon for the **Process Monitoring** workspace
Further information: "Settings for the Process Monitoring workspace", Page 268
- 3 Table with notes on the active NC program
Further information: "Notes on the NC program", Page 257
- 4 **Delete hints** button
You can use the **Delete hints** button to empty the table.
- 5 Information that this area is not monitored in the NC program

Notes on the NC program

In this area, the control shows a table with information about the active NC program. The table contains the following information:

Column or symbol	Meaning
Type	In the Type column, the control shows different types of notifications.
	Information, e.g. the number of monitoring sections
	Warning, e.g. if a monitoring section has been removed
	Error, e.g. if you should reset the recordings If you make changes within a monitoring section, that monitoring section can no longer be monitored. Therefore, you should reset the recordings and set new references so that machining is monitored again. Further information: "Settings for NC program window", Page 274 You can sort the table by information type by selecting the Type column.
Description	In the Description column, the control displays information about the information types, e.g.: <ul style="list-style-type: none"> ■ Changes to the NC program ■ Cycles contained in the NC program ■ Interruptions, e.g. M0 or M1
Program line	If the information depends on an NC block number, the control displays the program name and the NC block number.

Strategy range

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the strategy range.



Strategy range in the **Process Monitoring** workspace

The **Process Monitoring** workspace shows the following in the strategy range:

- 1 **Monitoring options** icon
Further information: "Monitoring options column", Page 269
- 2 **Settings** icon for the **Process Monitoring** workspace
Further information: "Settings for the Process Monitoring workspace", Page 268
- 3 **Settings** icon for the monitoring tasks
Further information: "Monitoring tasks", Page 261
- 4 Show or hide warning and error limits
Further information: "Icons", Page 255
- 5 Monitoring tasks
Further information: "Monitoring tasks", Page 261

- 6 The control shows the following information and functions:
- Name of the monitoring section, if applicable
If **AS** is defined in the NC program with the optional syntax element, the control displays the name.
If no name is defined, the control displays **MONITORING SECTION**.
Further information: User's Manual for Programming and Testing
 - Range of NC block numbers of the monitoring section in square brackets
Start and end of the monitoring section in the NC program
 - **Original strategy** or **Save strategy as template** button
Further information: "Strategy template", Page 259
 - Selection menu for strategy template
Further information: "Strategy template", Page 259

Strategy template

A strategy template includes one or more monitoring tasks, including the defined settings.

You can choose between the following strategy templates using a selection menu:

Strategy template	Meaning
MinMaxTolerance	<p>This strategy template includes the following monitoring tasks:</p> <ul style="list-style-type: none"> ■ MinMaxTolerance Further information: "Monitoring task MinMaxTolerance", Page 262 ■ SignalDisplay Further information: "Monitoring task SignalDisplay", Page 266 ■ SpindleOverride Further information: "Monitoring task SpindleOverride", Page 266 ■ FeedOverride Further information: "Monitoring task FeedOverride", Page 267
StandardDeviation	<p>This strategy template includes the following monitoring tasks:</p> <ul style="list-style-type: none"> ■ StandardDeviation Further information: "Monitoring task StandardDeviation", Page 265 ■ SignalDisplay Further information: "Monitoring task SignalDisplay", Page 266 ■ SpindleOverride Further information: "Monitoring task SpindleOverride", Page 266 ■ FeedOverride Further information: "Monitoring task FeedOverride", Page 267
User-defined	In this strategy template, you can compile the monitoring tasks yourself.

If you modify a strategy template, you can overwrite the modified strategy template by clicking the **Save strategy as template** button. The control overwrites the currently selected strategy template.



Since you cannot restore the as-delivered state of the strategy templates yourself, only overwrite the **User-defined** template.

The machine manufacturer can use the optional machine parameter **ProcessMonitoring** (no. 133700) to restore the as-delivered state of the strategy templates.

In the settings of the **Process Monitoring** workspace, you can define which strategy template the control selects by default after creating a new monitoring section.

Further information: "Settings for the Process Monitoring workspace", Page 268

Monitoring tasks

You can change the settings and responses of the monitoring tasks for each monitoring section. You can also remove a monitoring task for a monitoring section or add one using the plus sign.

The **Process Monitoring** workspace contains the following monitoring tasks:

- **MinMaxTolerance**

With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references including the percentage deviation.

Further information: "Monitoring task MinMaxTolerance", Page 262

- **StandardDeviation**

With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including expansion by the factor σ .

Further information: "Monitoring task StandardDeviation", Page 265

- **SignalDisplay**

With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

Further information: "Monitoring task SignalDisplay", Page 266

- **SpindleOverride**

With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

Further information: "Monitoring task SpindleOverride", Page 266

- **FeedOverride**

With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

Further information: "Monitoring task FeedOverride", Page 267

The values of the monitoring tasks set in the as-delivered state are considered recommended output values, which you can adapt if necessary in special machining situations.

In each monitoring task, the control shows the current processing and the selected references as a graph. The time axis is specified in seconds.

Monitoring task MinMaxTolerance

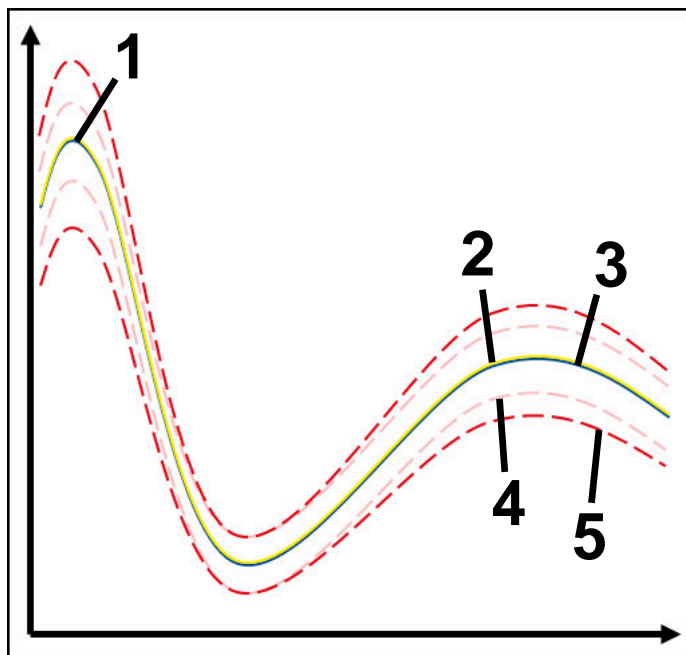
With **MinMaxTolerance**, the control monitors whether the current machining operation is within the range of the selected references including the percentage deviation.

The percentage deviation takes into account the tool wear.

The application cases of **MinMaxTolerance** are significant process faults, e.g. during small series production:

- Tool breakage
- Missing tool
- Changed position or size of the workpiece blank

The control requires at least one recorded machining operation as a reference. If you do not select a reference, this monitoring task is inactive and does not draw a graph.



- 1 — First good reference
- 2 — Second good reference
- 3 — Third good reference
- 4 — Limits consisting of the tunnel width
- 5 — Limits consisting of a percentage expansion of the tunnel width

Further information: "Records of monitoring sections", Page 272

If, for example, you have a recording that is only just acceptable due to tool wear, you can also use an alternative application with this monitoring task.

Further information: "Alternative application with acceptable reference", Page 264

Settings for MinMaxTolerance

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

Percentage expansion of tunnel width

- **Static tunnel width**

Upper and lower limits, based on references

- **Hold time**

Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**

If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

Further information: "Message menu on the information bar", Page 301

- **Stop NC program**

If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.

- **Lock current tool**

If the signal exceeds the warning limits for the defined hold time, the control blocks the tool in the tool management.

Further information: "Tool management ", Page 165

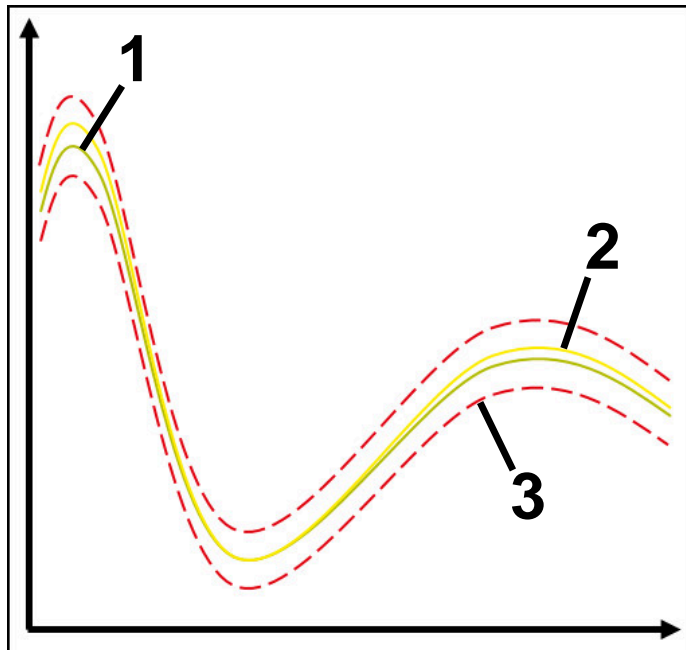
Alternative application with acceptable reference

If the control has recorded a machining operation that is only just acceptable, you can use an alternative application of the monitoring task **MinMaxTolerance**.

Select at least two references:

- An optimal reference
- A reference that is only just acceptable, e.g. showing a higher signal of the spindle load due to tool wear

The monitoring task checks whether the current machining operation is within the range of the selected references. For this strategy, select no deviation or a low percentage deviation, since the tolerance is already given by the different references.



- 1 ——— Optimal reference
- 2 ——— Reference only just acceptable
- 3 - - - Limits consisting of the tunnel width

Monitoring task StandardDeviation

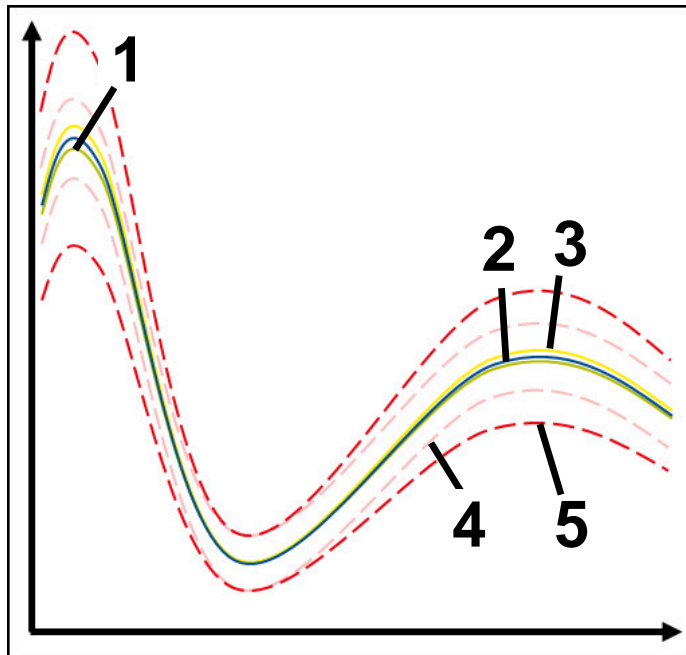
With **StandardDeviation**, the control monitors whether the current machining operation is within the range of the selected references, including expansion by the factor σ .

The application cases of **StandardDeviation** are process faults of all kinds, e.g. during series production:

- Tool breakage
- Missing tool
- Tool wear
- Changed position or size of the workpiece blank

The control requires at least three recorded machining operations as a reference. The references should include one optimal, one good and one acceptable machining operation. If you do not select the required references, this monitoring task is not active and does not draw a graph.

Further information: "Records of monitoring sections", Page 272



- 1 — Optimal reference
- 2 — Good reference
- 3 — Reference only just acceptable
- 4 — Limits consisting of the tunnel width
- 5 — Limits consisting of the expansion of the tunnel width multiplied by factor σ

Settings for StandardDeviation

You can use sliders to configure the following settings for this monitoring task:

- **A multiple of σ**
Expansion of the tunnel width multiplied by factor σ
- **Static tunnel width**
Upper and lower limits, based on references
- **Hold time**
Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**
If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.
Further information: "Message menu on the information bar", Page 301
- **Stop NC program**
If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.
- **Lock current tool**
If the signal exceeds the warning limits for the defined hold time, the control blocks the tool in the tool management.
Further information: "Tool management ", Page 165

Monitoring task SignalDisplay

With **SignalDisplay**, the control shows the process progress of all selected references and the current machining operation.

You can compare whether the current machining operation corresponds to the references. This allows you to visually check whether you can use the machining operation as a reference.

The monitoring task does not respond.

Monitoring task SpindleOverride

With **SpindleOverride**, the control monitors changes to the spindle override by the potentiometer.

The control uses the first recorded machining operation as a reference.

Settings for SpindleOverride

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

Accepted deviation of the override in percent compared to the first recording

- **Hold time**

Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**

If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

Further information: "Message menu on the information bar", Page 301

- **Stop NC program**

If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.

Monitoring task FeedOverride

With **FeedOverride**, the control monitors changes in the feed override by the potentiometer.

The control uses the first recorded machining operation as a reference.

FeedOverride settings

You can use sliders to configure the following settings for this monitoring task:

- **Accepted percentage difference**

Accepted deviation of the override in percent compared to the first recording

- **Hold time**

Maximum time in milliseconds for which the signal is permitted to be outside the defined deviation. Once this period has expired, the control triggers the defined response of the monitoring task.

You can activate or deactivate the following responses for this monitoring task:

- **Issue warning message**

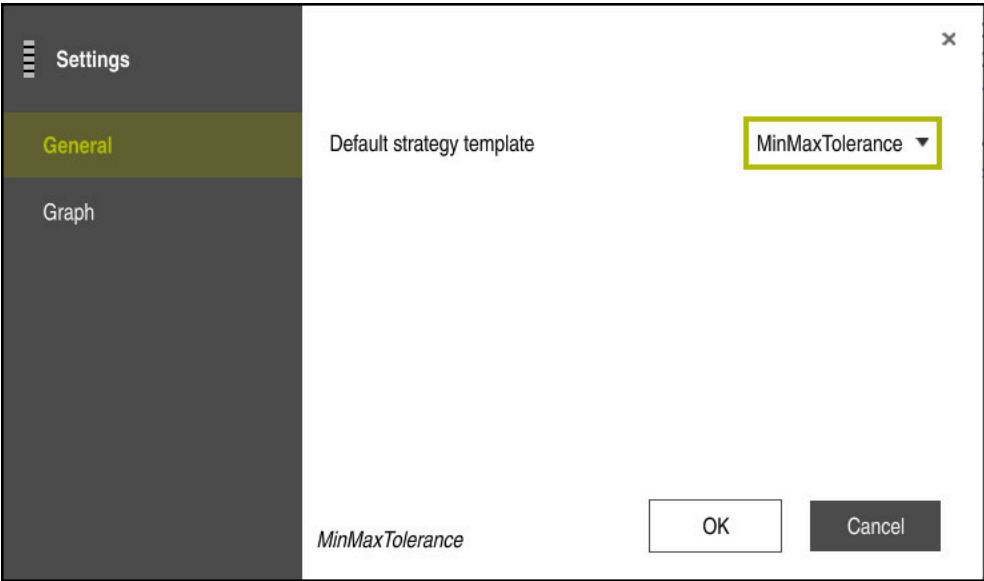
If the signal exceeds the limits for the defined hold time, the control shows a warning in the notification menu.

Further information: "Message menu on the information bar", Page 301

- **Stop NC program**

If the signal exceeds the warning limits for the defined hold time, the control stops the NC program.

Settings for the Process Monitoring workspace



Settings for the **Process Monitoring** workspace

General

In the **General** area, select which strategy template the control uses as the default:

- **MinMaxTolerance**
- **StandardDeviation**
- **User-defined**

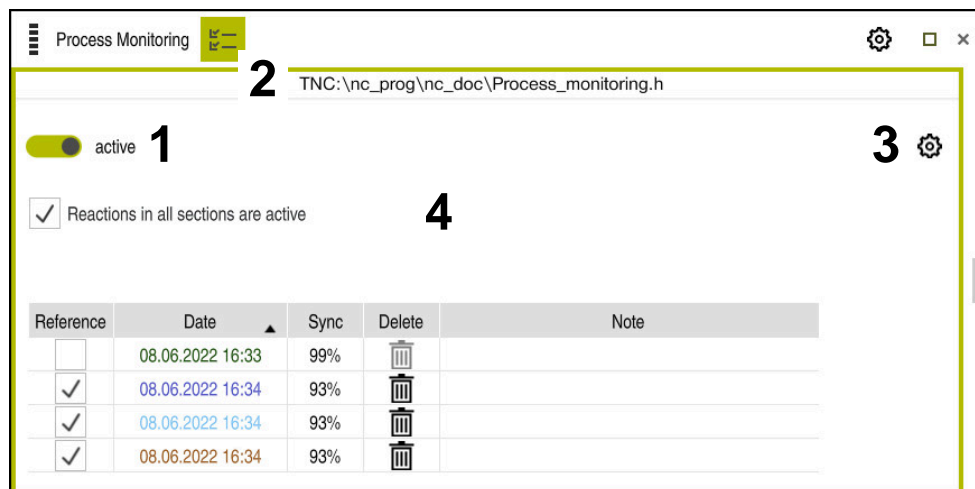
Further information: "Strategy template", Page 259

Graph

In the **Graph** area, you can select the following settings:

Setting	Meaning
Simultaneous-ly plotted refer-ences	<p>Select the maximum number of recordings that the control displays simultaneously as graphs in the monitoring tasks:</p> <ul style="list-style-type: none"> ■ 2 ■ 4 ■ 6 ■ 8 ■ 10 <p>If more references are selected than the control is to display, the control displays the last selected references as a graph.</p>
Preview [s]	<p>The control can run a selected reference as a preview during processing. The control shifts the time axis of the machining operation to the left.</p> <p>Select how many seconds of the reference the control will preview:</p> <ul style="list-style-type: none"> ■ 0 ■ 2 ■ 4 ■ 6 <p>Further information: "Records of monitoring sections", Page 272</p>

Monitoring options column



Monitoring options column in the global range

The **Monitoring options** column shows the following in the upper area regardless of the cursor position in the NC program:

- 1 Switch for activating or deactivating process monitoring for the entire NC program
- 2 Path of the current NC program
- 3 Open the **Settings** icon in the **Settings for NC program** window
Further information: "Settings for NC program window", Page 274
- 4 Checkbox for activating or deactivating the responses of all monitoring sections in the NC program

Depending on the cursor position in the NC program, the control provides the following areas:

- **Monitoring options** column in the global range
You can select references that are effective for all monitoring sections of the NC program.
Further information: "Monitoring options column in the global range", Page 270
- **Monitoring options** column within a monitoring section
You can define settings and select references that apply to the currently selected monitoring section.
Further information: "Monitoring options column within a monitoring section", Page 270

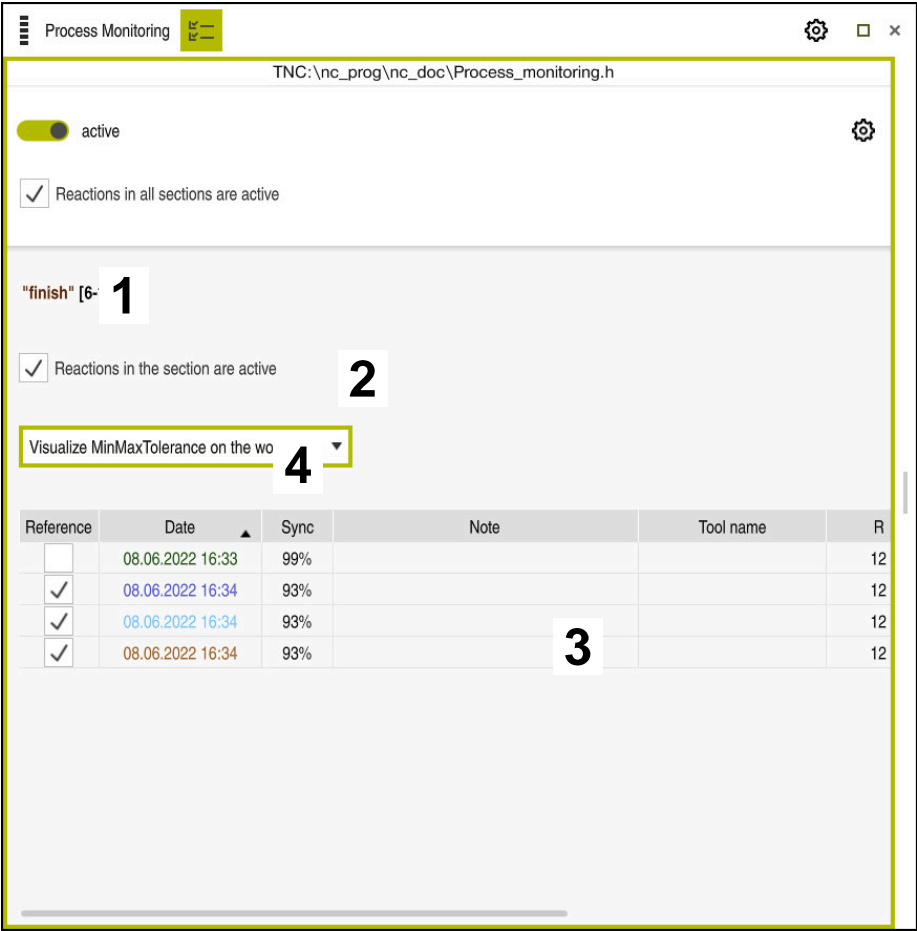
Monitoring options column in the global range

If the cursor is outside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column in the global area. In the global area, the control displays a table with the records of all monitoring sections of the NC program.

Further information: "Records of monitoring sections", Page 272

Monitoring options column within a monitoring section

If the cursor is inside a monitoring section in the NC program, the **Process Monitoring** workspace displays the **Monitoring options** column within the monitoring section. If the cursor is within the monitoring section, the control will leave this area gray.



Monitoring options column within the monitoring section

The **Monitoring options** column displays the following within the monitoring section:

- The control shows the following information and functions:
 - Name of the monitoring section, if applicable
If **AS** is defined in the NC program with the optional syntax element, the control displays the name.
If no name is defined, the control displays **MONITORING SECTION**.
Further information: User's Manual for Programming and Testing
 - Range of NC block numbers of the monitoring section in square brackets
Start and end of the monitoring section in the NC program

- 2 Checkbox for activating and deactivating the responses in the monitoring section
You can activate or deactivate the responses of the currently selected monitoring section.
- 3 Table with the recordings of the monitoring section
The recordings refer only to the monitoring section in which the cursor is currently located.
Further information: "Records of monitoring sections", Page 272
- 4 Selection menu for the process heat map
You can display a monitoring task as a process heat map in the **Simulation** workspace.
Further information: User's Manual for Programming and Testing

Records of monitoring sections

The contents and functions of the table with the recordings of the machining operations depend on the cursor position in the NC program.

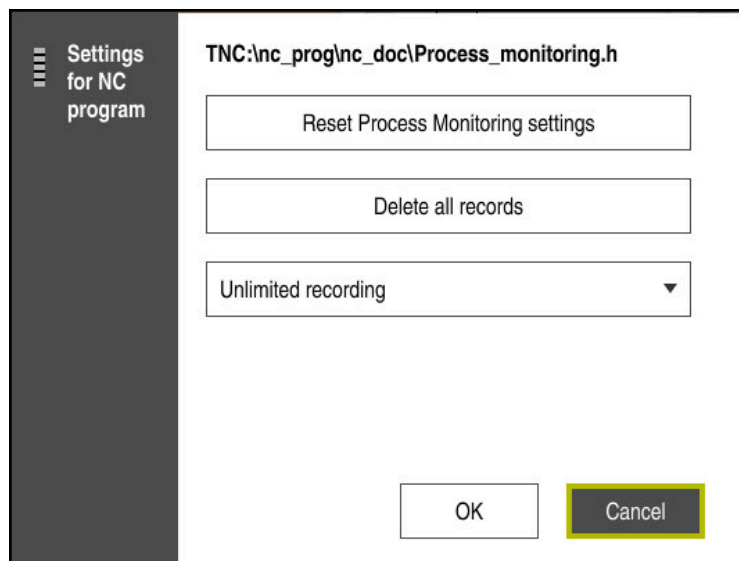
Further information: "Monitoring options column", Page 269

The table contains the following information about the monitoring section:

Column	Information or action
Reference	<p>If you activate the checkbox for a table row, the control uses this recording as a reference for the corresponding monitoring tasks.</p> <p>If you activate multiple table rows, the control uses all selected rows as references. If you select multiple references with greater deviation, the tunnel width also increases. You can select up to ten references at a time.</p> <p>The effect of the reference depends on the position of the cursor in the NC program:</p> <ul style="list-style-type: none"> ■ Within the monitoring section: <p>The reference only applies to the currently selected monitoring section.</p> <p>The control shows a dash in the global range in this table row for information. If a table row is marked as a reference in all strategy areas or in the global area, the control displays a check mark.</p> ■ Global area <p>The reference applies to all monitoring sections of the NC program.</p> <p>Mark recordings that have a provided satisfactory result, such as a clean surface, as a reference.</p> <p>You can only select a recording of a complete machining operation as a reference.</p>
Date	<p>Date and time of program start of each recorded machining operation</p> <p>If you select the Date column, the control sorts the table by date.</p>
Sync	<p>Quality of synchronization</p> <p>The quality is influenced by the following:</p> <ul style="list-style-type: none"> ■ Time delay, e.g. change of the feed rate override <p>If the potentiometer position of the feed rate override deviates from the reference machining process, the quality becomes worse.</p> ■ Local delay, e.g. due to a tool correction with DR <p>If the path of the tool center point TCP deviates from the reference machining operation, the quality becomes worse.</p> <p>The first row of the table is the reference for the quality of the following table rows.</p> <p>Further information: "Tool center point (TCP, tool center point)", Page 143</p> <p>If the quality is between 70% and 80%, the processing operation is still OK. You should manually check the recording in this area.</p>

Column	Information or action
Delete	<p>If you select the trash can icon, the control deletes the table row.</p> <p>You cannot delete the first row of the table because this row is used as a reference for the following functions:</p> <ul style="list-style-type: none"> ■ Sync column ■ SpindleOverride monitoring task ■ FeedOverride monitoring task <p>You delete all recordings including the first in the Settings for NC program window.</p> <p>Only in the global area</p>
Note	In the Note column, you can enter notes about the table row.
Tool name	<p>Name of the tool from the tool management</p> <p>Only within the monitoring section</p> <p>Further information: "Tool management ", Page 165</p>
R	<p>Radius of the tool from the tool management</p> <p>Only within the monitoring section</p> <p>Further information: "Tool management ", Page 165</p>
DR	<p>Delta value of the tool radius from the tool management</p> <p>Only within the monitoring section</p> <p>Further information: "Tool management ", Page 165</p>
L	<p>Length of the tool from the tool management</p> <p>Only within the monitoring section</p> <p>Further information: "Tool management ", Page 165</p>
CUT	<p>Number of cutting edges of the tool from the tool management</p> <p>Only within the monitoring section</p> <p>Further information: "Tool management ", Page 165</p>
CURR_TIME	<p>Tool life from the tool management at the start of the respective machining operation</p> <p>Only within the monitoring section</p> <p>Further information: "Tool management ", Page 165</p>

Settings for NC program window



Settings for NC program window

The **Settings for NC program** window provides the following settings:

- **Reset Process Monitoring settings**
- **Delete all records**, including the first table row
- Select the maximum number of recordings of machining operations in the table:
 - **Limit to 5 recordings**
 - **Limit to 10 recordings**
 - **Limit to 50 recordings**
 - **Limit to 200 recordings**
 - **Unlimited recording**

If the number of machining operations exceeds the maximum number, the control overwrites the last machining operation.

Further information: "Records of monitoring sections", Page 272

Notes

- If you use different sizes of workpiece blanks, set process monitoring to a more tolerant setting or start the first monitoring section after pre-machining.
- If the spindle load is too low, the control may not detect any difference from idling, e.g. for a tool with a small diameter.
- If you remove and add a monitoring task again, the previous recordings remain.

Basics of operation

- You can zoom in or out of the graph horizontally by dragging or scrolling.
- If you drag or swipe with the left mouse button held down, you can move the graph.
- You can align the graph by selecting an NC block number. The control marks the selected NC block number in green within the monitoring task.

Further information: "Common gestures for the touchscreen", Page 70

13

**Opening CAD Files
with the CAD-
Viewer**

13.1 Fundamentals

Application

CAD-Viewer enables you to open following standardized CAD data formats directly on the control:

File	Type	Format
Step	.STP and .STEP	■ AP 203 ■ AP 214
IGES	.IGS and .IGES	■ Version 5.3
DXF	.DXF	■ R10 to 2015
STL	STL	■ Binary ■ ASCII

The **CAD-Viewer** runs as a separate application on the third desktop of the control.

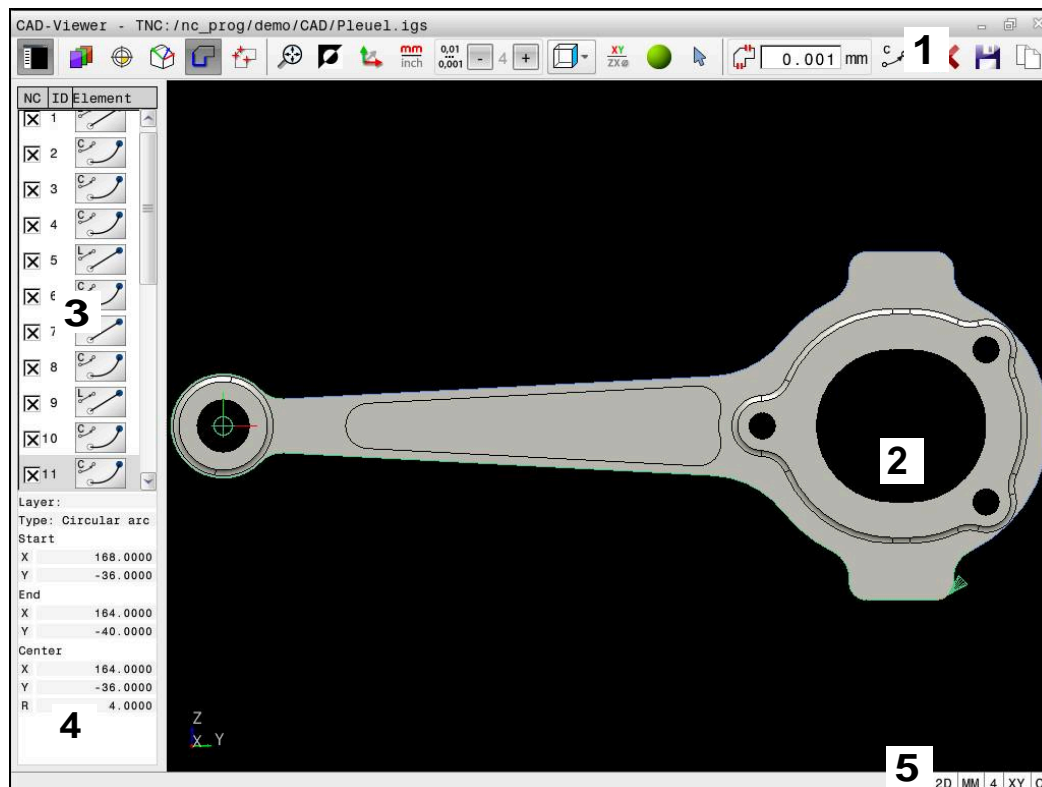
Related topics

- Creating 2D sketches on the control

Further information: User's Manual for Programming and Testing

Description of function

Screen layout



CAD file open in **CAD-Viewer**












CAD Viewer consists of the following areas:








- 1 Menu bar
Further information: "Menu bar icons", Page 277
- 2 Graphics window
The CAD model is displayed in the graphics window.
- 3 Sidebar window
The Sidebar window displays information on the active function, e.g. available layers or the position of the workpiece preset.
- 4 Element Information window
Further information: "Element Information window", Page 279
- 5 Status bar
The status bar contains the active settings.

Menu bar icons

The menu bar contains the following icons:

Icon	Function
	Show sidebar Show or hide the Sidebar window
	Display the layer Display the layer(s) in the Sidebar window Further information: "Layer", Page 280

Icon	Function
	Preset Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset Further information: "Workpiece preset in the CAD model", Page 281
	Datum Set the datum The datum has been set Further information: "Workpiece datum in the CAD model", Page 284
	Contour Select contour (option 42) Further information: "Applying contours and positions to NC programs with CAD Import (option 42)", Page 286
	Positions Select drilling positions (option 42) Further information: "Applying contours and positions to NC programs with CAD Import (option 42)", Page 286
	3D mesh Create a 3D mesh (option 152) Further information: "Generating STL files with 3D mesh (option 152)", Page 292
	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) of the output. Further information: "Applying contours and positions to NC programs with CAD Import (option 42)", Page 286
	Number of decimal places Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. Further information: "Applying contours and positions to NC programs with CAD Import (option 42)", Page 286 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

Icon	Function
	Axes Select the working plane: <ul style="list-style-type: none"> ■ XY ■ YZ ■ ZX ■ ZXØ <p>In the ZXØ working plane, you can select turning contours (option 50).</p> <p>If you take over a contour or position, the control will output the NC program in the selected working plane.</p> <p>Further information: "Applying contours and positions to NC programs with CAD Import (option 42)", Page 286</p>
	Toggle a 3D model between a solid model and a wire-frame model.
	"Select, add, or remove contour elements" mode
	<div>  The icon shows the current mode. Clicking the icon activates the next mode. </div>
	Further information: "Applying contours and positions to NC programs with CAD Import (option 42)", Page 286
	Undo

Element Information window

In the Element Information window, the following information is displayed for the selected element of the CAD file:

- Associated layer
- Element type
- Point type:
 - Point coordinates
- Line type:
 - Coordinates of the starting point
 - Coordinates of the end point
- Circular arc or circle type:
 - Coordinates of the starting point
 - Coordinates of the end point
 - Coordinates of the center point
 - Radius

Layer

CAD files usually contain several layers. The designer uses these layers to create groups of various types of elements, e.g. the actual workpiece contour, dimensions, auxiliary and design lines, shadings, and texts.

The CAD file to be processed must contain at least one layer. Elements not assigned to a layer are automatically moved by the control to the "anonymous" layer.

With the **Display the layer** icon, the control shows all the layers of the file in the List view window. Use the checkbox in front of the name to show and hide the individual layers.

When you open a CAD file in the **CAD-Viewer**, all layers available are shown.

If you hide unnecessary layers, the graphic becomes clearer.

Notes

- The control does not support binary DXF format. Save the DXF file in ASCII format in the CAD or drawing program.
- Before loading the file into the control, ensure that the name of the file contains only permitted characters.

Further information: User's Manual for Programming and Testing

- When you select a layer in the list view window, you can use the space bar to show and hide the layer.

13.2 Workpiece preset in the CAD model

Application

The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

Related topics

- Presets in the machine

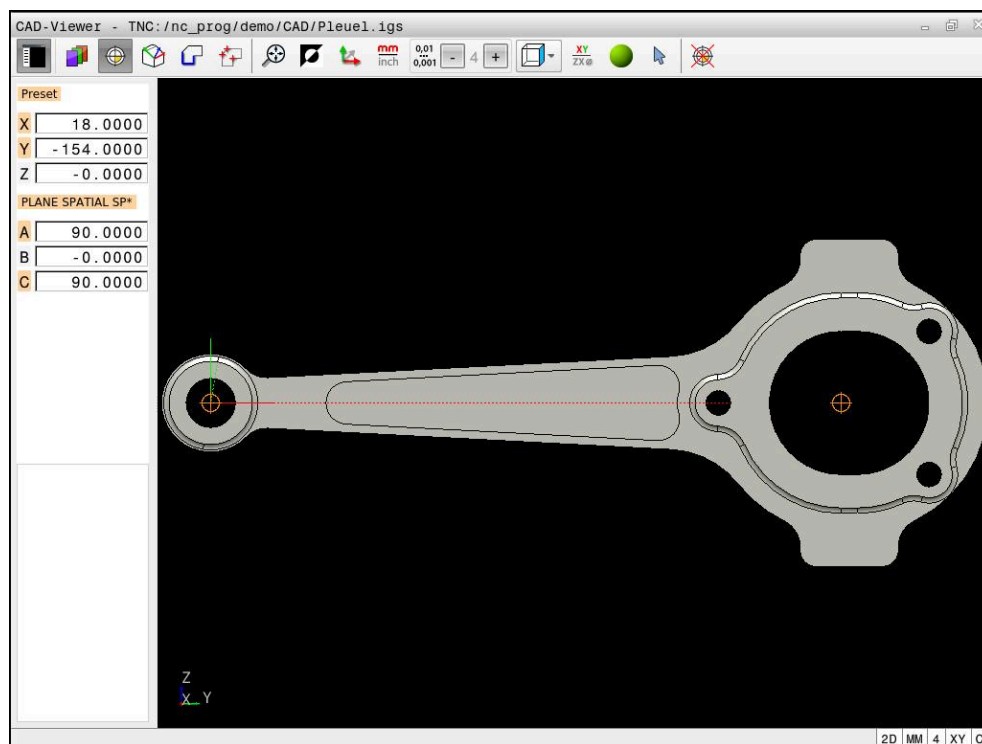
Further information: "Presets in the machine", Page 137

Description of function

When you select the **Preset** icon, the control displays the following information in the list view window:

- Distance between the defined preset and the drawing datum
- Orientation of the coordinate system with respect to the drawing

The control displays values not equal to 0 in orange.



Workpiece preset in the CAD model

You can position the preset at the following locations:

- By directly inputting numerical values into the List View window
- For straight lines:
 - Starting point
 - Center
 - End point
- For circular arcs:
 - Starting point
 - Center
 - End point
- For full circles:
 - At the quadrant transitions
 - At the center
- At the intersection between:
 - Two straight lines, even if the point of intersection is actually on the extension of one of the lines
 - Straight line and circular arc
 - Straight line and full circle
 - Two circles (regardless of whether a circular arc or a full circle)

If you have set a workpiece preset, the control displays the **Preset** icon in the menu bar with a yellow quadrant.

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;origin = X... Y... Z...
```

```
5 ;origin_plane_spatial = SPA... SPB... SPC...
```

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 Set the workpiece preset or workpiece datum and align the coordinate system



- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
Further information: "Common gestures for the touchscreen", Page 70
- The following contents also apply to the workpiece datum. In this case, select the **Datum** icon to begin with.

Set workpiece preset or workpiece datum on individual element

To set the workpiece preset on an individual element:



- ▶ Select **Preset**
- ▶ Position the cursor on the desired element
- ▶ If you are using a mouse, the control for the element displays selectable presets using gray icons.
- ▶ Click the icon at the desired position
- ▶ The control sets the workpiece preset to the selected position. The control turns the icon green.
- ▶ Align the coordinate system, if required

Set the workpiece preset or workpiece datum at the intersection of two elements

You can set the workpiece preset at intersection points of straight lines, full circles and arcs.

To set the workpiece preset at the intersection of two elements:



- ▶ Select **Preset**
- ▶ Click on the first element
- > The control highlights the element in color.
- ▶ Click on the second element
- > The control sets the workpiece preset at the intersection of the two elements. The control marks the workpiece preset with a green icon.
- ▶ Align the coordinate system, if required



- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Aligning the coordinate system

The following conditions must be met in order to align the coordinate system:

- Preset has been defined
- There are elements next to the preset that can be used for the desired alignment

To align the coordinate system:

- ▶ Select an element in the positive direction of the X axis
- > The control aligns the X axis.
- > The control changes the angle **C** in the list view window.
- ▶ Select an element in the positive direction of the Y axis
- > The control aligns the Y and Z axes.
- > The control changes the angles **A** and **C** in the list view window.

13.3 Workpiece datum in the CAD model

Application

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a tilting operation.

Related topics

- Presets in the machine

Further information: "Presets in the machine", Page 137

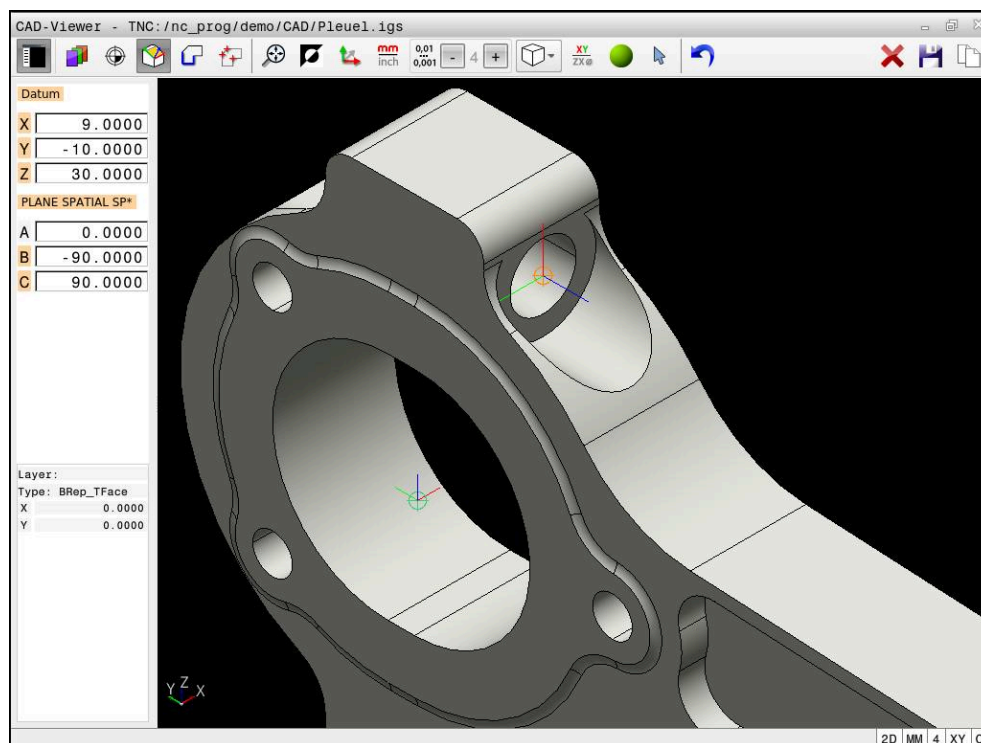
Description of function

When you select the **Datum** icon, the control displays the following information in the list view window:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the coordinate system

You can set a workpiece datum and also move it further by entering values directly in the list view window.

The control displays values not equal to 0 in orange.



Workpiece datum for tilted machining

The datum with the orientation of the coordinate system can be set at the same positions as a preset.

Further information: "Workpiece preset in the CAD model", Page 281

If you have set a workpiece datum, the control displays the **Datum** icon in the menu bar with a yellow area.

Further information: "Set the workpiece preset or workpiece datum and align the coordinate system", Page 283

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

4 TRANS DATUM AXIS X... Y... Z...

5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

4 ;TRANS DATUM AXIS X... Y... Z...

5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX

13.4 Applying contours and positions to NC programs with CAD Import (option 42)

Application

You can open CAD files directly on the control in order to extract contours and machining positions from it. 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 contain only **L** and **CC/C** blocks.

Related topics

- Using point tables

Further information: User's Manual for Machining Cycles

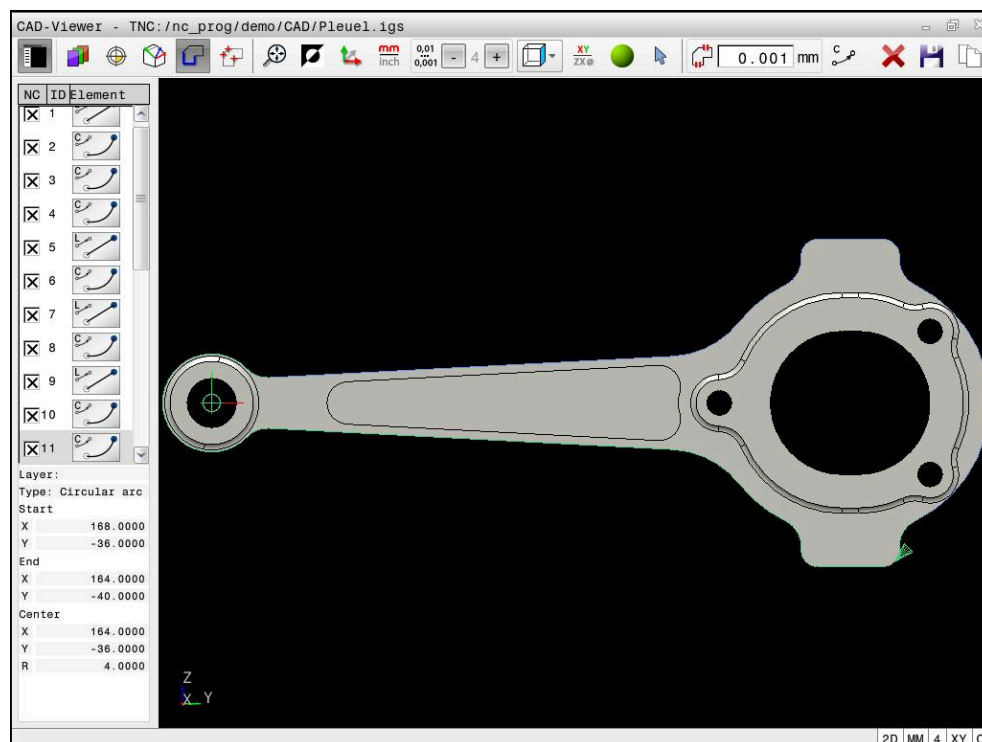
Requirement

- CAD Import (software option 42)

Description of function

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to the software tools (e.g., **Leafpad** or **Gnumeric**).










Further information: "Opening files with additional software", Page 506



CAD model with marked contour

Icons in the CAD Import

With the CAD Import, the control shows the following additional functions in the menu bar:

Icon	Function
	Delete entire list
	Save entire list content to a file
	Copy entire list contents to clipboard
	Set the transition tolerance The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred when the drawing was made. The default setting is 0.001 mm
 	C or CR Arc mode defines whether circular arcs are output in C format or CR format (e.g., for cylinder surface interpolation) in the NC program.
	Show connections between two positions Specifies whether the control should display the tool path as a dashed line during the selection of machining positions
	Apply path optimization The control optimizes the tool traverse movement to produce shorter traverse movements between the machining positions. When the icon is pressed again, the optimization is reset
	Find circles according to diameter range. Load center coordinates to the position list The control opens a pop-up window in which you can filter holes (full circles) based on their size

Applying contours

The following elements can be selected as a contour:

- Line segment
- Circle
- Circular arc
- Polyline
- Any curves (e.g., splines, ellipses)

You can also use the CAD viewer (option 50) to select contours for turning. The icon is grayed out if option 50 is not enabled. Before selecting a turning contour, you must set the preset on the rotary axis. If you select a turning contour, it is saved with Z and X coordinates. In addition, all X coordinate values in turning contours are transferred as diameter values, i.e. the drawing dimensions for the X axis are doubled. All contour elements below the rotary axis cannot be selected and are highlighted gray.

Linearization

During linearization, a contour is divided into individual positions. The CAD Import creates a straight line **L** for each position. With the CAD Import, you can therefore also apply contours that cannot be programmed with the path functions of the control, e.g. splines.

The **CAD-Viewer** linearizes all of the contours that are not in the XY plane. The finer the resolution, the more accurately the control will display the contours.

Applying positions

You can also use the CAD Import to save positions, e.g. for holes.

Three possibilities are available in the pattern generator for defining machining positions:

- Single selection
- Multiple selection within a range
- Multiple selection using search filters

Further information: "Select positions", Page 290

The following file types are available:









- Point table (**.PNT**)
- Klartext conversational language program (**.H**)

If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).

Multi-selection filter settings

After you have used the quick selection function to mark hole positions, a pop-up window appears in which the smallest diameter found is to the left and the largest diameter to the right. With the buttons just below the diameter display you can adjust the diameter so that you can load the hole diameters that you want.

The following buttons are available:

Icon	Filter setting of smallest diameter
	Display the smallest diameter found (default setting)
	Display the next smaller diameter found
	Display the next larger diameter found
	Display the largest diameter found. The control sets the filter for the smallest diameter to the value set for the largest diameter
Icon	Filter setting of largest diameter
	Display the smallest diameter found. The control sets the filter for the largest diameter to the value set for the smallest diameter
	Display the next smaller diameter found
	Display the next larger diameter found
	Display the largest diameter found (default setting)

13.4.1 Selecting and saving a contour



- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.
Further information: "Common gestures for the touchscreen", Page 70
- Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.

Selecting a contour with existing contour elements

To select and save a contour with existing contour elements:



- ▶ Select **Contour**
- ▶ Place the cursor on the first contour element
- ▶ The control shows the suggested direction of rotation as a dashed line.
- ▶ If necessary, move the cursor towards the more distant end point.
- ▶ The control changes the suggested direction of rotation.
- ▶ Select the contour element
- ▶ The selected contour element is displayed in blue and is marked in the Sidebar window.
- ▶ Other contour elements are shown in green.



The control suggests the contour that deviates least from the suggested direction. To change the suggested contour path, you can select paths independently of the existing contour elements

- ▶ Select the last desired contour element
- ▶ All contour elements up to the selected element are shown in blue and are marked in the Sidebar window.
- ▶ Select **Save entire list content to a file**
- ▶ The **Define file name for contour program** window opens.
- ▶ Enter the desired name
- ▶ Select the path to the storage location
- ▶ Select **Save**
- ▶ The selected contour is saved as an NC program.



- Alternatively, you can use the **Copy entire list contents to clipboard** icon to copy the selected contour to the clipboard and then paste it into an existing NC program.
- If you select an element with the CTRL key pressed, it is deselected for export.

Selecting paths independent of existing contour elements

To select a path independent of existing contour elements:



- ▶ Select **Contour**



- ▶ Select **Select**
 - The icon changes, and the control activates the **Add** mode.
 - ▶ Place the cursor relative to the desired contour element
 - The control displays selectable points:
 - End point or center point of a line or curve
 - Quadrant transitions or center of a circle
 - Points of intersection between existing elements
 - ▶ Select the desired point
 - ▶ Select more contour elements



If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line. If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.

Saving the contour as a workpiece blank definition (option 50)

For a workpiece blank definition in turning mode, a closed contour is required.

NOTICE

Danger of collision!

Closed contours must completely lie inside the workpiece blank definition. Otherwise, the system will follow closed contours also along the rotary axis when machining, causing collisions.

- ▶ Select or program only those contour elements that are actually required, e.g. within the definition of a finished part.

To select a closed contour:



- ▶ Select **Contour**
 - ▶ Select all required contour elements
 - ▶ Select the starting point of the first element
 - The control closes the contour.

13.4.2 Select positions



- The following instructions apply to the use of a mouse. You can also perform these steps with touch gestures.

Further information: "Common gestures for the touchscreen", Page 70

- Deselecting, deleting, and saving of elements works in the same way for taking over contours and positions.
"Selecting and saving a contour"

Individual selection

To select individual positions, e.g. holes:



- ▶ Select **Positions**
- ▶ Position the cursor on the desired element
- The control shows the circumference and center point of the element in orange.
- ▶ Select the desired element
- The control highlights the selected element in blue and displays it in the list view window.

Multiple selection by area

To select multiple positions within an area:



- ▶ Select **Positions**
- ▶ Select **Select**
- The icon changes, and the control activates the **Add** mode.
- ▶ Drag a box around the area while holding down the left mouse button
- The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameter found.
- ▶ Change the filter settings as needed
- ▶ Press **OK**
- The control highlights all positions of the selected diameter range in blue and shows them in the list view window.
- The control shows the traversing distance between the positions.

Multiple selection by search filter

To select multiple positions using a search filter:



- ▶ Select **Positions**
- ▶ **Find circles according to diameter range. Select Find circles according to diameter range. Load center coordinates to the position list**
- The control opens the **Find circle centers after diameter range** window and shows the smallest and largest diameter found.
- ▶ Change the filter settings as needed
- ▶ Press **OK**
- The control highlights all positions of the selected diameter range in blue and shows them in the list view window.
- The control shows the traversing distance between the positions.

Notes

- Set the correct unit of measure, because the CAD file does not contain this information.
- 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 also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.

Notes on applying contours

- If you double-click a layer in the list view window, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.
- Select the first contour element such that approach without collision is possible.
- You can even select a contour if the designer has saved the lines on different layers.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.
As an alternative, you can switch to the Remove mode:



13.5 Generating STL files with 3D mesh (option 152)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective files of fixtures and tool holders, for example, or to position STL files generated from the simulation for another machining operation.

Related topics

- Fixture Monitoring (option 40)
- Export simulated workpiece as STL file
- Using an STL file as workpiece blank

Further information: User's Manual for Programming and Testing

Requirement

- Software option 152, CAD Model Optimizer

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, e.g. small holes in a solid or self-intersections of a surface.

Option	Meaning
Repairs	<p>Indicates whether the original model has been repaired or not. If it has been repaired, the control will indicate the type of repair, e.g. Yes: Hole Int Shells.</p> <p>This indication consists of the following items:</p> <ul style="list-style-type: none"> ■ Hole CAD-Viewer closed holes in the 3D model. ■ Int CAD-Viewer removed self-intersections. ■ Shells CAD-Viewer joined multiple separate solids.

In order to use STL files for control functions, the saved files must meet the following requirements:





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

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD-Viewer provides the following functions:

Symbol	Function
	<p>Allowed simplification</p> <p>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.</p>
	<p>Remove holes <= diameter</p> <p>Use this function to remove holes and pockets up to the specified diameter from the original model.</p>
	<p>Only optimized mesh shown</p> <p>To be able to evaluate the deviations, use this function to toggle superimposition of the view containing the optimized triangle mesh with the original mesh from the original file.</p>
	<p>Save</p> <p>Use this function to save the simplified 3D model with the selected settings as an STL file.</p>

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

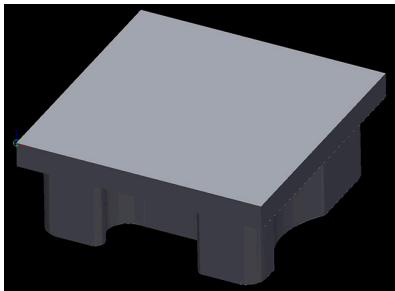
Further information: User's Manual for Programming and Testing



- ▶ Select the **Files** operating mode



- ▶ Select the exported STL file
- ▶ The control opens the CAD files in **CAD-Viewer**.
- ▶ Select **Preset**
- ▶ In the Sidebar window, the control displays information on the position of the preset.
- ▶ Enter the value of the new preset under **Preset**, e.g. **Z-40**
- ▶ Confirm your input
- ▶ Orient the coordinate system by specifying values under **PLANE SPATIAL SP***, e.g. **A+180** and **C+90**
- ▶ Confirm your input



- ▶ Select **3D mesh**
- ▶ The control opens the **3D mesh** mode and simplifies the 3D model using the default settings.
- ▶ Further simplify the 3D model using the **3D mesh** mode functions, if required.

Further information: "Functions for the simplified model", Page 294



- ▶ Select **Save**
- ▶ The control opens the **Define file name for 3D mesh** menu.
- ▶ Enter the desired name
- ▶ Select **Save**
- ▶ The control saves the STL file positioned for rear-face machining.



The resulting file can then be used for rear-face machining with the **BLK FORM FILE** function.

Further information: User's Manual for Programming and Testing

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

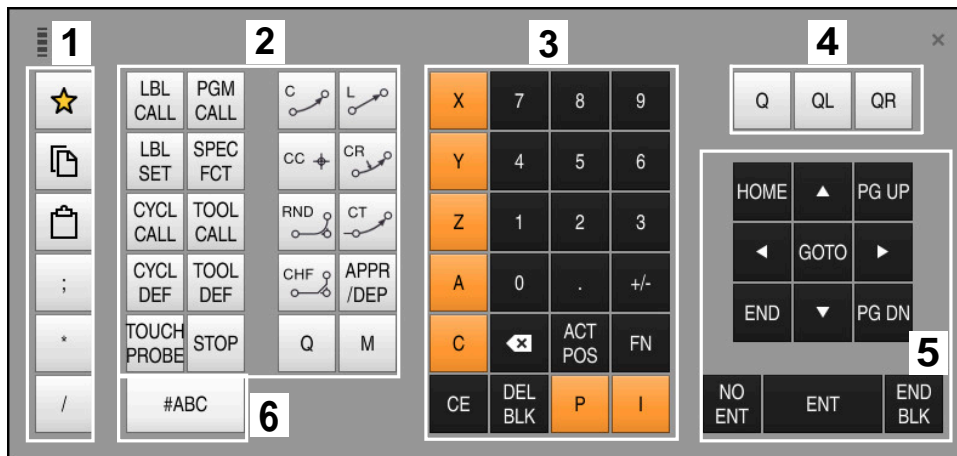
The control opens NC input mode by default after the start procedure.

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.

NC input areas



Virtual keyboard in NC input mode

NC input mode contains the following areas:

- 1 File functions
 - Define favorites
 - Copy
 - Paste
 - Add comment
 - Add structure item
 - Hide NC block
- 2 NC functions
- 3 Axis keys and numerical input
- 4 Q parameters
- 5 Navigation and dialog keys
- 6 Switch to text input



If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:

- **Q**
- **QL**
- **QR**

Areas of text input

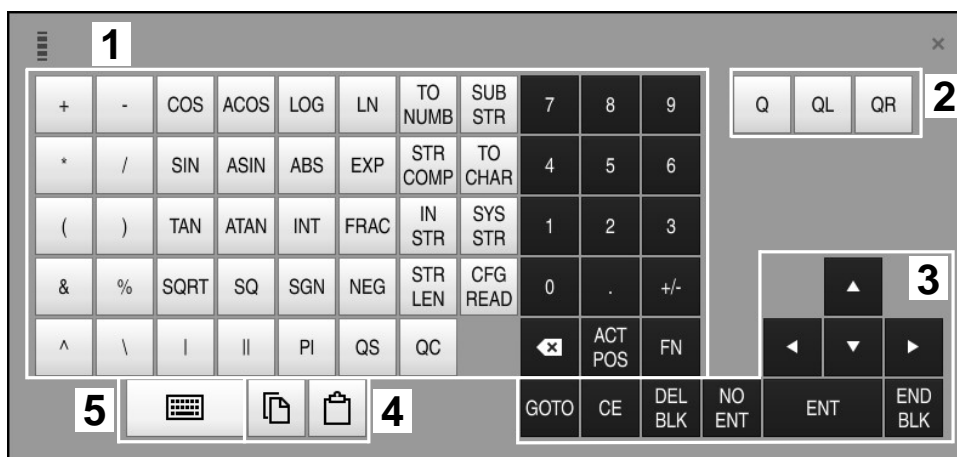


Virtual keyboard in text input mode

The text input contains the following areas:

- 1 Input
- 2 Navigation and dialog keys
- 3 Copying and pasting
- 4 Switch to formula input

Areas of formula input



Virtual keyboard in formula input mode

The formula input contains the following areas:

- 1 Input
- 2 Q parameters
- 3 Navigation and dialog keys
- 4 Copying and pasting
- 5 Switch to NC input

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:

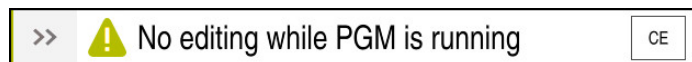
Symbol	Message type	Meaning
	Error Question type	The control displays a dialog with several options you can select from. You cannot delete this error message: you can only choose one of the possible responses. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.
	Reset error	The control must be restarted. This message cannot be cleared.
	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Information	You can continue without clearing the message. You can clear the information at any time.
	Note	You can continue without clearing the message. The control displays the note until you press the next valid key.
	No pending messages	

The message menu is collapsed by default.

The control displays messages upon various events, for example:

- Logical errors in the NC program
- Impossible contour elements
- Improper touch-probe inserts
- Hardware updates

Content



Collapsed message menu on the information bar

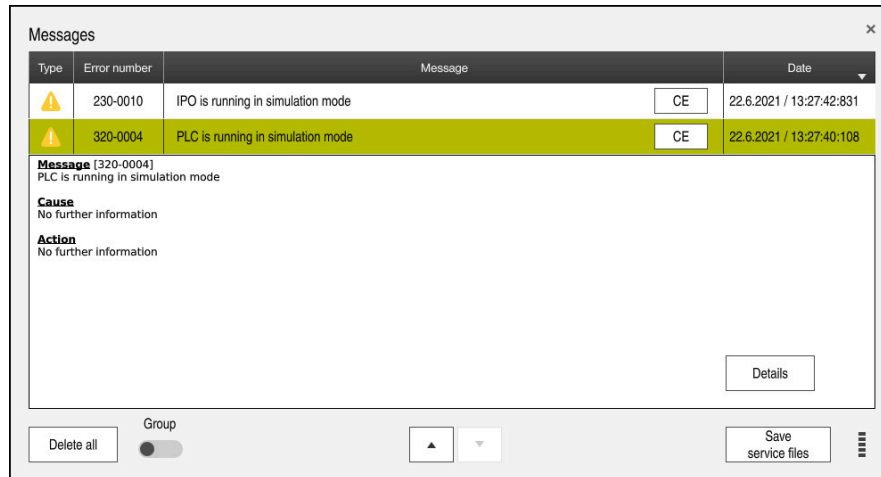
When the control displays a new message, the arrow to the left of the message blinks. Click or tap this arrow to confirm acknowledgment of the message; the control then minimizes the message.

The control displays the following information in the collapsed message menu:

- Message type
- Message
- Quantity of pending errors, warnings, and informational messages

Detailed messages

If you tap or click the symbol or within the message, the control expands the message menu.



Expanded message menu with pending messages

The control displays all pending messages in chronological order.

The message menu shows the following information:

- Message type
- Error number
- Message
- Date
- Additional information (cause, action)

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** 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 delete all messages of a group.

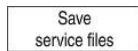
Service file

Use the **Save service files** button to create a service file.

A service file can be of use to the service technician during troubleshooting. 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.

14.2.1 Creating a service file

To create a service file:



- ▶ Expand the message menu
- ▶ Select **Save service files**
- > The control opens the **Save service file** window.
- ▶ Enter the file name
- ▶ Press **OK**
- > The control saves the service file in the **TNC:\service** directory.

15

**Touch Probe
Functions in the
Manual Operating
Mode**

15.1 Fundamentals

Application

The touch probe functions allow you to set presets on the workpiece, measure the workpiece, and determine and compensate for workpiece misalignment.

Related topics

- Automatic touch probe cycles
Further information: User's Manual for Measuring Cycles for Workpieces and Tools
- Preset table
Further information: "Preset table", Page 403
- Datum table
Further information: User's Manual for Programming and Testing
- Reference systems
Further information: "Reference systems", Page 178
- Preassigned variables
Further information: User's Manual for Programming and Testing

Requirements

- Calibrated workpiece touch probe
Further information: "Calibrating the workpiece touch probe", Page 319

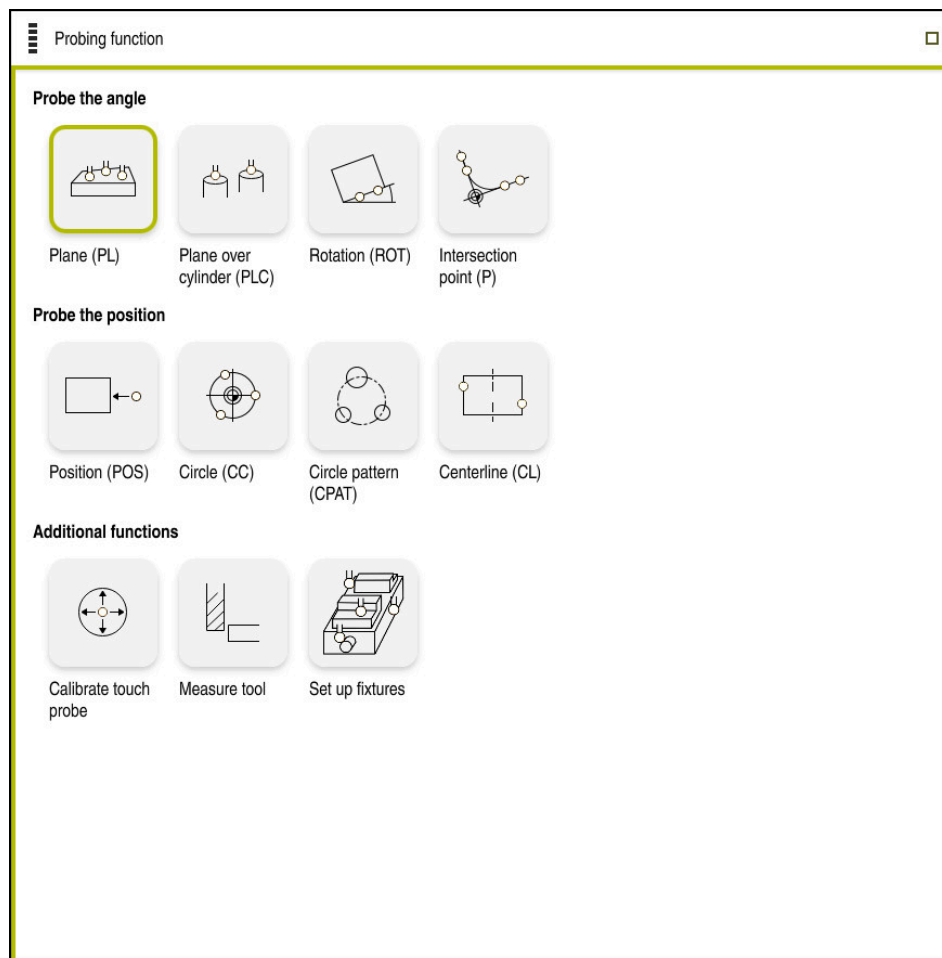
Description of function

The control provides the following functions for setting up the machine in the **Manual** operating mode in the **Setup** application:

- Define the workpiece preset
- Determine and compensate for workpiece misalignment
- Calibrate the workpiece touch probe
- Calibrate the tool touch probe
- Measure the tool

Within the functions, the control provides the following probing methods:

- Manual probing method
You position and start individual probing processes manually within a touch probe function.
 - Automatic probing method
You manually position the touch probe to the first probing point before the start of the probing routine and fill out a form with the individual parameters for the respective touch probe function. When you start the touch probe function, the control automatically positions and automatically performs probing.
- Further information:** "Setting a preset in a linear axis", Page 313
- Further information:** "Determining the circle center point of a stud using the automatic probing method ", Page 315







Probing function workspace

Overview

The touch probe functions are structured in the following groups:

Probe the angle

The **Probe the angle** group contains the following touch probe functions:

Button	Function
Plane (PL) 	Use the Plane (PL) function to determine the solid angle of a plane. You then save the values in the preset table or align the plane.
Plane over cylinder (PLC) 	Use the Plane over cylinder (PLC) function to probe one or two cylinders, each at two different heights. The control calculates the solid angle of a plane from the points probed. You then save the values in the preset table or align the plane.
Rotation (ROT) 	Use the Rotation (ROT) function to determine the skew of a workpiece using a straight line. Then save the determined skew as a basic transformation or offset in the preset table. Further information: "Determining and compensating the rotation of a workpiece", Page 316
Intersection point (P) 	Use the Intersection point (P) function to probe four probing objects. The probing objects can be either positions or circles. The control determines the intersection of the axes and the skew of the workpiece from the objects that have been probed. You can set the intersection point as a preset. You can transfer the determined skew to the preset table as a basic transformation or as an offset.



The control interprets a basic transformation as a basic rotation, and an offset as a table rotation.

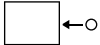



Further information: "Preset table", Page 403

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 326




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 313
Circle (CC) 	The Circle (CC) function is used to determine the coordinates of a circle center point, e.g. for a hole or for a stud. Further information: "Determining the circle center point of a stud using the automatic probing method ", Page 315
Circle pattern (CPAT) 	The Circle pattern (CPAT) function is used to determine the center point coordinates of a circle pattern.
Centerline (CL) 	The Centerline (CL) function is used to determine the center point of a ridge or slot.

Additional functions group









The **Additional functions** group contains the following touch probe functions:

Button	Function
Calibrate touch probe 	The Calibrate touch probe function is used to determine the length and radius of a workpiece touch probe. Further information: "Calibrating the workpiece touch probe", Page 319
Measure tool 	The Measure tool function allows you to measure tools by scratching. In this function, the control supports milling tools, drilling tools and turning tools.
Set up fixtures 	The Set up fixtures function is used to determine the position of a clamping device in the machine space using a workpiece touch probe. Further information: "Spannmittel in Kollisionsüberwachung einmessen", Page

Buttons



General buttons in the touch probe functions

The following buttons are available, depending on the selected touch probe function:

Button	Function
	Finish the active touch probe function
	Open the Change the preset window Select the workpiece preset in the Change the preset window and adapt arbitrary values to the preset table. Further information: "Preset table", Page 403
	Display help graphics for the selected touch probe function
	Select the probing direction
	Apply the actual position
	Approaching and probing points on a straight surface manually
	Approaching and probing points on a stud or in a hole manually
	Approaching and probing points on a stud or in a hole automatically After the last touching process and if the opening angle contains the value 360°, the control positions the workpiece touch probe back to the position it had prior to starting the probing function.

Calibration buttons




The control offers the following ways for calibrating a 3D touch probe:

Button	Function
	Calibrating the length of a 3D touch probe
	Calibrating the radius of a 3D touch probe
Apply calibration data	Transferring values from the calibration process into tool management

Further information: "Calibrating the workpiece touch probe", Page 319

You can calibrate a 3D touch probe by using a calibration standard, e.g. a calibrating ring.

The control provides the following options:

Button	Function
	Measure the radius and the center offset using a calibration ring
	Measure the radius and the center offset using a stud or a calibration pin
	Measure the radius and the center offset using a calibration sphere Optionally calibrating the tool touch probe in 3D (option 92) Further information: User's Manual for Programming and Testing Further information: "3D calibration (option 92)", Page 320

Buttons in the Working plane is inconsistent! window

If the positions of the rotary axes do not match the tilting situation in the **3-D rotation** window, the control opens the **Working plane is inconsistent!** window.

The control offers the following functions in the **Working plane is inconsistent!** window:

Button	Function
3-D ROT Apply status	The 3-D ROT Apply status function transfers the position of the rotary axes into the 3-D rotation window. Further information: "3-D rotation window (option 8)", Page 199
3-D ROT Ignore status	The 3-D ROT Ignore status function makes the control calculate the probing results, assuming that the rotary axes are in their zero position.
Align the rotary axes	The Align the rotary axes function aligns the rotary axes to the active tilting situation in the 3-D rotation window.

Buttons for measured values

After executing a touch probe function, you select the desired control reaction.

The control offers the following functions:

Button	Function
Compensate the active preset	The Compensate the active preset function transfers the measuring result into the active line of the preset table. Further information: "Preset table", Page 403
Write the datum	The Write the datum function transfers the measuring result into a desired line of the datum table. Further information: User's Manual for Programming and Testing
Align rotary table	The Align rotary table function aligns the rotary axes mechanically according to the measuring result.

Log file of touch probe cycles

After executing the respective touch-probe cycle, the control writes the measured values to the TCHPRMAN.html file.

You can check the readings of past measurements in the **TCHPRMAN.html** file.

If you have not defined a path in the machine parameter

FN16DefaultPath (no. 102202), the control will store the TCHPRMAN.html file directly under **TNC:**.

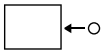
If you run several touch probes cycles in a row, the control stores the measured values below each other.

15.1.1 Setting a preset in a linear axis

To probe the preset in any axis:



- ▶ Select the **Manual** operating mode



- ▶ Call the workpiece touch probe as a tool

- ▶ Select the **Setup** application

- ▶ Select the **Position (POS)** touch probe function

- The control opens the **Position (POS)** touch probe function.



- ▶ Select **Change the preset**

- The control opens the **Change the preset** window.

- ▶ Select the desired row of the preset table

- The control highlights the selected line in green.



- ▶ Press **Apply**

- The control activates the selected line as the workpiece preset.

- ▶ Using the axis keys, position the workpiece touch probe at the desired probing position, e.g. above the workpiece in the workspace



- ▶ Select the probing direction, e.g. **Z-**



- ▶ Press the **NC start** key

- The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.

- The control shows the measurement results.

- ▶ In the **Nominal value** area, enter the new preset of the probed axis, e.g. **1**

Compensate the
active preset

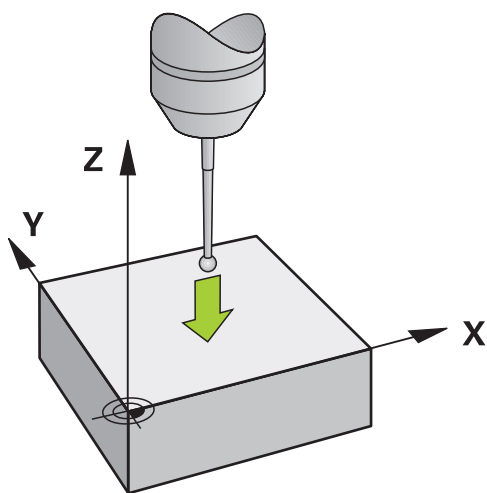
- ▶ Select **Compensate the active preset**
- The control enters the defined nominal value in the preset table.



When you have completed the probing process in the first axis, you can probe up to two additional axes using the **Position (POS)** probing function.



- ▶ Select **Exit probing**
- The control closes the **Position (POS)** probing function.



15.1.2 Determining the circle center point of a stud using the automatic probing method

To probe a circle center point:



- ▶ Select the **Manual** operating mode

- ▶ Call the workpiece touch probe as a tool

Further information: "Manual operation application", Page 130



- ▶ Select the **Setup** application



- ▶ Select **Circle (CC)**

- ▶ The control opens the **Circle (CC)** probing function.

- ▶ If necessary, select another preset for the probing process



- ▶ Select measuring method **A**



- ▶ Select **Type of contour**, e.g. stud

- ▶ Enter **Diameter**, e.g. 60 mm

- ▶ Enter **Starting angle**, e.g. -180°

- ▶ Enter **Angular length**, e.g. 360°

- ▶ Position the 3D touch probe at the desired probing position next to the workpiece and below the workpiece surface



- ▶ Select the probing direction, e.g. **X+**

- ▶ Turn the feed rate potentiometer to zero



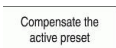
- ▶ Press the **NC start** key

- ▶ Slowly turn on the feed rate potentiometer

- ▶ The control executes the touch probe function based on the data entered.

- ▶ The control shows the measurement results.

- ▶ In the **Nominal value** area, enter the new preset of the scanned axes, e.g. **0**



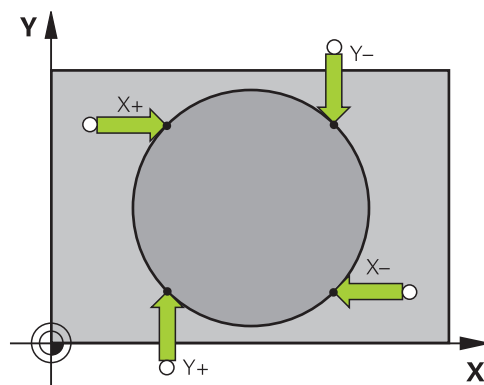
- ▶ Select **Compensate the active preset**

- ▶ The control sets the preset to the entered nominal value.



- ▶ Select **Exit probing**

- ▶ The control closes the **Circle (CC)** probing function.



15.1.3 Determining and compensating the rotation of a workpiece

To probe the rotation of a workpiece:



- ▶ Select the **Manual** operating mode



- ▶ Call the 3D touch probe as a tool



- ▶ Select the **Setup** application



- ▶ Select **Rotation (ROT)**

➤ The control opens the **Rotation (ROT)** probing function.

➤ If necessary, select another preset for the probing process



- ▶ Position the 3D touch probe at the desired probing position in the workspace

- ▶ Select the probing direction, e.g. **Y+**



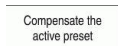
- ▶ Press the **NC start** key

➤ The control executes the first probing process and limits the subsequently selectable probing directions.

- ▶ Position the 3D touch probe at the second probing position in the workspace

- ▶ Press the **NC start** key

➤ The control executes the probing process and then shows the measurement results.



- ▶ Select **Compensate the active preset**

➤ The control transfers the determined basic rotation to the **SPC** column of the active line of the preset table.

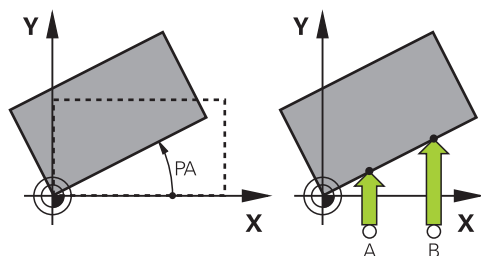


Depending on the tool axis, the measurement result can also be written to another column of the preset table, e.g. **SPA**.



- ▶ Select **Exit probing**

➤ The control closes the **Rotation (ROT)** probing function.



15.1.4 Using touch probe functions with mechanical probes or dial gages

If your machine does not have an electronic 3D touch probe, you can use all manual touch probe functions with manual probing methods with mechanical buttons or with scratching.

For this, the control provides the **Accept position** button.

To determine a basic rotation with a mechanical probe:



- ▶ Select the **Manual** operating mode



- ▶ Replace tool, e.g. analog 3D probe or feeler lever gage
- ▶ Select the **Setup** application
- ▶ Select the **Rotation (ROT)** probing function



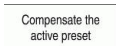
- ▶ Select the probing direction, e.g. **Y+**
- ▶ Move the mechanical probe to the first position to be captured by the control.



- ▶ Select **Accept position**
- The control saves the current position.
- ▶ Move the mechanical probe to the next position to be captured by the control.



- ▶ Select **Accept position**
- The control saves the current position.



- ▶ Select **Compensate the active preset**
- The control transfers the determined basic rotation to the active line of the preset table.



The determined angles have different effects depending on whether they are transferred as an offset or as a basic rotation to the corresponding table.

Further information: "Comparison of offset and 3D basic rotation", Page 326



- ▶ Select **Exit probing**
- The control closes the **Rotation (ROT)** probing function.

Notes

- When you use a non-contacting tool touch probe, use touch probe functions by the third-party manufacturer, e.g. with a laser touch probe. Refer to your machine manual.
- The accessibility of the pallet preset table in the touch probe functions depends on the machine manufacturer's configuration. Refer to your machine manual.
- The use of touch probe functions deactivates the global program settings (GPS, option 44) temporarily.

Further information: "Global Program Settings (GPS, option 44)", Page 238

- You can use the manual touch probe functions only with restrictions in turning mode (option 50).
- You must calibrate the touch probe separately in turning mode. The factory default setting of the worktable may vary between milling mode and turning mode, which is why you must calibrate the touch probe without any center offset in turning mode. You can create a tool index for storing the additionally calibrated tool data in the same tool.

Further information: "Indexed tool", Page 146

- When probing while the guard door is open and spindle orientation to probing direction is active, the number of spindle revolutions is limited. When the maximum permitted number of spindle revolutions is reached, the direction of spindle rotation changes and the control may no longer orient the spindle on the shortest path.
- If you try to set a preset in a locked axis, the control will issue either a warning or an error message, depending on what the machine manufacturer has defined.
- When writing into an empty line of the preset table, the control automatically fills the other columns with values. To define a preset completely, you must determine the values in all axes and write them into the preset table.
- If no tool touch probe is inserted, the actual position can be captured with **NC START**. The control displays a warning that no probing movement is carried out in that case.
- Recalibrate the workpiece touch probe in the cases below:
 - Initial configuration
 - Broken stylus
 - Stylus replacement
 - Change in the probe feed rate
 - Irregularities caused, for example, when the machine heats up
 - Change of active tool axis

Definition

Spindle tracking

If the **Track** parameter in the touch probe table is active, the control orients the workpiece probing system so that the same position is always used for probing. By deflecting in the same direction, you can reduce the measurement error to the repeatability of the workpiece probing system. This behavior is called spindle tracking.

15.2 Calibrating the workpiece touch probe

Application

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

During 3D calibration, you determine the angle-dependent deflection behavior of a workpiece touch probe in any probing direction (option 92).

Related topics

- Calibrate the workpiece touch probe automatically

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

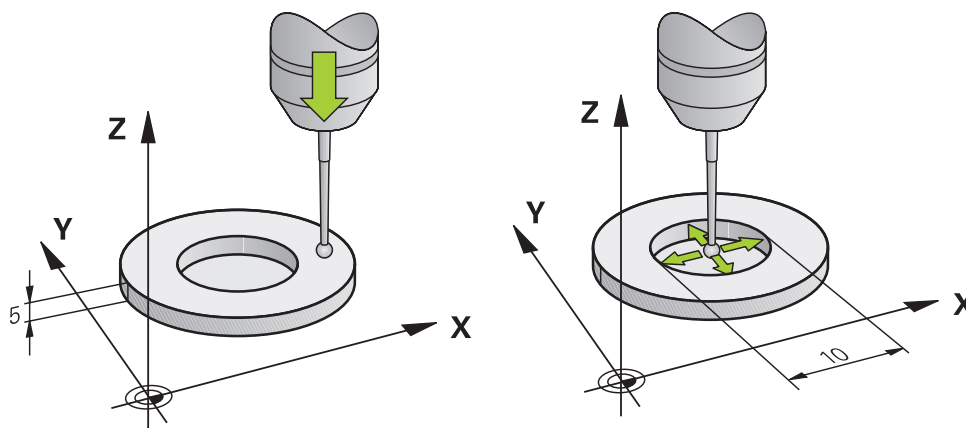
- Touch probe table

Further information: "Touch probe table tchprobe.tp", Page 392

- 3D radius compensation depending on the contact angle (option 92)

Further information: User's Manual for Programming and Testing

Description of function



During calibration, the control finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The effective length of the workpiece touch probe refers to the tool carrier preset.

Further information: "Tool carrier reference point", Page 141

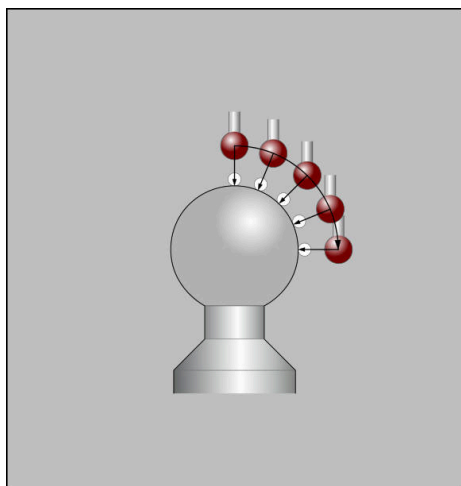
You can calibrate the workpiece touch probe with various tools. For example, the workpiece touch probe can be calibrated using an overmilled surface in length and a calibration ring in the radius. This creates a reference between the workpiece touch probe and the tools in the spindle. In this procedure, measured tools and the calibrated workpiece touch probe correspond using the tool presetting device.

3D calibration (option 92)

In addition to calibrating with a calibration sphere, the control also enables the touch probe to be calibrated dependent on the angle. For this purpose the control probes the calibration sphere in a quarter circle in the perpendicular. The 3D calibration data specifies the deflection behaviour of the touch probe in any probing direction.

The control saves the deviations in a compensation value table ***.3DTC** in the folder **TNC:\system\3D-ToolComp**.

The control creates a specific table for each calibrated touch probe. In the tool table the **DR2TABLE** column is automatically referenced to this.



3D calibration

Reversal measurement

When calibrating the ball-tip radius, the control executes an automatic probing routine. In the first run the control finds the midpoint of the calibration ring or stud (approximate measurement) and positions the touch probe in the center. Then, in the actual calibration process (fine measurement), the radius of the ball tip is ascertained. If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

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.

15.2.1 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 194

- ▶ Set preset in the tool axis, e.g. **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. **Z=0**
- ▶ Switch off spindle, e.g. with **M5**
- ▶ Replace the tool touch probe
- ▶ Select the **Setup** application
- ▶ Select **Calibrate touch probe**



- ▶ Select the **Length calibration** measurement method
- The control displays the current calibration data.
- ▶ Enter the reference surface position, e.g. **0**
- ▶ Position the workpiece touch probe close to the surface of the overmilled area



Check that the area to be probed is flat and free of chips before you start the touch probe function.



- ▶ Press the **NC start** key
- The control performs the probing process and then automatically retracts the workpiece touch probe to the starting point.
- ▶ Check results
- ▶ Select **Apply calibration data**
- The control transfers the calibrated length of the 3D touch probe to the tool table.
- ▶ Select **Exit probing**
- The control closes the **Calibrate touch probe** function.

Apply calibration data



15.2.2 Calibrating the radius of the workpiece touch probe

To calibrate a workpiece touch probe using a setting ring in the radius:

- ▶ Clamp the setting ring on the machine table, e.g. with clamps



- ▶ Select the **Manual** operating mode
- ▶ Position the 3D touch probe in the hole of the setting ring



Make sure that the stylus tip is completely recessed into the calibration ring. This causes the control to probe with the largest point of the stylus tip.



- ▶ Select the **Setup** application
- ▶ Select **Calibrate touch probe**



- ▶ Select **Radius** measurement method



- ▶ Select **Setting ring** calibration standard



- ▶ Enter the diameter of the ring gauge
- ▶ Enter the start angle
- ▶ Enter the number of touch points
- ▶ Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.
- ▶ Check results
- ▶ Select **Apply calibration data**
- > The control stores the calibrated radius of the 3D touch probe in the tool table.
- ▶ Select **Exit probing**
- > The control closes the **Calibrate touch probe** function.

Apply calibration data



15.2.3 3D calibration of workpiece touch probe (option 92)

To calibrate a workpiece touch probe using a calibration sphere in the radius:

- ▶ Clamp the setting ring on the machine table, e.g. with clamps



- ▶ Select the **Manual** operating mode
- ▶ Position the workpiece touch probe centrally above the sphere
- ▶ Select the **Setup** application
- ▶ Select **Calibrate touch probe**



- ▶ Select **Radius** measurement method



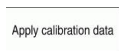
- ▶ Select the **Calibration sphere** calibration standard

- ▶ Enter the diameter of the sphere
- ▶ Enter the start angle
- ▶ Enter the number of touch points



- ▶ Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine. The control calculates the effective stylus tip radius. If probing from opposite orientations is possible, the control calculates the center offset.

- ▶ Check results



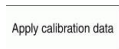
- ▶ Select **Apply calibration data**
- > The control stores the calibrated radius of the 3D touch probe in the tool table.
- > The control shows the **3D calibration** measurement method.
- ▶ Select the **3D calibration** measurement method



- ▶ Enter the number of touch points



- ▶ Press the **NC Start** key
- > The 3D touch probe probes all required touch points in an automatic probing routine.



- ▶ Select **Apply calibration data**
- > The control saves the deviations in a compensation value table under **TNC:\system\3D-ToolComp**.



- ▶ Select **Exit probing**
- > The control closes the **Calibrate touch probe** function.

Instructions for calibration

- In order to be able to determine ball-tip center misalignment, the control needs to be specially prepared by the machine manufacturer.
- If you press the **OK** button after the calibration process, the control accepts the calibration values for the active touch probe. The updated tool data then becomes immediately effective, and it is not necessary to repeat the tool call.
- HEIDENHAIN only guarantees the proper operation of the touch probe cycles in conjunction with HEIDENHAIN touch probes.
- If you want to calibrate using the outside of an object, you need to pre-position the touch probe above the center of the calibration sphere or calibration pin. Ensure that the probing points can be approached without collisions.
- The control saves the effective length and effective radius of the touch probe in the tool table. The control saves the touch probe center offset in the touch probe table. The control uses the **TP_NO** parameter to link the data from the touch probe table with the data from the tool table.

Further information: "Touch probe table tchprobe.tp", Page 392

15.3 Suppressing touch probe monitoring

Application

If you move a workpiece touch probe too close to the workpiece, you can accidentally deflect the workpiece touch probe. You cannot retract a deflected workpiece touch probe in the monitored state. You can retract a deflected workpiece touch probe by suppressing touch probe monitoring.

Description of function

If the control does not receive a stable signal from the probe, the button displays **Suppress touch probe monitoring**.

As long as touch-probe monitoring is switched off, the control displays the error message **The touch probe monitor is deactivated for 30 seconds**. This error message remains active only for 30 seconds.

15.3.1 Deactivating touch probe monitoring

To deactivate touch probe monitoring:



- ▶ Select the **Manual** operating mode
- ▶ Select **Suppress touch probe monitoring**
- The control disables touch-probe monitoring for 30 seconds.
- ▶ If required, move the touch probe so that the control receives a stable signal from it.

Notes

NOTICE

Danger of collision!

While touch-probe monitoring is deactivated, the control will not perform collision checking. Thus, you must ensure that the touch probe can be positioned safely. There is a risk of collision if you choose the wrong direction of traverse!

- Carefully move the axes in the **Manual** operating mode

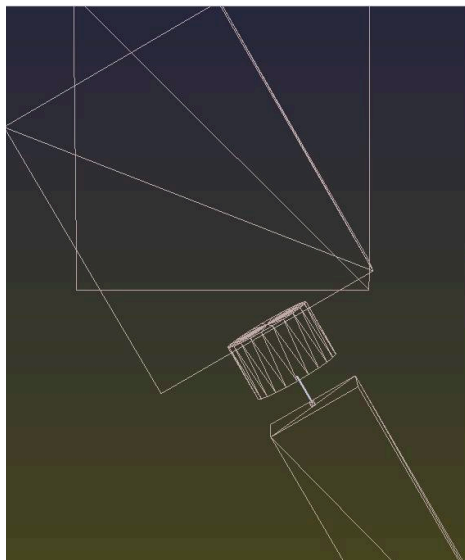
If the touch probe sends a stable signal within the 30 seconds, then touch-probe monitoring reactivates itself automatically and the error message is cleared.

15.4 Comparison of offset and 3D basic rotation

The following example shows how the two functions differ.

Offset

Initial state



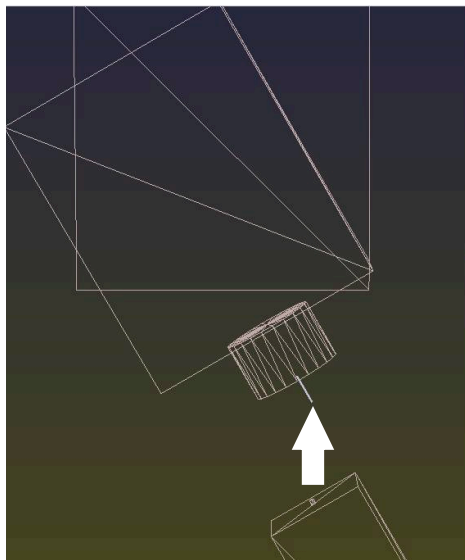
Position display:

- Actual position
- **B** = 0
- **C** = 0

Preset table:

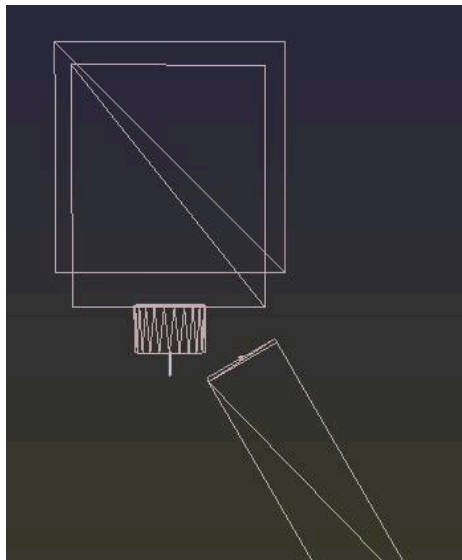
- **SPB** = 0
- **B_OFFS** = -30
- **C_OFFS** = +0

Movement in +Z without tilting



3D basic rotation

Initial state



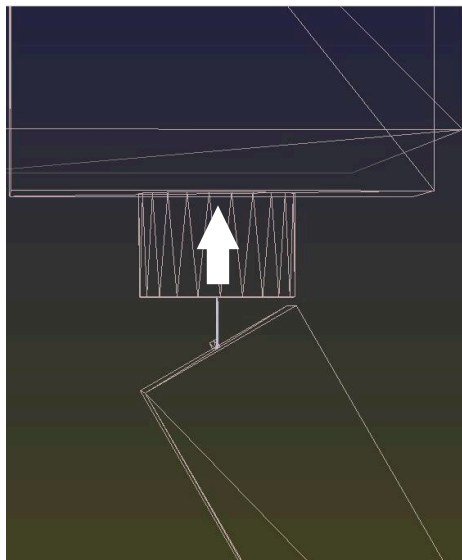
Position display:

- Actual position
- **B** = 0
- **C** = 0

Preset table:

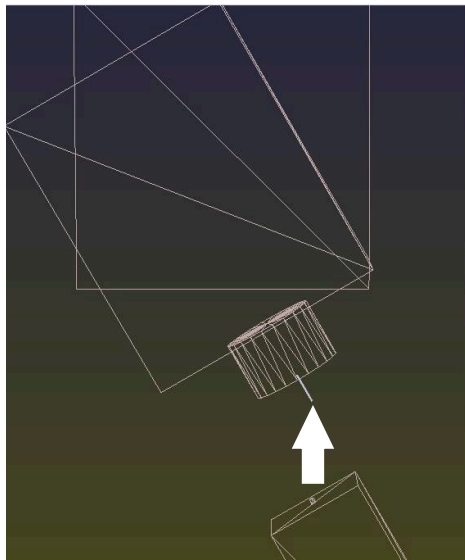
- **SPB** = -30
- **B_OFFS** = +0
- **C_OFFS** = +0

Movement in +Z without tilting



Offset

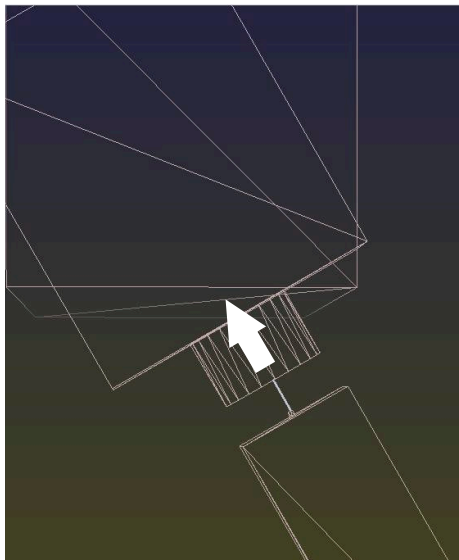
Movement in +Z with tilting

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.

16

Application MDI

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 memorizes modally effective program information.

Related topics

- Creating NC programs

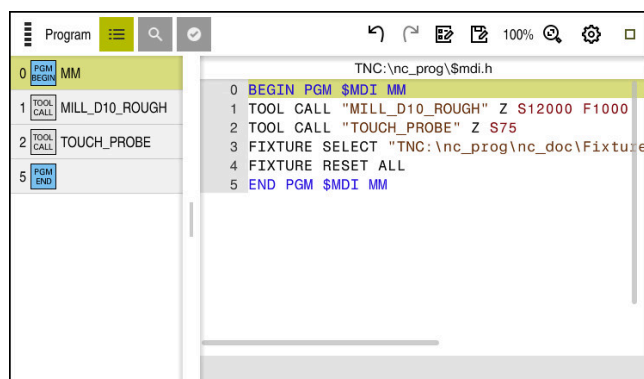
Further information: User's Manual for Programming and Testing

- Running NC programs

Further information: "Program Run", Page 333

Description of function

If you program using the millimeter unit of measurement, the control will use the NC program **\$mdi.h** by default. If you program using the inch unit of measurement, the control will use the NC program **\$mdi_inch.h**.



Program workspace in the **MDI** application

The **MDI** application provides the following workspaces:

- **GS** (option 44)

Further information: "Global Program Settings (GPS, option 44)", Page 238

- **Help**

- **Positions**

Further information: "Positions workspace", Page 93

- **Program**

Further information: User's Manual for Programming and Testing

- **Simulation**

Further information: User's Manual for Programming and Testing

- **Status**

Further information: "Status workspace", Page 101

- **Keyboard**

Further information: "Virtual keyboard of the control bar", Page 298

Buttons

In the **MDI** application, the function bar provides the following buttons:

Button	Meaning
Klartext programming	If this switch is active, then you are using dialog-guided programming. If this switch is not active, then you are programming in the text editor. Further information: User's Manual for Programming and Testing
Insert NC function	The control opens the Insert NC function window. Further information: User's Manual for Programming and Testing
Q info	The control opens the Q parameter list window, where you can see and edit the current values and descriptions of the variables. Further information: User's Manual for Programming and Testing
GOTO block number	Mark an NC block to be run without considering any previous NC blocks Further information: User's Manual for Programming and Testing
/ Skip Off/On	Mark NC blocks with a / character. NC blocks marked with a / character will be ignored during program run as soon as the Skip / switch is active. Further information: User's Manual for Programming and Testing
Skip /	If this switch is active, the control ignores NC blocks marked with a / character. Further information: User's Manual for Programming and Testing
; Comment Off/On	Insert or remove a ; character in front of an NC block. If an NC block begins with a ; character, then the block is a comment. Further information: User's Manual for Programming and Testing
FMAX	You activate a feed-rate limitation and define the value. Further information: "Feed rate limit F MAX", Page 337
Editing	The control opens the context menu. Further information: User's Manual for Programming and Testing
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run. Further information: "Interrupting, stopping or canceling program run", Page 338
Reset program	If you select Internal stop , the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Modally effective program information

In the **MDI** application, you always run the NC blocks in **Single Block** mode. After the control has run an NC block, the program run is considered to be interrupted.

Further information: "Interrupting, stopping or canceling program run", Page 338

The block numbers of all NC blocks that you have successively run are shown in green.

The control saves the following data in this state:

- The last tool that was called
- Current coordinate transformations (e.g., datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

Notes

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
 - Cursor movement to another NC block
 - The jump command **GOTO** to another NC block
 - Editing an NC block
 - Modifying the values of variables by using the window **Q parameter list**
 - Switching the operating modes
 - ▶ Restore the contextual reference by repeating the required NC blocks
-
- In the **MDI** application, you can create and execute NC programs step by step. Then you can use **Save as** to save the current contents with a different file name.
 - The following functions are not available in the **MDI** application:
 - Calling an NC program with **PGM CALL**, **SEL PGM**, or **CALL SELECTED PGM**
 - Test Run in the **Simulation** workspace
 - **Manual traverse** and **Approach position** while program run is interrupted
 - **Block scan** function

17

Program Run

17.1 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 one block at a time or in full sequence.

You also execute pallet tables in this operating mode.

Related topics

- Executing individual NC blocks in the **MDI** application
Further information: "Application MDI", Page 329
- Creating NC programs
Further information: User's Manual for Programming and Testing
- Pallet tables
Further information: User's Manual for Programming and Testing

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- Copy the NC program and all called files to the **TNC:** drive

Description of function



The following information also applies to pallet tables and job lists.

When you select a new NC program or when an NC program has been completely executed, the cursor is at the beginning of the program.

If you want to start machining at a different NC block, you first need to select the desired NC block by using the **Block scan** function.

Further information: "Block scan for mid-program startup", Page 342

By default, the control runs NC programs in Full Sequence mode after the **NC Start** key has been pressed. In this mode, the control runs an NC program continuously up to its end, or up to a manual or programmed interruption.

In the **Single Block** mode, you execute each NC block separately by pressing the **NC Start** key.

The control shows the status of the machining process with the **Control-in-operation** icon in the status overview.

Further information: "Status overview on the control bar", Page 99

The **Program Run** operating mode provides the following workspaces:



- **GS** (option 44)
Further information: "Global Program Settings (GPS, option 44)", Page 238
- **Positions**
Further information: "Positions workspace", Page 93
- **Program**
Further information: User's Manual for Programming and Testing
- **Simulation**
Further information: User's Manual for Programming and Testing
- **Status**
Further information: "Status workspace", Page 101
- **Process Monitoring**
Further information: User's Manual for Programming and Testing

When you open a pallet table, the control will display the **Job list** workspace. You cannot edit this workspace.

Further information: User's Manual for Programming and Testing

Icons and buttons

The **Program Run** operating mode contains the following icons and buttons:

Icon or button	Meaning
	Open file With Open file you can open a file, for example an NC program. If you open a file, the control closes the file that was already open.
	Execution cursor The execution cursor shows which NC block is currently being run or is marked for running.
Single Block	If this switch is active, then you run each NC block separately with the NC Start key. If Single Block mode is selected, then the operating mode's icon in the control bar changes.
Q info	The control opens the Q parameter list window, where you can see and edit the current values and descriptions of the variables. Further information: User's Manual for Programming and Testing
Compensation tables	The control opens a selection menu with the following tables: <ul style="list-style-type: none"> ■ D ■ T-CS ■ WPL-CS Further information: "Compensation during program run", Page 351
FMAX	You activate a feed-rate limitation and define the value. Further information: "Feed rate limit F MAX", Page 337
GOTO Cursor	The control marks the currently selected table row for execution. Active only if a pallet table is open (option 22) Further information: User's Manual for Programming and Testing
Skip /	If this switch is active, the control ignores NC blocks marked with a / character. Further information: User's Manual for Programming and Testing
Pause at M1	If this switch is active, the control pauses program run at the next NC block with M1 . Further information: User's Manual for Programming and Testing
GOTO block number	Mark an NC block to be run without considering any previous NC blocks Further information: User's Manual for Programming and Testing
Manual traverse	During a program run interruption you can move the axes manually. If Manual traverse is active, then the operating mode's icon in the control bar changes. Further information: "Manual traverse during an interruption", Page 341
Edit	If this switch is active, then you can edit the pallet table. Active only if a pallet table is open Further information: User's Manual for Programming and Testing
3D ROT	During a program run interruption you can move the axes manually in the tilted working plane (option 8). Further information: "Manual traverse during an interruption", Page 341

Icon or button	Meaning
Approach position	Return to contour after manual traverse of the machine axes during an interruption Further information: "Returning to the contour", Page 349
Block scan	The Block scan function allows you to start program run at any desired NC block. The control takes the preceding parts of the NC program up to this NC block into account mathematically; for example, whether the spindle was switched on with M3 . Further information: "Block scan for mid-program startup", Page 342
Open in the editor	The control opens the NC program in the Editor operating mode. Active only if an NC program is open Further information: User's Manual for Programming and Testing
Internal stop	If an NC program is interrupted due to an error or a stop, the control activates this button. Use this button to abort program run.
Reset program	If you select Internal stop , the control activates this button. The control places the cursor back to the beginning of the program and resets any modally effective program information as well as the program run-time.

Feed rate limit F MAX

The **F MAX** button allows you to reduce the feed rate for all operating modes. The reduction applies to all rapid traverse and feed rate movements. The value you have entered remains active across power cycles.

The **FMAX** button is available in the **MDI** application and in **Editor** operating mode.

When you select the **FMAX** button in the function bar, the control will open the **Feedrate + FMAX** window.

If a feed rate limit is active, the control highlights the **FMAX** button in color and displays the defined value.

You deactivate the feed rate limit by entering a value of 0 in the **Feedrate + FMAX** window.

Interrupting, stopping or canceling program run

There are several ways to stop a program run:

- Interrupt the program run (e.g., with the miscellaneous function **M0**)
- Stop the program run (e.g., with the **NC stop** key)
- Cancel the program run (e.g., with the **NC Stop** key in combination with the **INTERNAL STOP** button)
- Terminate the program run (e.g., with the miscellaneous functions **M2** or **M30**)

During major errors, the control automatically aborts the program run (e.g., during a cycle call with stationary spindle).

Further information: "Message menu on the information bar", Page 301

If you run your NC program in **Single Block** mode or in the **MDI** application, the control will switch to the interrupted state after the execution of each NC block.

The control shows the current program run status with the **Control-in-operation** icon.

Further information: "Status overview on the control bar", Page 99

You can execute, e.g., the following functions in an interrupted or canceled state:

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

NOTICE

Danger of collision!

Certain manual interactions may lead to the control losing the modally effective program information (i.e., the contextual reference). Loss of this contextual reference may result in unexpected and undesirable movements. There is a risk of collision during the subsequent machining operation!

- ▶ Do not perform the following interactions:
 - Cursor movement to another NC block
 - The jump command **GOTO** to another NC block
 - Editing an NC block
 - Modifying the values of variables by using the window **Q parameter list**
 - Switching the operating modes
- ▶ Restore the contextual reference by repeating the required NC blocks

Programmed interruptions

You can set interruptions directly in the NC program. The control interrupts the program run in the NC block containing one of the following inputs:

- Programmed stop **STOP** (with and without miscellaneous function)
- Programmed stop **M0**
- Conditional stop **M1**



Refer to your machine manual.

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

Resuming program run

After stopping the program with the **NC Stop** key or a programmed interruption, you can resume program run by pressing the **NC Start** key.

After canceling the program run with an **Internal stop**, you must start the program run at the beginning of the NC program or use the **Block scan** function.

After an interruption of the program run within a subprogram or program section repeat, you need to use the **Block scan** function for mid-program startup.

Further information: "Block scan for mid-program startup", Page 342

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 349



The saved data remains active until it is reset (e.g., by selecting a program).

Notes

NOTICE

Danger of collision!

Program cancellation, manual intervention, or forgotten resetting of NC functions or transformations can lead to the control performing unexpected or undesirable movements. This can lead to workpiece damage or collision.

- ▶ Rescind all programmed NC functions and transformations within the NC program
- ▶ Run a simulation before executing an NC program
- ▶ Check both the general as well as the additional status display for NC functions and transformations, such as an active basic rotation, before executing an NC program
- ▶ Carefully verify the NC program in the **Single Block** mode

NOTICE

Danger of collision!

With NC software 81762x-16, the TNC7 does not support ISO programming. There is a risk of collision during execution due to the missing support.

- ▶ Use Klartext NC programs exclusively.

- In the **Program Run** operating mode, the control marks active files with the status **M**, such as a selected NC program or tables. If you open such a file in another operating mode, the controls shows the status on the tab of the application bar.
- When moving an axis, the control checks whether the defined rotational speed has been reached. The control does not check the rotational speed in positioning blocks with **FMAX** as feed rate.
- You can adjust the feed rate and the spindle speed during program run with the potentiometers.
- If you modify the workpiece reference point during a program run interruption, you must re-select the NC block to resume.

Further information: "Block scan for mid-program startup", Page 342

- HEIDENHAIN recommends switching the spindle on with **M3** or **M4** after every tool call. That way you avoid problems during program run, such as when restarting after an interruption.
- The settings in the **GS** workspace have an effect on program run, such as handwheel superimpositioning (option 44).

Further information: "Global Program Settings (GPS, option 44)", Page 238

Definitions

Abbreviation	Definition
GS (global program settings)	Global program settings
ACC (active chatter control)	Active Chatter Control

17.1.2 Manual traverse during an interruption

Application

During a program run interruption you can move the machine axes manually.

Use the **Tilt the working plane (3D ROT)** window to assign the reference system for traversing the axes (option 8).

Related topics




- Manual traverse of machine axes
Further information: "Moving the machine axes", Page 132
- Manual tilting of the working plane (option 8)
Further information: User's Manual for Programming and Testing

Description of function

When you select **Manual traverse**, you can move the axes with the axis keys of the control.

Further information: "Using axis keys to move the axes ", Page 132

In the **Tilt the working plane (3D ROT)** window, you can select the following functions:

Icon	Function	Meaning
	M-CS machine	Traversing in the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 179
	W-CS workpiece	Traversing in the workpiece coordinate system W-CS Further information: "Workpiece coordinate system W-CS", Page 184
	WPL-CS working plane	Traversing in the working plane coordinate system WPL-CS Further information: "Working plane coordinate system WPL-CS", Page 186
	T-CS tool	Traversing in the tool coordinate system T-CS Further information: "Working plane coordinate system WPL-CS", Page 186

When you select one of the functions, the control will display the associated icon in the **Positions** workspace. The control additionally shows the active coordinate system on the **3D ROT** button.

If **Manual traverse** is active, then the operating mode's icon in the control bar changes.

Notes

NOTICE

Danger of collision!

During a program interruption, you can move the axes manually (e.g., in order to retract from a hole when the working plane is tilted). There is a risk of collision if the **3-D ROT** setting is incorrect!

- ▶ It is better to use the **T-CS** function
- ▶ Use a low feed rate

- On some machines, you may have to press the **NC Start** key while **Manual traverse** is active in order to enable the axis keys.
Refer to your machine manual.

17.1.3 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: User's Manual for Programming and Testing
- Pallet tables and job lists
Further information: User's Manual for Programming and Testing

Requirement

- The function must be enabled by your machine manufacturer.
The **Block scan** function must be enabled and configured by your machine manufacturer.

Description of function

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

- **Internal stop** button
- Emergency stop
- Power failure

If, while restarting, the control finds a saved point of interruption, then it outputs a message. You can then execute a block scan directly to the point of interruption. The control displays the message when you switch to **Program Run** operating mode for the first time.

You have the following options for a block scan:

- Block scan in the main program, with repetitions if necessary
Further information: "Performing a single-level block scan", Page 345
- Multi-level block scan in subprograms and touch probe cycles
Further information: "Performing a multi-level block scan", Page 346
- Block scan in a point table
Further information: "Block scan in point tables", Page 347
- Block scan in pallet programs
Further information: "Block scan in pallet tables", Page 348

At the start of the block scan, the control resets the data, as with a selection of a new NC program. During a block scan you can activate or deactivate **Single Block** mode.

Block scan window

Block scan window with the interruption point saved and the **Point table** area opened

The **Block scan** window provides the following data:

Row	Meaning
Pallet number	Row number in the pallet table
Program	Path of the active NC program
Block number	Number of the NC block at which program run should start Use the search icon to select the NC block in the NC program.
Repetitions	Number of the repetition for mid-program startup if the desired NC block is located within a program-section repeat.
Last pallet number	Pallet number that is active at the time of interruption Select the interruption point by using the Select last button.
Last program	Path of the NC program that is active at the time of interruption Select the interruption point by using the Select last button.
Last block	Number of the NC block that was active at the time of interruption Select the interruption point by using the Select last button.
Point file	Path of the point table In the Point table area
Point number	Row in the point table In the Point table area

Performing a single-level block scan

To start in an NC program by using a single-level block scan:



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



- ▶ Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.
- ▶ Enter the **Program** as needed
- ▶ Enter the **Block number**
- ▶ Enter the **Repetitions** as needed
- ▶ If required, use **Select last** to start at a saved interruption point



- ▶ Press the **NC Start** key
- The control starts the block scan and calculates up to the entered NC block.
- If you have changed the machine status, the control will display 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 will display the **Axis sequence for return to contour:** window.



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



You can also position the axes individually in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 350



- ▶ 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 345



- ▶ Activate the **Single Block** switch as needed



- ▶ Press the **NC Start** key to execute individual NC blocks as needed



- ▶ Select **Continue block scan**



- ▶ Define the NC block for mid-program startup
- ▶ Press the **NC Start** key
- ▶ The control starts the block scan and calculates up to the entered NC block.

- ▶ If you have changed the machine status, the control will display 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 will display the **Axis sequence for return to contour:** window.



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



You can also position the axes individually in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 350



- ▶ Select **Continue block scan** again as needed
- ▶ Repeat the steps



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

Block scan in point tables

To start in a point table:



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



- ▶ Select **Block scan**
- The control opens the **Block scan** window. The fields **Program**, **Block number** and **Repetitions** contain the current values.

- ▶ Select **Point table**

- The control opens the **Point table** area.

- ▶ **Point file:** Enter the path of the point table

- ▶ **Point number:** Select the row number of the point table for mid-program startup



- ▶ Press the **NC Start** key

- The control starts the block scan and calculates up to the entered NC block.

- If you have changed the machine status, the control will display 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 will display the **Axis sequence for return to contour:** window.



- ▶ Press the **NC Start** key

- Using the displayed positioning logic, the control moves to the required positions.



You can also position the axes individually in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 350



If you would like to use the block scan function to start in a point pattern, then use the same procedure. Define the desired point for mid-program startup in the **Point number** field. The first point in the point pattern has the number 0.

Further information: User's Manual for Machining Cycles

Block scan in pallet tables

To start in a pallet table:



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



- ▶ Select **Block scan**
- The control opens the **Block scan** window.
- ▶ **Pallet number:** Enter the row number of the pallet table
- ▶ Enter the **Program** as needed
- ▶ Enter the **Block number**
- ▶ Enter the **Repetitions** as needed
- ▶ If required, use **Select last** to start at a saved interruption point



- ▶ Press the **NC Start** key
- The control starts the block scan and calculates up to the entered NC block.
- If you have changed the machine status, the control will display 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 will display the **Axis sequence for return to contour:** window.



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



You can also position the axes individually in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 350



If the program run of a pallet table has been canceled, the control will suggest the most recently selected NC block of the most recently executed NC program as a point of interruption.

Notes

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

The **Block scan** function skips over the programmed touch probe cycles. As a result, the result parameters contain no values or, possibly, incorrect values. If the subsequent machining operation uses these result parameters, then there is a risk of collision!

- ▶ Use the **Block scan** function at multiple levels

- The control only displays the dialogs required by the process in the pop-up window.
- A **Block scan** always takes place in a workpiece-oriented manner, even if you selected a tool-oriented machining method. After the block scan, the control continues working again in accordance with the selected machining method.
Further information: User's Manual for Programming and Testing
- Even after an internal stop, the control shows the number of repetitions on the **LBL** tab of the **Status** workspace.
Further information: "LBL tab", Page 105
- 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.

17.1.4 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

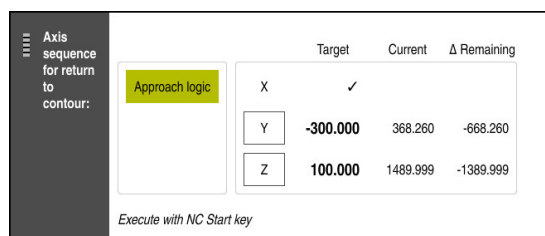
- Manual traverse during program run interruptions
Further information: "Manual traverse during an interruption", Page 341
- **Block scan** function
Further information: "Block scan for mid-program startup", Page 342

Description of function

If you have selected the **Manual traverse** button, this button will change to **Approach position**.

When you select **Approach position**, the control will open the **Axis sequence for return to contour:** window.

Axis sequence for return to contour: window



Axis sequence for return to contour: window

In the **Axis sequence for return to contour:** window, the control displays all of the axes that are not yet located at the correct position for program execution.

The control suggests a positioning logic for the sequence of the traversing movements. If the tool is located in the tool axis below the position to be approached, then the control offers the tool axis as the first traverse direction. You can also traverse the axes in a self-selected sequence.

Further information: "Positioning the axes in a self-selected sequence", Page 350

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 351

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 checkmark under **Target**.
- ▶ Position the remaining axes
- When all axes have reached their positions, the control closes the window.

Positioning manual axes

To position manual axes:

Approach
position

- ▶ 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.

Definition

Manual axis

Manual axes are non-driven axes that need to be positioned by the machine operator.

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: User's Manual for Programming and Testing
- Editing compensation tables in the NC program
Further information: User's Manual for Programming and Testing
- Contents and creation of compensation tables
Further information: User's Manual for Programming and Testing
- Contents and preparing a datum table
Further information: User's Manual for Programming and Testing
- Activating a datum table in the NC program
Further information: User's Manual for Programming and Testing

Description of function

The control opens the selected tables in **Tables** operating mode.

The changed values do not take effect until the compensation or the datum has been activated again.

17.2.1 Opening tables from within Program Run operating mode

To open the compensation tables from within **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 **Tables** operating mode.

Notes

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- ▶ Make sure to confirm any changes made to the table immediately, e.g., by pressing the **ENT** key
- ▶ Activate the datum or compensation value in the NC program again
- ▶ Carefully test the NC program after changing the table values

- When you open a table in **Program Run** operating mode, the control will display the status **M** on the tab of the table. This status indicates that this table is active for program run.
- The clipboard allows you to transfer axis positions from the position display to the datum table.

Further information: "Status overview on the control bar", Page 99

17.3 Retract application

Application

The **Retract** application allows you to disengage the tool from the workpiece after an interruption in power (e.g., retraction of a tap engaged in the workpiece).

You can also retract a tool when the working plane is tilted or retract an inclined tool.

Requirement

- This application must be enabled by your machine manufacturer.
The machine parameter **retractionMode** (no. 124101) allows the machine manufacturer to define whether the control will display the **Retract** switch during start-up.

Description of function

The **Retract** application provides the following workspaces:

- **Retract**
Further information: "Retract workspace", Page 354
- **Positions**
Further information: "Positions workspace", Page 93
- **Status**
Further information: "Status workspace", Page 101

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** switch if the following conditions apply during start-up:

- Power interrupted
- No control voltage for the relay
- **Move to ref. point** application

If you have activated a feed rate limit before the power failure occurred, this feed rate limit will still be active. When you select the **Retract** button, the control will display a pop-up window: This window allows you to deactivate the feed rate limit.

Further information: "Feed rate limit F MAX", Page 337

Retract workspace

The **Retract** workspace provides the following contents:

Row	Meaning
Traversing mode	Traverse mode for retraction: <ul style="list-style-type: none"> ■ Machine axes: Traverse in the machine coordinate system M-CS ■ Tilted system: Traverse in the working plane coordinate system WPL-CS (option 8) ■ Tool axis: Traverse in the tool coordinate system T-CS (option 8) ■ Thread: Traverse in the tool coordinate system T-CS with compensating movements of the spindle Further information: "Reference systems", Page 178
Kinematics	Name of the active machine kinematics
A, B, C	Current position of the rotary axes Effective for Tilted system traverse mode
Thread pitch	Thread pitch from the PITCH column of tool management Effective for Thread traverse mode
Direct. of rotation	Direction of rotation of the thread-turning tool: <ul style="list-style-type: none"> ■ Right-hand thread ■ Left-hand thread Effective for Thread traverse mode
Coordinate system for handwheel superimposition	Coordinate system in which handwheel superimpositioning takes effect Effective for Tool axis traverse mode

The control selects the mode of traverse and the associated parameters automatically. If the traverse mode or the parameters have not been correctly preselected, you are able to reset them manually.

Note

NOTICE

Caution: Danger to the tool and workpiece!

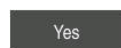
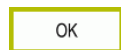
A power failure during the machining operation can cause uncontrolled "coasting" or braking of the axes. In addition, if the tool was in effect prior to the power failure, then the axes cannot be referenced after the control has been restarted. For non-referenced axes, the control takes over the last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

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

Example

The power failed while a thread-cutting cycle was being performed in the tilted working plane. You have to retract the tap:

- ▶ Switch on the power supply for control and machine
- > The control starts the operating system. This process may take several minutes.
- > The control displays the **Power interrupted** dialog in the **Start/Login** workspace



- ▶ Activate the **Retract** switch
- ▶ Press **OK**
- > The control compiles the PLC program.
- ▶ Switch the machine control voltage on
- > The control checks the functioning of the emergency stop circuit
- > The control opens the **Retract** application and displays the **Assume position values?** window.
- ▶ Compare the displayed position values with the actual position values
- ▶ Select **OK**
- > The control closes the **Assume position values?** window
- ▶ Select the **Thread** traverse mode as needed
- ▶ Enter the thread pitch as needed
- ▶ Enter the direction of rotation as needed
- ▶ Select **Retract**
- ▶ Retract the tool with the axis keys or the handwheel
- ▶ Select **End retraction**
- > The control opens the **End retraction?** window and prompts you to answer a confirmation request.
- ▶ If the tool was correctly retracted, select **Yes**
- > The control closes the **End retraction?** window and the **Retract** application.

18

Tables

18.1 Tables operating mode

Application

In the **Tables** operating mode you can open various tables and edit them as necessary.

Description of function

If you select **Add**, the control will display the **Quick selection** and **Open File** workspaces.

In the **Quick selection** workspace, you can open some tables directly.

Further information: User's Manual for Programming and Testing

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: User's Manual for Programming and Testing

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application.

You can open the **Table** and **Form** workspaces in every application.

Further information: "Table workspace", Page 359

Further information: "Form workspace for tables", Page 363

You can select various functions by using the context menu (e.g., **Copy**).

Further information: User's Manual for Programming and Testing

Buttons

The **Tables** operating mode provides the following buttons in the function bar:

Button	Meaning
Activate the preset	Activate the selected row of the preset table as a preset. Further information: "Preset table", Page 403
Undo	Undo the last change
Redo	Redo the last undone change
GOTO record	The control opens the GOTO jump instruction window. The control jumps to the row number you have defined.
Edit	If the switch is active, you can edit the table.
Insert tool	The control opens the Insert tool window that allows you to add a new tool to tool management. Further information: "Tool management ", Page 165 When you select the Append check box, the control will insert the tool below the last row of the table.
Insert line	The control inserts a row at the end of the table.
Reset row	The control resets all data contained in the row.
Delete tool	The control deletes the tool selected in tool management Further information: "Tool management ", Page 165
Delete row	The control deletes the currently selected row.
T INSPECT	The control inspects a tool.
T OUT	The control unloads a tool.
T IN	The controls loads a tool.

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.



If the **Edit** switch is active, you can edit the contents in both the **Table** workspace and the **Form** workspace.

18.2 Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. The control displays a column with filters and a search function on the left side of some tables.

Description of function

The screenshot shows a software interface titled 'Table' with a search bar and a filter 'Filter: all tools > all tool types'. On the left is a sidebar with icons for various tool types: all tools, tools in magazines, all tool types, milling tools, drilling tools, tapping tools, threadmilling tools, turning tools, touchprobes, dressing tools, grinding tools, and undefined tools. The main area displays a table with the following data:

T	P	NAME	TYP
0		NULLWERKZEUG	MILL_R
1	1.1	MILL_D2_ROUGH	MILL_R
2	1.2	MILL_D4_ROUGH	MILL_R
3	1.3	MILL_D6_ROUGH	MILL_R
4	1.4	MILL_D8_ROUGH	MILL_R
5	1.5	MILL_D10_ROUGH	MILL_R
6	0.0	MILL_D12_ROUGH	MILL_R
7	1.7	MILL_D14_ROUGH	MILL_R
8	1.8	MILL_D16_ROUGH	MILL_R
9	1.9	MILL_D18_ROUGH	MILL_R
10	1.10	MILL_D20_ROUGH	MILL_R
11	1.11	MILL_D22_ROUGH	MILL_R
12	1.12	MILL_D24_ROUGH	MILL_R
13	1.13	MILL_D26_ROUGH	MILL_R
14	1.14	MILL_D28_ROUGH	MILL_R

At the bottom of the table, there are labels: 'Tool name?' on the left, 'Min:' in the center, and 'Max:' on the right.

Table workspace

In **Tables** operating mode, the **Table** workspace is open in every application by default.






The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column.

If the table allows it, you can also edit the table contents in this workspace.

Icons and shortcuts

The **Table** workspace provides the following icons or shortcuts:

Icon or shortcut	Function
	Open the filters Further information: "Filters in the Table workspace", Page 361
	Open the search function Further information: "Search column in the Table workspace", Page 362
100%	Font size of the table <div> When you select the percent value, the control will display icons for increasing and decreasing the font size.</div>
	Set the font size of the table to 100%
	Open the settings in the Tables window Further information: "Settings in the Table workspace", Page 362
CTRL+A	Mark all rows
CTRL+BLANK	Mark the active row or end the marking function
SHIFT+↑	Additionally mark the row above
SHIFT+↓	Additionally mark the row below

Filters in the Table workspace

You can filter the tool tables and the **Pocket table**.

Filtering in Tool management

The following filter options are available for tool management:

- **All tools**
- **Magazine tools**

Depending on the filter you have selected (all tools or magazine tools), you can additionally filter by tool type in this area:

- **All types**
- **Milling cutters**
- **Drills**
- **Taps**
- **Thread cutters**
- **Lathe tools**
- **Touch probes**
- **Dressing tools**
- **Grinding tools**
- **Undefined tools**

Filters in the Pocket table

The following filter options are available for the pocket table:

- **All magazines**
- **Main magazine**
- **Spindle**

Depending on the filter you have selected (all magazines, main magazine, or spindle), you can additionally filter by pockets in this area:

- **All pockets**
- **Empty pockets**
- **Assigned pockets**

Search column in the Table workspace

You can search the **Tool management** and **Pocket table** 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, such as **Contains** or **Equal to (=)**
Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

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**

General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control will move the cursor synchronously in the **Form** workspace.

Column sequence area

Tables window

In the **Column sequence** area, you define the view for each table.

The **Use standard format** switch allows you to show all columns in the default sequence.

The **Number of frozen columns** switch allows you to define how many columns the control will freeze to the left border of the screen. These columns will remain visible when you navigate further to the right within the table.

The control lists all table columns vertically below each other. Use the switch to select for each column whether it will be shown or hidden.

The control displays a line below the selected number of frozen columns. The control will freeze the columns above this line.

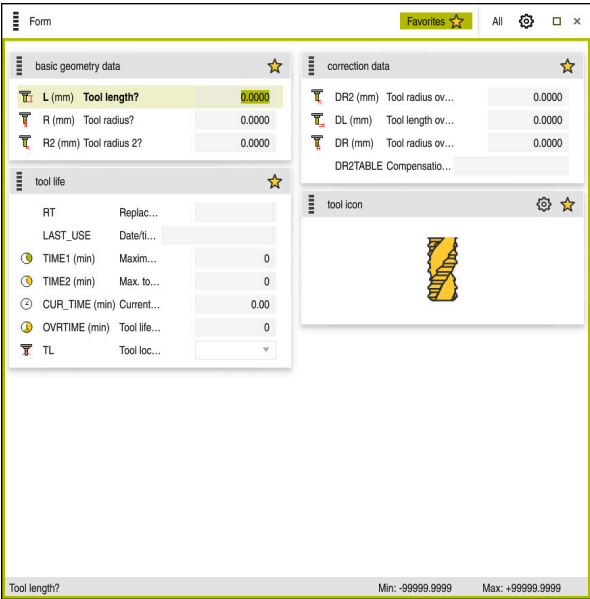
When you select a column, the control will display up and down arrows. Use these arrows to change the sequence of the columns.

18.3 Form workspace for tables

Application

In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.

Description of function



Form workspace in the **Favorites** view

The control displays the following information for each column:



- Icon of the column as needed
- Name of the column
- Unit of measure as needed
- Column description
- Current value

If an input is invalid, the control will display an icon ahead of the input field. When you tap this icon, the control will show the cause of the error (e.g., **Too many characters**).

The control displays the contents of specific tables in groups within the **Form** workspace. In the **All** view, the control shows all groups. Use the **Favorites** function to select individual groups, in order to configure a customized view. Use the handle to arrange the groups.

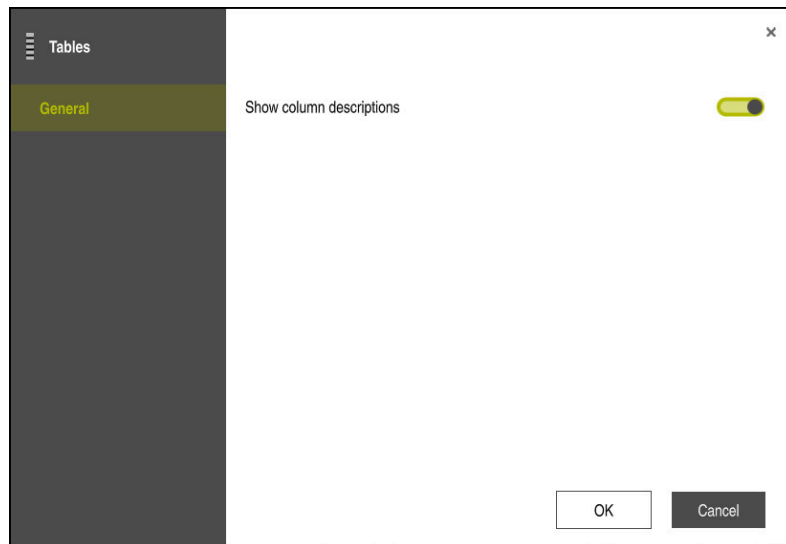
Icons

The **Table** workspace provides the following icons:

Icon or shortcut	Function
	Open the settings in the Tables window Further information: "Settings in the Form workspace", Page 365
	Favorite

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the column descriptions. The selected setting is modally effective.



18

Tool Tables

18.4.1 Overview

This chapter describes the tool tables of the control.

- Tool table **tool.t**

Further information: "Tool table tool.t", Page 368

- Turning tool table **toolturn.trn** (option 50)

Further information: "Turning tool table toolturn.trn (option 50)", Page 377

- Grinding tool table **toolgrind.grd** (option 156)

Further information: "Grinding tool table toolgrind.grd (option 156)", Page 382

- Dressing tool table **tooldress.drs** (option 156)

Further information: "Dressing tool table tooldress.drs (option 156)", Page 389

- Touch probe table **tchprobe.tp**

Further information: "Touch probe table tchprobe.tp", Page 392

You can edit the tools, except for the touch probes, in tool management only.

Further information: "Tool management ", Page 165

18.4.2 Tool table tool.t

Application

The tool table **tool.t** contains the data specific to drilling and milling tools. The tool table also contains all tool data that are independent of the technology, such as the tool life **CUR_TIME**.

Related topics

- Editing tool data in tool management

Further information: "Tool management ", Page 165

- Tool data required for milling or drilling tools







Further information: "Tool data for milling and drilling tools", Page 154




Description of function






The file name of the tool table is **tool.t** and this table must be stored in the folder **TNC:\table**.



The **tool.t** tool table provides the following parameters:




Parameter	Meaning
T	<p>Tool number?</p> <p>Row number in the tool table</p> <p>The tool number allows identifying each tool unambiguously, e.g. for calling a tool.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>You can define an index after the period.</p> <p>Further information: "Indexed tool", Page 146</p> <p>This parameter applies to all tools, regardless of technology.</p> <p>Input: 0.0...32767.9</p>





Parameter	Meaning
NAME	Tool name? The tool name allows identifying a tool, e.g. for calling a tool. Further information: User's Manual for Programming and Testing You can define an index after the period. Further information: "Indexed tool", Page 146 This parameter applies to all tools, regardless of technology. Input: Text width 32
L 	Tool length? Length of tool, with respect to the tool carrier preset Further information: "Tool carrier reference point", Page 141 Input: -99999.9999...+99999.9999
R 	Tool radius? Tool radius, with respect to the tool carrier reference point Further information: "Tool carrier reference point", Page 141 Input: -99999.9999...+99999.9999
R2 	Tool radius 2? Corner radius for the exact definition of the tool for three-dimensional radius compensation, graphic representation and collision monitoring of e.g. ball-nose cutter or toroid cutters. Further information: User's Manual for Programming and Testing Input: -99999.9999...+99999.9999
DL 	Tool length oversize? Delta value of tool length as a compensation value in connection with touch probe cycles. The control enters corrections automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter L Input: -999.9999...+999.9999
DR 	Tool radius oversize? Delta value of tool radius as a compensation value in connection with touch probe cycles. The control enters corrections automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to parameter R Input: -999.9999...+999.9999
DR2 	Tool radius oversize 2? Delta value of tool radius 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to parameter R2 Input: -999.9999...+999.9999


Parameter	Meaning
TL 	Tool locked? Tool is enabled or locked for machining: <ul style="list-style-type: none"> ■ No value entered: Enabled ■ L: Locked The control locks the tool after exceeding maximum tool age TIME1 , maximum tool age 2 TIME2 or after exceeding one of the parameters for automatic tool measurement. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: No value, L
RT	Replacement tool? Number of the replacement tool If the control calls a tool in a TOOL CALL and the tool is not available or locked, the control inserts the replacement tool. If M101 is active and the current tool age CUR_TIME exceeds the TIME2 value, the control locks the tool and inserts the replacement tool at a suitable location. Further information: User's Manual for Programming and Testing If the replacement tool is not available or locked, the control inserts the replacement tool of the replacement tool. You can define an index after the period. Further information: "Indexed tool", Page 146 If you define the value 0, the control will not use a replacement tool. This parameter applies to all tools, regardless of technology. Selection by means of a selection window Input: 0.0...32767.9
TIME1 	Maximum tool age? Maximum tool age in minutes If the current tool age CUR_TIME exceeds the TIME1 value, the control locks the tool and displays an error message when the tool is called the next time. The behavior depends on the machine. Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: 0...99999
TIME2 	Max. tool age for TOOL CALL? Maximum tool age 2 in minutes The control inserts a replacement tool in the cases below: <ul style="list-style-type: none"> ■ When the current tool age CUR_TIME exceeds the TIME2 value, the control locks the tool. The control no longer inserts the tool when the tool is called. If a replacement tool RT is defined and available in the magazine, the control inserts the replacement tool. If no replacement tool is available, the control will display an error message. ■ If M101 is active and the current tool age CUR_TIME exceeds the TIME2 value, the control locks the tool and inserts the replacement tool RT at a suitable location. Further information: User's Manual for Programming and Testing This parameter applies to all tools, regardless of technology. Input: 0...99999



Parameter	Meaning
CUR_TIME 	Current tool age? <p>The current tool age equals the time during which the tool is cutting a workpiece. The control counts this time automatically and enters the current tool age in minutes.</p> <p>This parameter applies to all tools, regardless of technology.</p> <p>Input: 0...99999.99</p>
TYPE	Tool type? <p>Depending on the selected tool type, the control displays the suitable tool parameters in the Form workspace of tool management.</p> <p>Further information: "Tool types", Page 150</p> <p>Further information: "Tool management ", Page 165</p> <p>This parameter applies to all tools, regardless of technology.</p> <p>Selection by means of a selection window</p> <p>Input: MILL, MILL_R, MILL_F, BALL, TORUS, DRILL, TAP, CENT, TURN, TCHP, REAM, CSINK, TSINK BOR, BCKBOR, GF, GSF, EP, WSP, BGF, ZBGF, GRIND, and DRESS</p>
DOC	Tool description <p>This parameter applies to all tools, regardless of technology.</p> <p>Input: Text width 32</p>
PLC	PLC status? <p>Tool information for the PLC</p> <p>Refer to your machine manual.</p> <p>This parameter applies to all tools, regardless of technology.</p> <p>Entry: %00000000...%11111111</p>
LCUTS 	Tooth length in the tool axis? <p>Length of cutting edge for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring.</p> <p>Input: -99999.9999...+99999.9999</p>
LU 	Usable length of the tool? <p>Usable length of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of e.g. necks of end mills.</p> <p>Input: 0.0000...999.9999</p>
RN 	Neck radius of the tool? <p>Neck radius for the exact definition of the tool for graphic simulation and collision monitoring of e.g. neck of end mills or side milling cutters.</p> <p>Only if the useful length LU is longer than the LCUTS length of the cutting edge can the tool contain a neck radius RN.</p> <p>Input: 0.0000...999.9999</p>
ANGLE 	Maximum plunge angle? <p>Maximum plunge angle of the tool for reciprocating plunge-cutting in the cycles.</p> <p>Input: -360.00...+360.00</p>

Parameter	Meaning
CUT 	<p>Number of teeth?</p> <p>Number of teeth of the tool for automatic tool measurement or cutting data calculation.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>Further information: User's Manual for Programming and Testing</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: 0...99</p>
TMAT 	<p>Tool material?</p> <p>Tool material from the tool material table TMAT.tab for cutting data calculation.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>Selection by means of a selection window</p> <p>Input: Text width 32</p>
CUTDATA 	<p>Cutting data table?</p> <p>Select cutting data table with the *.cut or *.cutd file extensions for cutting data calculation.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>Selection by means of a selection window</p> <p>Entry: Text width 20</p>
LTOL 	<p>Wear tolerance: length?</p> <p>Permitted tool length deviation in wear detection for automatic tool measurement.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>If the entered value is exceeded, the control locks the tool in column TL.</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: 0...9.9999</p>
RTOL 	<p>Wear tolerance: radius?</p> <p>Permitted tool radius deviation in wear detection for automatic tool measurement.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>If the entered value is exceeded, the control locks the tool in column TL.</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: 0...9.9999</p>

Parameter	Meaning
R2TOL	<p>Wear tolerance: Radius 2?</p> <p>Permitted tool radius 2 deviation in wear detection for automatic tool measurement.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>If the entered value is exceeded, the control locks the tool in column TL.</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: 0...9.9999</p>
DIRECT 	<p>Cutting direction?</p> <p>Cutting direction of the tool for automatic tool measurement with a rotating tool:</p> <ul style="list-style-type: none"> ■ -: M3 ■ +: M4 <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: -, +</p>
R-OFFS 	<p>Tool offset: radius?</p> <p>Position of tool upon length measurement, offset between the center of the tool touch probe and the tool center for automatic tool measurement.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: -99999.9999...+99999.9999</p>
L-OFFS 	<p>Tool offset: length?</p> <p>Position of tool upon radius measurement, distance between the top edge of the tool touch probe and the tool tip for automatic tool measurement.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>Is added to the machine parameter offsetToolAxis (no. 122707)</p> <p>This parameter applies to the following tools, regardless of technology:</p> <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) <p>Input: -99999.9999...+99999.9999</p>

Parameter	Meaning
LBREAK 	Breakage tolerance: length? Permitted tool length deviation in breakage detection for automatic tool measurement. Further information: User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column TL . This parameter applies to the following tools, regardless of technology: <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) Input: 0...3.2767
RBREAK 	Breakage tolerance: radius? Permitted tool radius deviation in breakage detection for automatic tool measurement. Further information: User's Manual for Measuring Cycles for Workpieces and Tools If the entered value is exceeded, the control locks the tool in column TL . This parameter applies to the following tools, regardless of technology: <ul style="list-style-type: none"> ■ Milling and drilling tools ■ Turning tools (option 50) Input: 0.0000...0.9999
NMAX 	Maximum speed [rpm] Limitation of spindle speed for the programmed value including control by the potentiometer. Input: 0...999999
LIFTOFF	Lift-off allowed? Automatic tool lift-off with active M148 or allow FUNCTION LIFTOFF : <ul style="list-style-type: none"> ■ Y: Activate LIFTOFF ■ N: Deactivate LIFTOFF Further information: User's Manual for Programming and Testing Selection by means of a selection window Input: Y, N
TP_NO	Number of the touch probe Number of touch probe in the touch probe table tchprobe.tp Further information: "Touch probe table tchprobe.tp", Page 392 Input: 0...99
T-ANGLE 	Point angle Point angle of the tool for exact definition of the tool for graphical simulation, automatic calculation within cycles and collision monitoring of e.g. drills. Further information: User's Manual for Machining Cycles Input: -180...+180
LAST_USE	Date/time of last tool call Time of last tool presence in the spindle This parameter applies to all tools, regardless of technology. Input: 00:00:00 01.01.1971...23:59:59 31.12.2030

Parameter	Meaning
PTYP	Tool type for pocket table? Tool type for evaluation in the pocket table Further information: "Pocket table tool_p.tch", Page 396 Refer to your machine manual. This parameter applies to all tools, regardless of technology. Input: 0...99
AFC	Feedback-control strategy Control setting for adaptive feed control (AFC) (option 45) from the AFC.tab table. Further information: "Adaptive Feed Control (AFC, option 45)", Page 230 Selection by means of a selection window Entry: Text width 10
ACC	ACC active? Activate or deactivate active chatter control (ACC) (option 145): <ul style="list-style-type: none"> ■ Y: Activate ■ N: Deactivate Further information: "Active Chatter Control (ACC, option 145)", Page 237 Selection by means of a selection window Input: Y, N
PITCH 	Tool thread pitch? Thread pitch of the tool for automatic calculations within cycles. A positive sign means a right-hand thread. Further information: User's Manual for Machining Cycles Input: -9.9999...+9.9999
AFC-LOAD	Reference power for AFC [%] Tool-dependent reference power for AFC (option 45). The input in percent refers to the rated spindle power. The control immediately uses the value given for feedback control, meaning a teach-in cut is dropped. Calculate the value beforehand with a teach-in step. Further information: "AFC teach-in cut", Page 235 Input: 1.0...100.0
AFC-OVLD1	AFC overload warning level [%] Cut-related tool wear monitoring for AFC (option 45). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. Further information: "Monitoring tool wear and tool load", Page 236 Input: 0.0...100.0
AFC-OVL2	AFC overload switch-off level [%] Cut-related tool load monitoring for AFC (option 45). The input in percent refers to the reference power. The value 0 deactivates the monitoring function. An empty field has no effect. Further information: "Monitoring tool wear and tool load", Page 236 Input: 0.0...100.0

Parameter	Meaning
KINEMATIC	<p>Tool-carrier kinematics</p> <p>Assigning a tool carrier for exact definition of the tool for graphical simulation and collision monitoring.</p> <p>Further information: "Tool carrier management", Page 169</p> <p>Selection by means of a selection window</p> <p>This parameter applies to all tools, regardless of technology.</p> <p>Entry: Text width 20</p>
DR2TABLE	<p>Compensation val. table for DR2</p> <p>Assigning a compensation value table *.3drc for 3D tool radius compensation depending on the contact angle (option 92). This allows the control to compensate for, e.g., inaccuracies in the shape of a ball-nose cutter or the deflection behavior of a touch probe.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>Selection by means of a selection window</p> <p>Entry: Text width 16</p>
OVRTIME 	<p>Tool life expired</p> <p>Time in minutes during which the tool may be used beyond the tool life defined in column TIME1.</p> <p>The machine manufacturer defines the function of this parameter. The machine manufacturer defines how the control uses the parameter when searching for tool names. Refer to your machine manual.</p> <p>This parameter applies to all tools, regardless of technology.</p> <p>Input: 0...99</p>
RCUTS 	<p>Width of the indexable insert</p> <p>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.</p> <p>Input: 0...99999.9999</p>

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 395

- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- Assign unique tool names!

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

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



Refer to your machine manual.

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

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

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

- In the machine parameter **offsetToolAxis** (no. 122707), the machine manufacturer defines the distance between the upper edge of the tool touch probe and the tool tip.

The parameter **L-OFFS** is added to this defined distance.

- In the machine parameter **zeroCutToolMeasure** (no. 122724) the machine manufacturer defines whether the control takes the parameter **R-OFFS** into account for automatic tool measurement.

18.4.3 Turning tool table toolturn.trn (option 50)

Application

The turning tool table **toolturn.trn** contains the data specific to turning tools.

Related topics

- Editing tool data in tool management
Further information: "Tool management ", Page 165
- Tool data required for turning tools
Further information: "Tool data for turning tools (option 50)", Page 156
- Milling-turning operations on the control
Further information: User's Manual for Programming and Testing
- General tool data, regardless of the technology
Further information: "Tool table tool.t", Page 368





Requirements






- Software option 50: Combined milling-turning
- Turning tool is defined in **TYPE** column of tool management
Further information: "Tool types", Page 150










Description of function


The file name of the turning tool table is **toolturn.trn** and this table must be stored in the folder **TNC:\table**.

The **toolturn.trn** turning tool table provides the following parameters:

Parameter	Meaning
T	<p>Row number in the turning tool table</p> <p>The tool number allows identifying each tool unambiguously, e.g. for calling a tool.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>You can define an index after the period.</p> <p>Further information: "Indexed tool", Page 146</p> <p>The row number must match the tool number in the tool.t tool table.</p> <p>Input: 0.0...32767.9</p>
NAME	<p>Tool name?</p> <p>The tool name allows identifying a tool, e.g. for calling a tool.</p> <p>Further information: User's Manual for Programming and Testing</p> <p>You can define an index after the period.</p> <p>Further information: "Indexed tool", Page 146</p> <p>Input: Text width 32</p>
ZL 	<p>Tool length 1?</p> <p>Length of the tool in the Z direction, with respect to the tool carrier preset</p> <p>Further information: "Tool carrier reference point", Page 141</p> <p>Input: -99999.9999...+99999.9999</p>
XL 	<p>Tool length 2?</p> <p>Length of the tool in the X direction, with respect to the tool carrier preset</p> <p>Further information: "Tool carrier reference point", Page 141</p> <p>Input: -99999.9999...+99999.9999</p>
YL 	<p>Tool length 3?</p> <p>Length of the tool in the Y direction, with respect to the tool carrier preset</p> <p>Further information: "Tool carrier reference point", Page 141</p> <p>Input: -99999.9999...+99999.9999</p>
DZL 	<p>Oversize in tool length 1?</p> <p>Delta value of tool length 1 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece.</p> <p>Further information: User's Manual for Measuring Cycles for Workpieces and Tools</p> <p>Is added to the parameter ZL</p> <p>Input: -99999.9999...+99999.9999</p>

Parameter	Meaning
DXL 	Oversize in tool length 2? Delta value of tool length 2 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter XL Input: -99999.9999...+99999.9999
DYL 	Tool length oversize 3? Delta value of tool length 3 as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter YL Input: -99999.9999...+99999.9999
RS 	Cutting edge radius? The control takes into account the cutter radius for tool tip radius compensation. Further information: User's Manual for Programming and Testing In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. Further information: User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters TO , T-ANGLE , and P-ANGLE . Input: 0...99999.9999
DRS 	Cutter radius oversize? Delta value of cutter radius as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to the parameter RS Input: -999.9999...+999.9999
TO 	Tool orientation? From the tool orientation, the control determines the position of the tool tip and, depending on the selected tool type, additional information such as the tool angle direction. This information is necessary, for example, for calculating the cutter radius compensation, milling cutter radius compensation, plunge angle, etc.. In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. Further information: User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters RS , T-ANGLE , and P-ANGLE . Input: 1...19

Parameter	Meaning
SPB-INSERT 	Angular offset? Angular offset for recessing tools Input: -90.0...+90.0
ORI 	Angle of spindle orientation? Angle of tool spindle for aligning the turning tool Input: -360.000...+360.000
T-ANGLE 	Tool angle In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. Further information: User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters RS , TO , and P-ANGLE . Input: 0...179.999
P-ANGLE 	Point angle In turning cycles, the control takes into account the cutter geometry to prevent damage to the defined contour. If the contour cannot be machined completely, the control will display a warning. Further information: User's Manual for Machining Cycles For the cutter geometry, the control also considers the parameters RS , TO , and T-ANGLE . Input: 0...179.999
CUTLENGTH  	Cutting length of recessing tool Length of the cutting edge of a turning or recessing tool The control monitors the length of the cutting edge in the turning cycles. If the cutting depth programmed in the turning cycle is greater than the length of the cutting edge defined in the tool table, then the control will display a warning and will automatically reduce the cutting depth. Further information: User's Manual for Machining Cycles Input: 0...99999.9999
CUTWIDTH  	Width of recessing tool The control uses the width of a recessing tool for calculations within cycles. Further information: User's Manual for Machining Cycles Input: 0...99999.9999
DCW 	Oversize f. recessing tool width Delta value of recessing tool width as a compensation value in connection with touch probe cycles. The control enters compensation values automatically after measuring the workpiece. Further information: User's Manual for Measuring Cycles for Workpieces and Tools Is added to parameter CUTWIDTH Input: -99999.9999...+99999.9999

Parameter	Meaning
TYPE 	Type of turning tool Depending on the selected turning tool type, the control displays the suitable tool parameters in the Form workspace of tool management. Further information: "Types within the turning tools", Page 151 Further information: "Tool management ", Page 165 Selection by means of a selection window Input: ROUGH, FINISH, THREAD, RECESS, BUTTON, and RECTURN
WPL-DX-DIAM	Compensation value for the workpiece diameter Compensation value for the workpiece diameter with respect to the working plane coordinate system (WPL CS). Further information: "Working plane coordinate system WPL-CS", Page 186 Input: -99999.9999...+99999.9999
WPL-DZL	Compensation value for the workpiece length Compensation value for the workpiece length with respect to the working plane coordinate system (WPL CS). Further information: "Working plane coordinate system WPL-CS", Page 186 Input: -99999.9999...+99999.9999

Notes

- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- Geometry values from the tool table **tool.t**, such as length **L** or radius **R**, are not effective with turning tools.
- Assign unique tool names!
If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
 - Tool that is in the spindle
 - Tool that is in the magazine



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

- Tool that is defined in the tool table but is currently not in the magazine
If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!
Further information: "Creating a tool table in inches", Page 395
- The columns **WPL-DX-DIAM** and **WPL-DZL** are deactivated in the default configuration.
In the machine parameter **columnKeys** (no. 105501), the machine manufacturer activates the columns **WPL-DX-DIAM** and **WPL-DZL**. The names of the columns may be different, however.

18.4.4 Grinding tool table toolgrind.grd (option 156)

Application

The grinding tool table **toolgrind.grd** contains the data specific to grinding tools.

Related topics

- Editing tool data in tool management
Further information: "Tool management ", Page 165
- Tool data required for grinding tools
Further information: "Tool data for grinding tools (option 156)", Page 158
- Grinding operations on milling machines
Further information: User's Manual for Programming and Testing
- Tool table for dressing tools
Further information: "Dressing tool table tooldress.drs (option 156)", Page 389
- General tool data, regardless of the technology
Further information: "Tool table tool.t", Page 368

Requirements


- Software option 156: Jig grinding
- Grinding tool is defined in **TYPE** column of tool management
Further information: "Tool types", Page 150








Description of function







The file name of the grinding tool table is **toolgrind.grd** and this table must be stored in the folder **TNC:\table**.

The **toolgrind.grd** grinding tool table provides the following parameters:








Parameter	Meaning
T	Tool number Row number in the grinding tool table The tool number allows identifying each tool unambiguously, e.g. for calling a tool. Further information: User's Manual for Programming and Testing You can define an index after the period. Further information: "Indexed tool", Page 146 The row number must match the tool number in the tool.t tool table Input: 0...32767
NAME	Name of grinding wheel The tool name allows identifying a tool, e.g. for calling a tool. Further information: User's Manual for Programming and Testing You can define an index after the period. Further information: "Indexed tool", Page 146 Input: Text width 32

Parameter	Meaning
TYPE 	Type of grinding wheel Depending on the selected grinding tool type, the control displays the suitable tool parameters in the Form workspace of tool management. Further information: "Types within the grinding tools", Page 152 Further information: "Tool management ", Page 165 Selection by means of a selection window Input: GRIND_M, GRIND_MS, GRIND_MT, GRIND_S, GRIND_A, and GRIND_P
R-OVR 	Radius of grinding wheel Outermost radius of grinding tool After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.000000...999.999999
L-OVR 	Overhang of grinding wheel Length up to the outermost radius of the grinding tool, with respect to the tool carrier preset After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.000000...999.999999
LO 	Overall length Absolute length of the grinding tool, with respect to the tool carrier preset After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.000000...999.999999
LI 	Length to the inner edge Length up to the inner edge, with respect to the tool carrier preset After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.000000...999.999999
B 	Width Width of the grinding tool After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.000000...999.999999
G 	Depth Depth of grinding wheel After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.000000...999.999999
ALPHA	Angle for the slant After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.00000...90.00000

Parameter	Meaning
GAMMA	Angle for the corner After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 45.00000...180.00000
RV 	Radius at the edge for L-OVR After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.00000...999.99999
RV1 	Radius at the edge for LO After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.00000...999.99999
RV2 	Radius at the edge for LI After initial dressing, you can no longer edit this parameter. Further information: User's Manual for Machining Cycles Input: 0.00000...999.99999
dR-OVR 	Compensation of the radius Delta value of the radius for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter R-OVR Input: -999.999999...+999.999999
dL-OVR 	Compensation of the overhang Delta value of the overhang for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter L-OVR Input: -999.999999...+999.999999
dLO 	Compensation of the total length Delta value of the total length for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter LO Input: -999.999999...+999.999999
dLI 	Compensation of the length to the inner edge Delta value of the length up to the inner edge for tool compensation The control uses this value only for machining, and not for dressing! After dressing and measuring the grinding tool, the control will automatically enter the compensation value. Is added to the parameter LI Input: -999.999999...+999.999999

Parameter	Meaning
R_SHAFT 	Radius of the tool shank Input: 0.00000...999.99999
R_MIN 	Min. permissible radius If, after dressing, the actual radius is below the minimum permissible radius defined here, the control will display an error message. Input: 0.00000...999.99999
B_MIN 	Min. permissible width If, after dressing, the actual width is below the minimum permissible width defined here, the control will display an error message. Input: 0.00000...999.99999
V_MAX 	Maximum permissible cutting speed Cutting speed limit This value cannot be exceeded by programming a higher value or by using the potentiometer. Input: 0.000...999.999
V	Current cutting speed Currently no function Input: 0.000...999.999
W	Tilt angle Currently no function Input: -90.00000...90.0000
W_TYPE	Tilted toward inner or outer edge Currently no function Input: -1, 0, +1
KIND	Type of machining (internal/external grinding) Currently no function Input: 0, 1
HW	Wheel has a relief cut Grinding wheel with or without relief cut: <ul style="list-style-type: none"> ■ 0: No relief cut ■ 1: With relief cut Selection by means of a selection window Input: 0, 1
HWA 	Angle for relief cut on the outer edge Input: 0.00000...45.00000
HWI 	Angle for relief cut on the inner edge Input: 0.00000...45.00000

Parameter	Meaning
INIT_D_OK	<p>Initial dressing performed</p> <p>Initial dressing is the first dressing operation performed on the grinding wheel. If the following requirements are fulfilled, the control will set the parameter INIT_D_OK to 1:</p> <ul style="list-style-type: none"> ■ Grinding tool is defined ■ Initial dressing performed <p>If the parameter INIT_D_OK is set to 1, the control will disable the parameters for defining the grinding tool.</p> <p>If you set the parameter INIT_D_OK to 0, the control will re-enable the editing of the parameters. In this case, the control will have to perform initial dressing of the tool again.</p> <p>Further information: User's Manual for Machining Cycles</p> <p>Input: 0, 1</p>
INIT_D_PNR	<p>Dresser location for initial dressing</p> <p>Dressing location used for initial dressing</p> <p>Input: 0...9999</p>
INIT_D_DNR	<p>Dresser number for initial dressing</p> <p>Number of the dresser used for initial dressing</p> <p>Input: 0...32767</p>
MESS_OK	<p>Measure the grinding wheel</p> <p>Currently no function</p> <p>Input: 0, 1</p>
STATE	<p>Setup status</p> <p>Currently no function</p> <p>Input: %0000000000000000...%1111111111111111</p>
A_NR_D	<p>Dresser number (diameter dressing)</p> <p>Number of dresser used for dressing the diameter</p> <p>Input: 0...32767</p>
A_NR_A	<p>Dresser number (outer edge dressing)</p> <p>Number of dresser used for dressing the outer edge</p> <p>Input: 0...32767</p>
A_NR_I	<p>Dresser number (inner edge dressing)</p> <p>Number of dresser used for dressing the inner edge</p> <p>Input: 0...32767</p>
DRESS_N_D	<p>Dressing counter for diameter (specification)</p> <p>Specified number of dressing cycle calls that will be skipped until the next dressing of the diameter.</p> <p>Input: 0...999</p>
DRESS_N_A	<p>Dressing counter for outer edge (specification)</p> <p>Specified number of dressing cycle calls that will be skipped until the next dressing of the outer edge.</p> <p>Input: 0...999</p>

Parameter	Meaning
DRESS_N_I 	Dressing counter for inner edge (specification) Specified number of dressing cycle calls that will be skipped until the next dressing of the inner edge. Input: 0...999
DRESS_N_D_ACT 	Current dressing counter of the diameter Current number of dressing cycles that have been skipped since the last dressing of the diameter. Input: 0...999
DRESS_N_A_ACT 	Current dressing counter of the outer edge Current number of dressing cycles that have been skipped since the last dressing of the outer edge. Input: 0...999
DRESS_N_I_ACT 	Current dressing counter of the inner edge Current number of dressing cycles that have been skipped since the last dressing of the inner edge. Input: 0...999
AD 	Retraction amount at the diameter The control uses this parameter when using a cycle for dressing. Further information: User's Manual for Machining Cycles Input: 0.00000...999.99999
AA 	Retraction amount at the outer edge The control uses this parameter when using a cycle for dressing. Further information: User's Manual for Machining Cycles Input: 0.00000...999.99999
AI 	Retraction amount at the inner edge The control uses this parameter when using a cycle for dressing. Further information: User's Manual for Machining Cycles Input: 0.00000...999.99999
FORM	Wheel shape Currently no function Input: 0.00...99.99
A_PL	Chamfer length at outside Currently no function Input: 0.00000...999.99999
A_PW	Chamfer angle at outside Currently no function Input: 0.00000...89.99999
A_R1	Corner radius at outside Currently no function Input: 0.00000...999.99999
A_L	Length of outside Currently no function Input: 0.00000...999.99999

Parameter	Meaning
A_HL	Length of relief cut, wheel depth at outside Currently no function Input: 0.00000...999.99999
A_HW	Angle of relief cut at outside Currently no function Input: 0.00000...45.00000
A_S	Side depth at outside Currently no function Input: 0.00000...999.99999
A_R2	Angle of departure at outside Currently no function Input: 0.00000...999.99999
A_G	Reserve at outside Currently no function Input: 0.00000...999.99999
I_PL	Chamfer length at inside Currently no function Input: 0.00000...999.99999
I_PW	Chamfer angle at inside Currently no function Input: 0.00000...89.99999
I_R1	Corner radius at inside Currently no function Input: 0.00000...999.99999
I_L	Length of inside Currently no function Input: 0.00000...999.99999
I_HL	Length of relief cut, wheel depth at inside Currently no function Input: 0.00000...999.99999
I_HW	Angle of relief cut at inside Currently no function Input: 0.00000...45.00000
I_S	Side depth at inside Currently no function Input: 0.00000...999.99999
I_R2	Angle of departure at inside Currently no function Input: 0.00000...999.99999
I_G	Reserve at inside Currently no function Input: 0.00000...999.99999

Notes

- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with grinding tools.
- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.
- Measure the grinding tool after dressing so that the control enters the correct delta values.
- Assign unique tool names!
If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
 - Tool that is in the spindle
 - Tool that is in the magazine



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

- Tool that is defined in the tool table but is currently not in the magazine
If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- The control shows delta values from the tool management graphically in the simulation. For delta values from the NC program or from compensation tables, the control only changes the position of the tool in the simulation.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

Further information: "Creating a tool table in inches", Page 395

18.4.5 Dressing tool table **tooldress.drs** (option 156)

Application

The dressing tool table **tooldress.drs** contains the data specific to dressing tools.

Related topics

- Editing tool data in tool management
Further information: "Tool management ", Page 165
- Tool data required for dressing tools
Further information: "Tool data for dressing tools (option 156)", Page 162
- Initial dressing
Further information: User's Manual for Machining Cycles
- Grinding operations on milling machines
Further information: User's Manual for Programming and Testing
- Tool table for grinding tools
Further information: "Grinding tool table toolgrind.grd (option 156)", Page 382
- General tool data, regardless of the technology
Further information: "Tool table tool.t", Page 368

Requirements






- Software option 156: Jig grinding
- Dressing tool is defined in **TYPE** column of tool management






Further information: "Tool types", Page 150

Description of function

The file name of the dressing tool table is **tooldress.drs** and this table must be stored in the folder **TNC:\table**.

The **tooldress.drs** dressing tool table provides the following parameters:

Parameter	Meaning
T	<p>Row number in the dressing tool table</p> <p>The tool number allows identifying each tool unambiguously, e.g. for calling a tool.</p> <p>Further information: User's Manual for Programming and Testing You can define an index after the period.</p> <p>Further information: "Indexed tool", Page 146</p> <p>The row number must match the tool number in the tool.t tool table.</p> <p>Input: 0.0...32767.9</p>
NAME	<p>Name of dressing tool</p> <p>The tool name allows identifying a tool, e.g. for calling a tool.</p> <p>Further information: User's Manual for Programming and Testing You can define an index after the period.</p> <p>Further information: "Indexed tool", Page 146</p> <p>Input: Text width 32</p>
ZL 	<p>Tool length 1</p> <p>Length of the tool in the Z direction, with respect to the tool carrier preset</p> <p>Further information: "Tool carrier reference point", Page 141</p> <p>Input: -99999.9999...+99999.9999</p>
XL 	<p>Tool length 2</p> <p>Length of the tool in the X direction, with respect to the tool carrier preset</p> <p>Further information: "Tool carrier reference point", Page 141</p> <p>Input: -99999.9999...+99999.9999</p>
YL 	<p>Tool length 3</p> <p>Length of the tool in the Y direction, with respect to the tool carrier preset</p> <p>Further information: "Tool carrier reference point", Page 141</p> <p>Input: -99999.9999...+99999.9999</p>
DZL 	<p>Tool length oversize 1</p> <p>Delta value of tool length 1 for tool compensation Is added to the parameter ZL</p> <p>Input: -99999.9999...+99999.9999</p>
DXL 	<p>Tool length oversize 2</p> <p>Delta value of tool length 2 for tool compensation Is added to the parameter XL</p> <p>Input: -99999.9999...+99999.9999</p>

Parameter	Meaning
DYL 	Tool length oversize 3 Delta value of tool length 3 for tool compensation Is added to the parameter YL Input: -99999.9999...+99999.9999
RS 	Tool tip radius Input: 0.0000...99999.9999
DRS 	Cutter radius oversize Delta value of the cutter radius for tool compensation Is added to the parameter RS Input: -999.9999...+999.9999
TO 	Tool orientation The control uses the tool orientation to determine the position of the tool's cutting edge. Input: 1...9
CUTWIDTH	Width of tool (plate, roll) Tool width of the tool types dressing plate and dressing roll Input: 0.0000...99999.9999
TYPE 	Type of dressing tool Depending on the selected dressing tool type, the control displays the suitable tool parameters in the Form workspace of tool management. Further information: "Types within the dressing tools", Page 152 Further information: "Tool management ", Page 165 Selection by means of a selection window Input: DIAMOND, SPINDLE, PLATE, and ROLL
N-DRESS	Speed of the tool (dressing spindle) Shaft speed of a dressing spindle or dressing roll Input: 0.0000...99999.9999

Notes

- When dressing a grinding tool, the tool must not be assigned a tool carrier kinematic model.
- Geometry values from the tool table **tool.t**, such as length or radius, are not effective with dressing tools.
- Assign unique tool names!
If you define identical tool names for multiple tools, the control looks for the tool in the following sequence:
 - Tool that is in the spindle
 - Tool that is in the magazine



Refer to your machine manual.

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

- Tool that is defined in the tool table but is currently not in the magazine
If the control, for example, finds multiple available tools in the tool magazine, it inserts the tool with least remaining tool life.
- If you want to archive tool tables, save them with different file names and the corresponding file extension.
- Use the machine parameter **unitOfMeasure** (no. 101101) to define inches as the unit of measure. This does not automatically change the unit of measure in the tool table!

Further information: "Creating a tool table in inches", Page 395

18.4.6 Touch probe table **tchprobe.tp**

Application

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



Related topics





- Editing tool data in tool management
Further information: "Tool management ", Page 165
- Touch probe functions
Further information: "Touch Probe Functions in the Manual Operating Mode", Page 305
- Programmable touch probe cycles
Further information: User's Manual for Measuring Cycles for Workpieces and Tools

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**.

The touch probe table **tchprobe.tp** provides the following parameters:

Parameter	Meaning
NO	Sequential number of touch probe You use this number to assign the touch probe to the data in the tool management column TP_NO . Input: 1...99
TYPE	Selection of the touch probe? <div>  <div>  <p>The TS 642 touch probe makes the following values available:</p> <ul style="list-style-type: none"> ■ 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. </div> </div> Input: TS120, TS220, TS249, TS260, TS440, TS444, TS460, TS630, TS632, TS640, TS642-3, TS642-6, TS649, TS740, TS 760, KT130, OEM
CAL_OF1	TS center misalignmt. ref. axis? [mm] Offset of the touch probe axis to the spindle axis in the main axis Input: -99999.9999...+99999.9999
CAL_OF2	TS center misalignmt. aux. axis? [mm] Offset of the touch probe axis to the spindle axis in the secondary axis Input: -99999.9999...+99999.9999
CAL_ANG	Spindle angle for calibration? Prior to calibrating or probing, the control aligns the touch probe with the spindle orientation angle (if possible). Input: 0.0000...359.9999
F	Probing feed rate? [mm/min] In the machine parameter maxTouchFeed (no. 122602), the machine manufacturer defines the maximum probing feed rate. If F is greater than the maximum probing feed rate, then the maximum probing feed rate will be used. Input: 0...9999
FMAX	Rapid traverse in probing cycle? [mm/min] Feed rate at which the control pre-positions the touch probe and positions it between the measuring points Input: -99999...+99999
DIST	Maximum measuring range? [mm] If the stylus is not deflected in a probing process within the defined value, the control will display an error message. Input: 0.00100...99999.99999

Parameter	Meaning
SET_UP 	Set-up clearance? [mm] Distance of touch probe from the defined touch point when pre-positioning The smaller this value is, the more exactly you must define the touch point position. Safety clearances defined in the touch probe cycle are added to this value. Input: 0.00100...99999.99999
F_PREPOS 	Pre-position at rapid? ENT/NOENT Speed for pre-positioning: <ul style="list-style-type: none"> ■ FMAX_PROBE: Pre-position at the speed from FMAX ■ FMAX_MACHINE: Pre-position at machine rapid traverse Input: FMAX_PROBE, FMAX_MACHINE
TRACK 	Probe oriented? Yes=ENT/No=NOENT Orienting the infrared touch probe in each probing process: <ul style="list-style-type: none"> ■ ON: The control orients the touch probe in the defined probing direction. In this way, the stylus is always deflected in the same direction, improving measuring accuracy. ■ OFF: The control will not orient the touch probe. If you change the TRACK parameter, you must recalibrate the touch probe. Input: ON, OFF
SERIAL 	Serial number? The control automatically edits this parameter of touch probes with an EnDat interface. Input: Text width 15
REACTION	Reaction? EMERGSTOP=ENT/NCSTOP=NOENT As soon as touch probes with a collision protection adapter detect a collision, they react by resetting the ready signal. Reaction to resetting the ready signal: <ul style="list-style-type: none"> ■ NCSTOP: Interrupt NC program ■ EMERGSTOP: Emergency stop, quick braking of the axes Input: NCSTOP, EMERGSTOP

Editing the touch probe table

To edit the touch probe table:



- ▶ Select the **Tables** operating mode



- ▶ Select **Add**
- The control opens the **Quick selection** and the **Open File** workspaces.



- ▶ Select the **tchprobe.tp** file in the **Open File** workspace
- ▶ Select **Open**











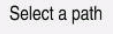

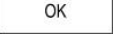


- The control opens the **Touch probes** application.
- ▶ Activate **Edit**
- ▶ Select the desired value
- ▶ Edit the value

Notes

- You can also edit the touch probe table values in the tool management.
- If you want to archive tool tables or use them for simulation, save them with different file names and the corresponding file extension.
- In the machine parameter **overrideForMeasure** (no. 122604), the machine manufacturer defines whether you will be allowed to change the feed rate with the feed-rate potentiometer during probing.

18.4.7 Creating a tool table in inches

To create a tool table in inches:

- | | |
|---|--|
|  | ▶ Select the Manual operating mode |
|  | ▶ Select T |
|  | ▶ Select the tool T0 |
|  | ▶ Press the NC Start key |
| | ➢ The control removes the current tool and does not insert a new tool. |
| | ▶ Restart the control |
| | ▶ Do not acknowledge Power interrupted |
|  | ▶ Select the Files operating mode |
| | ▶ Open the TNC:\table folder |
| | ▶ Rename the original file (e.g., tool.t as tool_mm.t) |
|  | ▶ Select the Tables operating mode |
|  | ▶ Select Add |
|  | ▶ Select Create new table |
| | ➢ The control opens the Create new table window. |
| | ▶ Select a folder with a corresponding file extension (e.g., t) |
|  | ▶ Select Select a path |
| | ➢ The control opens the Save as window. |
| | ▶ Select the table folder |
| | ▶ Enter a name (e.g., tool) |
|  | ▶ Select Create |
|  | ▶ Select OK |
| | ➢ The control opens the Tool table tab in Tables operating mode. |
| | ▶ Restart the control |
|  | ▶ Acknowledge Power interrupted with the CE key |
|  | ▶ Select the Tool table tab in Tables operating mode |
| | ➢ The control uses the newly created table as a tool table. |

18.5 Pocket table tool_p.tch

Application

The **tool_p.tch** pocket table provides the pocket assignment of the tool magazine. The control needs the pocket table in order to change the tool.

Related topics

- Tool call
Further information: User's Manual for Programming and Testing
- Tool table
Further information: "Tool table tool.t", Page 368

Requirement

- The tool is defined in the tool management.
Further information: "Tool management ", Page 165

Description of function

The file name of the pocket table is **tool_p.tch** and this table must be stored in the folder **TNC:\table**.

The **tool_p.tch** pocket table provides the following parameters:

Parameter	Meaning
P	Pocket number? Pocket number of the tool in the tool magazine Input: 0.0...99.9999
T	Tool number? Row number of the tool from the tool table Further information: "Tool table tool.t", Page 368 Input: 1...99999
TNAME	Tool name? Name of the tool from the tool table When you define the tool number, the control will automatically load the tool name. Further information: "Tool table tool.t", Page 368 Input: Text width 32
RSV	Reserve pocket? When a tool is in the spindle, the control reserves the pocket of this tool in the box magazine. To reserve the pocket for the tool: <ul style="list-style-type: none"> ■ No value entered: Pocket is not reserved ■ R: Pocket is reserved Input: No value, R
ST	Special tool? Define the tool as a special tool (e.g., with oversize tools): <ul style="list-style-type: none"> ■ No value entered: No special tool ■ S: Special tool Input: No value, S

Parameter	Meaning
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: <ul style="list-style-type: none"> ■ No value entered: No fixed pocket ■ F: Fixed pocket Input: No value, F
L	Locked pocket? To lock a pocket for tools (e.g., the pockets next to special tools): <ul style="list-style-type: none"> ■ No value entered: Do not lock ■ L: Lock Input: No value, L
DOC	Pocket comment? The control automatically loads the tool comment from the tool table. Further information: "Tool table tool.t", Page 368 Input: Text width 32
PLC	PLC status? Information about this tool pocket, which is transferred to the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: %00000000...%11111111
P1 ... P5	Value? The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: -99999.9999...+99999.9999
PTYP	Tool type for pocket table? Tool type for evaluation in the pocket table The machine manufacturer defines the function of this parameter. Refer to your machine manual. Input: 0...99
LOCKED_ABOVE	Lock pocket above? Box magazine: Lock the pocket above This parameter depends on the machine. Refer to your machine manual. Input: 0...99999
LOCKED_BELOW	Lock pocket below? Box magazine: Lock the pocket below This parameter depends on the machine. Refer to your machine manual. Input: 0...99999
LOCKED_LEFT	Lock pocket at left? Box magazine: Lock the pocket at left This parameter depends on the machine. Refer to your machine manual. Input: 0...99999

Parameter	Meaning
LOCKED_RIGHT	Lock pocket at right? Box magazine: Lock the pocket at right This parameter depends on the machine. Refer to your machine manual. Input: 0...99999
LAST_USE	LAST_USE The control automatically loads the date and time of the last tool call from the tool table. Further information: "Tool table tool.t", Page 368 Refer to your machine manual. Entry: Text width 20
S1	S1 Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: Text width 16
S2	S2 Value for evaluation in the PLC The machine manufacturer defines the function of this parameter. Refer to your machine manual. Entry: Text width 16

18.6 Tool usage file

Application

The control saves information about the tools of an NC program in a tool usage file (e.g., all the required tools and the tool usage times). The control needs this file for the tool usage test.

Related topics

- Using the tool usage test
Further information: "Tool usage test", Page 172
- Working with pallet tables
Further information: User's Manual for Programming and Testing
- Tool data from the tool table
Further information: "Tool table tool.t", Page 368

Requirements

- **Generate tool-usage file** is enabled by your machine manufacturer
The machine manufacturer uses the machine parameter **createUsageFile** (no. 118701) to define whether the **Generate tool-usage file** function is enabled.
Further information: "Creating the tool usage file", Page 173
- The **Generate tool-usage file** function setting is set to either **once** or **always**
Further information: "Channel settings", Page 449

Description of function

The tool usage file provides the following parameters:

Parameter	Meaning
NR	Row number in the tool usage file Input: 0...99999
TOKEN	In the TOKEN column, the control uses one word to show which information is contained in the respective row: <ul style="list-style-type: none"> ■ TOOL: Data per tool call; listed in chronological order ■ TTOTAL: All data of a tool; listed in alphabetical order ■ STOTAL: Called NC programs; listed in chronological order ■ TIMETOTAL: Total tool usage time of an NC program ■ TOOLFILE: Path of the tool table This enables the control during the tool usage test to detect whether you have performed the simulation with the tool table tool.t Input: Text width 17
TNR	Tool number If the control has not yet inserted a tool, the column contains the value -1 . Input: -1...32767
IDX	Tool index Input: 0...9
NAME	Tool name Input: Text width 32
TIME	Tool usage time in seconds Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Input: 0...9999999
WTIME	Total tool usage time in seconds Total time between the tool changes, during which the tool is cutting a workpiece Input: 0...9999999
RAD	Sum of the tool radius R and the delta radius DR from the tool table Input: -999999.9999...999999.9999
BLOCK	NC block number of the tool call Input: 0...999999999
PATH	Path of the NC program, the pallet table, or the tool table Input: Text width 300
T	Tool number, including the tool index If the control has not yet inserted a tool, the column contains the value -1 . Input: -1...32767.9

Parameter	Meaning
OVRMAX	Maximum feed-rate override If you only simulate the machining operation, then the control will enter the value 100 . Input: 0...32767
OVRMIN	Minimum feed rate override If you only simulate the machining operation, then the control will enter the value -1 . Input: -1...32767
NAMEPRG	Type of tool definition during a tool call: <ul style="list-style-type: none"> ■ 0: The tool number is programmed ■ 1: The tool name is programmed Input: 0, 1
LINENR	Row number of the pallet table in which the NC program is defined Input: -1...99999

Note

The control creates dependency files (*.dep); for example, the tool-usage file in order to perform a tool usage test.

In the machine parameter **dependentFiles** (no. 122101) the machine manufacturer defines whether the control displays dependency files.

18.7 T usage order (option 93)

Application

In the **T usage order** table, the control displays the tool call sequence in an NC program. Before starting the program, you can see, for example, when a manual tool change will take place.

Requirements

- Software option 93: Extended Tool Management
- Tool-usage file has been created

Further information: "Creating the tool usage file", Page 173

Further information: "Tool usage file", Page 398

Description of function

When you select an NC program in **Program Run** operating mode, the control will automatically create the **T usage order** table. The control displays the table in the **T usage order** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot 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 146 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 146 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: <ul style="list-style-type: none"> ■ 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 396 ■ T no. missing: Tool is not defined in the tool management Further information: "Tool management ", Page 165
T PROG	Number or name of the programmed tool, including an index as needed Further information: "Indexed tool", Page 146
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 398
TOOL TIME	Estimated time of tool change
M3/M4 TIME	Tool usage time from the TIME column of the tool usage file (in seconds) Time during which the tool is cutting a workpiece (excluding rapid traverse movements) Further information: "Tool usage file", Page 398
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 396

18.8 Tooling list (option 93)

Application

In the **Tooling list** table, the control displays information about all the tools called within an NC program. Before starting the program, you can check, for example, whether all tools are contained in the magazine.

Requirements

- Software option 93: Extended Tool Management
- Tool-usage file has been created

Further information: "Creating the tool usage file", Page 173

Further information: "Tool usage file", Page 398

Description of function

When you select an NC program in **Program Run** operating mode, the control will automatically create the **Tooling list** table. The control displays the table in the **Tooling list** application in **Tables** operating mode. The control lists all the tools called within the active NC program and all the tools called within called NC programs in chronological order. You cannot 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 146 May differ from the programmed tool (e.g., when a replacement tool is used)
TOOL INFO	The control displays the following tool information: <ul style="list-style-type: none"> ■ 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 396 ■ T no. missing: Tool is not defined in the tool management Further information: "Tool carrier management", Page 169
T PROG	Number or name of the programmed tool, including an index as needed Further information: "Indexed tool", Page 146
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 398
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 396

18.9 Preset table

Application

The **preset.pr** preset table allows you to manage presets, such as the position and misalignment of a workpiece in the machine. The active row in the preset table is used as a workpiece preset in the NC program and as the coordinate origin of the workpiece coordinate system **W-CS**.

Further information: "Presets in the machine", Page 137

Related topics

- Setting and activating presets

Further information: "Preset management", Page 193

Description of function

The preset table is stored in the **TNC:\table** directory by default and is named **preset.pr**. In the **Tables** operating mode, the preset table is open by default.



Refer to your machine manual.

The machine tool builder 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.

Preset table icons

The preset table contains the following icons:

Icon	Function
	Active row
	Write-protected row

Preset table parameters

The preset table contains the following parameters:

Parameter	Meaning
NO	Number of preset table row Input: 0...99999999
DOC	Comment Entry: Text width 16
X	X coordinate of preset Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 182 Input: -99999.99999...+99999.99999
Y	Y coordinate of preset Basic transformation relating to the basic coordinate system B-CS Further information: "Basic coordinate system B-CS", Page 182 Input: -99999.99999...+99999.99999

Parameter	Meaning
Z	<p>Z coordinate of preset</p> <p>Basic transformation relating to the basic coordinate system B-CS</p> <p>Further information: "Basic coordinate system B-CS", Page 182</p> <p>Input: -99999.99999...+99999.99999</p>
SPA	<p>Spatial angle of preset in the A axis</p> <p>Basic transformation relating to the basic coordinate system B-CS, the preset contains a 3D basic rotation in tool axis Z.</p> <p>Further information: "Basic coordinate system B-CS", Page 182</p> <p>Input: -99999.999999...+99999.999999</p>
SPB	<p>Spatial angle of preset in the B axis</p> <p>Basic transformation relating to the basic coordinate system B-CS, the preset contains a 3D basic rotation in tool axis Z.</p> <p>Further information: "Basic coordinate system B-CS", Page 182</p> <p>Input: -99999.999999...+99999.999999</p>
SPC	<p>Spatial angle of preset in the C axis</p> <p>Basic transformation relating to the basic coordinate system B-CS, the preset contains a basic rotation in tool axis Z.</p> <p>Further information: "Basic coordinate system B-CS", Page 182</p> <p>Input: -99999.999999...+99999.999999</p>
X_OFFS	<p>Position of the X axis for the preset</p> <p>Offset relating to the machine coordinate system M-CS</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>Input: -99999.99999...+99999.99999</p>
Y_OFFS	<p>Position of the Y axis for the preset</p> <p>Offset relating to the machine coordinate system M-CS</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>Input: -99999.99999...+99999.99999</p>
Z_OFFS	<p>Position of the Z axis for the preset</p> <p>Offset relating to the machine coordinate system M-CS</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>Input: -99999.99999...+99999.99999</p>
A_OFFS	<p>Axis angle of the A axis for the preset</p> <p>Offset relating to the machine coordinate system M-CS</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>Input: -99999.999999...+99999.999999</p>
B_OFFS	<p>Axis angle of the B axis for the preset</p> <p>Offset relating to the machine coordinate system M-CS</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>Input: -99999.999999...+99999.999999</p>
C_OFFS	<p>Axis angle of the C axis for the preset</p> <p>Offset relating to the machine coordinate system M-CS</p> <p>Further information: "Machine coordinate system M-CS", Page 179</p> <p>Input: -99999.999999...+99999.999999</p>

Parameter	Meaning
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 179 Input: -99999.99999...+99999.99999
V_OFFS	Position of the V axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 179 Input: -99999.99999...+99999.99999
W_OFFS	Position of the W axis for the preset Offset relating to the machine coordinate system M-CS Further information: "Machine coordinate system M-CS", Page 179 Input: -99999.99999...+99999.99999
ACTNO	Active workpiece preset The control automatically enters 1 in the active row. Input: 0, 1
LOCKED	Write-protection of the table row Entry: Text width 16



Refer to your machine manual.

In the optional machine parameter **CfgPresetSettings** (no. 204600), the machine manufacturer can block the setting of a preset in individual axes.

Write-protection for table rows

The **LOCKED** column allows protecting any rows of the preset table against overwriting. You can protect the row with or without a password.

The control displays an icon at the beginning of write-protected rows.

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.

- ▶ Protecting table rows without a password is recommended
- ▶ Note down your passwords

18.9.1 Activating write protection

Protecting table rows without a password

To protect a table row without a password:



- ▶ Activate the **Edit** switch
- ▶ Select the **LOCKED** column in the desired row
- ▶ Enter **L**
- ▶ Confirm your input
- The control protects the row from being edited and displays an icon at the beginning of the row.

Protecting table rows with a password

NOTICE
<p>Caution: Data may be lost!</p> <p>Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.</p> <ul style="list-style-type: none"> ▶ Protecting table rows without a password is recommended ▶ Note down your passwords

To protect a table row with a password:



- ▶ Activate the **Edit** switch
- ▶ Select the **LOCKED** column in the desired row
- ▶ Enter the password
- ▶ Confirm your input
- The control displays the characters **###** in the **LOCKED** column.
- The control protects the row from being edited and displays an icon at the beginning of the row.

18.9.2 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** switch
- ▶ Select the **LOCKED** column in the desired row
- ▶ Enter **L** again
- ▶ Confirm your input
- The control enables the row for editing and removes the icon at the beginning of the row.

Unlocking table rows that are protected with a password

NOTICE

Caution: Data may be lost!

Rows protected by a password can be unlocked by entering the selected password exclusively. Forgotten passwords cannot be reset. This locks the protected rows permanently.

- ▶ Protecting table rows without a password is recommended
- ▶ Note down your passwords

To unlock a table row that is protected with a password:



- ▶ Activate the **Edit** switch
- ▶ Select the **LOCKED** column in the desired row
- ▶ Delete ###
- ▶ Enter the password
- ▶ Confirm your input
- > The control enables the row for editing and removes the icon at the beginning of the row.

18.9.3 Creating a preset table in inches

If you define inches as the unit of measure in the machine parameter **unitOfMeasure** (no. 101101), the unit of measure of the preset table will not be adjusted automatically.

To create a preset table in inches:



- ▶ Select the **Files** operating mode

- ▶ Open the **TNC:\table** folder

- ▶ Rename the **preset.pr** file (e.g., as **preset_inch.pr**)

- ▶ Select the **Tables** operating mode



- ▶ Select **Add**



- ▶ Select **Create new table**
- > The control opens the **Create new table** window.



- ▶ Select the **pr** folder



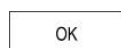
- ▶ Select the desired **prototype**
- > The control opens the **Save as** window.

- ▶ Select the **table** folder

- ▶ Enter the name **preset.pr**



- ▶ Select **Create**



- ▶ Select **OK**
- > The control opens the **Presets** tab in **Tables** operating mode.

- ▶ Restart the control



- ▶ Select the **Presets** tab in **Tables** operating mode
- > The control uses the newly created table as a preset table.

Notes

NOTICE

Caution: Significant property damage!

Undefined fields in the preset table behave differently from fields defined with the value **0**: Fields defined with the value **0** overwrite the previous value when activated, whereas with undefined fields the previous value is kept.

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

- To optimize the file size and the processing speed, keep the preset table as short as possible.
- New rows can be inserted only at the end of the preset table.
- The machine manufacturer uses the optional **initial** machine parameter (no. 105603) to define a default value for each column of a new row.
- If the unit of measure of the preset table is not identical with the unit of measure defined in the machine parameter **unitOfMeasure** (no. 101101), the control displays a message in the diagnostics bar when in the **Tables** operating mode.
- The control may feature a pallet preset table, depending on the machine. When a pallet preset is active, the presets in the preset table are referenced to this pallet preset.

Further information: User's Manual for Programming and Testing

18.10 Tables for AFC (option 45)

18.10.1 Basic AFC settings in AFC.tab

Application

In the **AFC.TAB** table, you can enter the feed rate control settings to be used by the control. This table must be saved in the **TNC:\table** directory.

Related topics

- Programming AFC

Further information: "Adaptive Feed Control (AFC, option 45)", Page 230

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The data in this table are default values that are copied into a file belonging to the respective NC program during a teach-in cut. The values act as the basis for feedback control.

Further information: "Description of function", Page 412



If you define a tool-specific feedback-control reference power using the **AFC-LOAD** column in the tool table, the control generates the associated file for the relevant NC program without a teach-in cut. The file is created shortly before feedback control becomes effective.

Parameter

The **AFC.tab** table provides the following parameters:




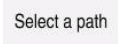
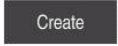
Parameter	Meaning
NR	Row number in the table Input: 0...9999
AFC	Name of the control setting Enter this name in the AFC tool management column. It specifies the assignment of the control parameters to the tool. Input: Text width 10
FMIN	Feed rate at which the control will perform an overload reaction Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) If the AFC.TAB columns FMIN and FMAX each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. Further information: "Monitoring tool wear and tool load", Page 236 Input: 0...999
FMAX	Maximum feed rate within the material up to which the control can automatically increase the feed rate Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) If the AFC.TAB columns FMIN and FMAX each have a value of 100%, Adaptive Feed Control is deactivated, but cut-related tool wear monitoring and tool load monitoring remain active. Further information: "Monitoring tool wear and tool load", Page 236 Input: 0...999
FIDL	Feed rate at which the control will traverse the tool outside of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: 0...999
FENT	Feed rate at which the control will move the tool into and out of the material Enter the value in percent of the programmed feed rate Not necessary in turning mode (option 50) Input: 0...999

Parameter	Meaning
OVL	<p>Desired reaction of the control to overload:</p> <ul style="list-style-type: none"> ■ M: Execution of a macro defined by the machine manufacturer ■ S: Immediate NC stop ■ F: NC stop once the tool has been retracted ■ E: Just display an error message on the screen ■ L: Disable active tool ■ -: No overload reaction <p>If the maximum spindle power is exceeded for more than one second and the feed rate falls below the defined minimum during that time, the control will conduct an overload reaction.</p> <p>In conjunction with the cut-related tool wear monitoring function, the control will only evaluate the options M, E, and L!</p> <p>Input: M, S, F, E, L, or -</p>
POUT	<p>Spindle power at which the control will detect that the tool exits the workpiece</p> <p>Enter the value in percent of the learned reference load</p> <p>Recommended input value: 8%</p> <p>In turning mode: Minimum load Pmin for tool monitoring (option 50)</p> <p>Input: 0...100</p>
SENS	<p>Sensitivity (aggressiveness) of feedback control</p> <p>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.</p> <p>In turning mode: Activate the monitoring of the minimum load Pmin (option 50):</p> <ul style="list-style-type: none"> ■ 1: Evaluate Pmin ■ 0: Do not evaluate Pmin <p>Input: 0...999</p>
PLC	<p>Value that the control will transfer to the PLC at the beginning of a machining step</p> <p>The machine manufacturer defines whether and which function will be performed by the control.</p> <p>Input: 0...999</p>

Creating an AFC.tab table

You need to create the table only if the table is missing in the **tables** folder.

To create the **AFC.tab** table:

- | | |
|---|---|
|  | ▶ Select the Tables operating mode |
|  | ▶ Select Add |
| | > The control opens the Quick selection and the Open File workspaces. |
|  | ▶ Select Create new table |
| | > The control opens the Create new table window. |
| | ▶ Select the tab folder |
| | ▶ Select the AFC.TAB format |
|  | ▶ Select Select a path |
| | > The control opens the Save as window. |
| | ▶ Select a folder |
| | ▶ Enter a name |
|  | ▶ Select Create |
| | > The control opens the table. |

Notes

- If there is no AFC.TAB table in the **TNC:\table** directory, the control uses a permanently defined, internal control setting for the teach-in cut. If, alternatively, a tool-dependent reference power value exists, the control uses it immediately. HEIDENHAIN recommends using the AFC.TAB table in order to ensure safe and well-defined operation.
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., +). Due to SQL commands, these characters can cause problems when data are input or read.

Further information: User's Manual for Programming and Testing

18.10.2 AFC.DEP settings file for teach-in cuts

Application

With a teach-in cut, the control at first copies the basic settings for each machining step, as defined in the AFC.TAB table, to a file called **<name>.H.AFC.DEP**. **<name>** is 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 408
- Setting up and using AFC
 "Adaptive Feed Control (AFC, option 45)"

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

Each row in the **<name>.H.AFC.DEP** file stands for a machining section, that you start with **FUNCTION AFC CUT BEGIN** and complete with **FUNCTION AFC CUT END**. You can edit all data of the **<name>.H.AFC.DEP** file for optimization purposes. If you have optimized the values from the AFC.TAB table, the control places a * in front of these control settings in the AFC column.

Further information: "Basic AFC settings in AFC.tab", Page 408

In addition to the contents from the **AFC.tab** table, the **AFC.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed (not editable)
IDX	Index of the tool with which the machining step was performed (not editable)
N	Difference for tool call: <ul style="list-style-type: none"> ■ 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: <ul style="list-style-type: none"> ■ L: In the next program run, a teach-in cut will be recorded for this machining step. The control will overwrite any existing values in this line ■ C: The teach-in cut was completed successfully. The next program run can be conducted with automatic feed control
AFC	Name of the control setting

Notes

- Note that the **<name>.H.AFC.DEP** file is locked against editing as long as the NC program **<name>.H** is running.

The control does not remove the editing lock until one of the following functions has been executed:

- **M2**
- **M30**
- **END PGM**
- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

18.10.3 Log file AFC2.DEP

Application

The control stores various pieces of information for each machining step of a teach-in cut in the **<name>.H.AFC2.DEP** file. **<name>** is the name of the NC program for which you have recorded the teach-in cut. During feedback control, the control updates the data and performs various evaluations.

Related topics

- Setting up and using AFC

Further information: "Adaptive Feed Control (AFC, option 45)", Page 230

Requirement

- Adaptive Feed Control (AFC, software option 45)

Description of function

The **AFC2.DEP** file provides the following information:

Column	Function
NR	Number of the machining step
TOOL	Number or name of the tool with which the machining step was performed
IDX	Index of the tool with which the machining step was performed
SNOM	Nominal spindle speed [rpm]
SDIFF	Maximum difference of the spindle speed in % of the nominal speed
CTIME	Machining time (tool in effect)
FAVG	Average feed rate (tool in effect)
FMIN	Smallest occurring feed factor. The control shows the value as a percentage of the programmed feed rate
PMAX	Maximum recorded spindle power during machining. The control shows the value as a percentage of the spindle's rated power.
PREF	Reference load of the spindle. The control shows the value as a percentage of the spindle's rated power.
OVLD	Overload reaction performed by the control: <ul style="list-style-type: none"> ■ M: A macro defined by the machine tool builder has been run ■ S: Immediate NC stop was conducted ■ F: NC stop was conducted after the tool was retracted ■ E: An error message was displayed ■ L: The current tool was locked ■ -: There was no overload reaction
BLOCK	Block number at which the machining step begins



During feedback control, the control determines the current machining time as well as the resulting time saving in percent. The control enters the results of the evaluation between the key words **total** and **saved** in the last line of the log file. Where the time balance is positive, the percentage value is also positive.

Note

- In the machine parameter **dependentFiles** (no. 122101), the machine manufacturer defines whether the control will display the dependent files in the file manager.

18.10.4 Editing tables for AFC

You can open and, if necessary, edit the tables for AFC during program run. The control offers only the tables for the active NC program.

To open a table for AFC:



AFC settings

- ▶ Select the **Program Run** operating mode
- ▶ Select **AFC settings**
- The control displays a selection menu. The control shows all existing tables for this NC program.
- ▶ Select a file, e.g. **AFC.TAB**
- The control opens the file in the **Tables** operating mode.

19

**Electronic
Handwheel**

19.1 Fundamentals

Application

If you want to approach a position in the machine's working space while the guard door is open or if you execute a small infeed movement, you can use the electronic handwheel. The electronic handwheel allows you to traverse the axes and perform various functions provided by the control.

Related topics

- Incremental jog positioning
Further information: "Incremental jog positioning of axes", Page 133
- Handwheel superimpositioning with GPS (option 44)
Further information: "Function Handwheel superimp.", Page 247
- Handwheel superimpositioning with **M118**
Further information: User's Manual for Programming and Testing
- Virtual tool axis **VT**
Further information: User's Manual for Programming and Testing
- Touch probe functions in **Manual** operating mode
Further information: "Touch Probe Functions in the Manual Operating Mode", Page 305

Requirement

- Electronic handwheel (e.g., HR 550FS)
The control supports the following electronic handwheels:
 - HR 410: Cable-bound handwheel without display
 - HR 420: Cable-bound handwheel with display
 - HR 510: Cable-bound handwheel without display
 - HR 520: Cable-bound handwheel with display
 - HR 550FS: Wireless handwheel with display, data transmission via radio

Description of function

You can use electronic handwheels in **Manual** or **Program Run** operating mode.

The HR 520 and HR 550FS portable handwheels feature a display that allows the control to show different types of information. You can use the handwheel soft keys for setup functions, such as the setting of presets or the activation of miscellaneous functions.

Once you have activated the handwheel with the handwheel activation key or the **Handwheel** switch, you can operate the control only by using the handwheel. If you press the axis keys in this state, the control will display the message **Handwheel active: Handwheel-1, MB0**.

If more than one handwheel is connected to a control, you can activate or deactivate a handwheel only by pressing the handwheel activation key on the respective handwheel. You need to deactivate the active handwheel in order to be able to select another handwheel.

Functions in Program Run operating mode

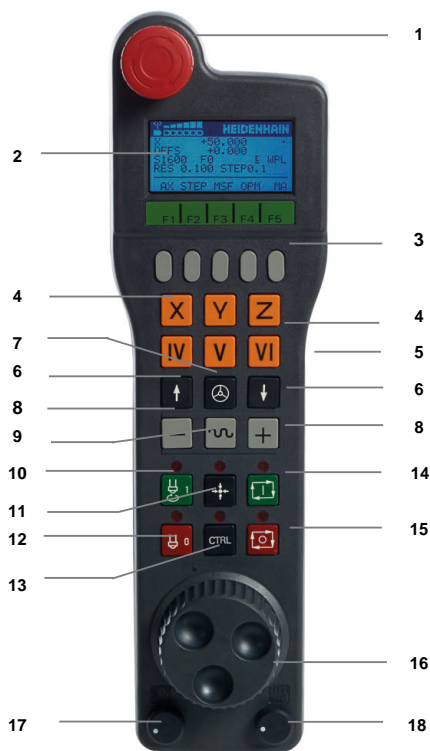
You can perform the following functions in **Program Run** operating mode:

- The **NC Start** key (**NC Start** handwheel key)
- The **NC Stop** key (**NC Stop** handwheel key)
- After the **NC Stop** key has been pressed: Internal stop (handwheel soft keys **MOP** and then **Stop**)
- After the **NC STOP** key has been pressed: Traverse manual axes (handwheel soft keys **MOP** and then **MAN**)
- Return to the contour after axes were manually traversed during an interruption of the program run (handwheel soft keys **MOP** and then **REPO**). The handwheel soft keys are used for operating.

Further information: "Returning to the contour", Page 349

- Switch on/off the "Tilt working plane" function (handwheel soft keys **MOP** and then **3-D**)

Operating elements of an electronic handwheel

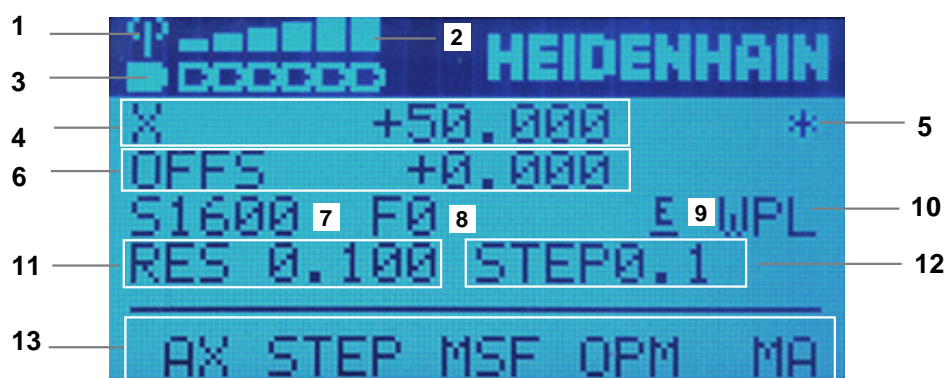


An electronic handwheel provides the following operating elements:

- 1 **EMERGENCY STOP** key
- 2 Handwheel display for status and for selecting functions
- 3 Handwheel soft keys
- 4 Axis keys; can be exchanged by the machine manufacturer depending on the axis configuration
- 5 Permissive button
The permissive button is on the rear side of the handwheel.
- 6 Arrow keys for defining the handwheel resolution
- 7 Handwheel activation key

- 8 Axis-direction key
Key for the direction of the traverse motion
- 9 Rapid traverse override for the traverse motion
- 10 Spindle switch-on (machine-dependent function, key can be exchanged by the machine manufacturer)
- 11 **Generate NC block** key (machine-dependent function, key can be exchanged by the machine manufacturer)
- 12 Spindle switch-off (machine-dependent function, key can be exchanged by the machine manufacturer)
- 13 **CTRL** key for special functions (machine-dependent function, key can be exchanged by the machine manufacturer)
- 14 **NC START** key (machine-dependent function, key can be exchanged by the machine manufacturer)
- 15 **NC Stop** key
Machine-dependent function; key can be exchanged by the machine manufacturer
- 16 Handwheel
- 17 Spindle speed potentiometer
- 18 Feed rate potentiometer
- 19 Cable connection; not required for the HR 550FS wireless handwheel

Contents of an electronic handwheel display



The display of an electronic handwheel consists of the following areas:

- 1 Handwheel is in the docking station or radio mode is active
Only with HR 550FS wireless handwheel
- 2 Field strength
Six bars = maximum field strength
Only with HR 550FS wireless handwheel
- 3 Charging condition of battery
Six bars = maximum charging condition. A bar moves from the left to the right during recharging.
Only with HR 550FS wireless handwheel
- 4 **X+50.000**: Position of the selected axis

- 5 *: STIB (control in operation); program run has been started or axis is in motion
- 6 Handwheel superimpositioning from **M118** or the Global Program Settings GPS (option 44)
 Further information: "Function Handwheel superimp.", Page 247
 Further information: User's Manual for Programming and Testing
- 7 **S1000:** Current spindle speed
- 8 Feed rate at which the selected axis is moving
- 9 **E:** Error message
 If an error message appears on the control, the handwheel display shows the message **ERROR** for three seconds. Then the letter **E** is shown in the display as long as the error is pending on the control.
- 10 Active setting in the **3-D rotation** window:
 - **VT:** Tool axis function
 - **WP:** Basic rotation function
 - **WPL:** 3D ROT function**Further information:** "3-D rotation window (option 8)", Page 199
- 11 Handwheel resolution
 Distance that the selected axis moves per handwheel revolution
 Further information: "Handwheel resolution", Page 420
- 12 Incremental jog active or inactive
 If the function is active, the control will display the active traverse step.
- 13 Soft-key row
 The soft key row provides the following functions:
 - **AX:** Select the machine axis
 Further information: "Creating a positioning block", Page 422
 - **STEP:** Incremental jog positioning
 Further information: "Incremental jog positioning", Page 422
 - **MSF:** Execute various functions of **Manual** operating mode (e.g., entering the feed rate **F**)
 Further information: "Entering miscellaneous functions M", Page 421
 - **OPM:** Select the operating mode
 - **MAN:** Manual operating mode
 - **MDI:** MDI application in **Manual** operating mode
 - **RUN:** Program Run operating mode
 - **SGL:** Single Block mode of **Program Run** operating mode
 - **MA:** Switching the magazine pockets

Handwheel resolution

The handwheel sensitivity determines which path an axis takes per revolution of the handwheel. The handwheel sensitivity results from the defined handwheel speed of the axis and the speed level used internally by the control. The speed level describes a percentage of the handwheel speed. The control calculates a specific handwheel sensitivity value for each speed level. The resulting handwheel sensitivity values are directly selectable with the handwheel arrow keys (only if incremental jog is not active).

The handwheel speed indicates the increment (e.g., 0.01 mm) traversed per handwheel detent position. You can change the handwheel speed with the handwheel's arrow keys.

If you have defined a handwheel speed of 1, the following handwheel resolutions are available:

Resulting handwheel sensitivity levels in mm/revolution and degrees/revolution:
0.0001/0.0002/0.0005/0.001/0.002/0.005/0.01/0.02/0.05/0.1/0.2/0.5/1

Resulting handwheel sensitivity levels in in/revolution:
0.000127/0.000254/0.000508/0.00127/0.00254/0.00508/0.0127/0.0254/0.0508/0.127/0.254/0.508

Examples for resulting handwheel sensitivity values:

Defined handwheel speed	Speed level	Resulting handwheel sensitivity
10	0.01 %	0.001 mm/revolution
10	0.01 %	0.001 degrees/revolution
10	0.0127 %	0.00005 in/revolution

Effect of the feed-rate potentiometer when handwheel is active

NOTICE

Caution: Possible damage to the workpiece!

When toggling between the machine operating panel and the handwheel, the feed rate may be reduced. This can cause visible marks on the workpiece.

- Make sure to retract the tool before toggling between the handwheel and the machine operating panel.

The settings of the feed-rate potentiometer on the handwheel may differ from those on the machine operating panel. When you activate the handwheel, the control will automatically activate the feed-rate potentiometer of the handwheel. When you deactivate the handwheel, the control will automatically activate the feed-rate potentiometer of the machine operating panel.

In order to make sure that the feed rate does not increase while you switch over between the potentiometers, the feed rate is either frozen or reduced.

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

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

19.1.1 Entering spindle speed S

To enter the spindle speed **S** by using an electronic handwheel:

- ▶ Press the **F3 (MSF)** handwheel soft key
- ▶ Press the **F2 (S)** handwheel soft key
- ▶ Select the desired speed by pressing the **F1** or **F2** key
- ▶ Press the **NC Start** key
- > The control activates the entered spindle speed.



If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10. By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

19.1.2 Entering the feed rate F

To enter the feed rate **F** by using an electronic handwheel:

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



If you press and hold the **F1** or **F2** key, the control will increase the counting increment by a factor of 10 each time it reaches a decimal value of 0. By additionally pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

19.1.3 Entering miscellaneous functions M

To enter a miscellaneous function by using an electronic handwheel:

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

Further information: User's Manual for Programming and Testing

19.1.4 Creating a positioning block



Refer to your machine manual.

Your machine tool builder can assign any function to the **Generate NC block** handwheel key.

To create a positioning block by using an electronic handwheel:



- ▶ Select the **Manual** operating mode

- ▶ Select the **MDI** application
- ▶ If necessary, select the NC block after which the positioning block should be inserted
- ▶ Activate the handwheel



- ▶ Press the **Generate NC block** key on the handwheel
- The control inserts a straight line **L**, including all of the axis positions.

19.1.5 Incremental jog positioning

Incremental jog positioning allows you to move the selected axis by a preset value.

To incrementally position an axis by using an electronic handwheel:

- ▶ Press the handwheel soft key F2 (**STEP**)
- ▶ Press the handwheel soft key 3 (**ON**)
- The control activates incremental jog positioning.
- ▶ Set the desired jog increment by using the **F1** or **F2** keys



The smallest possible increment is 0.0001 mm (0.00001 inches). The largest possible increment is 10 mm (0.3937 inches)..

- ▶ Confirm the selected jog increment by pressing the handwheel soft key F4 (**OK**)
- ▶ Use the **+** or **-** handwheel key to move the active handwheel axis in the corresponding direction
- The control moves the active axis by the entered increment every time the handwheel key is pressed.



If you press and hold the **F1** or **F2** key, each time it reaches a decimal value 0 the control increases the counting increment by a factor of 10. By also pressing the **CTRL** key, you can increase the counting increment by a factor of 100 when pressing **F1** or **F2**.

Notes

DANGER

Caution: hazard to the user!

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

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

NOTICE

Caution: Danger to the tool and workpiece!

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

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

- The machine manufacturer can provide additional functions for the HR5xx handwheels.
Refer to your machine manual.
- You can use the axis keys to activate the **X**, **Y**, and **Z** axes, as well as three other axes that can be defined by the machine manufacturer. Your machine manufacturer can also place the virtual axis **VT** on one of the free axis keys.

19.2 HR 550FS wireless handwheel

Application

With the HR 550FS wireless handwheel and its radio transmission characteristics, you can move farther away from the machine operating panel than with other handwheels. The HR 550FS wireless handwheel thus provides an important benefit, in particular for large machines.

Description of function

The HR 550FS wireless handwheel comes fitted with a rechargeable battery. The battery starts charging when you place the handwheel into the holder.

The HRA 551FS handwheel holder and the HR 550FS handwheel together form one function unit.



HR 550FS handwheel



HRA 551FS handwheel holder

The HR 550FS handwheel can be operated by battery for up to eight hours before it needs recharging. A completely discharged handwheel takes approx. three hours for a full charge. When you do not use the HR 550FS, always place it into the handwheel holder. This charges the handwheel battery constantly and a direct connection with the emergency-stop circuit is provided.

When the handwheel is in its holder, it provides the same functionality as during radio mode. This allows you to use a completely discharged handwheel.



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

If the control has triggered an emergency stop, you must reactivate the handwheel.

Further information: "Reactivating the handwheel", Page 428

If you happen to get close to the limit of the transmission range, the HR 550FS will set off a vibrating alarm. If this occurs, you must reduce the distance to the handwheel holder.

Note**⚠ DANGER****Caution: hazard to the user!**

Wireless handwheels, due to their rechargeable batteries and the influence of other wireless devices, are more susceptible to interference than cable-bound connections are. Ignoring the requirements for and information about safe operation leads to endangerment of the user, for example during installation or maintenance work.

- ▶ Check the radio connection of the handwheel for possible overlapping with other wireless devices
- ▶ Switch off the wireless handwheel and the handwheel holder after an operating time of 120 hours at the latest so that the control can run a functional test when it is restarted
- ▶ If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the handwheel holders (such as with color-coded stickers)
- ▶ If more than one wireless handwheel is being used in a workshop, then ensure an unambiguous assignment between the handwheels and the respective machine (such as with a functional test)

19.3 Configuration of wireless handwheel window

Application

The **Configuration of wireless handwheel** window shows the connection data of the HR 550FS wireless handwheel and provides various functions for optimizing the radio connection, such as setting the radio channel.

Related topics

- Electronic handwheel
Further information: "Electronic Handwheel", Page 415
- HR 550FS wireless handwheel
Further information: "HR 550FS wireless handwheel", Page 423

Description of function

Use the **Set up wireless handwheel** menu item to open the **Configuration of wireless handwheel** window. The menu item is in the **Machine settings** group of the **Settings** application.

Areas of the Configuration of wireless handwheel window

Configuration area

In the **Configuration** area, the control displays different types of information about the connected wireless handwheel, such as the serial number.

Statistics area

In the **Statistics** area, the control displays information about the transmission quality.

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

A high value under **Max. successive lost** is an indication of a limited quality of reception. If the control repeatedly displays values greater than 2 during normal operation of the wireless handwheel within the desired range of use, there is a high risk of undesired disconnection.

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

Further information: "Setting the radio channel", Page 428

Further information: "Selecting the transmission power", Page 427

Status area

In the **Status** area, the control displays the current status of the handwheel, such as **HANDWHEEL ONLINE** and pending error messages concerning the connected handwheel.

19.3.1 Assigning a handwheel to a handwheel holder

In order to assign a handwheel to a handwheel holder, the handwheel holder must be connected to the control hardware.

To assign a handwheel to a handwheel holder:

- ▶ Place the handwheel into the handwheel holder



- ▶ Select the **Home** operating mode



- ▶ Select the **Settings** application



- ▶ Select the **Machine settings** group



- ▶ Double-tap or double-click the **Set up wireless handwheel** menu item
- > The control opens the **Configuration of wireless handwheel** window.
- ▶ Select the **Connect HW** button
- > The control saves the serial number of the inserted wireless handwheel and displays it in the configuration window to the left of the **Connect HW** button.
- ▶ Select the **END** button
- > The control saves the configuration.

19.3.2 Selecting the transmission power

If you reduce the transmission power, the range of the wireless handwheel will decrease.

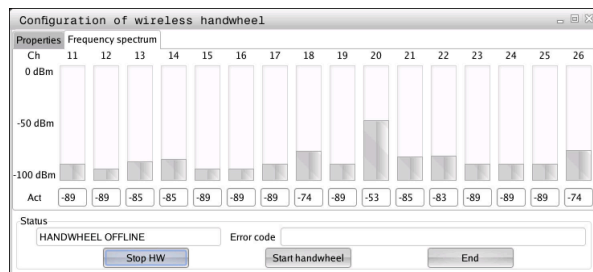
To set the transmission power of the handwheel:



- ▶ Open the **Configuration of wireless handwheel** window
- ▶ Select the **Set power** button
- > The control displays the three available power settings.
- ▶ Select the desired transmission power setting
- ▶ Select the **END** button
- > The control saves the configuration.

19.3.3 Setting the radio channel

If the wireless handwheel is started automatically, then the control tries to select the radio channel providing the best radio signal.



To set the radio channel manually:



- ▶ Open the **Configuration of wireless handwheel** window
- ▶ Select the **Frequency spectrum** tab
- ▶ Select the **Stop HW** button
- ▶ The control stops the connection to the wireless handwheel and determines the current frequency spectrum for all 16 available channels.
- ▶ Note the number of the channel with the least amount of radio traffic



The smallest bar indicates the channel with the least amount of radio traffic.

- ▶ Select the **Start handwheel** button
- ▶ The control restores the connection to the wireless handwheel.
- ▶ Select the **Properties** tab
- ▶ Select the **Select channel** button
- ▶ The controls shows all available channel numbers.
- ▶ Select the number of the channel with the least amount of radio traffic
- ▶ Select the **END** button
- ▶ The control saves the configuration.

19.3.4 Reactivating the handwheel

To reactivate the handwheel:



- ▶ Open the **Configuration of wireless handwheel** window
- ▶ Use the **Start handwheel** button to reactivate the wireless handwheel
- ▶ Select the **END** button

20

Touch Probes

20.1 Setting up touch probes

Application

The **Device configuration** window allows you to create and manage all the workpiece and tool touch probes of the control.

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

Related topics

- Creating a workpiece touch probe with cable or infrared transmission by using the touch probe table
Further information: "Touch probe table tchprobe.tp", Page 392
- Creating a tool touch probe with cable or infrared transmission by using the machine parameter **CfgTT** (no. 122700)
Further information: "Machine parameters", Page 492

Description of function

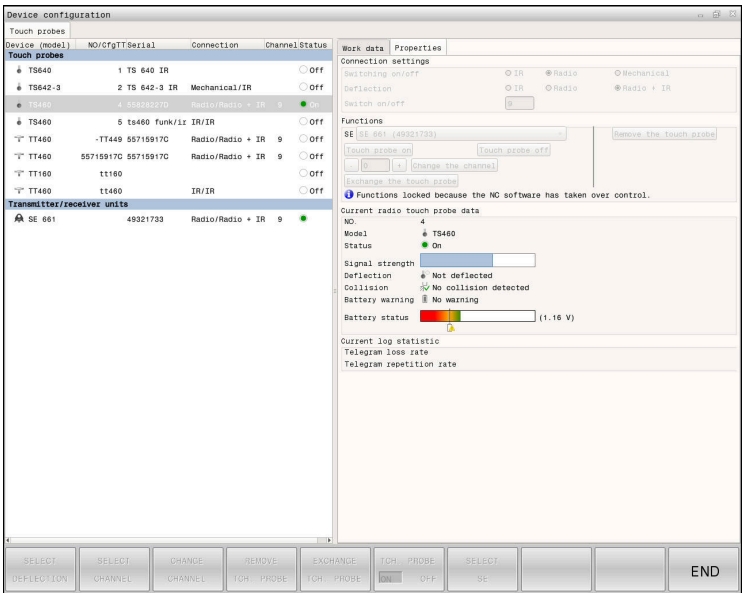
You open the **Device configuration** window in the **Machine settings** group of the **Settings** application. Double-tap or double-click the **Set up touch probes** menu item.

Further information: "Settings Application", Page 445

Touch probes with radio transmission can be created and managed only in the **Device configuration** window.

In order for the control to recognize the touch probe with radio transmission, you will require an **SE 661** transceiver with EnDat interface.

You define the new values in the **Work data** area.



Areas of the Device configuration window

Touch probes area

In the **Touch probes** area, the control displays all of the defined workpiece and tool touch probes, as well as the transceiver units. All other areas provide detailed information about the selected entry.

Work data area

For a workpiece touch probe, the control displays the values from the touch probe table in the **Work data** area.

For a tool touch probe, the control displays the values from the machine parameter **CfgTT** (no. 122700).

You can select and edit the displayed values. Under **Touch probes**, the control displays information about the active value (e.g., selection options). You can change the values of the tool touch probes only after entering the code number 123.

Properties area

In the **Properties** area, the control displays the connection data and diagnostic functions.

For touch probes with radio connection, the control displays the following information in **Current radio touch probe data**:

Display	Meaning
NO.	Number in the touch probe table
Model	Type of touch probe
Status	Touch probe active or inactive
Signal strength	Display of the signal strength in the bar graphic The control shows the currently best-known connection as a complete bar
Deflection	Stylus deflected or not deflected
Collision	Collision or no collision recognized
Battery status	Display of the battery quality If the charge is less than the displayed bar, then the control outputs a warning.

The **Switching on/off** connection setting is preset based on the type of touch probe. Under **Deflection**, you can select how the touch probe is to transmit the signal when probing.

Deflection	Meaning
IR	Infrared probe signal
Radio	Radio probe signal
Radio + IR	The control selects the probe signal



If you activate the touch probe's radio connection by using the connection setting **Switch on/off**, then the signal will be retained even after a tool change. You need to use this connection setting to deactivate the radio connection.

Buttons

The control provides the following buttons:

Button	Function
CREATE TS ENTRY	Create a new workpiece touch probe You define the new values in the Work data area.
CREATE TT ENTRY	Create a new tool touch probe You define the new values in the Work data area.
SELECT DEFLECTION	Select the probe signal
SELECT CHANNEL	Select the radio channel Select the channel with the best radio transmission and pay attention to overlaps with other machines or wireless handwheels.
CHANGE CHANNEL	Change the radio channel
REMOVE TCH. PROBE	Delete the touch probe data The control deletes the entry from the Device configuration window and from the touch probe table or the machine parameters.
EXCHANGE TCH. PROBE	Save a new touch probe in the current row The control automatically overwrites the serial number of the replaced touch probe with the new number.
SELECT SE	Select the SE transceiver
SELECT POWER	Select the strength of the infrared signal You only need to change the signal strength if there is interference.
SELECT POWER	Select the strength of the radio signal You only need to change the signal strength if there is interference.

Note

In the machine parameter **CfgHardware** (no. 100102), the machine manufacturer defines whether the control will show or hide the touch probes in the **Device configuration** window. Refer to your machine manual.

21

**Embedded
Workspace
and Extended
Workspace**

21.1 Embedded Workspace (option 133)

Application

You use Embedded Workspace to operate a Windows PC and display its screen contents on the control's user interface. You use Remote Desktop Manager (option 133) to connect the Windows PC.

Related topics

- Remote Desktop Manager (option 133)

Further information: "Remote Desktop Manager window (option 133)", Page 478

- Using Extended Workspace to operate a Windows PC through an additional connected monitor

Further information: "Extended Workspace", Page 436

Requirements

- Established RemoteFX connection to the Windows PC through Remote Desktop Manager (option 133)
- Connection defined in the machine parameter **CfgRemoteDesktop** (no. 133500)
In the optional machine parameter **connections** (no. 133501), the machine manufacturer enters the name of the RemoteFX connection.
Refer to your machine manual.

Description of function

Embedded Workspace is available on the control as an operating mode and as a workspace. If the machine manufacturer does not define a name, then the operating mode and workspace are both named **RDP**.

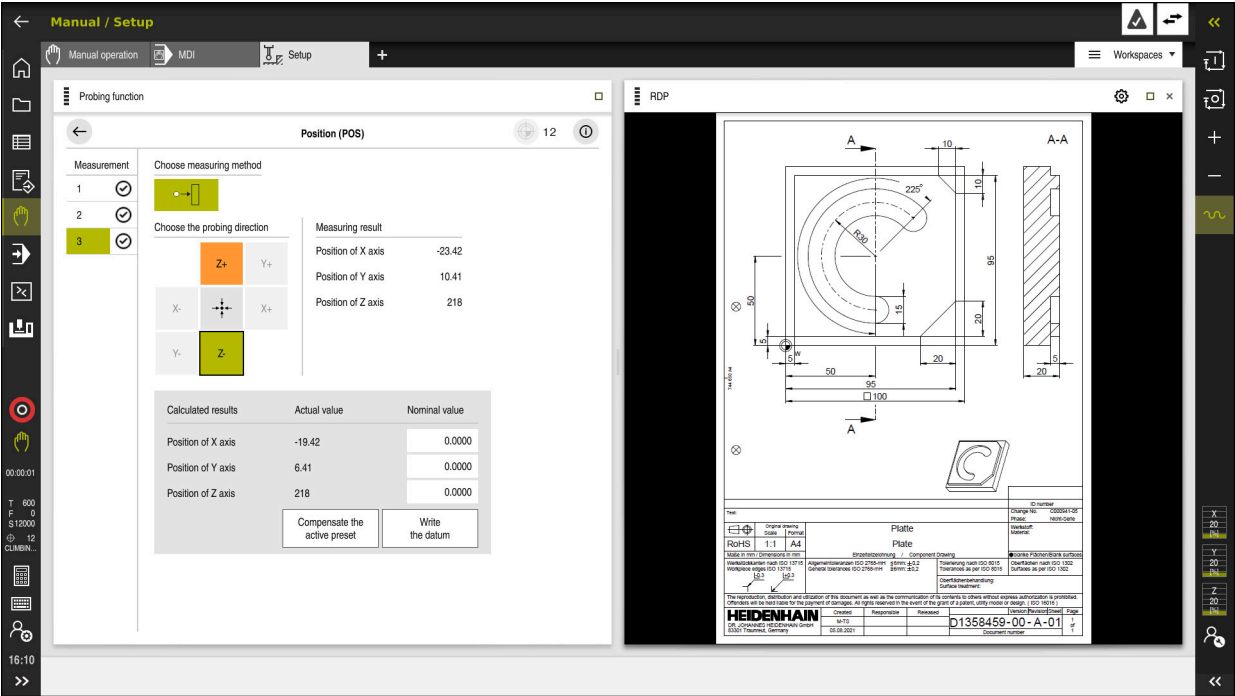
Entries cannot be made through the Windows PC as long as the RemoteFX connection is active. This avoids the problem of conflicting operation.

Further information: "Windows Terminal Service (RemoteFX)", Page 479

If you open Embedded Workspace as an operating mode, the control displays a full-screen version of the Windows PC user interface in it.

If you open Embedded Workspace as a workspace, you can change the size and position of the workspace as you wish. The control rescales the user interface of the Windows PC after each modification.

Further information: "Workspaces", Page 66



Embedded Workspace as workspace with opened PDF file

RDP settings window

If Embedded Workspace is open as a workspace, you can open the **RDP settings** window.

The **RDP settings** window contains the following buttons:

Button	Meaning
Reconnect	<p>If the control could not establish a connection to the Windows PC, for example due to a timeout, press this button to try again.</p> <p>The control can also display this button in the operating mode and the workspace.</p>
Adjust resolution	<p>With this button the control rescales the user interface of the Windows PC to the size of the workspace.</p>

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 (option 133)

Further information: "Embedded Workspace (option 133)", Page 434

- ITC hardware expansion

Further information: "Hardware enhancements", Page 61

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 498

- Displaying and operating computers connected through Remote Desktop Managers (option 133)

Further information: "Remote Desktop Manager window (option 133)", Page 478

22

**Integrated
Functional Safety
(FS)**

Application

The safety concept of integrated functional safety (FS) for machines with HEIDENHAIN controls offers supplementary software safety functions in addition to the mechanical safety features of the machine. For example, the integrated safety concept automatically reduces the feed rate when you perform operations with open guard doors. The machine manufacturer can modify or expand the FS safety concept.

Requirements

- Software option 160 (Integrated functional safety (FS), basic version) or software option 161 (Integrated functional safety (FS), full version)
- Software options 162 to 166 or software option 169, if necessary
Whether you need these software options depends on the machine's number of drives.
- The machine manufacturer must adapt the FS safety concept to the machine.

Description of function

Every machine tool user is exposed to certain risks. While protective devices can prevent access to dangerous locations, the user must also be able to work on the machine without this protection (e.g. guard door opened).

Safety functions

To ensure that the requirements for operator protection are met, integrated functional safety (FS) provides standardized safety functions. The machine manufacturer uses the standardized safety functions for implementing functional safety (FS) for the machine in question.

You can track the active safety functions in the axis status of functional safety (FS).

Further information: "Axis status menu item", Page 441

Description	Meaning	Short description
SS0, SS1, SS1D, SS1F, SS2	Safe Stop	Safe stopping of drives using different methods
STO	Safe Torque Off	The power supply to the motor is interrupted. Provides protection against unexpected start of the drives
SOS	Safe Operating Stop	Safe operating stop. Provides protection against unexpected start of the drives
SLS	Safely Limited Speed	Safely limited speed. Prevents the drives from exceeding the specified speed limits when the protective door is opened
SLP	Safely Limited Position	Safely limited position. Monitors safe axes to keep them within the limit values of a defined area
SBC	Safe Brake Control	Dual-channel control of the motor holding brakes

Safety-related operating modes of functional safety (FS)

Functional safety (FS) of a control offers various safety-related operating modes. The safety-related operating mode with the lowest number has the highest safety level.

Depending on how the machine manufacturer implements them, the following safety-related operating modes are available:



Refer to your machine manual.

The machine manufacturer must adapt the safety-related operating modes to each machine.


Icon	Safety-related operating mode	Short description
SOM 1	Operating mode SOM_1	Safe operating mode 1: Automatic mode, production mode
SOM 2	Operating mode SOM_2	Safe operating mode 2: Set-up mode
SOM 3	Operating mode SOM_3	Safe operating mode 3: Manual intervention; only for qualified operators
SOM 4	Operating mode SOM_4 This function must be enabled and adapted by the machine manufacturer.	Safe operating mode 4: Advanced manual intervention, process monitoring, only for qualified users

Functional safety (FS) in the Positions workspace

On a control with functional safety (FS), the control displays the monitored operating statuses of the speed **S** and feed rate **F** in the **Positions** workspace. If a safety function is triggered while in a monitored state, the control stops the feed movement and the spindle or reduces the speed (e.g., if a guard door is opened).

Further information: "Axis display and position display", Page 94

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.

Start/Login Settings Help FS Functional safety Workspaces

Overview

DS ID	Key name	Accepted	CRC	Active
59	CfgSafety	✗	0xdfe9682f	✓
60	CfgPtcSafety	✗	0x77c03a9b	✓
58	CfgAuParSafety HSE-V9_X_K00_E00	✗	0x96765968	✓
62	CfgMotParSafety HSE-V9_X_K00_E00	✗	0x55e79e2b	✓
85	CfgAuParSafety HSE-V9_Y_K00_E00	✓	0xd43e109f	✓
64	CfgMotParSafety HSE-V9_Y_K00_E00	✓	0x42531a0	✓
65	CfgAuParSafety HSE-V9_Z_K00_E00	✓	0xd8299386	✓
66	CfgMotParSafety HSE-V9_Z_K00_E00	✓	0x99fa2d8	✓
67	CfgAuParSafety HSE-V9_B_K00_E00	✓	0x498b3c9e	✓
68	CfgMotParSafety HSE-V9_B_K00_E00	✓	0x2ca6d1d3	✓
69	CfgAuParSafety HSE-V9_C_K00_E00	✓	0xd8a0584a9	✓
70	CfgMotParSafety HSE-V9_C_K00_E00	✓	0x3f20c9d	✓
71	CfgAuParSafety HSE-V9_U_K00_E00	✓	0x4a21405b	✓
72	CfgMotParSafety HSE-V9_U_K00_E00	✓	0xd68f5508	✓

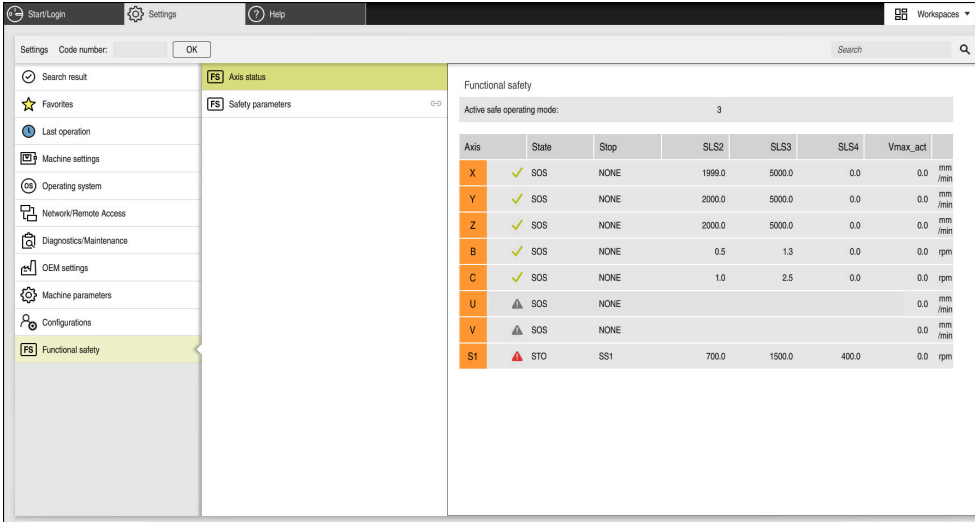
FS config. overview

Functional safety application

Axis status menu item

In the **Axis status** menu item of the **Settings** application, the control provides the following information about the status of the individual axes:

Field	Meaning
Axis	Configured axes of the machine
State	Active safety function
Stop	Stop reaction Further information: "Functional safety (FS) in the Positions workspace", Page 439
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 screenshot shows the 'Settings' application with the 'Functional safety' menu item selected. The 'Axis status' sub-menu is active, displaying a table of functional safety parameters. The table includes columns for Axis, State, Stop, SLS2, SLS3, SLS4, and Vmax_act. The 'Active safe operating mode' is set to 3. The table lists parameters for axes X, Y, Z, B, C, U, V, and S1.

Axis	State	Stop	SLS2	SLS3	SLS4	Vmax_act	
X	✓ SOS	NONE	1999.0	5000.0	0.0	0.0	mm/min
Y	✓ SOS	NONE	2000.0	5000.0	0.0	0.0	mm/min
Z	✓ SOS	NONE	2000.0	5000.0	0.0	0.0	mm/min
B	✓ SOS	NONE	0.5	1.3	0.0	0.0	rpm
C	✓ SOS	NONE	1.0	2.5	0.0	0.0	rpm
U	⚠ SOS	NONE				0.0	mm/min
V	⚠ SOS	NONE				0.0	mm/min
S1	⚠ STO	SS1	700.0	1500.0	400.0	0.0	rpm

Axis status menu item in the **Settings** application

Test status of the axes




In order for the control to ensure safe operation of the axes, it checks all monitored axes when the machine is switched on.

The control checks whether the position of an axis matches the position directly after shutdown. If a deviation is detected, the control marks the respective axis in the position display with a red warning triangle.


If checking of individual axes fails when starting the machine, you can check the axes manually.

Further information: "Checking axis positions manually", Page 443

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 443
	The axis is not monitored by functional safety (FS) or is not configured as a safe axis.


Feed-rate limiting with functional safety (FS)



Refer to your machine manual.
This function must be adapted by your machine manufacturer.

With the **FMAX** button you can prevent the SS1 reaction for safe stopping of drives when the guard door is opened.

By pressing the **FMAX** button, the control limits the speed of the axes and of the spindle(s) to the values defined by the machine manufacturer. The limitation depends on the active safety-related SOM_x operating mode. You can select the safety-related operating mode with the keylock switch.



In the safety-related operating mode SOM_1, the control stops the axes and spindles when the guard door is opened.

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 **Approach test position**
- ▶ The control displays the axes that have not been tested in the **Positions** workspace.
- ▶ Select the desired axis in the **Positions** workspace
- ▶ Press the **NC start** key



- ▶ The axis moves to the test position.
- ▶ After the test position has been reached, the control issues a message.
- ▶ Press the **permissive button** on the machine operating panel
- ▶ The control displays the axis as a tested axis.

NOTICE

Danger of collision!

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

- ▶ If necessary, move to a safe position before approaching the test positions
- ▶ Watch out for possible collisions

Notes



- Machine tools with HEIDENHAIN controls may be equipped with integrated functional safety (FS) or with external safety. This chapter refers exclusively to machines with integrated functional safety (FS).
- The machine manufacturer defines the behavior of speed-controlled FS-NC axes while the guard door is open in the machine parameter **speedPosCompType** (no. 403129). The machine manufacturer can e.g. allow switching-on of the spindle and thus enable scratching of the workpiece while the guard door is open. Refer to your machine manual.





23


Settings Application

23.1 Overview

The **Settings** application includes the following groups with menu items:

Icon	Category	Menu item
	Machine settings	<ul style="list-style-type: none"> ■ Machine settings Further information: "Machine settings menu item", Page 448 ■ General information Further information: "General information menu item", Page 450 ■ SIK Further information: "SIK menu item", Page 451 ■ Machine times Further information: "Machine times menu item", Page 452 ■ Set up touch probes Further information: "Setting up touch probes", Page 430 ■ Set up wireless handwheel Further information: "HR 550FS wireless handwheel", Page 423
	Operating system	<ul style="list-style-type: none"> ■ PKI Admin Manage certificates for the control (e.g., for OPC UA NC Server) Further information: "OPC UA NC Server (options 56 to 61)", Page 465 ■ Date/Time Further information: "Adjust system time window", Page 453 ■ Language/Keyboards Further information: "Conversational language of the control", Page 453 ■ About HEROS Further information: "Information on licensing and use", Page 55 ■ SELinux Further information: "SELinux security software", Page 455 ■ UserAdmin Currently no function ■ Current User Currently no function ■ Touchscreen configuration You can choose the touchscreen's sensitivity and whether touchpoints should be shown or hidden.

Icon	Category	Menu item
	Network/Remote Access	<ul style="list-style-type: none"> ■ Shares Further information: "Network drives on the control", Page 456 ■ Network Further information: "Ethernet interface", Page 459 ■ Printer Further information: "Printers", Page 471 ■ DNC Further information: "DNC menu item", Page 470 ■ OPC UA Further information: "OPC UA NC Server (options 56 to 61)", Page 465 ■ VNC Further information: "VNC menu item", Page 474 ■ Remote Desktop Manager Further information: "Remote Desktop Manager window (option 133)", Page 478 ■ Real VNC Viewer Define settings for external software accessing the control (e.g., for maintenance purposes); for network specialists ■ Firewall Further information: "Firewall", Page 484
	Diagnostics/Maintenance	<ul style="list-style-type: none"> ■ Terminal program Enter and execute console commands ■ HeLogging Define settings for internal diagnostic files ■ Portscan Further information: "Portscan", Page 487 ■ perf2 Check processor load and process load ■ RemoteService Further information: "Remote servicing", Page 488 ■ NC/PLC Restore Further information: "Backup and restore", Page 489 ■ TNCdiag Further information: "TNCdiag", Page 491 ■ NC/PLC Backup Further information: "Backup and restore", Page 489 ■ Touchscreen cleaning The control disables the touchscreen for 90 seconds.
	OEM settings	Settings for the machine manufacturer
	Machine parameters	<p>The group contains machine parameters that can be edited, depending on your rights (e.g., MPs for setters).</p> <p>Further information: "Machine parameters", Page 492</p>

Icon	Category	Menu item
	Functional safety	<ul style="list-style-type: none"> ■ Axis status Further information: "Axis status menu item", Page 441 ■ Safety parameters Further information: "Functional safety application", Page 440

23.2 Code numbers


Application

The top part of the **Settings** application contains the **Code number:** input field and the **unit of measure** switch. These settings are possible and accessible from every group.

Description of function

You can enable the following functions or areas with code numbers:

Code number	Function
123	Editing machine-specific user parameters Further information: "Machine parameters", Page 492
555343	Special functions for programming with variables Further information: User's Manual for Programming and Testing



The control indicates whether the caps lock key is pressed during entry. This helps to avoid incorrect entries.

23.3 Machine settings menu item

Application

In the **Machine settings** menu item of the **Settings** application, you can define the settings for simulation and program run.

Related topics

- Graphic settings for simulation
Further information: User's Manual for Programming and Testing

Description of function

Unit of measure area

In the **Unit of measure** area you can choose between mm and inch.

- Metric system: e.g. X = 15.789 (mm), the value is displayed to 3 decimal places
- Inch system: e.g. X = 0.6216 (inches), the value is displayed to 4 decimal places

If the display in inches is active, the control also displays the feed rate in inches/min. In an inch-based program, you must multiply the feed rate by 10.

Channel settings

The control displays the channel settings separately for the **Editor** operating mode and the **Manual** and **Program Run** operating modes.

You can define the following settings:

Setting	Meaning
Active kinematics	<p>Use the Active kinematics function to change the kinematics model for the machine and the simulation. This way you can test NC programs that, for example, are programmed for other machines.</p> <p>The control offers a selection menu with all available kinematics models. The machine manufacturer defines which kinematics models you can choose.</p> <p>The control displays the active kinematics model in the Machine mode of the Simulation workspace.</p>
Generate tool-usage file	<p>The control uses the tool-usage file to check tool usage.</p> <p>Further information: "Tool usage test", Page 172</p> <p>You select when the control should generate a tool-usage file:</p> <ul style="list-style-type: none"> ■ Never The control does not generate a tool-usage file. ■ Once The next time you simulate or run an NC program, the control will generate a tool-usage file once. ■ Always When you simulate or run an NC program, the control will generate a tool-usage file each time.

Traverse limits

Use the **Traverse limits** function to limit the possible traverse path of an axis. You can define traverse limits for each axis (e.g., to protect an indexing head from collision).

The **Traverse limits** function consists of a table with the following contents:

Column	Meaning
Axis	The TNC displays each axis of the active kinematics model in a row.
Status	If you have defined one or both limits, the control displays the contents Valid or Invalid .
Lower limit	You define the lower traverse limit of the axis in this column. You can enter up to four decimal places.
Upper limit	You define the upper traverse limit of the axis in this column. You can enter up to four decimal places.

The defined traverse limits are valid across power cycles of the control, until you delete all values from the table.

The following general conditions apply to the traverse limit values:

- The lower limit must be smaller than the upper limit.
- The upper and lower limit may not both equal 0.

Other conditions apply to traverse limits for modulo axes.

Further information: User's Manual for Programming and Testing

Notes

NOTICE

Danger of collision!

You can also select any stored kinematics model as the active machine kinematics. The control then executes all manual movements and machining operations using the selected kinematics. All subsequent axis movements pose a risk of collision!

- ▶ Use the **Active kinematics** function for the simulation only
- ▶ Use the **Active kinematics** function for selecting the active machine kinematics only if required

- In the optional machine parameter **enableSelection** (no. 205601), the machine manufacturer defines for each kinematics model whether the **Active kinematics** function can be selected.
- You can open the tool-usage file in the **Tables** operating mode.
Further information: "Tool usage file", Page 398
- If the control has generated a tool-usage file for an NC program, then there are contents in the **T usage order** and **Tooling list** tables (option 93).
Further information: "T usage order (option 93)", Page 400
Further information: "Tooling list (option 93)", Page 402

23.4 General information menu item

Application

In the **General information** menu item of the **Settings** application, the control provides the information about the control and the machine.

Description of function

Version information area

The control displays the following information:

Sub-area	Meaning
HEIDENHAIN	<ul style="list-style-type: none">■ Control model Designation of the control (managed by HEIDENHAIN)■ NC-SW Number of the NC software (managed by HEIDENHAIN)■ NCK Number of the NC software (managed by HEIDENHAIN)
PLC	PLC-SW Number or name of the PLC software (managed by the machine manufacturer)

The machine manufacturer can add further software numbers (e.g., that of a connected camera).

Info about machine manufacturer area

The control shows the contents of the optional machine parameter **CfgOemInfo** (no. 131700). The control displays this area only if the machine manufacturer defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 466

Machine information area (options 56 to 61)

The control shows the contents of the optional machine parameter **CfgMachineInfo** (no. 131600, options 56 to 61). The control displays this area only if the machine operator defines this machine parameter.

Further information: "Machine parameters in conjunction with OPC UA", Page 466

23.5 SIK menu item

Application

Use the **SIK** menu item of the **Settings** application to view control-specific information (e.g., the serial number and the available software options).

Related topics

- Software options on the control

Further information: "Software options", Page 48

Description of function

SIK information area

The control displays the following information:

- **Serial number**
- **Control model**
- **Performance class**
- **Features**
- **Status**

OEM key area

In the **OEM key** area the machine manufacturer can define a manufacturer-specific password for the control.

General key area

In the **General key** area the machine manufacturer can enable all software options once for a period of 90 days (e.g., for testing).

The control indicates the status of the general key:

Status	Meaning
NONE	The general key has not yet been used for this software version.
dd.mm.yyyy	Date up to which all software options will be available. Once the general key has expired, it cannot be used again.
EXPIRED	The general key has expired for this software version.

If the software version of the control is increased (e.g., by an update), then the **General key** can be used again.


Software options area

In the **Software options** area the control shows all available software options in a table.

Column	Meaning
#	Number of the software option
Option	Name of the software option
Expiration date	The machine manufacturer can enable software options for a limited time. In this case the control shows in this column the date through which the software option is available.
	The machine manufacturer uses the Set button to enable a software option. For enabled software options the control displays the text Enabled .

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** at the end of the row.

Definition

Abbreviation	Definition
SIK (System Identification Key)	SIK is the designation of the plug-in board for the control hardware. Each control can clearly be identified by the serial number of the SIK .

23.6 Machine times menu item

Application

In the **Machine times** menu item of the **Settings** application, the control shows the run times since initial setup.

Related topics

- Date and time of the control
Further information: "Adjust system time window", Page 453

Description of function

The control displays the following machine times:

Machine time	Meaning
Control on	Run time of the control since being put into service
Machine on	Run time of the machine tool since being put into service
Program Run	Run time of all program runs since being put into service



Refer to your machine manual.

The machine manufacturer can define up to 20 additional run times.

23.7 Adjust system time window

Application

In the **Adjust system time** window you can set the time zone, date and time manually or by means of an NTP server synchronization.

Related topics

- Run times of the machine tool

Further information: "Machine times menu item", Page 452

Description of function

The **Date/Time** menu item opens the **Adjust system time** window. The menu item is in the **Operating system** group of the **Settings** application.

The **Adjust system time** window consists of the following areas:

Area	Function
Set the time manually	Activate this check box to define the following data: <ul style="list-style-type: none"> ■ 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.

23.8 Conversational language of the control

Application

You use the **helocale** window to change the conversational language of the HEROS operating system and the machine parameters to change the NC conversational language of the control's user interface.

The HEROS conversational language only changes after a restart of the control.

Related topics

- Machine parameters of the control
 Further information: "Machine parameters", Page 492

Description of function

You can't define two different conversational languages for the operating system and control.

The **Language/Keyboards** menu item opens the **helocale** window. The menu item is in the **Operating system** group of the **Settings** application.

The **helocale** window consists of the following areas:

Area	Function
Language	Choose the HEROS conversational language from a selection menu Only if the machine parameter applyCfgLanguage (no. 101305) is defined as FALSE .
Keyboards	Select the language layout of the keyboard for HEROS functions

23.8.1 Changing the language

By default, the control assumes the NC conversational language for the HEROS conversational language.

To change the NC conversational language:

- ▶ Select the **Settings** application
- ▶ Enter the code number 123
- ▶ Select **OK**
- ▶ Select **Machine parameters**
- ▶ Double-tap or double-click **MPs for setters**
- > The control opens the **MPs for setters** application.
- ▶ Navigate to the machine parameter **ncLanguage** (no. 101301)
- ▶ Select the desired language
 - ▶ Select **Save**
 - > The control opens the **Configuration data changed. All changes.** window.
 - ▶ Select **Save**
 - > The control opens the notification menu and displays a "Question type" error.
 - ▶ Select **CLOSE CONTROL**
 - > The control restarts.
 - > Once the control has restarted, the NC conversational language and the HEROS conversational language are changed.



Note

Use the machine parameter **applyCfgLanguage** (no. 101305) to define whether the control assumes the setting for the NC conversational language for the HEROS conversational language.

- **TRUE** (default): The control assumes the NC conversational language. You can change the language only in the machine parameters.

Further information: "Changing the language", Page 454

- **FALSE:** The control assumes the HEROS conversational language. You can change the language only in the **helocale** window.

23.9 SELinux security software

Application

SELinux is an extension for Linux-based operating systems in the sense of Mandatory Access Control (MAC). The security software protects the system against the execution of unauthorized processes or functions, i.e. viruses and other malicious software.

The machine manufacturer defines the **SELinux** settings in the **Security Policy Configuration** window.

Related topics

- Security settings with firewall

Further information: "Firewall", Page 484

Description of function

The **SELinux** menu item opens the **Security Policy Configuration** window. The menu item is in the **Operating system** group of the **Settings** application.

The access control of **SELinux** is regulated as follows by default:

- The control executes only programs that are installed with the HEIDENHAIN NC software.
- Only explicitly selected programs can modify safety-relevant files, such as **SELinux** system files or HEROS boot files.
- New files created by other programs may not be run.
- USB data carriers can be deselected.
- Only two processes can run new files:
 - Software update: A software update from HEIDENHAIN can replace or modify system files.
 - SELinux configuration: The configuration of **SELinux** with the **Security Policy Configuration** window is usually password-protected by the machine manufacturer (refer to the relevant machine manual).

Note

HEIDENHAIN recommends using **SELinux** as additional protection against attacks from outside the network.

Definition

Abbreviation	Definition
MAC (mandatory access control)	MAC means that the control performs only explicitly permitted actions. SELinux is intended as protection in addition to the normal access restriction in Linux. Certain processes and actions can be performed only if the standard functions and access control of SELinux permit it.

23.10 Network drives on the control

Application

Use the **Mount Setup** window to connect network drives to the control. If a network drive is connected to the control, the control displays additional drives in the navigation column of the file management.

Related topics

- File management
Further information: User's Manual for Programming and Testing
- Network settings
Further information: "Ethernet interface", Page 459

Requirements

- Existing network connection
- Control and computer in same network
- Path and access data of drive to be connected are known

Description of function

The **Shares** menu item opens the **Mount Setup** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

You can also open the window with the **Mount network share** button of the **Files** operating mode.

Further information: User's Manual for Programming and Testing

You can define any number of network drives, but only seven can be connected at a time.

Network drive area

In the **Network drive** area, the control shows a list of all defined network drives, as well as the status of each drive.

The control displays the following buttons:

Button	Meaning
Mount	Connect a network drive The control selects the check box in the Mount column if an active connection exists.
Unmount	Disconnect a network drive
Auto	Automatically connect the network drive when the control is booting. The control selects the check box in the Auto column if an active automatic connection exists.
Add	Define a new connection Further information: "Mount assistant window", Page 458
Remove	Delete an existing connection
Copy	Copy connection Further information: "Mount assistant window", Page 458
Edit	Edit the connection settings Further information: "Mount assistant window", Page 458
Private network drive	User-specific connection if user administration is active The control selects the check box in the Privat column if a user-specific connection exists.

Status Log area

In the **Status Log** area, the control shows status information and error messages about connections.

Use the **Clear** button to delete the contents of the **Status Log** area.

Mount assistant window

In the **Mount assistant** window you define the settings for a connection with a network drive.

The **Add**, **Copy** and **Edit** buttons open the **Mount assistant** window.

The **Mount assistant** window contains tabs with the following settings:

Tab	Setting
Drive name	<ul style="list-style-type: none">■ Drive name: Network drive name in the file management of the control The control permits only capitals, with a colon (:) at the end.■ Private network drive When user administration is active, the connection is only visible to the user who created it.
Share type	Transfer protocol <ul style="list-style-type: none">■ Windows share (CIFS/SMB) or Samba server■ UNIX share (NFS)
Server and Share	<ul style="list-style-type: none">■ 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)	<ul style="list-style-type: none">■ Single Sign On When user administration is active, the control automatically connects an encrypted network drive when the user logs in.■ Windows user name:■ Ask for password? (not possible with the "Connect automatically" option) Select whether a password is required upon connecting.■ Password■ Password verification
Mounting options	Parameters for mount option "-o": Auxiliary parameters for the connection Further information: "Examples of Mounting options", Page 459
Check	The control displays a summary of the defined settings. You can check the settings and save them with Apply .

Examples of Mounting options

Enter options without a space, only separated by a comma

Options for SMB

Example	Meaning
domain=xxx	Name of the domain HEIDENHAIN recommends not to write the domain into the user name, but as an option.
vers=2.1	Protocol version

Options for NFS

Example	Meaning
rsz=8192	Packet size in bytes for data reception Input: 512...8192
wsz=4096	Packet size in bytes for data transmission Input: 512...8192
soft,timeo=3	Conditional Mount Time in tenths of a second after which the control will try to connect again
sec=ntlm	Authentication method ntlm Use this option if the control displays the Permission denied error message upon connecting.
nfsvers=2	Protocol version

Notes

- Have a network specialist configure the control.
- To avoid security gaps, prefer the current versions of the **SMB** and **NFS** protocols.

23.11 Ethernet interface

Application

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

Related topics

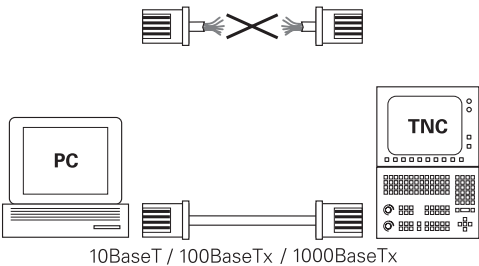
- Firewall settings
Further information: "Firewall", Page 484
- Network drives on the control
Further information: "Network drives on the control", Page 456
- External access
Further information: "DNC menu item", Page 470

Description of function

The control transfers data via the Ethernet interface using the following protocols:

- **CIFS** (common internet file system) or **SMB** (server message block)
The control supports versions 2, 2.1 and 3 of these protocols.
- **NFS** (network file system)
The control supports versions 2 and 3 of this protocol.

Connection options



You can integrate the Ethernet interface of the control into the network or connect it directly to a PC through the RJ45 connection X26. The connection is electrically isolated from the control electronics.

Use a Twisted Pair cable to connect the control to your network.

The maximum cable length permissible between the control and a node depends on the quality grade of the cable, the sheathing, and the type of network.

Ethernet connection icon

Icon	Meaning
	<p>Ethernet connection</p> <p>The control displays the icon at the bottom right in the taskbar.</p> <p>Further information: "Taskbar", Page 501</p> <p>When you click the icon, the control opens a pop-up window. The pop-up window contains the following information and functions:</p> <ul style="list-style-type: none">■ Connected networks You can disconnect the network connection. Select the network name to reconnect.■ Available networks■ VPN connections <p>Currently no function</p>

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.

23.11.1 Network settings window

Application

In the **Network settings** window you define the settings for the control's Ethernet interface.

 Have a network specialist configure the control.

Related topics

- Network configuration
Further information: "Network configuration with Advanced Network Configuration", Page 508
- Firewall settings
Further information: "Firewall", Page 484
- Network drives on the control
Further information: "Network drives on the control", Page 456

Description of function

Open the **Network settings** window via the **Network** menu item. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Network settings

Status

Interfaces

DHCP server

Ping/Routing

SMB release

Computer name

DE01PC23172-340595

Default gateway

10.7.127.254 on eth0

☒ Use proxy


Address:Port

Interfaces

Name	Connection	Connection status	Configuration name	Address
eth0	X26	CONNECTED	DHCP	10.7.113.38
eth1	X116	CONNECTED	DHCP-VBoxHostOnly_eth1	192.168.56.113

DHCP client

Name	IP address	MAC address	Type	Valid up to
------	------------	-------------	------	-------------

 The network interface does not have a static IP configuration.
The DHCP server will not be started.

OK

Apply

OEM authorization

Cancel

Network settings window

Status tab

The **Status** tab contains the following information and settings:

Domain	Information or Setting
Computer name	The control displays the name under which the control is visible in the company network. You can change the name.
Default gateway	The control shows the default gateway and the Ethernet interface being used.
Use proxy	You can define the address and the port of a proxy server in the network.
Interfaces	<p>The control shows an overview of available Ethernet interfaces. If there is no network connection, the table is empty.</p> <p>The control displays the following information in the table:</p> <ul style="list-style-type: none"> ■ 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 <p>Further information: "Interfaces tab", Page 463</p>
DHCP client	<p>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.</p> <p>The control displays the following information in the table:</p> <ul style="list-style-type: none"> ■ Name Host name and connection status of the device. The control shows the following connection status: <ul style="list-style-type: none"> ■ Green: Connected ■ Red: No connection ■ IP address Dynamically assigned network address of the device ■ MAC address Physical address of the device ■ Type Type of connection The control displays the following connection types: <ul style="list-style-type: none"> ■ TFTP ■ DHCP ■ Valid up to Time until which the IP address is valid without being renewed <p>The machine manufacturer can make settings for these devices. Refer to your machine manual.</p>

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 switch.
Connection	The control displays the number of the network connection.
Connection status	<p>The control displays the connection status of the Ethernet interface.</p> <p>The following connection statuses may be displayed:</p> <ul style="list-style-type: none"> ■ CONNECTED Connected ■ DISCONNECTED Connection separated ■ CONFIGURING The IP address is being fetched from the server ■ NOCARRIER No cable present
Configuration name	<p>You can execute the following functions:</p> <ul style="list-style-type: none"> ■ Select a profile for the Ethernet interface In the factory default setting, two profiles are available: <ul style="list-style-type: none"> ■ 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 508 ■ Reconnect the Ethernet interface with Reconnect ■ Edit the selected profile Further information: "Network configuration with Advanced Network Configuration", Page 508

The control additionally offers the following functions:

- **Set standard values**

The control resets all settings. The control restores the profiles that are available in the factory default setting.

- **Configuration name**

You can add, edit or remove profiles for the network connection.



If you have changed the profile of an active connection, the control will not update the profile being used. Reconnect the corresponding interface with **Reconnect**.

The control exclusively supports the **Ethernet** connection type.

Further information: "Network configuration with Advanced Network Configuration", Page 508

DHCP server tab

The machine manufacturer can use the **DHCP server** tab in the control to configure a DHCP server in the machine network. Using this server, the control can establish connections with other network components of the machine network, e.g. with industrial computers.

Refer to your machine manual.

Ping/Routing tab

You can check the network connection on the **Ping/Routing** tab.


The **Ping/Routing** tab contains the following information and settings:

Domain	Information or Setting
Ping	<p>Address:Port and Address:</p> <p>You can enter the IP address of the computer and possibly the port number for checking the network connection.</p> <p>Entry: Four numerical values separated by dots, possibly a port number separated by a colon e.g. 10.7.113.10:22</p> <p>As an alternative, you can enter the name of the computer whose connection you want to check.</p> <p>Starting and stopping the test</p> <ul style="list-style-type: none"> ■ Start button: starts the test The control displays status information in the ping field. ■ Stop button: stops the test
Routing	<p>The control displays status information of the operating system about the current routing for network administrators.</p>

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.

Notes

- Preferably restart the control after making changes in the network settings.
- The HEROS operating system manages the **Network settings** window. If you change the HEROS conversational language, you must restart the control for the change to take effect.

Further information: "Conversational language of the control", Page 453

23.12 OPC UA NC Server (options 56 to 61)

23.12.1 Fundamentals

Open Platform Communications Unified Architecture (OPC UA) describes a collection of specifications. These specifications are used to standardize machine-to-machine communication (M2M) in the field of industrial automation. OPC UA enables the data exchange across operating systems between products from different manufacturers, e.g. between a HEIDENHAIN control system and third-party software. Thus, OPC UA has become the data exchange standard for secure, reliable, manufacturer- and platform-independent industrial communication over the last years.

In 2016, the German Federal Office for Information Security (BSI) published a security analysis related to **OPC UA**. The specification analysis performed by the BSI determined that **OPC UA** provides a high level of security as compared to most other industrial protocols.

HEIDENHAIN follows the BSI recommendations and provides SignAndEncrypt, which exclusively features up-to-date IT security profiles. For this purpose, OPC UA-based industrial applications and the **OPC UA NC Server** exchange certificates for authentication. In addition, any transferred data is encrypted. This effectively prevents messages between the communication partners from being intercepted or altered.

Application

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology.

The **OPC UA NC Server** allows you to access the data and functions of the HEIDENHAIN NC information model exposed in the server address space.

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**

Requirements

- OPC UA NC Server (software options 56 to 61)
For OPC UA-based communication, the HEIDENHAIN control provides the **OPC UA NC Server**. For each OPC UA client to be connected, you need one of the six available software options (56 to 61).
- Firewall configured
Further information: "Firewall", Page 484
- OPC UA client supports the **Security Policy** and the authentication method of the **OPC UA NC Server**:
 - **Security Mode:** SignAndEncrypt
 - **Algorithm:** Basic256Sha256
 - **User Authentication:** X509 certificates

Description of function

Both standard and custom software can be used with the **OPC UA NC Server**. Compared to other established interfaces, significantly less development effort is required for OPC UA connection, thanks to the uniform communication technology. The control supports the following OPC UA functions:

- Write and read variables
- Subscribe to value changes
- Run methods
- Subscribe to events
- File system access to the **TNC:** drive
- File system access to the **PLC:** drive (a corresponding permission is required)

Machine parameters in conjunction with OPC UA

The **OPC UA NC Server** enables OPC UA client applications to query general machine information, such as the year of construction of the machine or its location. The following machine parameters are available for the digital identification of your machine:

- For users: **CfgMachineInfo** (no. 131700)
Further information: "Machine information area (options 56 to 61)", Page 451
- For the machine tool builder: **CfgOemInfo** (no. 131600)
Further information: "Info about machine manufacturer area", Page 451

Access to directories

The **OPC UA NC Server** enables read and write access to the **TNC:** and **PLC:** drives. The following actions are permitted:

- Creation and deletion of folders
- Reading, editing, copying, moving, creating, and deleting of files.

While the NC software is running, the files referenced in the following machine parameters are locked against write access:

- Tables referenced by the machine manufacturer in the machine parameter **CfgTablePath** (no. 102500)
- Files referenced by the machine manufacturer in the machine parameter **dataFiles** (no. 106303, branch **CfgConfigData** no. 106300)

The **OPC UA NC Server** enables access to the control even if the NC software is switched off. You can e.g. transfer automatically created service files at any time as long as the operating system is active.

NOTICE

Caution: potential damage to property!

The control does not automatically back up the files before editing or deletion. Files that are missing cannot be restored. The removal or editing of system-relevant files, such as the tool table, can negatively affect the control functions.

- System-relevant files must be edited only by authorized specialists

Required certificates

The **OPC UA NC Server** requires three different types of certificates. The server and the client need two of them, the application instance certificates, in order to establish a secure connection. The third certificate (user certificate) is required for authorization and for starting a session with specific user permissions.

The control automatically generates a two-level certificate chain referred to as the **Chain of Trust** for the server. This certificate chain consists of a self-signed root certificate (including a **revocation list**) and a certificate for the server that is created on the basis of the root certificate.

The client certificate must be added on the **Trusted** tab of the **PKI Admin** function. All other certificates should be added on the **Issuers** tab of the **PKI Admin** function for verification of the entire certificate chain.

User certificate

The control uses the HEROS functions **Current User** or **UserAdmin** for administration of the user certificate. When you initiate a session, the rights of the associated internal user are active.

To assign a user certificate to a user:

- ▶ Open the **Current User** HEROS function
- ▶ Select **SSH keys and certificates**
- ▶ Press the **Import certificate** soft key
- > The control opens a pop-up window.
- ▶ Select the certificate
- ▶ Select **Open**
- > The control imports the certificate.
- ▶ Press the **Use for OPC-UA** soft key

Self-generated certificates

You can also create and import all of the required certificates yourself.

Self-generated certificates must fulfill the following requirements:

- General requirements
 - File format: *.der
 - Signature with hash SHA256
 - Validity period of at most 5 years is recommended
- Client certificates
 - Host name of the client
 - Application URI of the client
- Server certificates
 - Host name of the control
 - Application URI of the server according to the following structure:
urn:<hostname>/HEIDENHAIN/OpcUa/NC/Server
 - Validity period of 20 years maximum

Note

OPC UA is a manufacturer/platform-independent, open communication standard. For this reason, an OPC UA client SDK is not included in the **OPC UA NC Server**.

23.12.2 OPC UA menu item (options 56 to 61)

Application

In the **OPC UA** menu item of the **Settings** application, you can set up the connections to the control and check the status of the OPC UA connections.

Description of function

Select the **OPC UA** menu item in the **Network/Remote Access** group.
The **OPC UA NC Server** area contains the following functions:

Function	Meaning
Status	Shows with an icon whether a connection with the OPC UA NC Server is active: <ul style="list-style-type: none">■ Green icon: connection active■ Gray icon: connection not active or software option not enabled
OPC UA connection wizard	Open the OPC UA NC Server connection wizard window Further information: "OPC UA connection wizard function (options 56 to 61)", Page 468
OPC UA license settings	Open the OPC UA NC Server - License Settings window Further information: "OPC UA license settings function (options 56 to 61)", Page 469
Host computer operation	Activate or deactivate host computer operation with a switch Further information: "DNC area", Page 470

23.12.3 OPC UA connection wizard function (options 56 to 61)

Application

For quick and easy setup of an OPC UA client application, you can use the **OPC UA NC Server connection wizard** window. This assistant guides you through the steps that are required to connect an OPC UA client application to the control.

Related topics

- Assigning an OPC UA client to a software option 56 to 61 with the **OPC UA NC Server - License Settings** window
- Managing certificates with the **PKI Admin** menu

Description of function

The **OPC UA connection wizard** function of the **OPC UA** menu item opens the **OPC UA NC Server connection wizard** window.

Further information: "OPC UA menu item (options 56 to 61)", Page 468

The assistant features the following steps:

- Export **OPC UA NC Server** certificates
- Import the certificates of the OPC UA client application
- Assign each of the available **OPC UA NC Server** software options to an OPC UA client application
- Import user certificates
- Assign user certificates to users
- Configure the firewall

If at least one of the options 56 to 61 is active, then, when booting for the first time, the control creates the server certificate as part of a self-generated certificate chain. The client application or the manufacturer of the application creates the client certificate. The user certificate is linked to the user account. Please contact your IT department.

Notes

- The **OPC UA NC Server connection wizard** also helps you create test or sample certificates for users and the OPC UA client application. Do not use the user and client application certificates created at the control for other purposes than development at the programming station.

23.12.4 OPC UA license settings function (options 56 to 61)

Application

Use the **OPC UA NC Server - License Settings** window to assign an OPC UA client-application to one of the software options 56 to 61.

Related topics

- Setting up an OPC UA client application with the **OPC UA connection wizard** function

Further information: "OPC UA connection wizard function (options 56 to 61)", Page 468

Description of function

After using the **OPC UA connection wizard** or the **PKI Admin** menu item to import a certificate of an OPC UA client application, you can choose the certificate from a selection window.

If you activate the **Active** check box for a certificate, the control uses a software option for the OPC UA client application.

23.13 DNC menu item

Application





With the **DNC** menu item you can grant or restrict access to the control (e.g., connections over a network).

Related topics

- Connecting network drives
Further information: "Network drives on the control", Page 456
- Setting up a network
Further information: "Ethernet interface", Page 459
- TNCremo
Further information: "PC software for data transfer", Page 504
- Remote Desktop Manager (option 133)
Further information: "Remote Desktop Manager window (option 133)", Page 478

Description of function

The **DNC** area contains the following icons:

Icon	Meaning
	External access to the control is active
	Add computer-specific connection
	Edit computer-specific connection
	Delete computer-specific connection

DNC area

In the **DNC** area you use switches to activate the following functions:

Switches	Meaning
DNC access permitted	Permit or block all accesses to the control through a network or a serial connection
TNCopt full access allowed	Depending on the machine, permit or block access for diagnostics or initial setup software
Host computer operation	<p>Pass command control to an external host computer, for example to transfer data to the control; or end host computer operation</p> <p>If host computer operation is active, the control displays the Host computer is active message in the info bar. You cannot use the Manual and Program Run operating modes.</p> <p>You cannot activate host computer operation while running an NC program.</p>

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	<ul style="list-style-type: none"> ■ Permit The control permits network access without confirmation. ■ Inquire The control asks for confirmation upon a network access attempt. You can choose whether to permit or block the access once or always. ■ Deny The control does not permit any network access
Type	<ul style="list-style-type: none"> ■ Com1 Serial interface 1 ■ Com2 Serial interface 2 ■ Ethernet Network connection
Active	If a connection is active, the control displays a green circle. If a connection is inactive, the control displays a gray circle.

Notes

- In the machine parameter **allowDisable** (no. 129202) the machine manufacturer defines whether the **Host computer operation** switch is available.
- In the optional machine parameter **denyAllConnections** (no. 123403) the machine manufacturer defines whether the control permits computer-specific connections.

23.14 Printers

Application

You add and manage printers through the **Printer** menu item in the **Heros Printer Manager** window.

Related topics

- Using the **FN 16: F-PRINT** function for printing
Further information: User's Manual for Programming and Testing

Requirement

- PostScript-capable printer
The control can communicate only with printers that understand PostScript emulation such as KPDL3. Some printers enable setting the PostScript emulation in the printer menu.
Further information: "Note", Page 474

Description of function

The **Printer** menu item opens the **Heross Printer Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application. You can print the following files:

- Text files
- Graphic files
- PDF files

Further information: User's Manual for Programming and Testing

Once you have added a printer, the control shows the **PRINTER:** drive in the file management. The drive contains one folder for each defined printer.

Further information: "Creating a printer", Page 474

There are various methods to start printing:

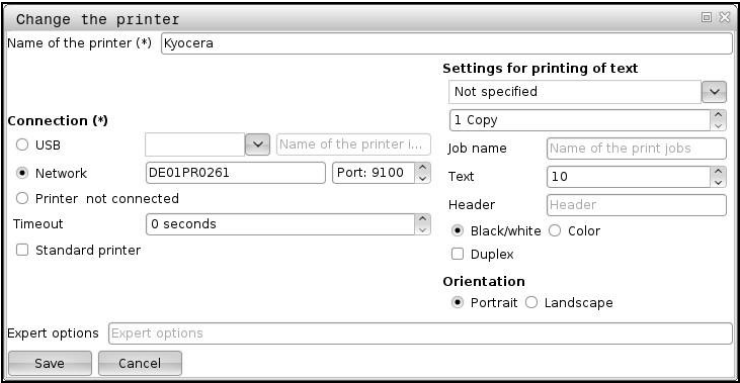
- Copying the file to be printed to the **PRINTER** drive
The file to be printed is automatically forwarded to the default printer and deleted from the directory after the print job has been executed.
You may also copy the file into the printer sub-directory if you wish to use a printer other than the default printer.
- Using the **FN 16: F-PRINT** function

Buttons

The **Heross Printer Manager** window contains the following buttons:

Button	Meaning
Create	Creates a printer
CHANGE	Adapts the properties of the selected printer
COPY	Creates a copy of the selected printer setting At first the copy has the same properties as the copied setting. This can be useful if printing both portrait and landscape formats on the same printer
DELETE	Deletes the selected printer
UP	Selects a printer
DOWN	
STATUS	Displays the status information of the selected printer
PRINT A TEST PAGE	Prints a test page on the selected printer

Change the printer window



For each printer, the following properties can be set:

Setting	Meaning
Name of the printer	Customizes the printer name
Connection	Selects the connection <ul style="list-style-type: none">■ USB: The control automatically displays the name■ Network: Network name or IP address of the printer Port for the network printer (default: 9001)■ Printer %1 not connected
Timeout	Delays the printing process The control delays the printing process by the pre-set number of seconds after the last change has been made to the file to be printed in PRINTER . Use this setting if the file to be printed is populated with FN functions, e.g. when probing.
Standard printer	Selects the default printer The control automatically assigns this setting to the first printer added.
Settings for printing of text	These settings are applicable when printing text documents: <ul style="list-style-type: none">■ Paper size■ Number of copies■ Job name■ Font size■ Header■ Print options (black and white, color, duplex)
Orientation	Portrait or landscape for all printable files
Expert options	Available only to authorized specialists

23.14.1 Creating a printer

To create a new printer:

- ▶ Enter the printer name in the name dialog
- ▶ Select **Create**
- > The control creates a new printer.
- ▶ Press **CHANGE**
- > The control opens the **Change the printer** window.
- ▶ Define the properties
- ▶ Select **Save**
- > The control applies the settings and displays the defined printer in the list.

Note

If your printer does not permit PostScript emulation, change the printer settings if possible.

23.15 VNC menu item

Application

VNC is software that shows the screen contents of a remote computer on a local computer, and also sends keyboard actions and mouse movements of the local computer to the remote computer.

Related topics




- Firewall settings
Further information: "Firewall", Page 484
- Remote Desktop Manager (option 133)
Further information: "Remote Desktop Manager window (option 133)", Page 478

Description of function

The **VNC** menu item opens the **VNC settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Buttons and icons

The **VNC settings** window contains the following buttons and icons:

Button and icon	Meaning
Add	Add new VNC viewer or client
Remove	Delete the selected client Only possible with manually entered clients.
Edit	Edit the configuration of the selected client
Update	Refresh view Required with connection attempts during which the dialog is open.
Set preferred owner of the focus	Activate the Preferred owner of the focus check box
	Another client owns the focus Mouse and keyboard are disabled
	You own the focus Entries can be made
	Prompt by another client to receive the focus Mouse and keyboard are disabled until the focus is assigned.

VNC participant settings area

In the **VNC participant settings** area, the control shows a list of all clients.

The control displays the following contents:

Column	Contents
Computer name	IP address or computer name
VNC	Connection of the client to the VNC viewer
VNC Focus	The client participates in the focus assignment
Type	<ul style="list-style-type: none"> ■ Manual Manually entered client ■ Denied This client is not permitted to connect. ■ Enable TeleService and IPC Client via a TeleService connection ■ DHCP Other computer that retrieves an IP address from this computer.

Global settings area

In the **Global settings** area you can define the following settings:

Function	Meaning
Enable TeleService and IPC	If the check box is selected, the connection is always permitted.
Password verification	Client must enter a password for verification The control opens a window when you select the check box. In this window you define the password for this client. The client must enter the password when establishing the connection.

Enabling other VNC area

In the **Enabling other VNC** area you can define the following settings:

Function	Meaning
Deny	Other VNC clients are not permitted.
Inquire	A dialog opens when another VNC client wants to connect. You must grant permission for this connection.
Permitted	Other VNC clients are permitted.

VNC Focus Settings area

In the **VNC Focus Settings** area, you can make the following settings:

Function	Meaning
Enabling VNC focus	Enables focus assignment for this system When the check box is inactive, the focus owner actively gives away the focus by using the focus symbol. The remaining clients can request the focus only after it was given away.
Reset the CapsLock key when changing the focus	When the check box is active and the focus owner has activated the CapsLock key, the CapsLock key is deactivated in a focus change. Only if the Enabling VNC focus check box is active
Enable Concurrency VNC Focus	When the check box is active, every client can request the focus at any time. The focus owner does not need to give away the focus before to enable that. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the Enabling VNC focus check box is active
Timeout Concurrency VNC Focus	Period of time after requesting the focus during which the focus owner can object to the focus change (at most 60 seconds). This period of time is set by moving a slider. When a client requests the focus, a pop-up window opens for all clients. If no client objects to the request within the pre-set period of time, the focus changes after the defined time limit. Only if the Enabling VNC focus check box is active



Activate the **Enabling VNC focus** check box only in connection with HEIDENHAIN devices provided especially for 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 with which another computer can be controlled over a network connection.

23.16 Remote Desktop Manager window (option 133)

Application

With Remote Desktop Manager you can display external computer units on the control screen that are connected via Ethernet, and operate them through the control. You can also shut down a Windows computer together with the control.

Related topics

- External access
Further information: "DNC menu item", Page 470

Requirement

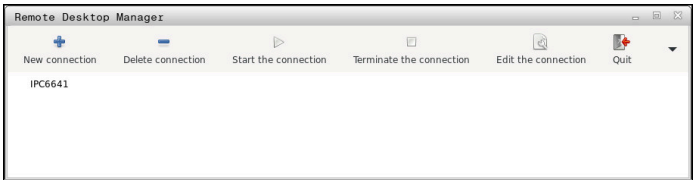
- Remote Desktop Manager (software option 133)
- Existing network connection
Further information: "Ethernet interface", Page 459

Description of function

The **Remote Desktop Manager** menu item opens the **Remote Desktop Manager** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

Remote Desktop Manager grants the following connection options:

- **Windows Terminal Service (RemoteFX):** Display the desktop of an external Windows computer on the control
Further information: "Windows Terminal Service (RemoteFX)", Page 479
- **VNC:** Display the desktop of an external Windows, Apple or Unix computer on the control
Further information: "VNC", Page 479
- **Switch-off/restart of a computer:** Automatically shut down a Windows computer together with the control
- **World Wide Web:** Only for authorized specialists
- **SSH:** Only for authorized specialists
- **XDMCP:** Only for authorized specialists
- **User-defined connection:** Only for authorized specialists



HEIDENHAIN offers the IPC 6641 as a Windows computer. With the IPC 6641 you can start and operate Windows-based applications directly from within the control. If the desktop of the external connection or the external computer is active, all inputs from the mouse and the alphabetic keyboard are transmitted there. When the operating system is shut down, the control automatically terminates all connections. Please note that only the connection is terminated, whereas the external computer or the external system is not shut down automatically.

Buttons

Remote Desktop Manager contains the following buttons:

Button	Function
New connection	Use the Edit the connection window to create a new connection Further information: "Establishing and starting a connection", Page 483
Delete connection	Delete the selected connection
Start the connection	Start the selected connection Further information: "Establishing and starting a connection", Page 483
Terminate the connection	Terminate the selected connection
Edit the connection	Use the Edit the connection window to modify the selected connection Further information: "Connection settings", Page 480
Exit	Close Remote Desktop Manager
Import connections	Restore the selected connection Further information: "Exporting and importing connections", Page 483
Export the connections	Back-up the selected connection Further information: "Exporting and importing connections", Page 483

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 482

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

With RemoteFX, a separate window opens for the screen of the external computer. The active desktop on the external computer is then locked and the user logged off. This prevents two users from accessing the control simultaneously.

VNC

You need an additional **VNC** server for your external computer when connecting through VNC. Install and configure the VNC server, e.g. TightVNC server, before establishing the connection.


VNC mirrors the screen of the external computer. The active desktop on the external computer is not locked automatically.

With a **VNC** connection you can shut down the external computer through the Windows menu. The computer cannot be restarted through the connection.

Connection settings

General settings

The following settings apply to all connection options:

Setting	Meaning	Intended use
Connection name	<p>Name of the connection in Remote Desktop Manager</p> <div>  <p>You can use the following characters in the name of the connection: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _</p> </div>	Required
Restarting after end of connection	<p>Behavior after disconnection:</p> <ul style="list-style-type: none"> ■ Always restart ■ Never restart ■ Always after an error ■ Ask after an error 	Required
Automatic starting upon login	Connect automatically when starting	Required
Add to favorites	<p>The control displays the connection's icon in the taskbar.</p> <p>Tap or click the icon to start the connection directly.</p>	Required
Move to the following workspace	<p>Number of the desktop for the connection; desktops 0 and 1 are reserved for the NC software.</p> <p>Default setting: Third desktop</p>	Required
Release USB mass memory	Permit access to connected USB mass memory devices	Required
Private connection	Connection can be seen and used only by its creator	Required
Computer	<p>Host name or IP address of the external computer</p> <p>HEIDENHAIN recommends the IPC6641.machine.net setting for the IPC 6641.</p> <p>The host name IPC6641 must be assigned to the IPC in the Windows operating system for this setting.</p>	Required
Password	Password of the user	Required
Entries in the Advanced options area	Available only to authorized specialists	Optional

Additional settings for Windows Terminal Service (RemoteFX)

The control offers the following additional connection settings for the **Windows Terminal Service (RemoteFX)** option:

Setting	Meaning	Intended use
User name	Name of the user	Required
Windows domain	Domain of the external computer	Optional
Full-screen mode or User-defined window size	Size of the connection window on the control	Required

Additional settings for VNC

The control offers the following additional connection settings for the **VNC** option:

Setting	Meaning	Intended use
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	Intended use
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

23.16.1 Configuring an external computer for Windows Terminal Service (RemoteFX)

To configure the external computer (e.g., in Windows 10 operating systems):

- ▶ Press the Windows key
- ▶ Select **Control Panel**
- ▶ Select **System and Security**
- ▶ Select **System**
- ▶ Select **Remote Settings**
- > The computer opens a pop-up window.
- ▶ Under **Remote Assistance**, enable **Allow Remote Assistance connections to this computer**
- ▶ In the **Remote Desktop** area, enable **Allow Remote Assistance connections to this computer**
- ▶ Press **OK** to confirm your settings

23.16.2 Establishing and starting a connection

To establish and start a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select **New connection**
- > The control displays a selection menu.
- ▶ Select a connection option
- ▶ Under **Windows Terminal Service (RemoteFX)**, select the operating system
- > The control opens the **Edit the connection** window.
- ▶ Define the connection settings
- Further information:** "Connection settings", Page 480
- ▶ Press **OK**
- > The control saves the settings and closes the window.
- ▶ Select connection
- ▶ Select **Start the connection**
- > The control starts the connection.

23.16.3 Exporting and importing connections

To export a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the desired connection
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Export the connections**
- > The control opens the **Select export file** window.
- ▶ Define the name of the saved file
- ▶ Select the target file
- ▶ Select **Save**
- > The control saves the connection data under the name defined in the window.

To import a connection:

- ▶ Open **Remote Desktop Manager**
- ▶ Select the right arrow icon in the menu bar
- > The control displays a selection menu.
- ▶ Select **Import connections**
- > The control opens the **Select file to import** window.
- ▶ Select file
- ▶ Select **Open**
- > The control creates the connection under the name that was defined originally in the **Remote Desktop Manager**.

Notes

NOTICE

Caution: Data may be lost!
If you do not shut down external computers properly, data may be irreversibly damaged or deleted.
► Configure the automatic shutdown of the Windows computer

- When you edit an existing connection, the control will automatically delete all impermissible characters from the name.

Notes in connection with the IPC 6641

- HEIDENHAIN assures a functioning connection between HEROS 5 and the IPC 6641. No guarantee is given for other combinations and connections.
- If you use the computer name **IPC6641.machine.net** to connect an IPC 6641, it is important to enter **.machine.net**.
With this entry, the control automatically searches the Ethernet interface **X116**, and not the interface **X26**; this reduces the time needed for access.

23.17 Firewall

Application

With the control you can set up a firewall for the primary network interface, and for a sandbox if needed. You can block incoming network traffic for specific senders and services.




Related topics

- Existing network connection
Further information: "Ethernet interface", Page 459
- SELinux security software
Further information: "SELinux security software", Page 455

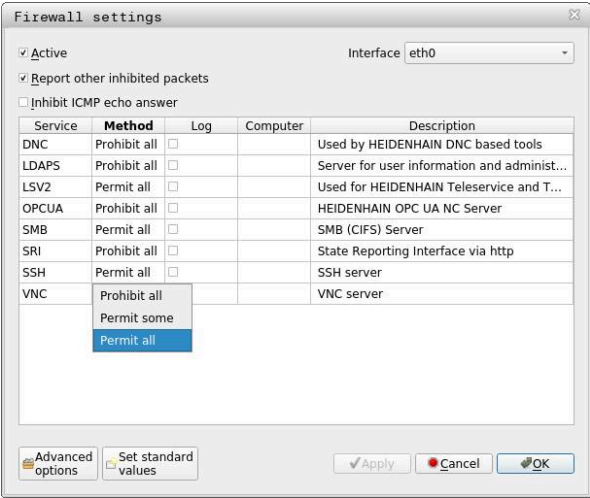
Description of function

The **Firewall** menu item opens the **Firewall settings** window. The menu item can be found in the **Network/Remote Access** group of the **Settings** application.

If you activate the firewall, the control displays an icon at the bottom right in the taskbar. The control displays the following icons, depending on the security level:


Icon	Meaning
	Firewall protection does not yet exist although it has been activated. Example: A dynamic IP address is used in the network interface configuration, but the DHCP server has not yet assigned an IP address. Further information: "DHCP server tab", Page 464
	Firewall active with medium security level.
	Firewall active with high security level. All services except for SSH are blocked.

Firewall settings



The **Firewall settings** window contains the following settings:

Setting	Meaning
Active	Activate or deactivate firewall
Interface	Select the interface <ul style="list-style-type: none">■ eth0: X26 of the control■ eth1: X116 of the control■ brsb0: Sandbox (optional) If a control has two Ethernet interfaces, then by default the DHCP server for the machine network is active for the second interface. With this setting you cannot activate the firewall for eth1 because the firewall and DHCP server mutually exclude each other.
Report other inhibited packets	Activate the firewall with a high security level All services except for SSH are blocked.
Inhibit ICMP echo answer	If this check box is selected, the control does not respond to a ping request.

Setting	Meaning
Service	<p>Brief designation of services configured with the firewall. You can change the settings even if the services are not started.</p> <ul style="list-style-type: none"> DNC DNC server using the RPC protocol for external applications that were developed with RemoTools SDK (port 19003) <div>  For more detailed information, consult the RemoTools SDK manual. </div> LDAPS Server with user data and configuration of user administration LSV2 Functionality for TNCremo, TeleService, and other HEIDENHAIN PC tools (port 19000) OPC UA Service provided by the OPC UA NC Server (port 4840). SMB Only incoming SMB connections, meaning a Windows share on the control. Outgoing SMB connections are not influenced, meaning a Windows share connected to the control. SSH SecureShell protocol (port 22) for secure LSV2 handling with active user administration; starting with HEROS 504 VNC Access to screen contents. If you block this service, then not even TeleService programs from HEIDENHAIN can access the control. If you block this service, the control displays a warning in the VNC settings window. Further information: "VNC menu item", Page 474
Method	<p>Configure accessibility</p> <ul style="list-style-type: none"> Prohibit all: Cannot be accessed by anyone Permit all: Can be accessed by everyone Permit some: Can be accessed only by specific clients In the Computer column you must define the computer for which access is permitted. If you do not define a computer, the control activates Prohibit all.
Log	<p>The control shows the following messages when transmitting network packets:</p> <ul style="list-style-type: none"> Red: Network packet blocked Blue: Network packet accepted
Computer	<p>IP address or host name of the computers with access rights. Separated by commas, if there are multiple computers</p> <p>The control converts the host name to an IP address when the control starts. If the IP address changes, you must restart the control or change the setting. The control issues an error message if it cannot convert the host name to an IP address.</p> <p>Only with the Permit some method</p>
Advanced options	Only for network specialists
Set standard values	Reset the settings to the default values recommended by HEIDENHAIN

Notes

- Have your network specialist check and, if necessary, change the standard settings.
- The firewall does not protect the second network interface **eth1**. Connect only trustworthy hardware to this interface, and do not use this interface for Internet connections.

23.18 Portscan

Application

With the **Portscan** function, the control checks all open, incoming TCP and UDP listen ports at defined intervals or when commanded. The control shows a message if a port is not listed.

Related topics

- Firewall settings
Further information: "Firewall", Page 484
- Network settings
Further information: "Network settings window", Page 461

Description of function

The **Portscan** menu item opens the **Portscan** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

The control searches for all open, incoming TCP and UDP listen ports on the system and compares them to the following whitelists:

- System-internal whitelists **/etc/sysconfig/portscan-whitelist.cfg** and **/mnt/sys/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with machine-manufacturer-specific functions: **/mnt/plc/etc/sysconfig/portscan-whitelist.cfg**
- Whitelist for ports with customer-specific functions: **/mnt/tnc/etc/sysconfig/portscan-whitelist.cfg**

Each whitelist contains the following information:

- Port type (TCP/UDP)
- Port number
- Offering program
- Comments (optional)

In the **Manual Execution** area, use the **Start** button to start the portscan manually. Use the **Automatic update on** function of the **Automatic Execution** to have the control automatically perform the portscan at a defined interval. You define the interval with a slider.

If the control performs the portscan automatically, then only ports listed in the whitelists may be open. The control shows a message window if a port is not listed.

23.19 Remote servicing

Application

In conjunction with the remote service setup tool, TeleService from HEIDENHAIN offers the possibility to establish encrypted end-to-end connections between a computer and a machine over the Internet.

Related topics

- External access

Further information: "DNC menu item", Page 470

- Firewall

Further information: "Firewall", Page 484

Requirements

- Existing Internet connection

Further information: "Network settings window", Page 461

- **LSV2** connection permitted by the firewall

Remote diagnosis via the TeleService PC software uses the **LSV2** service. By default, the control's firewall blocks all incoming and outgoing connections. For this reason you must permit a connection for this service.

You have the following options for permitting the connection:

- Deactivate the firewall
- Define the **Permit some** method for the **LSV2** service, and enter the name of the computer under **Computer**

Further information: "Firewall", Page 484

Description of function

The **Remote Service** menu item opens the **HEIDENHAIN remote maintenance** window. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You need a valid session certificate for the servicing session.

Session certificate

During installation of the NC software, a temporary certificate is automatically installed on the control. A new installation or update may only be carried out by a service technician from the machine manufacturer.

A new certificate must be installed if no valid session certificate is installed on the control. Clarify with the service technician which certificate is needed. The service technician will then provide you with a valid certificate file, if necessary, which you must then install.

Further information: "Installing a session certificate", Page 489

In order to start the servicing session, you must enter the session key from the machine manufacturer.

23.19.1 Installing a session certificate

To install a session certificate on the control:

- ▶ Select the **Settings** application
- ▶ Select **Network/Remote Access**
- ▶ Double-tap or double-click **Network**
- > The control opens the **Network settings** window.
- ▶ Select the **Internet** tab



The machine manufacturer defines the settings in the **Telemaintenance** field.

- ▶ Select **Add**
- > The control displays a selection menu.
- ▶ Select file
- ▶ Select **Open**
- > The control opens the certificate.
- ▶ Select **OK**
- ▶ Restart the control to load the settings

Notes

- If you've deactivated the firewall, you must reactivate it after the servicing session!
- If you permit the **LSV2** service in the firewall, the access security is ensured through the network settings. The network security is the responsibility of the machine manufacturer or the respective network administrator.

23.20 Backup and restore

Application

With the **NC/PLC Backup** and **NC/PLC Restore** functions you can back up and restore individual folders or the entire **TNC:** drive. You can save the backup files to various types of memory media.

Related topics

- File management, **TNC:** drive
- Further information:** User's Manual for Programming and Testing

Description of function

You open the backup function through the **NC/PLC Backup** menu item. The menu item is in the **Diagnostics/Maintenance** group of the **Settings** application.

You open the restore function through the **NC/PLC Backup** menu item.

The backup function creates a ***.tncbck** file. The restore function can restore these files as well as files from existing TNCbackup programs. If you double tap or click a ***.tncbck** file in the file manager, the control starts the restore function.

Further information: User's Manual for Programming and Testing

Within the backup function you can chose between the following types of backups:

- **Back up the "TNC:" partition**
Back-up all data on the **TNC:** drive
- **Back up the directory tree**
Back-up the selected folders and their subfolders on the **TNC:** drive
- **Back up the machine configuration**
Only for the machine manufacturer
- **Complete backup (TNC: and machine configuration)**
Only for the machine manufacturer

Backup and restore is subdivided into several steps. Navigate between these steps with the **FORWARD** and **BACK** buttons.

23.20.1 Backing up data

To back-up the data of the **TNC:** drive:

- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Backup**
- > The control opens the **Back up the "TNC:" partition** window.
- ▶ Specify the type of backup
- ▶ Select **Forward**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select any predefined exclusion rules or ones you have defined yourself
- ▶ Select **Forward**
- > The control generates a list of files for backing up.
- ▶ Check list
- ▶ Deselect files if necessary
- ▶ Select **Forward**
- ▶ Enter the name of the backup file
- ▶ Select the storage path
- ▶ Select **Forward**
- > The control generates the backup file.
- ▶ Confirm with **OK**
- > The control concludes the backup process and restarts the NC software.

23.20.2 Restoring data

NOTICE

Caution: Data may be lost!

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

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

To restore data:

- ▶ Select the **Settings** application
- ▶ Select **Diagnostics/Maintenance**
- ▶ Double-tap or double-click **NC/PLC Restore**
- > The control opens the **Restore data - %1** window.
- ▶ Select the archive to be restored
- ▶ Select **Forward**
- > The control generates a list of files for restoring.
- ▶ Check list
- ▶ Deselect files if necessary
- ▶ Select **Forward**
- ▶ If necessary, pause the control with **Stop NC software**
- ▶ Select **Extract archive**
- > The control restores the files.
- ▶ Confirm with **OK**
- > The control restarts the NC software.

Note

The TNCbackup PC program can also process ***.tncbck** files. TNCbackup is part of TNCremo.

23.21 TNCdiag

Application

The control displays status and diagnostic information of HEIDENHAIN components in the **TNCdiag** window.

Description of function



Only use this function after consultation with your machine manufacturer.



For more information, please refer to the **TNCdiag** documentation.

23.22 Machine parameters

Application

You can configure the behavior of the control with machine parameters. The control offers the applications **MPs for users** and **MPs for setters** for this. You can open the **MPs for users** application at any time without having to enter a code number.

The machine manufacturer defines which machine parameters are in which applications. HEIDENHAIN offers a standard scope of parameters for the **MPs for setters** application. The following contents describe only the standard scope of the **MPs for setters** application.

Related topics

- List of machine parameters in the **MPs for setters** application
Further information: "Machine parameters", Page 514

Requirements

- Code number 123
Further information: "Code numbers", Page 448
- Contents of the **MPs for setters** application are defined by the machine manufacturer

Description of function

The **MPs for setters** menu item opens the **MPs for setters** application. The menu item is in the **Machine parameters** group of the **Settings** application.

In the **Machine parameters** group the control shows only those menu items that you can choose with the current access rights.

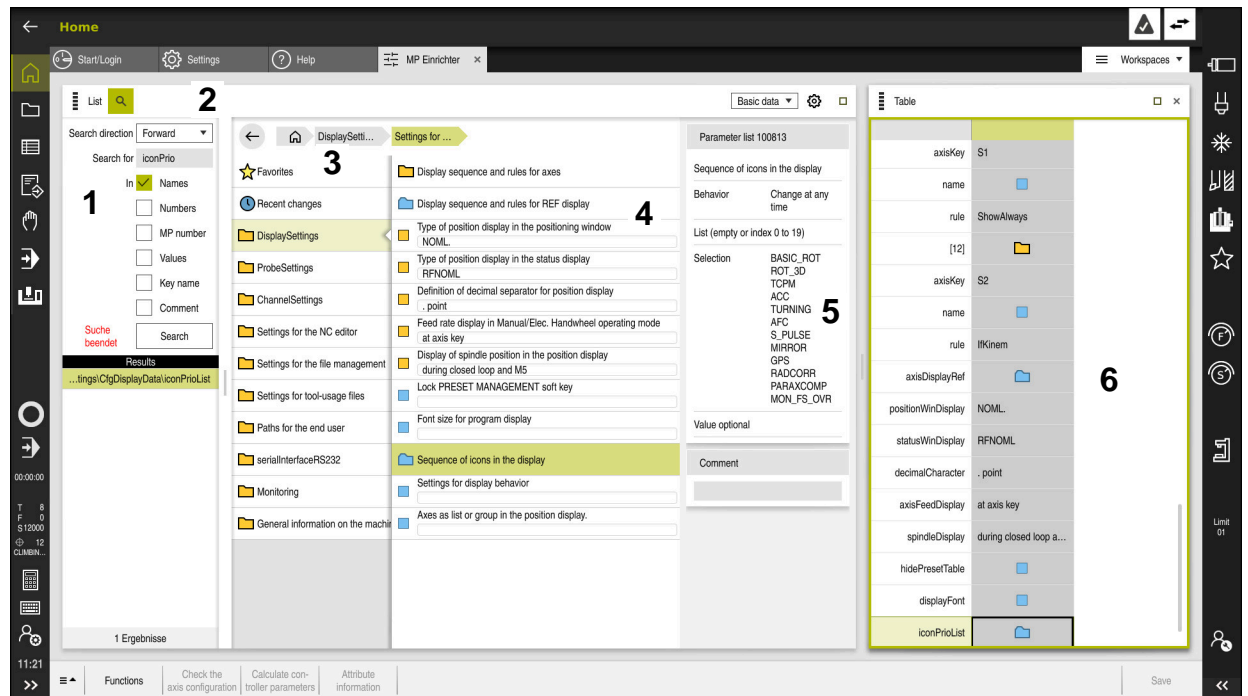
If you open an application for machine parameters, the control displays the configuration editor.

The configuration editor offers the following workspaces:

- **List**
- **Table**

You can't close the **List** workspace.

Areas of the configuration editor



MPs for setters application with selected machine parameter

The configuration editor shows the following areas:

1 Search column

You can search forward or backward with the following characteristics:

- Name
This is the language-neutral name used for machine parameters in the User's Manual.
- Number
This is the unique number used for machine parameters in the User's Manual.
- MP number of the iTNC 530
- Value
- Key name
Machine parameters for axes or channels exist more than once. In order to avoid ambiguity, each axis and each channel is identified with a key name (e.g., **X1**)
- Comment

The control displays the results.

2 Title bar of the List workspace

You can show and hide the **Search** column, use a selection menu to filter the contents, and open the **Configuration** window.

Further information: "Configuration window", Page 496

3 Navigation column

The control provides the following options for navigation:

- Navigation path
- Favorites
- 21 most recent changes
- Structure of the machine parameters

4 Content column

In the content column the control displays objects, machine parameters, or changes that you select using the search function or navigation column.

5 Information area

The control displays information about the selected machine parameter or change.

Further information: "Information area", Page 496

6 **Table** workspace











In the **Table** workspace the control shows the selected contents within the structure. In order to do so, in the **Configuration** window the **Synchronized navigation in list and table** switch must be set to active.

The control displays the following information:

- Name of the objects
- Icon of the objects
- Value of the machine parameters

Icons and buttons

The configuration editor contains the following icons and buttons:

Icon or button	Meaning
	Open the Configuration window Further information: "Configuration window", Page 496
	Select Recent changes
	Object exists <ul style="list-style-type: none"> ■ Data object ■ Directory ■ Parameter list
	Object empty
	Machine parameter exists
	Optional machine parameter does not exist
	Machine parameter invalid
	Machine parameter readable but not editable
	Machine parameter unreadable and uneditable
	Changes to the machine parameter not yet saved
Functions	Open the context menu Further information: User's Manual for Programming and Testing
Check the axis configuration	Only for the machine manufacturer
Calculate controller parameters	Only for the machine manufacturer
Attribute information	Only for the machine manufacturer
Save	The control opens a window with all of the changes since the most recent saving. You can save or discard the changes.

Configuration window

In the **Configuration** window you define the settings for displaying the machine parameters in the configuration editor.

The **Configuration** window consists of the following areas:

- **List**
- **Table**

The **List** area contains the following settings:

Setting	Meaning
Show MP descriptive texts	If the switch is active, the control displays a description of the machine parameter in the active conversational language. If the switch is not active, the control displays the language-neutral name of the machine parameter.
Show details	Hide or show the information area with this switch.

The **Table** area contains the following settings:

Setting	Meaning
Show details when table displayed	If the switch is active, the control shows the information area even if the Table workspace is open. If the switch is not active, the control shows the information area only if the Table workspace is closed.
Synchronized navigation in list and table	If the switch is active, the control always shows in the Table workspace the object that is marked in the List workspace, and vice versa. If the switch is not active, the contents of the two workspaces do not synchronize.

Information area

If you select contents from the favorites or the structure, the control will display some of the information below in the information area:

- Type of object, such as data object list or parameter, perhaps with number
- Descriptive text of machine parameter
- Information about the effect
- Permitted or required input
- Behavior, such as program run disabled
- MP number of the iTNC 530 for machine parameters
- Machine parameter optional

If you select content from one of the recent changes, the control will display the information below in the information area:

- Sequential number of the change
- Previous value
- New value
- Date and time of change
- Descriptive text of machine parameter
- Information about the effect

24

**HEROS Operating
System**

24.1 Fundamentals

HEROS is the fundamental basis for all NC controls from HEIDENHAIN. The HEROS operating system is based on Linux, and was adapted for the purposes of NC controls.

The TNC7 features the version HEROS 5.

24.2 HEROS menu

Application

In the HEROS menu the control shows information about the operating system. You can change settings or use HEROS functions.

By default you open the HEROS menu through the taskbar at the bottom edge of the screen

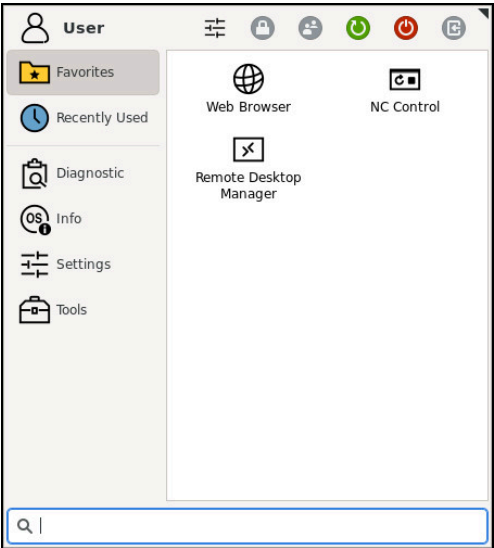
Related topics

- Opening HEROS functions through the **Settings** application
Further information: "Settings Application", Page 445

Description of function

You open the HEROS menu with the green DIADUR icon in the taskbar or with the **DIADUR** key.

Further information: "Taskbar", Page 501



Standard view of the HEROS menu

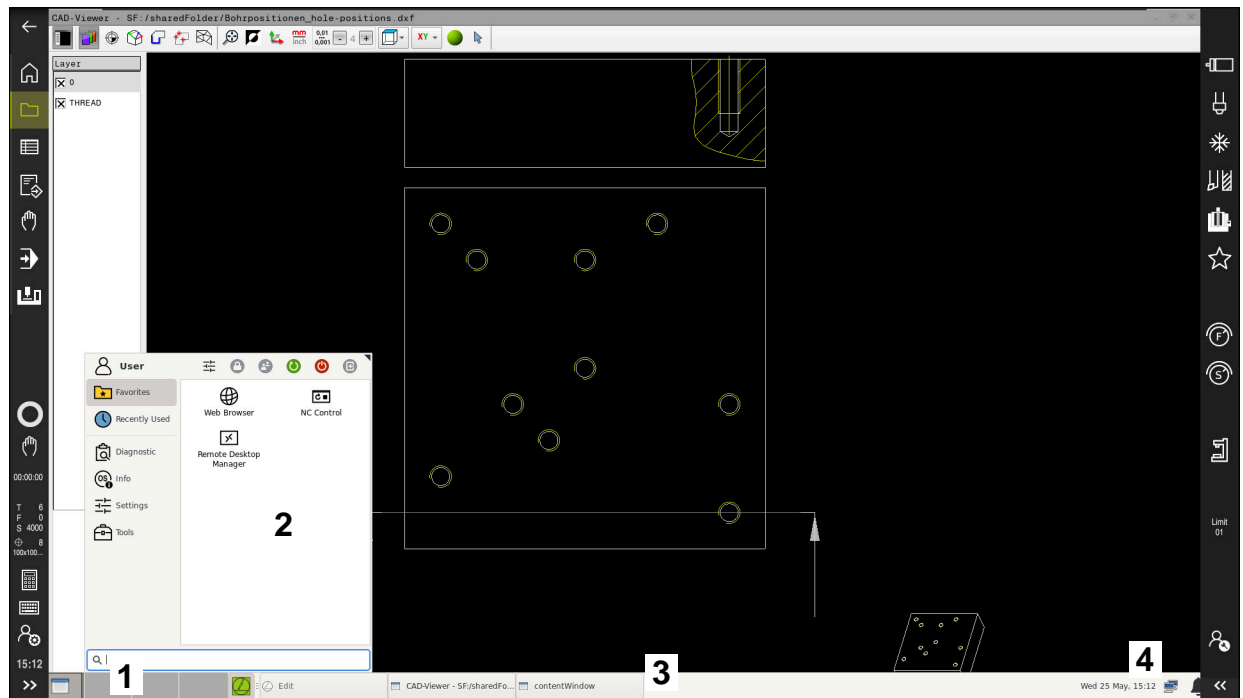
The HEROS menu contains the following functions:

Area	Function
Header	■ User name: Currently no function
	■ User-specific settings
	■ Lock display
	■ Switch user: Currently no function
	■ Restart
	■ Shut down
	■ Log out: Currently no function

Area	Function
Navigation	<ul style="list-style-type: none"> ■ Favorites ■ Recently used
Diagnostic	<ul style="list-style-type: none"> ■ GSmartControl: Available only to authorized specialists ■ HE Logging: Define settings for internal diagnostic files ■ HeMenu: Available only to authorized specialists ■ perf2: Check processor load and process load ■ Portscan: Test active connections Further information: "Portscan", Page 487 ■ Portscan OEM: Available only to authorized specialists ■ RemoteService: Start and stop remote maintenance Further information: "Remote servicing", Page 488 ■ 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 491
Settings	<ul style="list-style-type: none"> ■ screensaver ■ Current User: Currently no function ■ Date/Time Further information: "Adjust system time window", Page 453 ■ Firewall Further information: "Firewall", Page 484 ■ HePacketManager: Available only to authorized specialists ■ HePacketManager Custom: Available only to authorized specialists ■ Language/Keyboards Further information: "Conversational language of the control", Page 453 ■ License Settings Further information: "OPC UA NC Server (options 56 to 61)", Page 465 ■ Network Further information: "Ethernet interface", Page 459 ■ OEM Function Users: Currently no function ■ One Click Setup: Currently no function ■ OPC UA / PKI Admin Further information: "OPC UA NC Server (options 56 to 61)", Page 465 ■ Printer Further information: "Printers", Page 471 ■ SELinux Further information: "SELinux security software", Page 455 ■ Shares Further information: "Network drives on the control", Page 456 ■ UserAdmin: Currently no function ■ VNC Further information: "VNC menu item", Page 474 ■ WindowManagerConfig: Settings for the Window Manager Further information: "Window Manager", Page 502

Area	Function
Info	<ul style="list-style-type: none"> ■ About HeROS: Open information about the operating system of the control ■ About Xfce: Open information on the Window manager
Tools	<ul style="list-style-type: none"> ■ Switch-off: Shut-down or restart ■ Screenshot: Create screenshots ■ File Manager: Available only to authorized specialists ■ Document Viewer: Display and print files (e.g., PDF files) ■ Geeqie: Open, manage, and print graphics ■ Gnumeric: Open, edit, and print tables ■ IDS Camera Manager: Manage cameras connected to the control ■ keypad horizontal: Open virtual keyboard ■ keypad vertical: Open virtual keyboard ■ Leafpad: Open and edit text files ■ NC Control: Start or stop the NC software independently of the operating system ■ NC/PLC Backup Further information: "Backup and restore", Page 489 ■ NC/PLC Restore Further information: "Backup and restore", Page 489 ■ QupZilla: Alternative web browser for touch operation ■ Real VNC viewer: Define the settings for external software accessing the control for e.g. maintenance purposes ■ Remote Desktop Manager Further information: "Remote Desktop Manager window (option 133)", Page 478 ■ Ristretto: Open graphics ■ TNCguide: Open help files in CHM format ■ TouchKeyboard: Open keyboard for touch operation ■ Web Browser: Start the web browser ■ Xarchiver: Extract or compress directories
Searching	Full-text search of individual functions

Taskbar



CAD-Viewer opened in the third desktop with taskbar shown and active HEROS menu

The taskbar consists of the following areas:

- 1 Workspaces
- 2 HEROS menu

Further information: "Description of function", Page 498
- 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 484
 - Network status

Further information: "Ethernet interface", Page 459
 - Notifications
 - Shut down or restart the operating system

Window Manager

With the Window Manager you manage functions of the HEROS operating system as well as windows opened in the third desktop, such as the **CAD-Viewer**.

The control features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces.

The following functions are possible with the window manager:

- Display a taskbar for switching between various applications (user interfaces)
- Manage an additional desktop, on which special applications from your machine manufacturer can run
- Control the focus between NC software applications and those of the machine manufacturer
- You can change the size and position of pop-up windows. It is also possible to close, minimize and restore pop-up windows

If a window is opened in the third desktop, the control displays the **Window Manager** icon in the information bar. You can switch between the open applications by selecting the icon.

You can minimize the control's user interface by pulling down from the information bar. The TNC bar and the OEM bar remain visible.

Further information: "Areas of the control's user interface", Page 63

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 63

- 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.

24.3 Serial data transfer

Application

The TNC7 automatically uses the LSV2 transmission protocol for serial data transfer. All parameters of the LSV2 protocol are invariably fixed except for the baud rate in the machine parameter **baudRateLsv2** (no. 106606).

Description of function

The machine parameter **RS232** (no. 106700) allows you to define another transmission type (interface). The settings described below are effective only for the respective newly defined interface.

Further information: "Machine parameters", Page 492

In the machine parameters that then appear you can define the following settings:

Machine parameters	Setting
baudRate (no. 106701)	Data transfer rate (baud rate) Input: BAUD_110, BAUD_150, BAUD_300, BAUD_600, BAUD_1200, BAUD_2400, BAUD_4800, BAUD_9600, BAUD_19200, BAUD_38400, BAUD_57600, BAUD_115200
protocol (no. 106702)	Communications protocol <ul style="list-style-type: none"> ■ STANDARD: Standard data transmission, line-by-line ■ BLOCKWISE: Packet-based data transfer ■ RAW_DATA: Transmission without protocol (purely character-by-character) Input: STANDARD, BLOCKWISE, RAW_DATA
dataBits (no. 106703)	Data bits in each transferred character Input: 7 Bit, 8 Bit
parity (no. 106704)	Parity bit used to check for transmission errors <ul style="list-style-type: none"> ■ NONE: No parity, no error detection ■ EVEN: Even parity, error if the number of bits set is odd ■ ODD: Odd parity, error if the number of bits set is even Input: NONE, EVEN, ODD
stopBits (no. 106705)	The start bit and one or two stop bits enable the receiver to synchronize each transmitted character during serial data transmission. Input: 1 Stop-Bit, 2 Stop-Bits
flowControl (no. 106706)	By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking. <ul style="list-style-type: none"> ■ 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 <ul style="list-style-type: none"> ■ EXT: Minimum file system for printers or non-HEIDENHAIN transmission software ■ FE1: Communication with TNCserver or an external floppy disk unit If you require no special file system, this machine parameter is not needed. Input: EXT, FE1
bccAvoidCtrlChar (no. 106708)	The BCC is a block check character. The BCC is optionally added to a transfer block to simplify error detection. <ul style="list-style-type: none"> ■ TRUE: The BCC does not correspond to any control character ■ FALSE: Function not active Input: TRUE, FALSE

Machine parameters	Setting
rtsLow (no. 106709)	<p>This optional parameter determines the level of the RTS line in the idle state.</p> <ul style="list-style-type: none"> ■ TRUE: Level is LOW in idle state ■ FALSE: Level is HIGH in idle state <p>Input: TRUE, FALSE</p>
noEotAfterEtx (no. 106710)	<p>This optional parameter sets whether an EOT character (End of Transmission) is to be transmitted after receiving an ETX character (End of Text).</p> <ul style="list-style-type: none"> ■ TRUE: The EOT character is not sent ■ FALSE: The EOT character is sent <p>Input: TRUE, FALSE</p>

Example

In order to use the TNCserver PC software for data transfer, define the following settings in the machine parameter **RS232** (no. 106700):

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

TNCserver is part of the TNCremo software for PCs.
"PC software for data transfer"

24.4 PC software for data transfer

Application

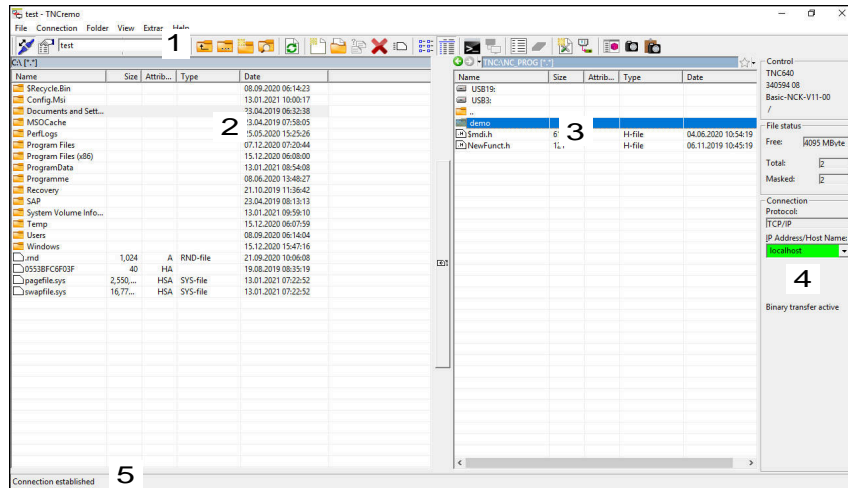
HEIDENHAIN offers the TNCremo software for connecting a Windows PC to a HEIDENHAIN control in order to transfer data.

Requirements

- PC operating system:
 - Windows 7
 - Windows 8
 - Windows 10
- PC RAM: 2 GB
- Free PC hard-disk space: 15 MB
- An available serial interface or connection to the control's network

Description of function

The TNCremo data transfer software provides the following areas:



- 1 Toolbar
This area provides the most important TNCremo functions.
- 2 File list of PC
In this area, TNCremo displays all of the folders and files of the connected drive (e.g., hard disk of a Windows PC or a USB flash drive).
- 3 File list of control
In this area, TNCremo displays all of the folders and files of the connected drive of the control.
- 4 Status display
In the status display, TNCremo shows information about the current connection.
- 5 Connection status
The connection status indicates whether a connection is currently active.



For more information, refer to the integrated help system of TNCremo. You can open the context-sensitive help function of the TNCremo software by pressing the **F1** key.

Notes

- When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user identification.
- You can download the current version of the TNCremo software from the **HEIDENHAIN website**.

24.5 Data backup

Application

If you create or modify files on the control, then you should back up these files periodically.

Related topics

- File management

Further information: User's Manual for Programming and Testing

Description of function

With the functions **NC/PLC Backup** and **NC/PLC Restore** you can create back-up files for specific directories or even an entire drive, and restore them as needed. You should store these backup files on an external storage medium.

Further information: "Backup and restore", Page 489

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 504

- External drive

You can transfer files from the control directly to an external drive.

Further information: "Network drives on the control", Page 456

- External data carriers

You can store the files on external data carriers or use external data carriers to transfer the files.

Further information: User's Manual for Programming and Testing

Notes

- You should back-up all machine-specific data, such as the PLC program or machine parameters. Consult your machine manufacturer about this.
- You must transmit files with the extensions PDF, XLS, ZIP, BMP, GIF, JPG and PNG in binary format from the PC to the control's hard disk.
- Backing up all files of the internal memory can take several hours. If required, perform the backup during a time when you don't need the machine.
- Periodically delete files that are no longer required. This ensures that the control has enough memory available for system files, such as the tool table.
- HEIDENHAIN recommends having the hard disk inspected after three to five years. After this time, and depending on the operating conditions (e.g., vibration loads), you must expect increased failure rates.

24.6 Opening files with additional software

Application

The control provides several additional software programs for opening and editing standard file types:




Related topics

- File types

Further information: User's Manual for Programming and Testing

Description of function

The control offers tools for the following file types:

File type	Tool
PDF	Document Viewer
XSLX (XSL) CSV	Gnumeric
INI A TXT	Leafpad
HTM/HTML	Web browser
<div>  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). </div>	
ZIP	Xarchiver
BMP GIF JPG/JPEG PNG	Ristretto or Geeqie
<div>  Ristretto can only open graphics files. Geeqie can also edit and print graphics. </div>	
OGG	Parole
<div>  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. </div>	

If you double-tap or double-click a file in the file manager, the control automatically starts the file with the correct tool. If more than one tool is possible for a file, the control displays a selection window.

The control opens the tools in the third desktop.

24.6.1 Opening tools

To open a tool:

- ▶ Select the HEIDENHAIN icon in the taskbar
- > The control opens the HEROS menu.
- ▶ Select **Tools**
- ▶ Select the tool, e.g. Leafpad
- > The control opens the tool in its own workspace.

Notes

- You can also open several tools from the **Desktop menu** workspace.
- Use the **ALT+TAB** key combination to switch between open workspaces.
- More information on how to use the various tools is provided within the respective tool under Help.
- After starting, the **web browser** checks at regular intervals whether updates are available.
If you want to update the **web browser**, then you must deactivate the SELinux security software during this time and establish a connection to the Internet. Reactivate SELinux after the update!

Further information: "SELinux security software", Page 455

24.7 Network configuration with Advanced Network Configuration

Application

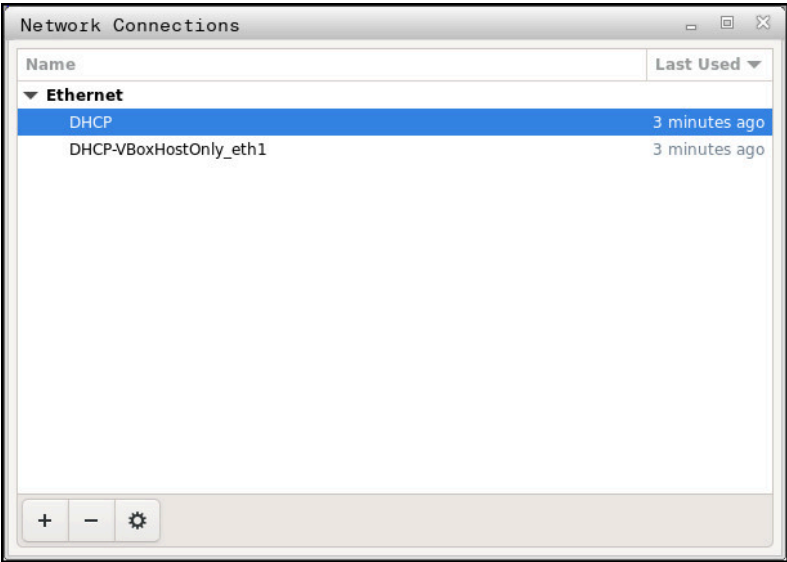
Using the **Advanced Network Configuration**, you can add, edit or remove profiles for the network connection.

Related topics

- Network settings
Further information: "Network settings window", Page 461

Description of function

When you select the **Advanced Network Configuration** application in the HEROS menu, the control opens the **Network connections** window.



Network connections window

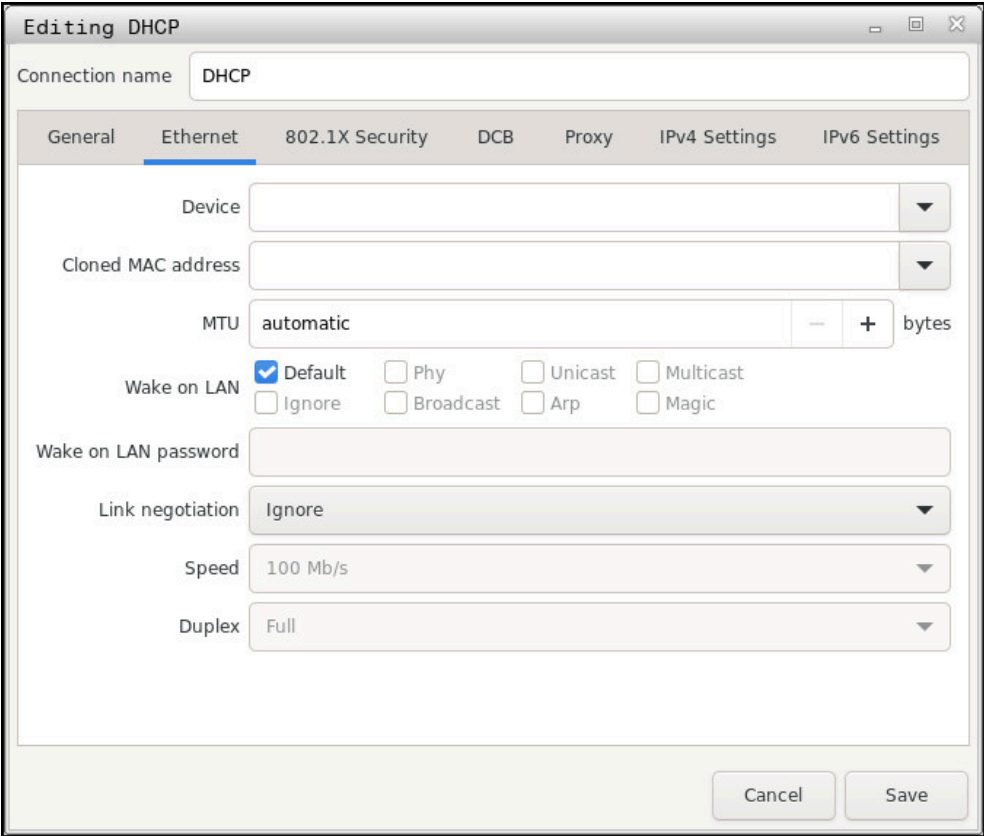
Symbols in the Network connections window

The following symbols are shown in the **Network connections** window:

Symbol	Function
+	Add network connection
—	Remove network connection
⚙	Edit network connection The control opens the Editing network connection window. Further information: "Editing network connection window", Page 509

24.7.1 Editing network connection window

In the **Editing network connection** window, the control shows the connection name of the network connection in the upper area. You can change the name.



Editing network connection window

General tab

The **General** tab contains the following settings:

Setting	Meaning
Connect automatically with priority	<p>If you are using several profiles, you can define an order of priority for the connection here.</p> <p>The control connects the network with the highest priority first.</p> <p>Input: -999...999</p>
All users may connect to this network	Here you can enable the selected network for all users.
Automatically connect to VPN	Currently no function
Metered connection	Currently no function

Ethernet tab

The **Ethernet** tab contains the following settings:

Setting	Meaning
Device	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
Wake on LAN	Currently no function
Wake on LAN password	Currently no function
Link negotiation	Here you have to configure the settings for the Ethernet connection: <ul style="list-style-type: none"> ■ 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: <ul style="list-style-type: none"> ■ 10 Mb/s ■ 100 Mb/s ■ 1 Gb/s ■ 10 Gb/s Only with selection Link negotiation Manual Selection by means of a selection window
Duplex	Here you have to select the duplex setting: <ul style="list-style-type: none"> ■ Half ■ Full Only with selection Link negotiation Manual Selection by means of a selection window

802.1X Security tab

Currently no function

DCB tab

Currently no function

Proxy tab

Currently no function

IPv4 Settings tab

The **IPv4 Settings** tab contains the following settings:

Setting	Meaning
Method	<p>Here you have to select a network connection method:</p> <ul style="list-style-type: none"> ■ 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
Additional static addresses	<p>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</p>
Additional DNS servers	<p>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 Methods Manual and Automatic (DHCP) addresses only</p>
Additional search domains	<p>Here you can add domains used by computer names. Separate multiple domains by commas. Only with Method Manual</p>
DHCP client ID	Currently no function
Require IPv4 addressing for this connection to complete	Currently no function

IPv6 Settings tab


Currently no function

25

Overviews

25.1 Pin layout and cables for data interfaces

25.1.1 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	Gray	6	6	Violet	4
7	RTS	4		5	7	Gray	8
8	CTR	5		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

25.1.2 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

25.2 Machine parameters

The following list shows the machine parameters that you can edit with the code number 123.

Related topics

















- Editing machine parameters with the **MPs for setters** application
Further information: "Machine parameters", Page 492





















25.2.1 List of user parameters











































Refer to your machine manual.


















- The machine manufacturer can make additional machine-specific parameters available as user parameters, so that you can configure the functions that are available.
- The machine manufacturer can adapt the structure and contents of the user parameters. The display on your machine may be different.

















Depiction in the configuration editor	MP number
 DisplaySettings Settings for screen displays	
 CfgDisplayData Settings for screen displays	100800
 axisDisplay Sequence and rules for displayed axes	100810
 x Key name of the axis	
 axisKey Key name of the axis	100810. [Index].01501
 name Axis designation	100810. [Index].01502
 rule Display rule for the axes	100810. [Index].01503
 axisDisplayRef Sequence and rules for display axes before crossing the reference marks	100811
 x Key name of the axis	
 axisKey Key name of the axis	100811. [Index].01501
 name Axis designation	100811. [Index].01502
 rule Display rule for the axes	100811. [Index].01503
 positionWinDisplay Type of position display in the positioning window	100803
 statusWinDisplay Type of position display in the status display	100804
 decimalCharacter Definition of decimal separator for position display	100805
 axisFeedDisplay Display of the feed rate in Manual Operation and El. Handwheel modes of operation	100806



















Depiction in the configuration editor		MP number
	spindleDisplay Display of spindle position in the position display	100807
	hidePresetTable Disable the PRESET MANAGEMENT soft key	100808
	displayFont Font size for program display in the operating modes Program Run Full Sequence, Program Run Single Block, and Positioning with Manual Data Input.	100812
	iconPrioList Sequence of icons in the display	100813
	compatibilityBits Settings for display behavior	100815
	axesGridDisplay Axes as list or group in the position display.	100806
	CfgPosDisplayPace Display step for the individual axes	101000
	xx Display step for position display in [mm] or [°]	
	displayPace Display step for position display in [mm] or [°]	101001
	displayPaceInch Display step for position display in [inch]	101002
	CfgUnitOfMeasure Definition of unit of measure in effect for display	101100
	unitOfMeasure Unit of measure for display and user interface	101101
	CfgProgramMode Format of the NC programs and cycle display	101200
	programInputMode MDI: Program entry in HEIDENHAIN Klartext format or ISO format	101201
	CfgDisplayLanguage Definition of the NC and PLC conversational language	101300
	ncLanguage NC conversational language	101301
	applyCfgLanguage Load the language of the NC control	101305
	plcDialogLanguage PLC conversational language	101302
	plcErrorLanguage PLC error message language	101303
	helpLanguage Language for online help	101304



















Depiction in the configuration editor		MP number
	CfgStartupData Behavior during control startup	101500
	powerInterruptMsg Acknowledge the Power interrupted message	101501
	opMode Operating mode that is switched to when the control has fully booted	101503
	subOpMode Submode to be activated for the operating mode entered in 'opMode'	101504
	CfgClockView Display mode for time of day	120600
	displayMode Display mode for time of day on the screen	120601
	timeFormat Time format of digital clock	120602
	CfgInfoLine Link row on/off	120700
	infoLineEnabled Enable/disable info line	120701
	CfgGraphics Settings for 3-D simulation graphics	124200
	modelType Model type of the 3D simulation graphics	124201
	modelQuality Model quality of the 3D simulation graphics	124202
	clearPathAtBlk Reset tool paths for new BLK FORM	124203
	extendedDiagnosis Write graphics journal files after restart	124204
	CfgPositionDisplay Settings for the digital readout	124500
	progToolCallDL Position display with TOOL CALL DL	124501
	CfgTableEditor Table editor configuration	125300
	deleteLoadedTool Behavior when deleting tools from the pocket table	125301
	indexToolDelete Behavior when deleting a tool's index entries	125302
	showResetColumnT Show the RESET T soft key	125303





















Depiction in the configuration editor		MP number
	CfgDisplayCoordSys Setting the coordinate systems for the display	127500
	transDatumCoordSys Coordinate system for the datum shift	127501
	CfgGlobalSettings GPS display settings	128700
	enableOffset Show offset in the GPS dialog	128702
	enableBasicRot Show an additive basic rotation in the GPS dialog	128703
	enableShiftWCS Show shift of W-CS in the GPS dialog	128704
	enableMirror Show mirror image in the GPS dialog	128712
	enableShiftMWCS Show shift of mW-CS in the GPS dialog	128711
	enableRotation Show rotation in the GPS dialog	128707
	enableFeed Show feed rate the GPS dialog	128708
	enableHwMCS M-CS coordinate system is selectable	128709
	enableHwWCS W-CS coordinate system is selectable	128710
	enableHwMWCS mW-CS coordinate system is selectable	128711
	enableHwWPLCS WPL-CS coordinate system is selectable	128712
	CfgRemoteDesktop Settings for Remote Desktop connections	100800
	connections List of Remote Desktop connections to be displayed	133501
	title Name of the OEM operating mode	133502
	dialogRes Name of a text	133502.00501
	text Language-sensitive text	133502.00502
	icon Path/name for optional icon graphic file	133503




















Depiction in the configuration editor		MP number
	locations List with positions where this Remote Desktop connection is displayed	133504
	x Operating mode	
	opMode Operating mode	133504. [Index].133401
	subOpMode Optional submode for the operating mode specified in 'opMode'	133504. [Index].133402
	ProbeSettings Configuration of the tool calibration	
	CfgTT Configuration of the tool calibration	122700
	TT140_x M function for spindle orientation	
	spindleOrientMode M function for spindle orientation	122704
	probingRoutine Probing routine	122705
	probingDirRadial Probing direction for tool radius measurement	122706
	offsetToolAxis Distance from lower edge of tool to upper edge of stylus	122707
	rapidFeed Rapid traverse in probing cycle for TT tool touch probe	122708
	probingFeed Probing feed rate for tool measurement with non-rotating tool	122709
	probingFeedCalc Calculation of the probing feed rate	122710
	spindleSpeedCalc Speed determination method	122711
	maxPeriphSpeedMeas Maximum permissible surface speed of the tool edge for radius measurement	122712
	maxSpeed Maximum permissible speed during tool measurement	122714

















Depiction in the configuration editor		MP number
	measureTolerance1 Maximum permissible measuring error for tool measurement with rotating tool (first measurement error)	122715
	measureTolerance2 Maximum permissible measuring error for tool measurement with rotating tool (second measurement error)	122716
	stopOnCheck NC stop during tool check	122717
	stopOnMeasurement NC stop during tool measurement	122718
	adaptToolTable Change the tool table during tool check and tool measurement	122719
	CfgTTRoundStylus Configuration of a round stylus	114200
	TT140_x Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum	
	centerPos Coordinates of the TT tool touch probe stylus contact center with respect to the machine datum	114201
	safetyDistToolAx Safety clearance around the probe contact of the TT tool touch probe for pre-positioning in the tool-axis direction	114203
	safetyDistStylus Safety zone around the stylus for pre-positioning	114204
	CfgTTRectStylus Configuration of a rectangular stylus	114300
	TT140_x Coordinates of the stylus center	
	centerPos Coordinates of the stylus center	114313
	safetyDistToolAx Set-up clearance above the stylus for pre-positioning	114317
	safetyDistStylus Safety zone around the stylus for pre-positioning	114318
	ChannelSettings Active kinematics	

Depiction in the configuration editor		MP number
	CH_xx Active kinematics	
	CfgActivateKinem Active kinematics	204000
	kinemToActivate Kinematics to be activated / Active kinematics	204001
	kinemAtStartup The kinematics to be activated during control start-up	204002
	CfgNcPgmBehaviour Specify the behavior of the NC program.	200800
	operatingTimeReset Reset the machining time when program starts.	200801
	plcSignalCycle PLC signal for the number of the pending machining cycle	200803
	CfgGeoTolerance Geometry tolerances	200900
	circleDeviation Permissible deviation of the radius	200901
	threadTolerance Permissible deviation in successive threads	200902
	moveBack Reserve for retraction movements	200903
	CfgGeoCycle Configuration of the fixed cycles	201000
	pocketOverlap Overlap factor for pocket milling	201001
	posAfterContPocket Traverse after machining the contour pocket	201007
	displaySpindleErr Display the Spindle is not rotating error message if M3/M4 is not active	201002
	displayDepthErr Display the Check the depth sign error message	201003
	apprDepCylWall Behavior when moving to wall of slot in the cylinder surface	201004
	mStrobeOrient M function for spindle orientation in machining cycles	201005

Depiction in the configuration editor		MP number
	suppressPlungeErr Do not show 'Plunging type is not possible' error message	201006
	restoreCoolant Behavior of M7 and M8 with Cycles 202 and 204	201008
	facMinFeedTurnSMAX Automatic feed rate reduction after attaining SMAX	201009
	suppressResMatlWar Do not show "Residual material" warning	201010
	CfgStretchFilter Geometry filter for filtering out linear elements	201100
	filterType Type of stretch filter	201101
	tolerance Maximum distance of the filtered to the unfiltered contour	201102
	maxLength Maximum length of the distance resulting from filtering	201103
	CfgThreadSpindle	113600
	sourceOverride Effective override potentiometer for feed rate during thread cutting	113603
	thrdWaitingTime Waiting time at reversal point in thread base	113601
	thrdPreSwitchTime Advanced switching time of spindle	113602
	limitSpindleSpeed Limit of spindle speed with Cycles 17, 207 and 18	113604
	CfgEditorSettings Settings for the NC editor	
	CfgEditorSettings Settings for the NC editor	105400
	createBackup Generate a backup file *.bak	105401
	deleteBack Behavior of the cursor after deletion of lines	105402
	cursorAround Behavior of the cursor on the first or last line	105403

Depiction in the configuration editor		MP number
	lineBreak Line break on NC blocks with more than one line	105404
	stdTNCHELP Activate help graphics when entering cycle data	105405
	toggleCyclDef Behavior of the cycle soft key row after a cycle entry	105406
	warningAtDEL Confirmation request when deleting an NC block.	105407
	maxLineGeoSearch Line number up to which a test of the NC program is to be run.	105408
	blockIncrement ISO programming: Block number increment	105409
	useProgAxes Specify programmable axes	105410
	enableStraightCut Allow or lock paraxial positioning blocks	105411
	maxLineCommandSrch Number of lines for searching for identical syntax elements	105412
	noParaxMode Allow/lock FUNCTION PARAXCOMP/PARAXMODE via soft key	105413
	CfgPgmMgt Settings for the file management	
	CfgPgmMgt Settings for the file management	122100
	dependentFiles Display of dependent files	122101
	CfgProgramCheck Settings for tool-usage files	
	CfgProgramCheck Settings for tool-usage files	129800
	autoCheckTimeOut Timeout for creation of tool-usage files	129803
	autoCheckPrg Create tool-usage file for NC program	129801
	autoCheckPal Create pallet-usage files	129802
	CfgUserPath Paths for the end user	
	CfgUserPath Paths for the end user	102200

Depiction in the configuration editor		MP number
	ncDir List of drives and/or directories	102201
	fn16DefaultPath Default output path for the FN16: F-PRINT function in the Program Run operating modes	102202
	fn16DefaultPathSim Default output path for the FN16: F-PRINT function in the Programming and Test Run operating modes	102203
	serialInterfaceRS232 Data record belonging to the serial port	
	CfgSerialPorts Data record belonging to the serial port	106600
	activeRs232 Enable the RS-232 interface in the program manager	106601
	baudRateLsv2 Data transfer rate for LSV2 communication in baud	106606
	CfgSerialInterface Definition of data records for the serial ports	106700
	RS232 Data transfer rate for communication in baud	
	baudRate Data transfer rate for communication in baud	106701
	protocol Communications protocol	106702
	dataBits Data bits in each transferred character	106703
	parity Type of parity checking	106704
	stopBits Number of stop bits	106705
	flowControl Type of data-flow checking	106706
	fileSystem File system for file operation via serial interface	106707
	bccAvoidCtrlChar Avoid control characters in the block check character (BCC)	106708
	rtsLow Idle state of the RTS line	106709
	noEotAfterEtx Behavior after reception of an ETX control character	106710

Depiction in the configuration editor		MP number
	Monitoring Monitoring settings for the user	
	CfgMonUser Monitoring settings for the user	129400
	enforceReaction The configured error reactions are enforced	129401
	showWarning Display warnings of monitoring tasks	129402
	CfgMonMbSection CfgMonMbSection defines monitoring tasks for a certain section of an NC program	02400
	tasks List of monitoring tasks to be performed	133701
	CfgMachineInfo General information of the machine operator	
	CfgMachineInfo General information of the machine operator	131700
	machineNickname Custom name (nickname) of the machine	131701
	inventoryNumber Inventory number or ID	131702
	image Photo or image of the machine	131703
	location Machine location	131704
	department Department or division	131705
	responsibility Responsible for the machine	131706
	contactEmail Contact email address	131707
	contactPhoneNumber Contact phone number	131708

25.3 Keycaps for keyboard units and machine operating panels
































































The keycaps with IDs 12869xx-xx and 1344337-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 361 (FS)







The keycaps with ID 679843-xx are suitable for use on the following keyboard units and machine operating panels:

- TE 360 (FS)









Keycaps for alphabetic keyboard

																		
ID 1286909	-08	-09	-10	-11	-12	-13	-14	-15	-16									
																		
ID 1286909	-17	-18	-19	-20	-21	-22	-23	-24	-25									
																		
ID 1286909	-26	-27	-28	-29	-30	-31	-32	-33	-34									
																		
ID 1286909	-35	-36	-	-38	-39	-	-41	-42	-43									
ID 1344337*)	-	-	-01*)	-	-	-02*)	-	-	-									
*) With tactile mark																		
																		
ID 1286909	-44	-45	-46	-47	-48	-49	-50	-51	-52									
																		
ID 1286909	-53	-54	-55	-56	-57	-58	-59	-60										
ID 679843	-	-	-	-F4	-	-	-F6	-										
																		
ID 1286911	-01	-02	-03	-04	-05													
																		
ID 1286914	-01	-03																
																		
ID 1286915	-01	-02	-03															
ID 1286917	-01																	




















Keycaps for operating aids

						
ID 1286909	-61	-62	-63	-64	-65	-66
ID 679843	–	-36	–	–	–	–











































Keycaps for operating modes

								
ID 1286909	-67	-68	-69	-70	-71	-72	-73	-74
ID 679843	–	–	-66	–	–	–	–	–











Keycaps for NC dialog

									
ID 1286909	-75	-76	-77	-78	-79	-80	-81	-82	-83
									
ID 1286909	-84	-85	-86	-87	-88	-89	-90	-91	-93
									
ID 1286909	-92								
ID 679843	-D6								









Keycaps for axis input and value input

									
	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange	Orange
ID 1286909	-94	-95	-96	-97	-98	-99	-0A	-4K	-4L
ID 679843	-C8	-D3	-53	-32	-31	–	–	-54	-88
									
ID 1286909	-0B	-0C	-0D	-0E	–	-0G	-0H	-2L	-2M
ID 1344337*)	–	–	–	–	-03*)	–	–	–	–
*) With tactile mark									
									
ID 1286909	-0K	-0L	-0M	-2N	-0P	-2P	-0R	-0S	-3N
									
ID 1286909	-3P	-3R	-3S	-4S	-4T	-0N	-3T	-3U	-3V
ID 679843	–	–	–	–	–	-E2	–	–	–
									
		Orange	Orange	Orange					
ID 1286909	-3W	–	–	–					
ID 679843	–	-55	-C9	-D4					
									
ID 1286914	-02	-04							

Keycaps for navigation

								
ID 1286909	-0T	-0U	-0V	-0W	–	-0Y	-0Z	-1A
ID 1344337*)	–	–	–	–	-04*)	–	–	–
*) With tactile mark								
								
ID 1286909	-1B	-1C						
ID 679843	-42	-41						

Keycaps for machine functions

									
ID 1286909	-1D	-1E	-1F	-1G	-1H	-1K	-1L	-1M	-1N
ID 679843	-09	-07	-05	-11	-13	-03	-16	-17	-06
									
ID 1286909	-1P	-1R	-1S	-1T	-1U	-1V	-1W	-1X	-1Y
ID 679843	-10	-14	-23	-22	-24	-29	-02	-21	-20
									
ID 1286909	-1Z	-2A	-2B	-2C	-2D	-2E	-2H	-2K	-2R
ID 679843	-25	-28	-01	-26	-27	-30	-57	-56	-04
									
ID 1286909	-	-2T	-2U	-2Z	-3A	-3E	-3F	-3G	-3H
ID 1344337*)	-05*)	-	-	-	-	-	-	-	-
ID 679843	-15	-08	-12	-59	-60	-40	-73	-76	-74
*) With tactile mark									
									
ID 1286909	-3L	-3M	-3X	-3Y	-3Z	-4A	-4B	-4C	-4D
ID 679843	-C6	-75	-46	-47	-F2	-67	-51	-68	-99
									
ID 1286909	-4E	-4F	-4H	-4M	-4N	-4P	-4R	-4U	-06
ID 679843	-B8	-B7	-45	-69	-70	-B2	-B1	-52	-18
									
ID 1286909	-07	-2F	-2G	-2V	-2W	-2X	-2Y	-3B	-3C
ID 679843	-19	-	-	-	-	-	-	-	-
									
ID 1286909	-3D	-3K	-4G	-	-	-	-	-	-
ID 679843	-	-	-	-43	-44	-91	-92	-93	-94

ID 679843	-B3	-B4	-B5	-B6	-B9	-C1	-C2	-C3	-C4
ID 679843	-C5	-D9	-E1	-61	-62	-63	-64	-A2	-A3
ID 679843	-95	-96	-A1	-C7	-A4	-A5	-A6	-A9	-E3
ID 679843	-E4	-E6	-E7	-E8	-48	-49	-50	-65	-71
ID 679843	-D8	-90	-89	-D7	-72	-F3	-97	-98	-E5

Other keycaps

			Orange	Green	Red				
ID 1286909	-01	-02	-05	-03	-04	-	-	-	-
ID 679843	-33	-34	-35	-	-	-38	-39	-A7	-A8
ID 679843	-D5	F5							

If you need keycaps with additional symbols, please contact HEIDENHAIN.

Index

3

3D basic rotation.....	195
3D calibration.....	320
3D-ROT menu.....	199

A

About the product.....	41
About the User's Manual.....	31
ACC.....	237
Accessories.....	61
Active Chatter Control (ACC).....	237
Adaptive Feed Control AFC.....	230
Additional documentation.....	33
Additional software.....	506
Additional status display.....	101
Additive basic rotation.....	242
Additive offset.....	241
AFC.....	230
Basic settings.....	408
Programming.....	232
Teach-in cut.....	235
Angle encoder.....	137
Application	
Functional safety.....	440
Manual operation.....	130
MDI.....	329
MPs for setters.....	492
MPs for users.....	492
Retract.....	352
Settings.....	445
Setup.....	305
Axes	
Moving.....	132
Referencing.....	124
Axis designation.....	136
Axis display.....	94
Axis key.....	132

B

Backup.....	489
Basic coordinate system.....	182
Basic rotation.....	195
B-CS.....	182
Block scan.....	342
Multi-level.....	346
Pallet table.....	348
Point table.....	347
Returning to the contour.....	349
Single-level.....	345

C

CAD file.....	275
CAD Import.....	286
Contour, saving.....	287
Position, saving.....	288

CAD-Viewer.....	275
Calibrating.....	319
Length.....	321
Radius.....	322
Calibration	
Deflection behavior.....	323
Cartesian coordinate system.....	179
CFG file.....	221
Chatter control.....	237
Code number.....	448
Collision monitoring.....	204
Activating.....	208
Fixtures.....	210
Compensation table	
Program run.....	351
Connecting cable.....	514
Connection	
Network.....	459
Network drive.....	456
Connection wizard.....	468
Contact.....	39
Control	
Powering off.....	126
Powering on.....	122
Control's user interface.....	63
Control-in-operation symbol.....	338
Conversational language.....	453
Coordinate system.....	178
Basics.....	179
Coordinate origin.....	179
CR2.....	144

D

Data backup.....	489, 506
Data interface.....	502
OPC UA.....	465
pin layout.....	514
Data transfer	
Software.....	504
Date and time.....	453
Datum table	
Program run.....	351
DCM.....	204
Activating.....	208
Fixtures.....	210
DNC.....	470
Dressing tool table.....	389
Columns.....	390
Dynamic Collision Monitoring (DCM).....	204

E

Embedded Workspace.....	434
Encoder.....	137
Error message.....	301
Error window.....	301
Ethernet interface.....	459, 514
Configuration.....	508

Setting.....	461
Extended Workspace.....	436
External access.....	470

F

FCL.....	55
Feature Content Level.....	55
Feed control.....	230
Feed factor.....	250
Feed rate limit.....	337
File	
Backing up.....	506
Tools.....	506
Firewall.....	484
First steps.....	81
Program run.....	89
Setup.....	87
Tool.....	83
Fixture monitoring.....	210
CFG file.....	212, 221
Integrating.....	213
M3D file.....	211
STL file.....	211
FreeTurn tool.....	148
Functional safety (FS).....	437
Functional safety (FS) operating modes.....	439

G

General status display.....	93
Gestures.....	70
Global Program Settings.....	238
Activating.....	240
Additive basic rotation.....	242
Additive offset.....	241
Feed factor.....	250
Handwheel superimpositioning.....	247
Mirroring.....	244
Overview.....	240
Resetting.....	241
Rotation.....	246
Shift.....	243
Shift mW-CS.....	245
GPS.....	238
Activating.....	240
Additive basic rotation.....	242
Additive offset.....	241
Feed factor.....	250
Handwheel superimpositioning.....	247
Mirroring.....	244
Overview.....	240
Resetting.....	241
Rotation.....	246
Shift.....	243
Shift mW-CS.....	245
Grinding tool table.....	382

Columns.....	382	Machine coordinate system.....	179	Tables.....	358
H		Machine datum.....	138	Operating modes	
Handwheel.....	415	Machine information.....	450	Overview.....	64
Operating elements.....	417	Machine parameters.....	492	Operating system.....	497
Wireless handwheel.....	423	List.....	515	P	
Handwheel mode.....	130	Overview.....	514	Parameter list.....	120
Handwheel superimpositioning		Machine settings.....	448	Pin layout	
Global Program Settings.....	247	Machine times.....	452	data interface.....	514
Virtual tool axis VT.....	248	Machining time.....	117	Place of operation.....	43
Hardware.....	56	Manual axis.....	351	Pocket table.....	396
HEROS.....	497	Manual operation.....	130	Portscan.....	487
HEROS function		Manual tilting, activating.....	199	Position display.....	94
Overview.....	498	Maximum feed rate.....	337	Mode.....	118
Settings application.....	445	M-CS.....	179	Status overview.....	99
HEROS menu.....	498	MDI.....	329	Position encoder.....	137
HEROS tool.....	506	Mesh.....	292	Positioning with Manual Data	
Host computer operation.....	470	Message.....	301	Input.....	329
I		Message menu.....	301	Powering off.....	126
Icons, miscellaneous.....	76	Mid-program startup.....	342	Powering on.....	122
I-CS.....	188	Mirroring		Powering on and off.....	121
Incremental jog positioning.....	133	GPS.....	244	Preset.....	193
Indexed tool.....	146	MOD menu.....	445	Activating.....	197
Input coordinate system.....	188	Overview.....	446	Inches.....	407
Integrated product help		Monitor.....	56	Scratching.....	194
TNCguide.....	36	Moving		Setting.....	196
Interface.....	63	Axis key.....	132	Preset management.....	193
Ethernet.....	459	Incremental jog.....	133	Preset table.....	403
OPC UA.....	465	N		Columns.....	403
J		NC fundamentals.....	136	Inches.....	407
Jog increment.....	133	Network.....	459	Write-protection.....	405
K		Configuration.....	508	Printer.....	471, 471
Keyboard.....	58	Setting.....	461	Process Monitoring.....	252
Formula.....	300	Network configuration.....	508	FeedOverride.....	267
NC functions.....	299	DCB.....	511	MinMaxTolerance.....	262
Text.....	300	Ethernet.....	511	SignalDisplay.....	266
Virtual.....	298	General.....	510	SpindleOverride.....	266
Keys.....	70	IPv4 Settings.....	512	StandardDeviation.....	265
Kinematics.....	448	IPv6 Settings.....	512	Workspace.....	253
KinematicsDesign.....	221	Proxy.....	511	Program Run.....	334
L		Security.....	511	Block scan.....	342
Language.....	453	Network drive.....	456	Canceling.....	338
Changing.....	454	Connecting.....	456	Compensation table.....	351
Conversational language		Network setting		Contextual reference.....	339
Changing.....	454	DHCP Server.....	464	Datum table.....	351
License settings.....	469	Ping.....	464	Global Program Settings.....	238
Licensing terms.....	55	Routing.....	464	Manual traverse.....	341
Linear encoder.....	137	SMB share.....	464	Retract.....	352
M		Status.....	462, 463	Returning to the contour.....	349
M92 datum M92-ZP.....	138	Notes, types of.....	34	Program run time.....	117
Machine		O		Proper and intended operation.....	42
Powering off.....	126	OPC UA NC Server.....	465	Q	
Powering on.....	122	Connection wizard.....	468	Q-parameter list.....	120
Machine axes, moving.....	132	License settings.....	469	Q parameters	
		Operating elements.....	70	Show.....	120
		Operating mode			
		Program Run.....	334		

R

Reference, traversing.....	124
Reference point.....	138
Reference system.....	178
Basic coordinate system.....	182
Input coordinate system.....	188
Machine coordinate system..	179
Tool coordinate system.....	190
Working plane coordinate system.....	186
Workpiece coordinate system.....	184
Remote Desktop Manager.....	478
External computer, shutting down.....	478
VNC.....	479
Windows Terminal Service....	479
Remote Service.....	488
Restarting.....	126
Restore.....	489
Retract.....	352
Returning to the contour.....	349
Rotation	
GPS.....	246
Run time	
Machine information.....	452
Program run.....	117

S

Safety precaution.....	44
Content.....	34
Scratching.....	194
Security software SELinux.....	455
SELinux.....	455
Service file.....	301
Creating.....	304
Setting	
Network.....	461
Settings.....	445
VNC.....	474
Settings application	
Overview.....	446
Setting up a vice.....	219
Setting up fixtures.....	213
Sequence.....	218
Vice.....	219
Shift.....	243
Shift mW-CS.....	245
SIK menu.....	451
Simulation status.....	116
Software number.....	48
Software option.....	48, 451
Split screen layout of User's Manual.....	33
Status display.....	91
Additional.....	101
Axis.....	94
General.....	93

Overview.....	92
Position.....	94
Simulation.....	116
technology.....	95
Status overview.....	99
Control bar.....	99
Control-in-operation symbol..	100
Step index.....	146
STL file	
Optimizing.....	292
System time.....	453

T

Table	
Preset table.....	403
Tool tables.....	367
Target group.....	32
Taskbar.....	501
TCP.....	143
T-CS.....	190
Telemaintenance.....	488
Tilting	
Manually.....	198
Time.....	453
Time zone.....	453
TIP.....	142
TLP.....	143
TNCdiag.....	491
TNCremo.....	504
Tool.....	139
Defining.....	165
Dressing tool.....	389
Exporting and importing.....	166
FreeTurn.....	148
Grinding tool.....	382
Overview.....	140
Preset.....	140
Table.....	367
Tool data, required.....	152
Touch probe.....	392
Turning tool.....	377
Tool carrier management.....	169
Tool carrier reference point.....	141
Tool center point TCP.....	143
Tool change position.....	138
Tool coordinate system.....	190
Tool data.....	145
Exporting.....	168
Importing.....	167
Required.....	152
Tool ID number.....	145
Tooling list.....	402
Tool location point TLP.....	143
Tool management.....	165
Tool name.....	145
Tool radius 2 center CR2.....	144
Tool rotation point TRP.....	144
Tool table.....	368

Columns.....	368
Inches.....	395
Input options.....	368
Tool tip TIP.....	142
Tool type	
Tool data, required.....	152
Tool types.....	150
Tool usage file.....	398
Tool usage test.....	172
Touch probe	
3D calibration.....	323
Calibrating.....	319
Length, calibrating.....	321
Radius, calibrating.....	322
Setting up fixtures.....	213
Setup.....	430
Touch probe cycle	
Manual.....	305
Touch probe data.....	393
Touch probe function.....	305
Overview.....	308
Touch probe monitoring.....	324
Touch probes	
Radio transmission.....	430
Touch probe table.....	392
Columns.....	393
Touchscreen.....	56
Traverse	
Handwheel.....	415
Traverse limit.....	448
TRP.....	144
Turning tool table.....	377
Columns.....	378
T usage order.....	400

U

Unit of measure.....	448
User aids.....	297
User interface of the control.....	63
User parameters.....	492
List.....	515

V

Virtual keyboard.....	298
VNC.....	474

W

W-CS.....	184
Window Manager.....	502
Wireless handwheel.....	423
Configuring.....	425
Working plane.....	136
Working plane, tilting	
Fundamentals.....	198
Head rotary axis.....	199
Manually.....	198
Table rotary axis.....	199
Working plane coordinate	

- system..... 186
- Workpiece coordinate system... 184
- Workpiece datum..... 138
- Workpiece preset..... 138, 193
- Workspaces..... 66
 - Overview..... 67
- WPL-CS..... 186
- Write-protection, preset table..... 405
- Write protection for preset table
 - Activating..... 406
 - Removing..... 406

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

E-mail: info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104

E-mail: service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101

E-mail: service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103

E-mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102

E-mail: service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106

E-mail: service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com

The Information Site for
HEIDENHAIN Controls

Klartext App

Klartext on your
mobile device

Google
Play Store

Apple
App Store



Touch probes from HEIDENHAIN

help you reduce non-productive time and improve the dimensional accuracy of the finished workpieces.

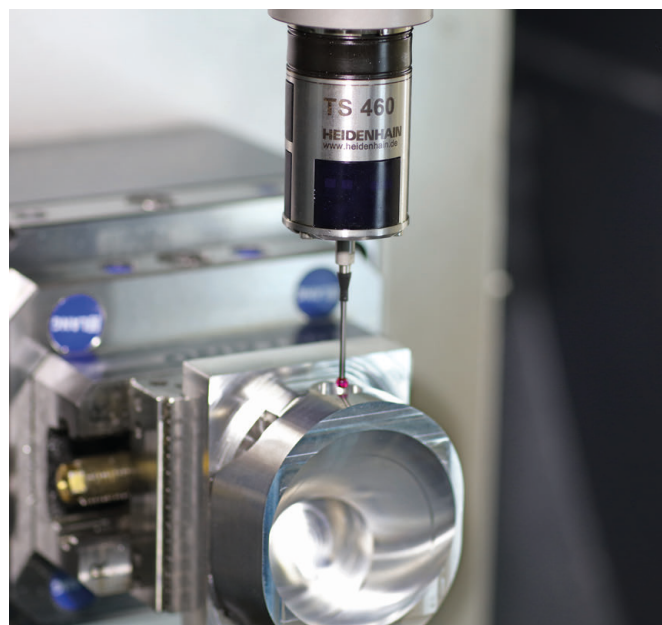
Workpiece touch probes

TS 150, TS 260, Signal transmission by cable
TS 750

TS 460, TS 760 Radio or infrared transmission

TS 642, TS 740 Infrared transmission

- Workpiece alignment
- Preset setting
- Workpiece measurement



Tool touch probes

TT 160 Signal transmission by cable

TT 460 Infrared transmission

- Tool measurement
- Wear monitoring
- Tool breakage detection

