



HEIDENHAIN

TNC7 go

User's Manual
Programming and Testing

NC Software
81762x-20

English (en)
10/2025

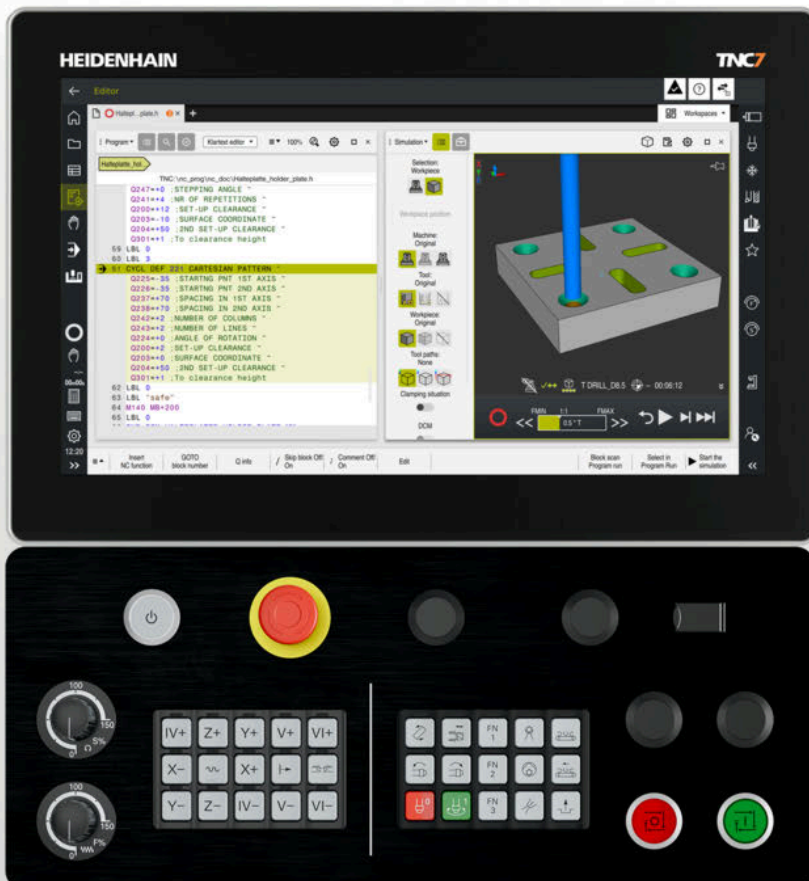


Table of contents

1	About the User's Manual.....	31
2	About the product.....	43
3	First steps.....	79
4	NC and programming fundamentals.....	105
5	Machining mode.....	149
6	Workpiece blank.....	151
7	Tools.....	157
8	Traverse movements.....	171
9	Programming techniques.....	177
10	Point definitions.....	205
11	Cycles for drilling, centering, and thread machining.....	229
12	Milling cycles.....	277
13	Coordinate transformation.....	307
14	Compensations.....	339
15	Files.....	347
16	Control functions.....	375
17	Miscellaneous functions.....	381
18	Programming with variables.....	391
19	ISO.....	469
20	User aids.....	489
21	The Simulation workspace.....	527
22	Touch probes.....	553
23	Touch-probe cycles for tools (#17 / #1-05-1).....	561
24	Tables.....	579
25	Overviews.....	615

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.....	36
1.5	User's Manual as integrated product aid: TNCguide.....	37
1.5.1	Searching in TNCguide.....	40
1.5.2	Copying NC examples to clipboard.....	41
1.6	Contacting the editorial staff.....	42

2	About the product.....	43
2.1	The TNC7 go.....	44
2.1.1	Proper and intended use.....	44
2.1.2	Data Architecture and Access Options.....	45
2.1.3	Intended place of operation.....	46
2.2	Safety precautions.....	47
2.3	Software.....	49
2.3.1	Software options.....	50
2.3.2	Information on licensing and use.....	52
2.4	Hardware.....	53
2.4.1	Touchscreen and keyboard unit.....	53
2.5	Areas of the control's user interface.....	57
2.6	Overview of the operating modes.....	59
2.7	Workspaces.....	61
2.7.1	Operating elements within the workspaces.....	61
2.7.2	Icons within the workspaces.....	62
2.7.3	Overview of workspaces.....	62
2.8	Operating elements.....	65
2.8.1	Common gestures for the touchscreen.....	65
2.8.2	Operating elements of the keyboard unit.....	65
2.8.3	Keyboard shortcuts for operating the control.....	72
2.8.4	Icons on the control's user interface.....	73
2.8.5	The Desktop menu workspace.....	76

3	First steps.....	79
3.1	Chapter overview.....	80
3.2	Switching the machine and the control on.....	81
3.3	Programming and simulating a workpiece.....	83
3.3.1	Programming the workpiece.....	83
3.3.2	Simulating a workpiece.....	102
3.4	Switching the machine off.....	104

4	NC and programming fundamentals.....	105
4.1	NC fundamentals.....	106
4.1.1	Programmable axes.....	106
4.1.2	Designation of the axes of milling machines.....	106
4.1.3	Position encoders and reference marks.....	107
4.1.4	Presets in the machine.....	108
4.2	Programming possibilities.....	109
4.2.1	Miscellaneous functions M	109
4.2.2	Subprograms and program-section repeats.....	109
4.2.3	Control structures.....	109
4.2.4	Programming with variables.....	109
4.2.5	Possible methods for editing.....	109
4.3	Programming fundamentals.....	114
4.3.1	Contents of an NC program.....	114
4.3.2	The Editor operating mode.....	116
4.3.3	The Program workspace.....	118
4.3.4	The Insert NC function window.....	133
4.3.5	Text mode.....	135
4.4	Working with cycles.....	137
4.4.1	General information on cycles.....	137
4.4.2	General information about touch probe cycles.....	145
4.4.3	Machine-specific cycles.....	147
4.4.4	Available cycle groups.....	148

5	Machining mode.....	149
5.1	Switching the operating mode with FUNCTION MODE.....	150

6	Workpiece blank.....	151
6.1	Defining a workpiece blank with BLK FORM.....	152
6.1.1	Cuboid workpiece blank with BLK FORM QUAD	153
6.1.2	Cylindrical workpiece blank with BLK FORM CYLINDER	154
6.1.3	STL file as workpiece blank with BLK FORM FILE	155

7	Tools.....	157
7.1	Fundamentals.....	158
7.2	Presets on the tool.....	159
7.2.1	Tool carrier reference point.....	159
7.2.2	Tool tip TIP	160
7.2.3	Tool center point (TCP, tool center point).....	160
7.2.4	Tool location point (TLP, tool location point).....	161
7.3	Tool call.....	162
7.3.1	Using TOOL CALL to call a tool.....	162
7.3.2	Cutting data.....	166
7.3.3	Tool pre-selection using TOOL DEF	169

8	Traverse movements.....	171
8.1	Fundamentals of coordinate definitions.....	172
8.1.1	Cartesian coordinates.....	172
8.1.2	Absolute input.....	173
8.1.3	Incremental entries.....	174
8.2	Fundamentals of traverse movements.....	175

9	Programming techniques.....	177
9.1	Subprograms and program section repeats with the label LBL.....	178
9.2	Nesting with LBL.....	182
9.2.1	Examples.....	183
9.3	Control structures.....	185
9.3.1	Fundamentals.....	185
9.3.2	Case analyses.....	185
9.3.3	Program loops.....	188
9.3.4	Advanced controlling of program loops.....	192
9.4	Selection functions.....	194
9.4.1	Overview of selection functions.....	194
9.4.2	Calling an NC program with CALL PGM	194
9.4.3	Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM	196
9.5	Cycle 12 PGM CALL.....	199
9.5.1	Cycle parameters.....	200
9.6	NC sequences for reuse.....	201

10	Point definitions.....	205
10.1	Point tables.....	206
10.1.1	Selecting the point table in the NC program with SEL PATTERN	207
10.1.2	Calling the cycle with a point table.....	207
10.2	Pattern definition with PATTERN DEF.....	208
10.2.1	Defining individual machining positions.....	210
10.2.2	Defining a single row.....	211
10.2.3	Defining an individual pattern.....	212
10.2.4	Defining an individual frame.....	213
10.2.5	Defining a full circle.....	215
10.2.6	Defining a pitch circle.....	216
10.2.7	Example: Using cycles in conjunction with PATTERN DEF.....	217
10.3	Pattern definition cycles.....	219
10.3.1	Overview.....	219
10.3.2	Cycle 220 POLAR PATTERN	220
10.3.3	Cycle 221 CARTESIAN PATTERN	223
10.3.4	Programming examples.....	226

11 Cycles for drilling, centering, and thread machining.....	229
11.1 Overview.....	230
11.2 Drilling.....	232
11.2.1 Cycle 200 DRILLING	232
11.2.2 Cycle 201 REAMING	236
11.2.3 Cycle 202 REAMING	238
11.2.4 Cycle 203 UNIVERSAL DRILLING	242
11.2.5 Cycle 205 UNIVERSAL PECKING	247
11.2.6 Cycle 241 SINGLE-LIP D.H.DRLNG	254
11.3 Countersinking and centering.....	262
11.3.1 Cycle 204 BACK BORING	262
11.3.2 Cycle 240 CENTERING	266
11.4 Tapping.....	269
11.4.1 Cycle 206 TAPPING	269
11.4.2 Cycle 207 RIGID TAPPING	272

12 Milling cycles.....	277
12.1 Overview.....	278
12.2 Milling pockets.....	279
12.2.1 Cycle 251 RECTANGULAR POCKET	279
12.2.2 Cycle 253 SLOT MILLING	284
12.3 Milling studs.....	289
12.3.1 Cycle 256 RECTANGULAR STUD	289
12.3.2 Programming examples.....	295
12.4 Milling planes.....	297
12.4.1 Cycle 233 FACE MILLING	297

13	Coordinate transformation.....	307
13.1	Reference systems.....	308
13.1.1	Overview.....	308
13.1.2	Basics of coordinate systems.....	309
13.1.3	Machine coordinate system M-CS	310
13.1.4	Basic coordinate system B-CS	311
13.1.5	Workpiece coordinate system W-CS	312
13.1.6	Input coordinate system I-CS	313
13.1.7	Tool coordinate system T-CS	314
13.2	NC functions for preset management.....	315
13.2.1	Overview.....	315
13.2.2	Activating the preset with PRESET SELECT	315
13.2.3	Copying the preset with PRESET COPY	317
13.2.4	Correcting the preset with PRESET CORR	318
13.3	Datum table.....	320
13.3.1	Activating a datum table in the NC program.....	321
13.4	Coordinate transformation cycles.....	322
13.4.1	Fundamentals.....	322
13.4.2	Cycle 8 MIRRORING	323
13.4.3	Cycle 11 SCALING FACTOR	324
13.4.4	Cycle 26 AXIS-SPECIFIC SCALING	326
13.4.5	Cycle 247 PRESETTING	327
13.4.6	Example: Coordinate conversion cycles.....	329
13.5	NC functions for coordinate transformation.....	331
13.5.1	Overview.....	331
13.5.2	Datum shift with TRANS DATUM	332
13.5.3	Mirroring with TRANS MIRROR	334
13.5.4	Scaling with TRANS SCALE	335
13.5.5	Resetting with TRANS RESET	337

14	Compensations.....	339
14.1	Tool compensation for tool length and tool radius.....	340
14.2	Tool radius compensation.....	344

15 Files.....	347
15.1 File management.....	348
15.1.1 Basic information.....	348
15.1.2 The Open File workspace.....	359
15.1.3 The Quick selection workspaces.....	360
15.1.4 The Document workspace.....	361
15.1.5 Adapting files.....	363
15.1.6 USB devices.....	364
15.2 The Text editor workspace.....	366
15.3 Programmable file functions.....	370

16	Control functions.....	375
16.1	Functions for controlling program run.....	376
16.1.1	Overview.....	376
16.1.2	Programmed dwell time with FUNCTION DWELL	376
16.1.3	Cyclic dwell time with FUNCTION FEED DWELL	377
16.2	Cycles with control function.....	378
16.2.1	Overview.....	378
16.2.2	Cycle 9 DWELL TIME	379
16.2.3	Cycle 13 ORIENTATION	380

17 Miscellaneous functions.....	381
17.1 Miscellaneous functions M	382
17.2 Overview of miscellaneous functions.....	383
17.3 Miscellaneous functions for coordinate entries.....	385
17.3.1 Traversing in the machine coordinate system M-CS with M91	385
17.3.2 Traversing in the M92 coordinate system with M92	387
17.4 Miscellaneous functions for path behavior.....	388
17.4.1 Reducing the feed rate for infeed movements with M103	388
17.4.2 Interpreting the feed rate as mm/rev with M136	388
17.4.3 Retracting in the tool axis with M140	389

18 Programming with variables.....	391
18.1 Overview of variable programming.....	392
18.2 Variables: Q, QL, QR, QS parameters and named parameters.....	393
18.2.1 Basics.....	393
18.2.2 The Q parameter list window.....	399
18.2.3 Preassigned Q parameters.....	402
18.2.4 The Basic arithmetic folder.....	404
18.2.5 The Trigonometric functions folder.....	407
18.2.6 The Circle calculation folder.....	408
18.2.7 The Jump commands folder.....	410
18.2.8 Special functions for programming with variables.....	412
18.2.9 NC functions for accessing tables with variables.....	423
18.2.10 Formulas in the NC program.....	427
18.3 String functions.....	432
18.3.1 Assigning text to a string parameter.....	434
18.3.2 Concatenating values of string parameters.....	435
18.3.3 Converting the value of a string parameter to number.....	435
18.3.4 Converting numerical values to texts.....	436
18.3.5 Copying a substring from a string parameter.....	436
18.3.6 Searching for a substring within QS parameter contents.....	436
18.3.7 Determining the number of characters in QS parameter contents.....	437
18.3.8 Comparing the lexical order of two texts.....	437
18.3.9 Applying the contents of a machine parameter.....	438
18.4 Format strings.....	439
18.5 Defining counters with FUNCTION COUNT.....	443
18.5.1 Example.....	444
18.6 Program defaults for cycles.....	445
18.6.1 Overview.....	445
18.6.2 Entering GLOBAL DEF definitions.....	446
18.6.3 Using GLOBAL DEF information.....	446
18.6.4 Global data valid everywhere.....	447
18.6.5 Global data for drilling operations.....	448
18.6.6 Global data for milling operations with pocket cycles.....	449
18.6.7 Global data for milling operations with contour cycles.....	449
18.6.8 Global data for positioning behavior.....	450
18.6.9 Global data for probing functions.....	450

18.7	Table access with SQL statements.....	451
18.7.1	Fundamentals.....	451
18.7.2	Binding a variable to a table column with SQL BIND	454
18.7.3	Reading out a table value with SQL SELECT	455
18.7.4	Executing SQL statements with SQL EXECUTE	457
18.7.5	Reading a line from a result set with SQL FETCH	461
18.7.6	Discarding changes to a transaction using SQL ROLLBACK	462
18.7.7	Completing a transaction with SQL COMMIT	464
18.7.8	Changing the row of a result set with SQL UPDATE	465
18.7.9	Creating a new row in the result set with SQL INSERT	467

19 ISO.....	469
19.1 Fundamentals.....	470
19.2 ISO syntax.....	472
19.3 Cycles.....	486
19.4 Klartext functions in ISO programming.....	487

20 User aids.....	489
20.1 The Help workspace.....	490
20.2 Virtual keyboard of the control bar.....	493
20.2.1 Opening and closing the virtual keyboard.....	495
20.3 The GOTO function.....	496
20.3.1 Selecting an NC block or row with GOTO	497
20.4 Adding comments.....	498
20.4.1 Adding a comment as an NC block.....	498
20.4.2 Adding a comment in an NC block.....	498
20.4.3 Commenting an NC block out or in.....	498
20.5 Hiding NC blocks.....	499
20.5.1 Hiding or showing NC blocks.....	499
20.6 Structuring of NC programs.....	500
20.6.1 Adding a structure item.....	500
20.7 The Structure column in the Program workspace.....	501
20.7.1 Editing an NC block using the structure.....	502
20.7.2 Marking NC blocks using the structure.....	503
20.8 The Search column in the Program and Text editor workspaces.....	504
20.8.1 Search for and replace syntax elements.....	508
20.9 Program comparison.....	509
20.9.1 Applying differences to the active NC program.....	510
20.10 Context menu.....	511
20.11 Calculator.....	516
20.11.1 Opening and closing the calculator.....	516
20.11.2 Actual position capture.....	517
20.11.3 Selecting a result from the history.....	517
20.11.4 Deleting the history.....	517
20.12 Cutting data calculator.....	518
20.12.1 Calculating the cutting data without tables.....	521
20.12.2 Calculating cutting data with tables.....	521
20.13 Issuing notifications with FUNCTION REPORT.....	522
20.14 Issuing text with FUNCTION PRINT.....	524

21 The Simulation workspace.....	527
21.1 Fundamentals.....	528
21.2 Pre-defined views.....	538
21.3 Exporting a simulated workpiece as STL file.....	539
21.3.1 Saving a simulated workpiece as STL file.....	540
21.4 Advanced checks in the simulation.....	541
21.5 Measuring function.....	542
21.5.1 Measuring the difference between the workpiece blank and the finished part.....	543
21.6 Cutout view in the simulation.....	544
21.6.1 Shifting the sectional plane.....	544
21.7 The Rewind function.....	545
21.7.1 Analyzing a simulation warning.....	546
21.8 Model comparison.....	547
21.9 Center of rotation in the simulation.....	549
21.9.1 Setting the center of rotation to a corner of the simulated workpiece.....	549
21.10 Simulation speed.....	550
21.11 Simulating an NC program up to a certain NC block.....	551
21.11.1 Simulating an NC program up to a certain NC block.....	552

22 Touch probes.....	553
22.1 Calibrating a tool touch probe (#17 / #1-05-1).....	554
22.1.1 Overview.....	554
22.1.2 Fundamentals.....	554
22.1.3 Cycle 480 CALIBRATE TT (#17 / #1-05-1).....	555
22.1.4 Cycle 484 CALIBRATE IR TT (#17 / #1-05-1).....	557

23 Touch-probe cycles for tools (#17 / #1-05-1)	561
23.1 Overview	562
23.2 Fundamentals	563
23.2.1 Application	563
23.2.2 Measuring a tool of length 0	563
23.2.3 Setting machine parameters	564
23.2.4 Entries in the tool table for milling tools	566
23.3 Measurement of milling cutters (#17 / #1-05-1)	568
23.3.1 Cycle 481 CAL. TOOL LENGTH (#17 / #1-05-1)	568
23.3.2 Cycle 482 CAL. TOOL RADIUS (#17 / #1-05-1)	571
23.3.3 Cycle 483 MEASURE TOOL (#17 / #1-05-1)	575

24 Tables.....	579
24.1 The Tables operating mode.....	580
24.1.1 Editing the contents of tables.....	582
24.2 The Create new table window.....	584
24.3 The Table workspace.....	586
24.4 The Form workspace for tables.....	594
24.4.1 Adding a column in the workspace.....	596
24.5 Importing and exporting of table contents.....	597
24.5.1 Exporting table contents.....	599
24.5.2 Importing table contents.....	600
24.6 Accessing table values.....	602
24.6.1 Fundamentals.....	602
24.6.2 Reading table values with TABDATA READ	603
24.6.3 Writing table values with TABDATA WRITE	603
24.6.4 Adding table values with TABDATA ADD	605
24.7 Freely definable tables *.tab.....	606
24.7.1 Changing table properties of freely definable tables.....	608
24.8 Point table *.pnt.....	609
24.8.1 Hiding individual points during machining.....	609
24.9 Datum table *.d.....	610
24.9.1 Editing a datum table.....	611
24.10 Tables for cutting data calculation.....	612

25	Overviews.....	615
25.1	Special functions defining the machine behavior.....	616

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
 - Using the Simulation mode to test the NC programs
 - Eliminating possible errors during program test

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

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



HEIDENHAIN offers separate information products for other target groups:

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

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

HEIDENHAIN training portal

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

1.2 Available user documentation

User's Manual

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

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

- As a PDF file containing a **complete edition** of the User's Manual with all contents
ID: 1441440-xx
https://content.heidenhain.de/doku/tnc_guide/html/en/
- As an HTML file with all contents for use as the integrated product aid **TNCguide**: directly on the control
https://content.heidenhain.de/doku/tnc_guide/html/en/
- As a printed version, sub-divided into the modules below:
 - The **Setup and Program Run** User's Manual contains all information needed for setting up the machine and for executing NC programs.
ID: 1441439-xx
 - The **Programming and Testing** User's Manual contains all information needed for creating and testing NC programs.
ID: 1441438-xx

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

Further information: "Proper and intended use", Page 44



Complete edition and TNCguide for the TNC7 go

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

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

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

Further information products for users

The following information products are available:

- The **Overview of New and Modified Software Functions** informs you about the innovations of specific software versions.
https://content.heidenhain.de/doku/tnc_guide/html/en/
- **Overview of the machine parameters, error numbers and system data**, providing the following functions:
 - Machine parameters of the **MPs for setters** application
 - Preassigned error numbers of the **FN 14: ERROR** NC function (ISO: **D14**)
 - System data readable with the **FN 18: SYSREAD** (ISO: **D18**) and **SYSSTR** NC functions
https://content.heidenhain.de/doku/tnc_guide/html/en/
- **HEIDENHAIN brochures** inform you about products and services from HEIDENHAIN (e.g., software options of the control).
HEIDENHAIN brochures
- The **NC Solutions** database offers solutions for frequently occurring tasks.
HEIDENHAIN NC solutions

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
Caution indicates danger to material or data. If you do not follow the avoidance instructions, the hazard could result in things other than personal injury, such as property damage .

Sequence of information in precautionary statements

All precautionary statements comprise the following four sections:

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

Informational notes

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

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



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



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



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

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 aid: TNCguide

Application

The integrated product aid **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 use", Page 44

Related topics

- The **Help** workspace

Further information: "The Help workspace", Page 490

Requirement

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

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

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



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

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

Description of function

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

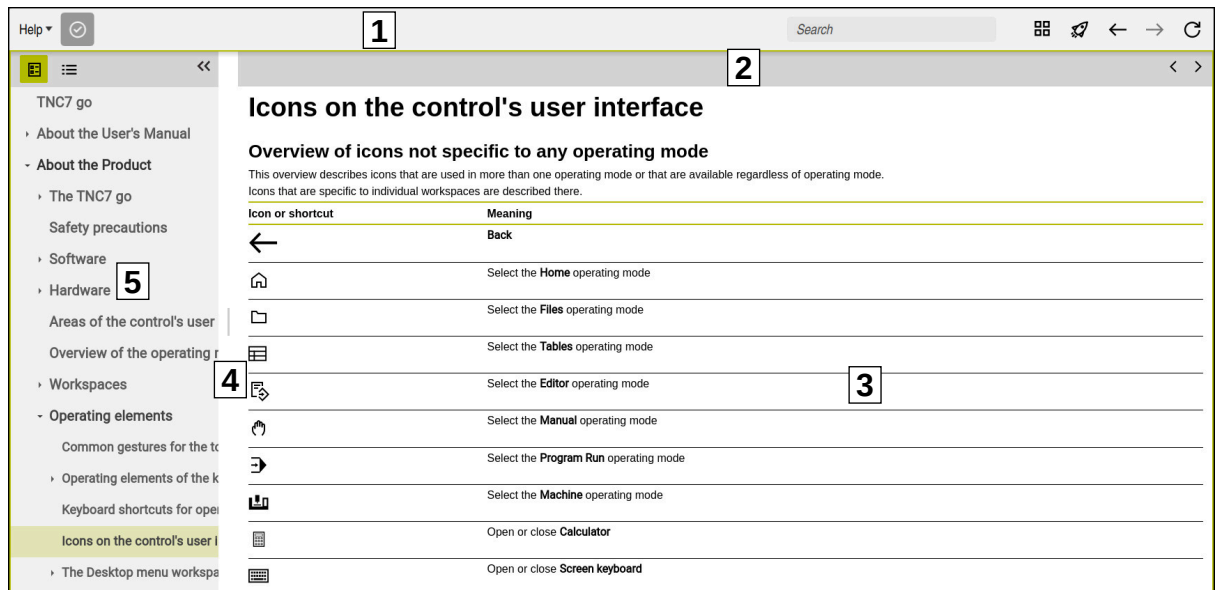
Further information: "The Help application", Page 38

Further information: "The Help workspace", Page 490

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

Further information: "Icons", Page 39

The Help application



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

TNCguide includes the following areas:

- 1 Title bar of the **Help** workspace
Further information: "The Help workspace", Page 39
- 2 Title bar of the integrated product aid **TNCguide**
Further information: "TNCguide ", Page 39
- 3 Content column of the **TNCguide**
- 4 Separator between the columns of the **TNCguide**
Adjust the column width by means of the separator.
- 5 Navigation column of the **TNCguide**

Icons

The Help workspace

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

Icon	Meaning
	Open or close the Search results column Further information: "Searching in TNCguide", Page 40
	Open Home page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., TNCguide). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function.
	Open Tutorials
	Navigate Navigate between the contents opened recently
	Refresh

TNCguide

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

Icon	Meaning
	Open Structure The structure consists of the content headings. The structure serves for main navigation within the documentation.
	Open Index The index consists of important keywords. The index serves as an alternative navigation within the documentation.
	Navigate Display previous or next page within the documentation
	Open or close Display or hide the navigation
	Copy Copy NC examples to the clipboard Further information: "Copying NC examples to clipboard", Page 41

Context-sensitive help

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

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

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

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

Further information: "The Help application", Page 38

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

Further information: "The Help workspace", Page 490

1.5.1 Searching in TNCguide

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

Use the search function as follows:

- ▶ Enter a string in **Search**



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

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

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
- ▶ Expand **Notes on using NC programs**
- ▶ Read and follow **Notes on using NC programs**

Further information: "Notes on using NC programs", Page 36



- ▶ Copy NC example to clipboard



- > The button switches colors while copying.
- > The clipboard contains the entire content of the copied NC example.
- ▶ Insert the NC example into the NC program
- ▶ Adapt the inserted content according to the **Notes on using NC programs**
- ▶ Use the Simulation mode to test the NC program

Further information: "The Simulation workspace", Page 527

1.6 Contacting the editorial staff

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

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

`tnc-userdoc@heidenhain.com`

2

About the product

2.1 The TNC7 go

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

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

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

Further information: User's Manual for Setup and Program Run

Definitions

Abbreviation	Definition
TNC	TNC is derived from the acronym CNC (computerized numerical control). The T (tip or touch) stands for the capability of entering NC programs directly at the control or to program them graphically using gestures.
7	The product number indicates the control generation. The range of functions depends on the enabled software options.
go	The addition go shows that the control is a workshop-oriented contouring control that enables you to program common milling and drilling operations right at the machine.

2.1.1 Proper and intended use

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

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



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

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



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

Further information: "Contacting the editorial staff", Page 42

2.1.2 Data Architecture and Access Options

Directories

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

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

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

Data acquisition

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

Data acquisition examples:

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



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

Data transfer and data access

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

Examples of data transfer:

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



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

Data access via file management:

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



For further details on file access, please refer to the section on the **Files** operating mode.

Please observe the safety instructions and information given by the machine manufacturer in order to avoid loss of data or jeopardize the functionality of your system.

2.1.3 Intended place of operation

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

Definitions

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

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

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!**

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

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

NOTICE**Danger of collision!**

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

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

NOTICE**Danger of collision!**

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

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

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

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

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

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

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

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

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 50

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



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

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

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



Refer to your machine manual.

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

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

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

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: User's Manual for Setup and Program Run

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

The software options are saved on the **SIK** (System Identification Key) plug-in board. The TNC7 go is equipped with a **SIK2** plug-in board, which means that the numbers of the software options differ from older controls, for example.

i The option numbers in parentheses given in the User's Manual show you that a function is not included in the standard range of available functions. The parentheses enclose the **SIK** and **SIK2** option numbers, separated by a slash, for example: (#18 / #3-03-1). The Technical Manual contains information about additional software options that are relevant to the machine manufacturer.

SIK2 definitions

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

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

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

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

Further information: User's Manual for Setup and Program Run

Overview



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

Further information: User's Manual for Setup and Program Run

Software option	Definition and application
Control Loop Qty. (#0-3 / #6-01-1*)	Additional control loop A control loop is required for each axis or spindle moved to a programmed nominal value by the control. Additional control loops are required, for example, for detachable and motor-driven tilting tables. If your control features a SIK2 , you can order this software option multiple times and enable up to control loops.
Touch Probe Functions (#17 / #1-05-1)	Touch-probe functions This software option is used to program and execute automatic probing processes. If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled. The software option includes the following functions: ■ Automatic compensation of workpiece misalignment ■ Automatic setting of workpiece presets ■ Automatic measurement of workpieces ■ Automatic measurement of tools The touch-probe functions reduce the setup effort and increase accuracy when machining workpieces.
HEIDENHAIN DNC (#18 / #3-03-1)	HEIDENHAIN DNC This software option enables external Windows applications to access data of the control via the TCP/IP protocol. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data HEIDENHAIN DNC is required in conjunction with external Windows applications.
OPC UA NC Server (#56-61 / #3-02-1*)	OPC UA NC Server These software options include OPC UA, a standardized interface for remote access to the control's data and functions. Potential fields of application include: ■ Connection to higher-level ERP or MES systems ■ Capture of machine and operating data Each software option enables one client connection. If more than one parallel connection is required, you need to enable multiple of these software options. If your control features the SIK2 , you can order this software option multiple times and enable up to ten connections. Further information: User's Manual for Setup and Program Run

Software option	Definition and application
CAD Model Optimizer (#152 / #1-04-1)	Optimization of CAD models This software option can be used, for example, to repair faulty files of fixtures and tool holders or to position STL files generated from the simulation for a different machining operation. Further information: User's Manual for Setup and Program Run
PTI 8 channels (#3-04-1*)	Process Tracking Interface PTI This software option provides an interface for recording control and process signals at a scanning interval as short as 3 ms. With PTI, you can record live information about axes and spindles. The signals can be used, among other things, to create a digital workpiece twin in order to meet documentation requirements in the aerospace industry. This software option is available only for controls with SIK2 . You can order this software option up to two times, each time selecting up to eight channels for recording. Further information: User's Manual for Setup and Program Run

 For more information, refer to the integrated help system of RemoTools SDK.
ID 340442-xx

2.3.2 Information on licensing and use

Open-source software

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

To get to the licensing terms on the control:



- ▶ Select the **Home** operating mode



- ▶ Select the **Settings** application



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

OPC UA

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

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

Further information: User's Manual for Setup and Program Run

2.4 Hardware

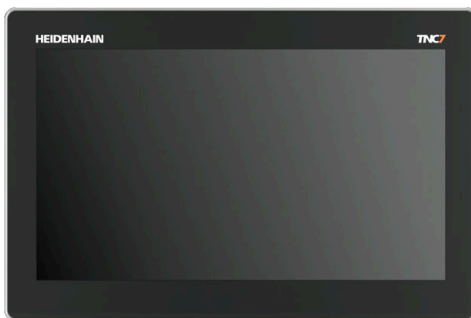
This User's Manual describes functions for setting up and operating the machine. These functions primarily depend on the installed software.

Further information: "Software", Page 49

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

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

2.4.1 Touchscreen and keyboard unit



16" MC 345

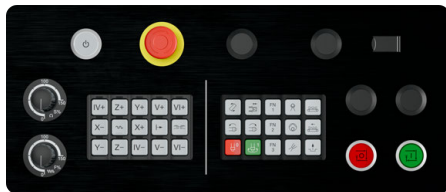
The TNC7 go is delivered with a 16-inch screen.

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

Further information: "Common gestures for the touchscreen", Page 65

Further information: "Operating elements of the keyboard unit", Page 65

The machine operating panel is machine-dependent.



MB 340 (FS)

Operating and cleaning the touchscreen

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

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

To navigate to this function:

Settings application ► Diagnostics/Maintenance group ► Touchscreen Cleaning

Further information: User's Manual for Setup and Program Run

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

The following cleaning agents are permitted for the screen:

- Glass cleaner
- Foaming screen cleaners
- Mild detergents

The following cleaning agents are prohibited for the screen:

- Aggressive solvents
- Abrasives
- Compressed air
- Steam cleaners



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

Cleaning the keyboard unit

Switch the control off before cleaning the keyboard unit.

NOTICE

Caution: risk of property damage

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

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

The following cleaners are permitted for the keyboard unit:

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

The following cleaning agents are prohibited for the keyboard unit:

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



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

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

To clean a trackball (if needed):

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



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

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

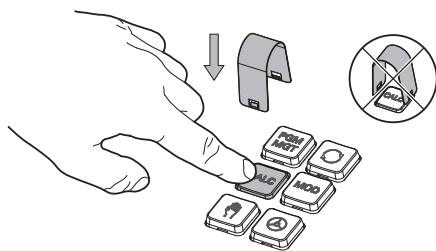
Exchanging keycaps

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



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

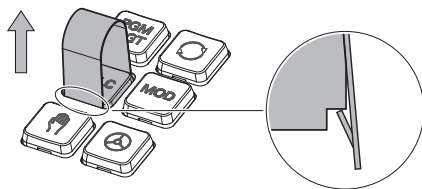
To exchange the keycaps:



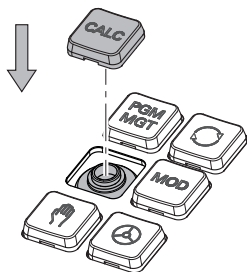
- Slide the keycap puller (ID 1325134-01) over the keycap until the grippers engage



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



- Pull off the keycap



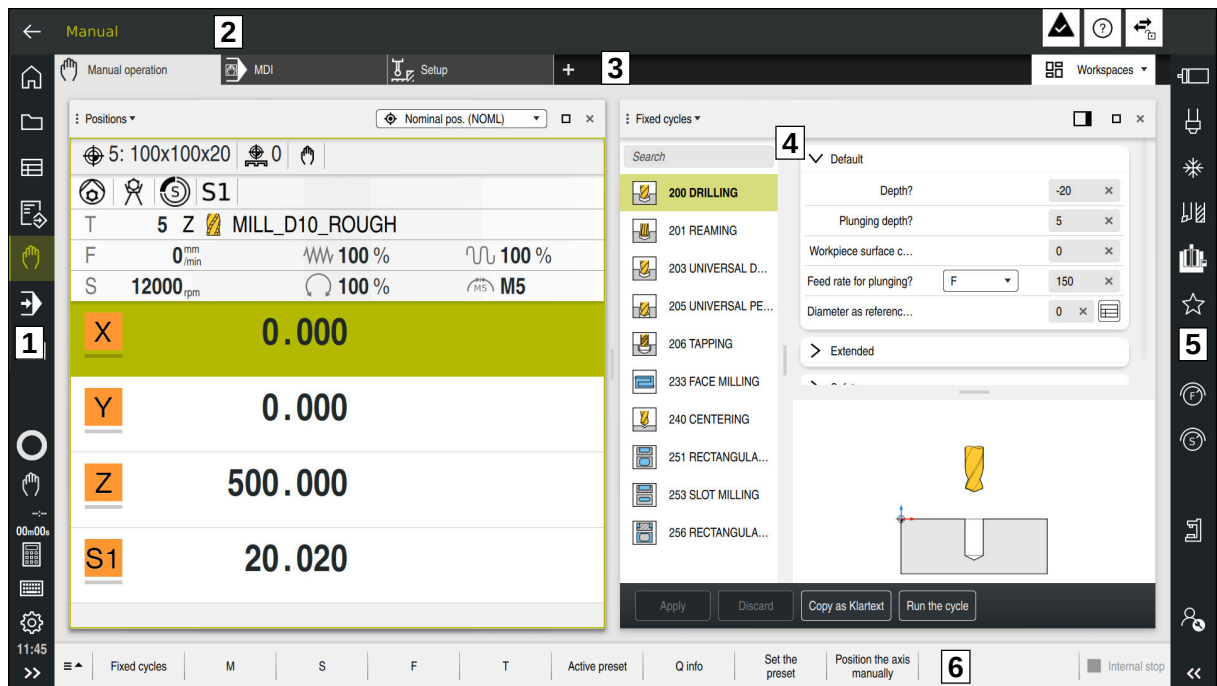
- Place the keycap onto the seal and push it down



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

- Verify proper seating and correct functionality

2.5 Areas of the control's user interface



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

The control's user interface shows the following areas:

- 1 TNC bar
 - **Back**
Use this function to go backwards in the application history since booting the control.
 - **Operating modes**
Further information: "Overview of the operating modes", Page 59
 - **Status overview**
Further information: User's Manual for Setup and Program Run
 - **Calculator**
Further information: "Calculator", Page 516
 - **Screen keyboard**
Further information: "Virtual keyboard of the control bar", Page 493
 - **Settings**
The Settings menu enables you to change the control interface:
 - **Left-hand mode**
The control swaps the positions of the TNC bar and the machine manufacturer bar.
 - **Dark Mode**
In the machine parameter **darkModeEnable** (no. 135501), the machine manufacturer defines whether the **Dark Mode** function is available for selection.

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

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

Further information: "Icons on the control's user interface", Page 73
- 3 Application bar
 - Tabs of opened applications

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

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

Further information: "Workspaces", Page 61
- 5 Machine manufacturer bar

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

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

With the buttons you activate individual functions of the control.



Refer to your machine manual.

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

- Operating modes
- Applications
- Workspaces
- Buttons

2.6 Overview of the operating modes

The control provides the following operating modes:

Icon	Operating modes	Further information
	The Home operating mode contains the following applications: ■ The Start/Login application During the startup process, the control is in the Start/Login application. ■ The Settings application ■ The Help application ■ Applications for machine parameters	See the User's Manual for Setup and Program Run Page 490 See the User's Manual for Programming and Testing See the User's Manual for Setup and Program Run
	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.	Page 348 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 580
	In the Editor operating mode you can do the following: ■ Create, edit and simulate NC programs	Page 116 See the User's Manual for Programming and Testing
	The Manual operating mode contains the following applications: ■ The Manual operation application ■ The MDI Application ■ The Setup application ■ The Move to ref. point application ■ The Retract application You can move the tool away from the workpiece, for example after a power failure.	See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run See the User's Manual for Setup and Program Run
	In the Program Run operating mode you produce workpieces by having the control execute NC programs either block-by-block or in full sequence.	See the User's Manual for Setup and Program Run
	In the Machine operating mode, the machine manufacturers define their own functions, such as diagnostic functions for spindle and axes, or other applications. Refer to your machine manual.	



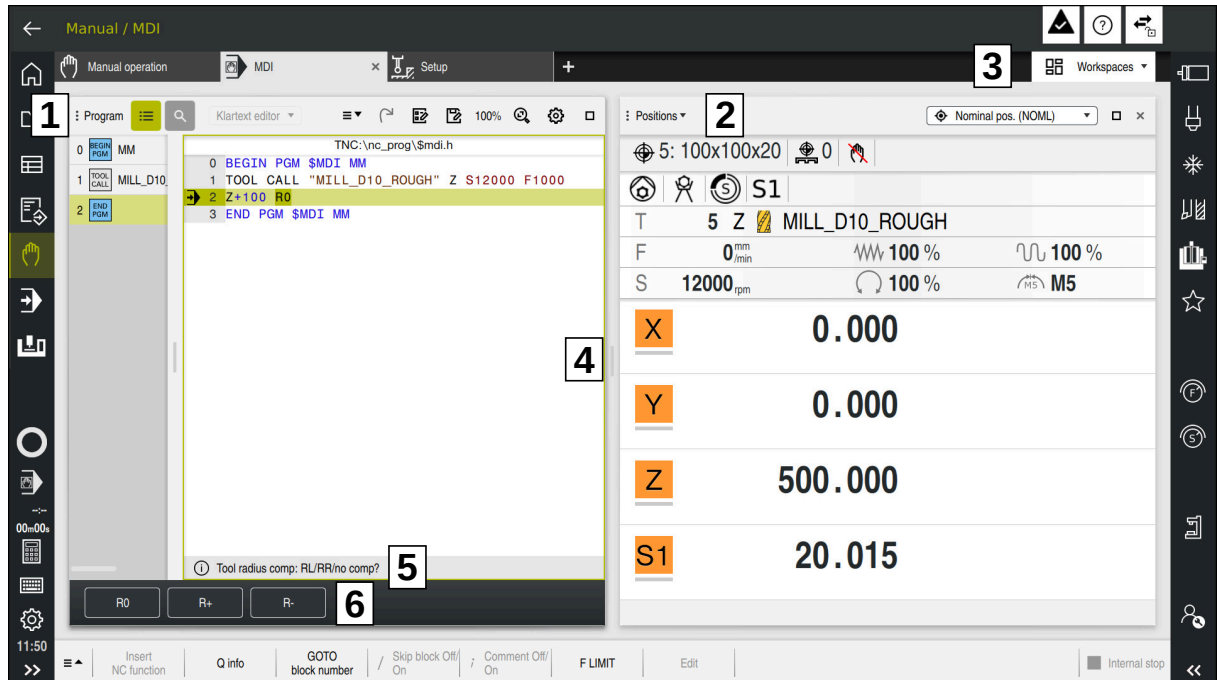
Refer to your machine manual.

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

- Operating modes
- Applications
- Buttons

2.7 Workspaces

2.7.1 Operating elements within the workspaces



The control in the **MDI** application with two open workspaces.

The control displays the following operating elements:

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

2.7.2 Icons within the workspaces

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

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

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

Icons that are specific to individual workspaces are described there.

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

Further information: "Overview of icons not specific to any operating mode", Page 73

2.7.3 Overview of workspaces

The control offers the following workspaces:

Workspace	Further information
Probing function (#17 / #1-05-1) In the Probing function workspace you set presets on the workpiece and . You can also calibrate the touch probe, measure tools.	See the User's Manual for Setup and Program Run
Fixed cycles In the Fixed cycles workspace, you can define a cycle the control will execute at the current position. The machine manufacturer defines the available cycles.	See the User's Manual for Setup and Program Run
Open File In the Open File workspace you select or create files, for example.	Page 359
Files In the file manager, the control displays drives, folders, and files. You can, for example, create or delete folders or files and can also connect drives. The Files workspace is part of the Files operating mode.	Page 348
Details In the Details workspace, the control displays information on the selected machine parameter or the last change you made.	See the User's Manual for Setup and Program Run
Document You can open files for viewing in the Document workspace, for example a technical drawing.	Page 361

Workspace	Further information
Settings In the Settings workspace, you can display and edit, if required, various settings of the control (e.g., set up the traverse limits). The Settings workspace is part of the Settings application.	See the User's Manual for Setup and Program Run
The Form for tables In the Form workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.	Page 594
Retract In the Retract workspace, you can disengage the tool after a power interruption.	See the User's Manual for Setup and Program Run
Graphic Positioning In the Graphic Positioning workspace, you can define a cycle the control will execute at multiple positions. You can define the machining positions graphically.	See the User's Manual for Setup and Program Run
Desktop menu In the Desktop menu workspace, the control displays selected control and HEROS functions.	Page 76
Help In the Help workspace, the control displays a help graphic for the current syntax element of an NC function or the integrated product aid TNCguide .	Page 490
List In the List workspace, the control shows the machine parameter structure; you might be able to edit some of the parameters.	See the User's Manual for Setup and Program Run
Positions In the Positions workspace, the control displays information about the status of various functions of the control and about current axis positions.	See the User's Manual for Setup and Program Run
Program The control displays the NC program in the Program workspace.	Page 118
Referencing On machines with incremental linear and angle encoders, the control shows in the Referencing workspace which axes need to be referenced.	See the User's Manual for Setup and Program Run
Quick selection In the Quick selection new table and Quick selection new file workspaces, you can create files or open existing files, depending on the active operating mode.	Page 360
Simulation In the Simulation workspace, the control shows the simulated or current movements, depending on the operating mode.	Page 527
Simulation status In the Simulation status workspace the control shows data based on the simulation of the NC program.	See the User's Manual for Setup and Program Run
Start In the Start workspace, the control shows the steps that are performed during startup.	Page 81

Workspace	Further information
Status In the Status workspace, the control shows the status and values of individual functions.	See the User's Manual for Setup and Program Run
Table In the Table workspace, the control shows the contents of a table. You can search in all tables and filter the table content.	Page 586
The Table for machine parameters In the Table workspace the control shows the machine parameters; you might be able to edit some of them.	See the User's Manual for Setup and Program Run
Keyboard In the Keyboard workspace, you can enter NC functions, letters and numbers, and also navigate.	Page 493
Text editor Use the Text editor workspace to create and edit text files.	Page 366
Overview In the Overview workspace, the control displays information on the status of individual functional safety (FS) safety functions.	See the User's Manual for Setup and Program Run



Refer to your machine manual.

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

- Operating modes
- Applications
- Workspaces
- Buttons

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:

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

2.8.2 Operating elements of the keyboard unit

Application

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

Further information: "Common gestures for the touchscreen", Page 65

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

Description of function

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



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

Further information: "Virtual keyboard of the control bar", Page 493

Keycaps for alphabetic keyboard

Key	Meaning
  	Enter texts (e.g., file names)
	Q
  	With an open NC program, enter a Q parameter formula in the Editor operating mode, or in the Manual operating mode open the Q parameter list window Further information: "The Q parameter list window", Page 399 By selecting the Q key multiple times, you can switch between Q , QL , and QR .
	Close the window or context menu, discard the current input
	Select the next element; for example, an input field, button, or selection option
[SHIFT] + [TAB]	Select the previous element
	Show the hidden NC block Further information: "Hiding or showing NC blocks", Page 499
	Hide or show NC blocks
	Create screenshot
	The DIADUR keys provide the following functions: ■ Left DIADUR key Open the HEROS menu
	Open context menu
  	Perform calculations for numerical values in input fields and table cells
  	

Keycaps for operating aids

Key	Meaning
	Open the Open File workspace in the Editor and Program Run operating modes Further information: "The Open File workspace", Page 359
	Currently no function
	Open and close the message menu Further information: User's Manual for Setup and Program Run
	Open and close the calculator Further information: "Calculator", Page 516
	Open the Settings application Further information: User's Manual for Setup and Program Run
	Open the online help Further information: "User's Manual as integrated product aid: TNCguide", Page 37

Operating modes



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

Key	Meaning
	Open the Manual operation application in the Manual operating mode Further information: User's Manual for Setup and Program Run
	Activate and deactivate the electronic handwheel in the Manual operating mode Further information: User's Manual for Setup and Program Run
	Open the Tool Management tab in the Tables operating mode Further information: User's Manual for Setup and Program Run
	Open the MDI application in the Manual operating mode Further information: User's Manual for Setup and Program Run
	Open the Program Run operating mode in Single Block mode Further information: User's Manual for Setup and Program Run
	Open the Program Run operating mode Further information: User's Manual for Setup and Program Run
	Open the Editor operating mode Further information: "The Editor operating mode", Page 116
	While the NC program is running, open the Simulation workspace in the Editor operating mode Further information: "The Simulation workspace", Page 527

Keycaps for NC dialog



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

Key	Meaning
	In the Insert NC function window, open the Setup folder in order to select a touch probe cycle
	In the Insert NC function window, open the Fixed cycles folder in order to select a cycle Further information: "Defining cycles", Page 140
	In the Insert NC function window, open the Cycle call folder to select a machining cycle. Further information: "Calling cycles", Page 143
	Program a jump label Further information: "Defining a label with LBL SET", Page 178
	Program a subprogram or a program section repeat Further information: "Calling a label with CALL LBL", Page 179
	Program an intentional stop Further information: "Miscellaneous functions M", Page 382
	Pre-select a tool in the NC program Further information: "Tool pre-selection using TOOL DEF", Page 169
	Call the tool in the NC program Further information: "Using TOOL CALL to call a tool", Page 162
	In the Insert NC function window, open the Special functions folder (e.g., to program a workpiece blank)
	In the Insert NC function window, open the Selection folder (e.g., to call an external NC program)

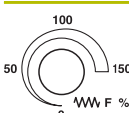
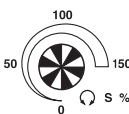
Keycaps for axis input and value input

Key	Meaning
 ... 	Select axes in the Manual operating mode, or enter them in the Editor operating mode
 ... 	Enter numbers (e.g., coordinate values)
	Insert a decimal separator during entry
	Invert algebraic sign of entered value
	Delete values during entry
	Open position display of the status overview to copy axis values Further information: "Editing NC functions", Page 111 In the Editor operating mode and the MDI application, take over the actual position of the defined axis
	In the Editor operating mode, open the FN folder in the Insert NC function window
	
	Clear entries or delete messages
	Delete NC block or cancel a dialog during programming
	Skip or remove optional syntax elements during programming
	Confirm entries and continue dialogs
	Conclude entry (e.g., finish an NC block)
	Switch between entry of incremental and absolute coordinates

Keycaps for navigation

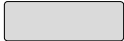
Key	Meaning
 	Position the cursor
 	
	■ Directly position the cursor by using the number of an NC block, table row or machine parameter ■ Open the selection menu while editing
	Jump to first line of an NC program or first column of a table
	Jump to last line of an NC program or last column of a table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

Potentiometers

Potentiometer	Function
	Increase or reduce the feed rate Further information: "Feed rate F", Page 167
	Increase or reduce the spindle speed Further information: "Spindle speed S", Page 166

2.8.3 Keyboard shortcuts for operating the control

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

Key	Designation
	SHIFT
	SPACE
	RETURN
	TAB
	UP
	DOWN
	RIGHT
	LEFT

2.8.4 Icons on the control's user interface

Overview of icons not specific to any operating mode

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

Icons that are specific to individual workspaces are described there.

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

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

Icon or shortcut	Meaning
	Show the Message menu
	Call context-sensitive help Further information: "Context-sensitive help", Page 40
	Secure connection configuration External access to the control is active; all connections are using a secure connection configuration.
	Non-secure connection configuration External access to the control is active and at least one connection uses a non-secure connection configuration.
	Automatic program start active Further information: User's Manual for Setup and Program Run
	Window manager Select active applications in the background (e.g., HEROS functions window)
	ITC is connected and Enabling VNC focus is set Further information: User's Manual for Setup and Program Run
	Both the control and the ITC show the following icons. The focus is on the currently used device You can use the mouse and keyboard for the local and the remote device.
	The focus is on the remote device You cannot use the mouse and keyboard on the local device.
	No focus assigned
	Inputs with the mouse and keyboard are locked until the focus has been assigned to the local or remote device.
Safety self-test	Self-test of the control is running
[CTRL] + [A]	Mark everything

2.8.5 The Desktop menu workspace

Application

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

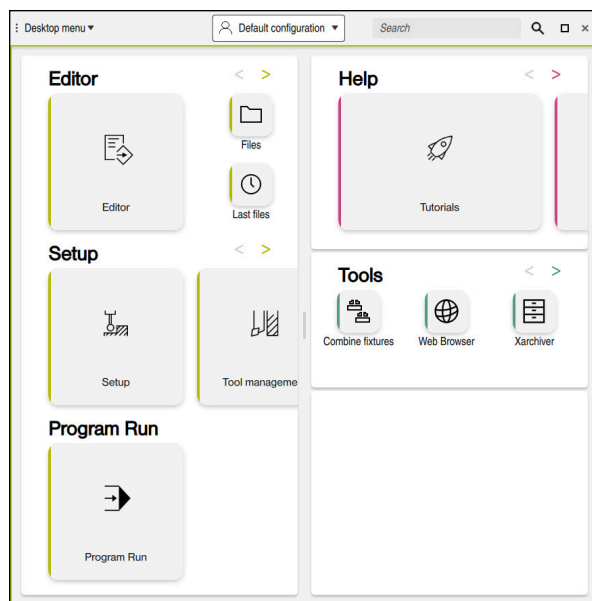
Description of function

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

- The **Active Configuration** selection menu
Using the selection menu, you can activate a configuration of the control interface.
- Full-text search
Search for functions in the workspace with the full-text search.
Further information: "Adding and removing favorites", Page 77

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

- **Control**
In this area you can open operating modes or applications.
Further information: "Overview of the operating modes", Page 59
Further information: "Overview of workspaces", Page 62
- **Tools**
In this area you can open some tools from the HEROS operating system.
Further information: User's Manual for Setup and Program Run
- **Help**
In this area you can open training videos or **TNCguide**.
Further information: "User's Manual as integrated product aid: TNCguide", Page 37
- **Favorites**
In this area you will find the favorites that you have chosen.
Further information: "Adding and removing favorites", Page 77



The **Desktop menu** workspace

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

Showing or hiding an area

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

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



Use the minus sign to hide an area.

Adding and removing favorites

Adding favorites

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

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



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

Removing favorites

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

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



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

3

First steps

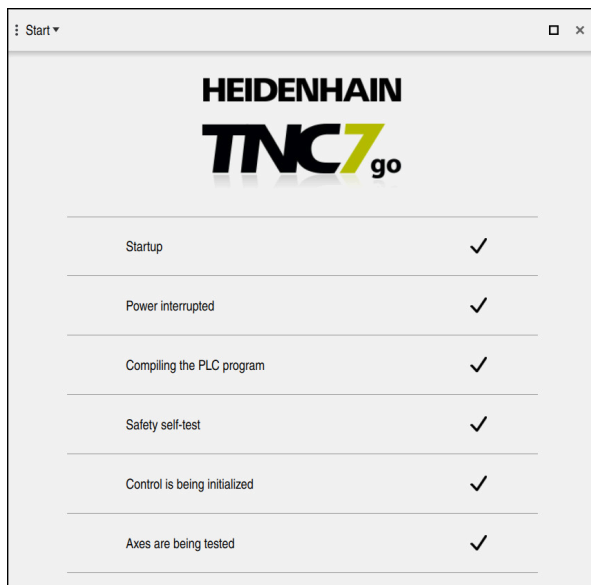
3.1 Chapter overview

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 and the control on
- Programming and simulating a workpiece
- Switching the machine off

3.2 Switching the machine and the control on



The **Start** workspace

DANGER

Caution: hazard to the user!

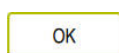
Machines and machine components always pose mechanical hazards. Electric, magnetic, or electromagnetic fields are particularly hazardous for persons with cardiac pacemakers or implants. The hazard starts when the machine is powered up!

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

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

To switch the machine on:

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



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



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

Further information: User's Manual for Setup and Program Run



- ▶ 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: User's Manual for Setup and Program Run

More detailed information

- Switching on and off

Further information: User's Manual for Setup and Program Run

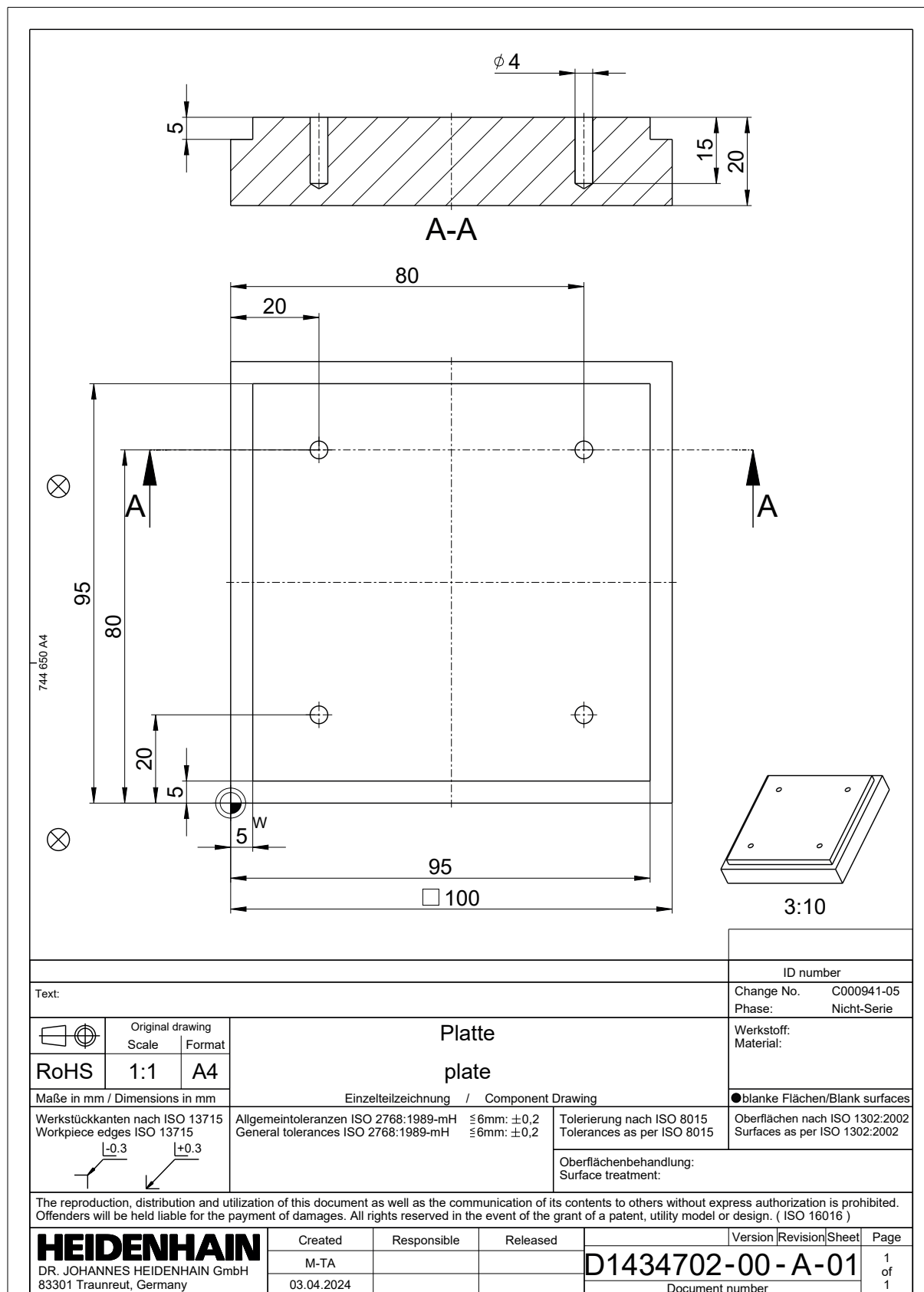
- Position encoders

Further information: "Position encoders and reference marks", Page 107

3.3 Programming and simulating a workpiece

3.3.1 Programming the workpiece

Sample task 1434702-00-A-01



Selecting the Editor operating mode

NC programs are always programmed in the **Editor** operating mode.

Requirement

- It must be possible to select the icon of the operating mode
In order to be able to select the **Editor** operating mode, the control must have already progressed enough during booting that the operating mode icon is no longer dimmed.

Selecting the Editor operating mode

To select the **Editor** operating mode:

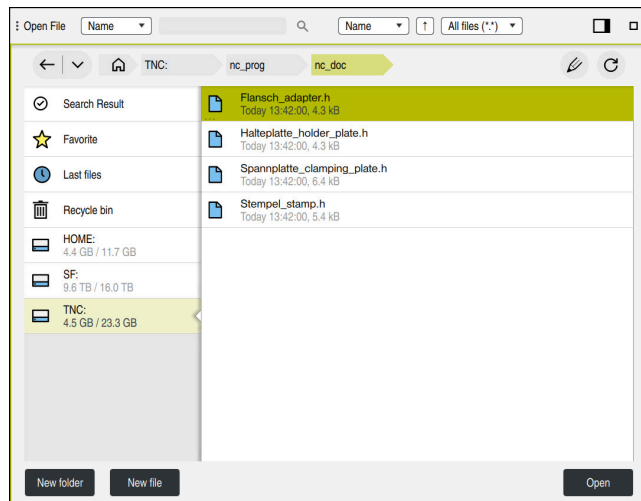


- ▶ Select the **Editor** operating mode
- > The control displays the **Editor** operating mode and the most recently opened NC program.

More detailed information

- The **Editor** operating mode
Further information: "The Editor operating mode", Page 116

Creating a new NC program



The **Open File** workspace in the **Editor** operating mode

To create an NC program in the **Editor** operating mode:



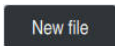
- ▶ Select **Add**
- The control shows the **Quick selection** and **Open File** workspaces.



- ▶ Select the desired drive in the **Open File** workspace



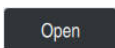
- ▶ Select a folder



- ▶ Select **New file**
- ▶ Enter a file name (e.g., 1434702.h)



- ▶ Confirm with the **ENT** key



- ▶ Select **Open**
- The control opens a new NC program and the **Insert NC function** window for definition of the workpiece blank.

More detailed information

- The **Open File** workspace
Further information: User's Manual for Setup and Program Run
- The **Editor** operating mode
Further information: "The Editor operating mode", Page 116

Configuring the control's user interface for programming

The **Editor** operating mode gives you several possibilities for writing an NC program.



The first steps describe the procedure when you are in the **Klartext editor** mode with the **Form** column open.

Opening the Form column

You can open the **Form** column only if an NC program is open.

To open the **Form** column:



- ▶ Select **Form**
- > The control opens the **Form** column

More detailed information

- Editing an NC program

Further information: "Possible methods for editing", Page 109

- The **Form** column

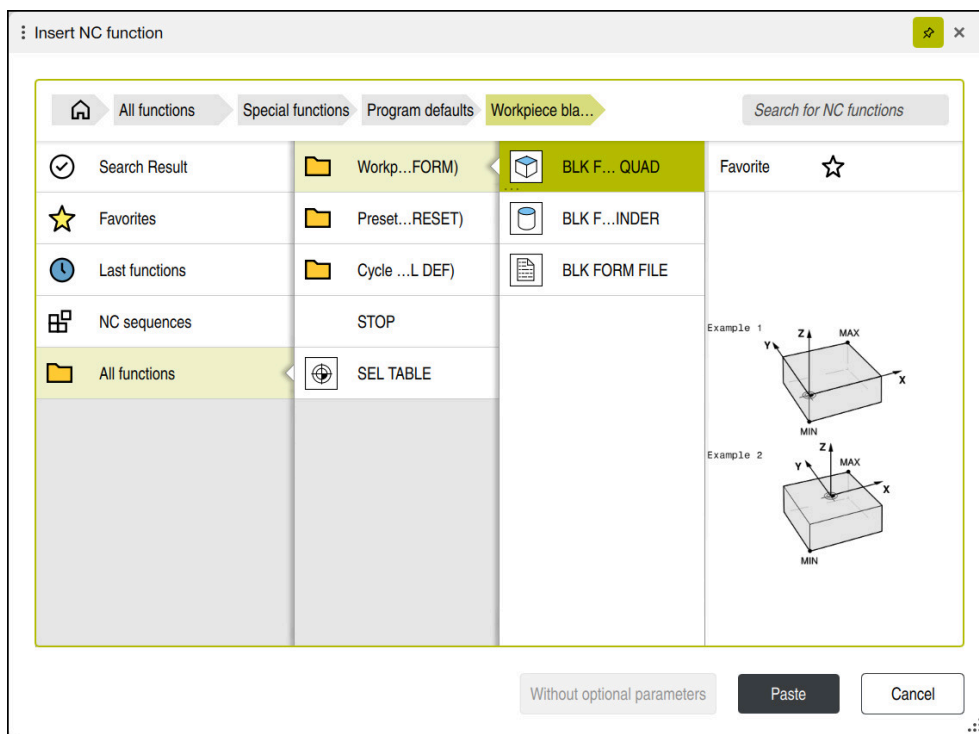
Further information: "The Form column in the Program workspace", Page 132

Defining the workpiece blank

For the NC program you can define a workpiece blank that the control then uses for the simulation. When you create an NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

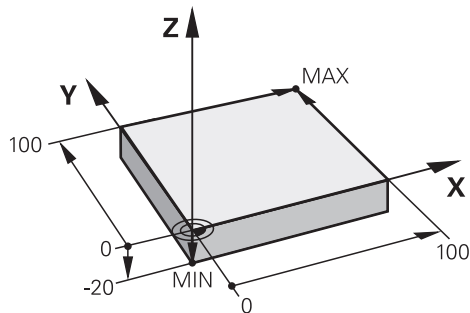


If you close the window without selecting a workpiece blank, you can use the **Insert NC function** button to select the workpiece blank definition later.



The **Insert NC function** window for workpiece blank definition

Defining a cuboid workpiece blank



Cuboid workpiece blank with minimum point and maximum point

You define a cuboid through a diagonal in space by entering the minimum point and maximum point relative to the active workpiece preset.



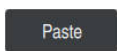
You can confirm the entries as follows:

- **ENT** key
- Right arrow key
- Click or tap the next syntax element

To define a cuboid workpiece blank:



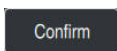
- ▶ Select **BLK FORM QUAD**



- ▶ Select **Paste**
- The control inserts the NC block for definition of the workpiece blank.



- ▶ Open the **Form** column
- ▶ Select the tool axis (e.g., **Z**)
- ▶ Confirm your input
- ▶ Enter the smallest X coordinate (e.g., **0**)
- ▶ Confirm your input
- ▶ Enter the smallest Y coordinate (e.g., **0**)
- ▶ Confirm your input
- ▶ Enter the smallest Z coordinate (e.g., **-20**)
- ▶ Confirm your input
- ▶ Enter the largest X coordinate (e.g., **100**)
- ▶ Confirm your input
- ▶ Enter the largest Y coordinate (e.g., **100**)
- ▶ Confirm your input
- ▶ Enter the largest Z coordinate (e.g., **0**)
- ▶ Confirm your input



- ▶ Select **Confirm**
- The control concludes the NC block.

Working spindle axis

X Y **Z**

Workpiece blank def.: MIN point

X 0 ✕

Y 0 ✕

Z -40 ✕

Workpiece blank def.: MAX point

X 100 ✕

Y 100 ✕

Z 0 ✕

Comment

;

Confirm Discard Delete line

The **Form** column with the defined columns

0 BEGIN PGM 1434702 MM

1 BLK FORM 0.1 Z X+0 Y+0 Z-20

2 BLK FORM 0.2 X+100 Y+100 Z+0

3 END PGM 1434702 MM



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

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

More detailed information

- Inserting the workpiece blank
Further information: "Defining a workpiece blank with BLK FORM", Page 152
- Reference points in the machine
Further information: "Presets in the machine", Page 108

Structure of an NC program

Using a uniform structure for an NC program offers the following advantages:

- Improved overview
- Quicker programming
- Fewer sources of error

Recommended structure for a contouring program



The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks.

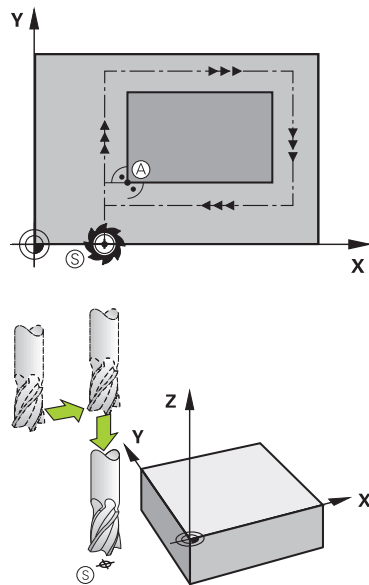
- 1 **BEGIN PGM** with selection of the unit of measure
- 2 Define the workpiece blank
- 3 Call the tool, with the tool axis and the technological data
- 4 Move the tool to a safe position, and switch the spindle on
- 5 Pre-position the tool in the working plane, near the first contour point
- 6 Pre-position the tool in the tool axis, turn coolant on if necessary
- 7 Approach the contour, activate tool radius compensation if necessary
- 8 Machine the contour
- 9 Depart from the contour, turn coolant off
- 10 Move the tool to a safe position
- 11 Conclude the NC program
- 12 **END PGM**

Contour approach and departure

When you program a contour, you need a starting point and end point outside the contour.

The following positions are necessary for contour approach and departure:

Help graphic



Position

Starting point

The following preconditions apply for the starting point:

- No tool radius compensation
- Approachable without danger of collision
- Near to the first contour point
- Extension of the first tool path

Approaching the starting point in the tool axis

Before approaching the first contour point, you must position the tool to the working depth in the tool axis.

First contour point

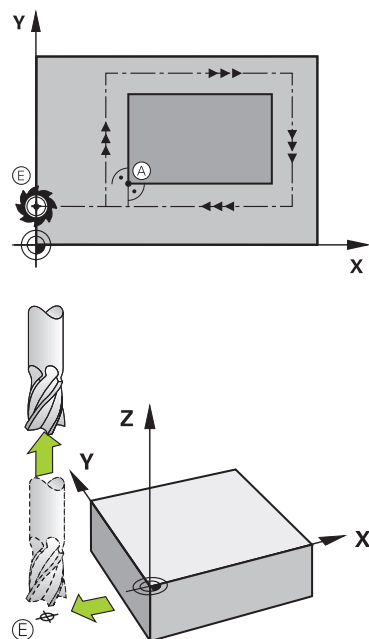
The control moves the tool from the starting point to the first contour point.

You need to program tool radius compensation for the tool movement to the first contour point.

End point

The following preconditions apply for the end point:

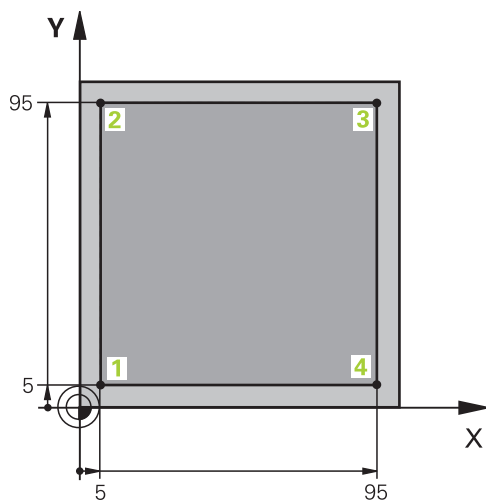
- Approachable without danger of collision
- Near to the last contour point
- Extension of the last tool path



Departing from the end point in the tool axis

Program the tool axis separately when departing from the end point.

Programming a simple contour



Workpiece to be programmed

The following texts show you how to mill once at a depth of 5 mm around the contour shown here. You have already defined the workpiece blank.

Further information: "Defining the workpiece blank", Page 86

After you have inserted an NC function, the control shows an explanation about the current syntax element in the dialog bar. You can enter the data directly in the form.



Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.

Call the tool

The **Form** column with the syntax elements of the tool call

To call a tool:

TOOL
CALL

- ▶ Select **TOOL CALL**
- ▶ Select **Number** in the form
- ▶ Enter the tool number (e.g., **16**)
- ▶ Select the tool axis **Z**
- ▶ Select the spindle speed **S**
- ▶ Enter the spindle speed (e.g., **6500**)
- ▶ Select **Confirm**
- > The control concludes the NC block.

Confirm

3 TOOL CALL 16 Z S6500



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

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

Move the tool to a safe position

Coordinates?

X Y **Z**

Z 250 x

Rotational direction

R0 R+ R-

Feed rate

FMAX ▾

M-Functions

M 3 x

M x

Comment

Confirm Discard Delete line

The **Form** column with the syntax elements of a straight line

To move the tool to a safe position:

- z

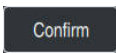
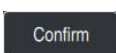
Confirm

 - ▶ Select **Z**
 - ▶ Enter a value (e.g., **250**)
 - ▶ Select tool radius compensation **R0**
 - The control applies **R0**, which means there is no tool radius compensation.
 - ▶ Select the **FMAX** feed rate
 - The control adopts **FMAX** for rapid traverse.
 - ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)
 - ▶ Select **Confirm**
 - The control concludes the NC block.

4 Z+250 R0 FMAX M3

Pre-positioning in the working plane

To pre-position in the working plane:

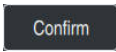
- 
 - ▶ Select **X**
 - ▶ Enter a value (e.g., **-20**)
 - ▶ Select the **FMAX** feed rate
- 
 - ▶ Select **Confirm**
 - ▶ The control concludes the NC block.
- 
 - ▶ Select **Y**
 - ▶ Enter a value (e.g., **-20**)
 - ▶ Select the **FMAX** feed rate
- 
 - ▶ Select **Confirm**
 - ▶ The control concludes the NC block.

5 X-20 FMAX

6 Y-20 FMAX

Pre-positioning in the tool axis

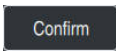
To pre-position in the tool axis:

- 
 - ▶ Select **Z**
 - ▶ Enter a value (e.g., **+5**)
 - ▶ Select the **FMAX** feed rate
 - ▶ The control adopts **FMAX** for rapid traverse.
- 
 - ▶ Select **Confirm**
 - ▶ The control concludes the NC block.

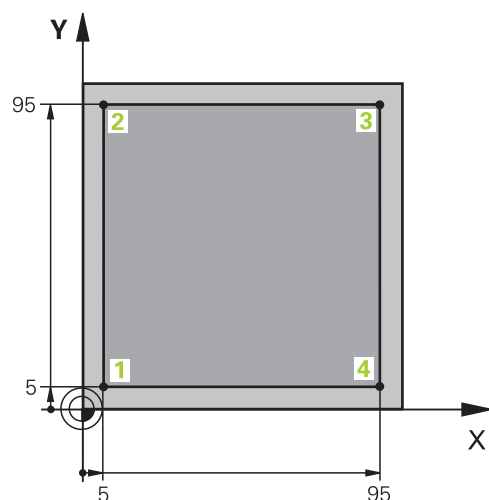
7 Z+5 FMAX

Infeed in the tool axis

To infeed in the tool axis:

- 
 - ▶ Select **Z**
 - ▶ Enter a value (e.g., **-5**)
 - ▶ Select the **F** feed rate
 - ▶ Enter the value for the positioning feed rate (e.g., **3000**)
 - ▶ If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)
- 
 - ▶ Select **Confirm**
 - ▶ The control concludes the NC block.

8 Z-5 F3000 M8

Approaching the contour

Workpiece to be programmed

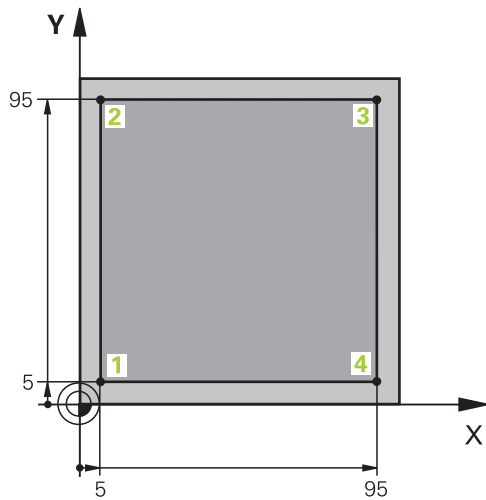
To approach the contour:

X

- ▶ Select **X**
- ▶ Enter the coordinates of the starting point **1** (e.g., **+5**)
- ▶ Select tool radius compensation **R-**
- ▶ The control shortens the traverse path by an amount equivalent to the tool radius.
- ▶ Select the feed rate **F**
- ▶ Enter the value for the machining feed rate (e.g., **700**)

9 X+5 R- F700

Machining a contour



Workpiece to be programmed

To machine the contour:

Y

- ▶ Select **Y**
- ▶ Enter the coordinates of the starting point **2** (e.g., **Y +95**)
- ▶ Select tool radius compensation **R+**
- ▶ The control extends the traverse path by an amount equivalent to the tool radius.
- ▶ Conclude the NC block with **Confirm**

Confirm

X

- ▶ Select **X**
- ▶ Enter the coordinates of the starting point **3** (e.g., **X +95**)
- ▶ Conclude the NC block with **Confirm**

Confirm

Y

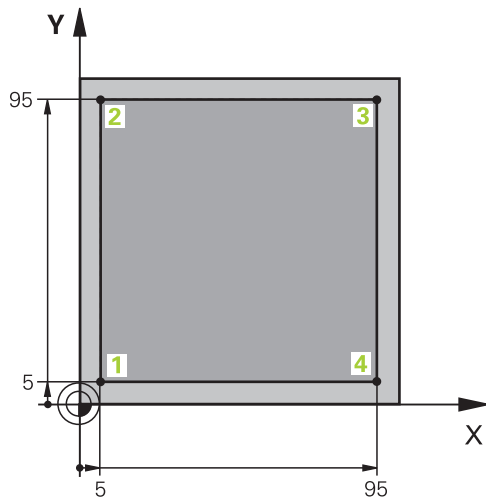
- ▶ Select **Y**
- ▶ Enter the coordinates of the starting point **4** (e.g., **Y +5**)
- ▶ Conclude the NC block with **Confirm**

Confirm

10 Y+95 R+

11 X+95

12 Y+5

Close the contour and retract the tool

Workpiece to be programmed

To close the contour and retract the tool:



- Select **X**
- Approach the coordinates of the starting point **1** and retract the tool (e.g., **5**)

13 X+5

More detailed information

- Tool call
Further information: "Using TOOL CALL to call a tool", Page 162
- Designation of the axes and the working plane
Further information: "Designation of the axes of milling machines", Page 106
- Miscellaneous functions
Further information: "Overview of miscellaneous functions", Page 383

Programming a machining cycle

The following texts show you how to machine holes with a depth of 15 mm with a standard drilling cycle. You have already defined the workpiece blank and created the outside contour.

Further information: "Sample task 1434702-00-A-01", Page 83

After you have inserted a cycle, you can define the associated values in the cycle parameters. You can program the cycle directly in the **Form** column.

Calling a tool

To call a tool:

TOOL
CALL

- ▶ Select **TOOL CALL**
- ▶ Select **Number** in the form
- ▶ Enter the tool number (e.g., **6**)
- ▶ Select the tool axis **Z**
- ▶ Select the spindle speed **S**
- ▶ Enter the spindle speed (e.g., **6500**)
- ▶ Select **Confirm**
- > The control concludes the NC block.

Confirm

15 TOOL CALL 6 Z S6500

Moving the tool to a safe position

The **Form** column with the syntax elements of a straight line

To move the tool to a safe position:

Z

- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- > The control applies **R0**, which means there is no tool radius compensation.
- ▶ Select the **FMAX** feed rate
- > The control adopts **FMAX** for rapid traverse.
- ▶ If needed, enter a miscellaneous function **M**, such as **M3** (turn spindle on)
- ▶ Select **Confirm**
- > The control concludes the NC block.

Confirm

16 Z+250 R0 FMAX M3

Define the cycle

To define the drilling cycle:



- ▶ Select **CYCLE DEF**
- The control opens the **Insert NC function window**.
- ▶ Select Cycle **200 DRILLING**



Paste

- ▶ Select **Insert**
- The control inserts the cycle.
- ▶ Enter the input values in the form



Confirm

- ▶ Select **Confirm**
- The control concludes the NC block.

17 CYCL DEF 200 DRILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-15	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q202=+5	;PLUNGING DEPTH ~
Q210=+0	;DWELL TIME AT TOP ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q395=+1	;DEPTH REFERENCE

Define the machining positions

To define the Machining positions:



- ▶ Select **SPEC FCT**
- ▶ Select **Contour/point machining**



- ▶ Select **Pattern**
- ▶ Select **PATTERN DEF POS**



Paste

- ▶ Select **Insert**
- The control inserts the cycle.
- ▶ Open the **Form** column
- ▶ Enter the four Machining positions in the form



Confirm

- ▶ Select **Confirm**
- The control concludes the NC block.

18 PATTERN DEF	
POS1	(X+20 Y+20 Z+0)
POS2	(X+20 Y+80 Z+0)
POS3	(X+80 Y+80 Z+0)
POS4	(X+80 Y+20 Z+0)

Calling a cycle

To call the cycle:

CYCL
CALL

- ▶ Select **CYCL CALL PAT**
- ▶ Select the **FMAX** feed rate
- ▶ If needed, enter a miscellaneous function **M**, such as **M8** (turn coolant on)
- ▶ Select **Confirm**
- > The control concludes the NC block.

19 CYCL CALL PAT FMAX M8

Moving the tool to a safe position

To move the tool to a safe position:

Z

- ▶ Select **Z**
- ▶ Enter a value (e.g., **250**)
- ▶ Select tool radius compensation **R0**
- ▶ Select the **FMAX** feed rate
- ▶ Enter a miscellaneous function **M** if required
- ▶ Select **Confirm**
- > The control concludes the NC block.
- ▶ Select **M**
- ▶ Define the miscellaneous function **M30**
- ▶ Select **Confirm**
- > The control concludes the NC block.

20 Z+250 R0 FMAX M30

21 M30

More detailed information

- Working with cycles

Further information: "Working with cycles", Page 137

Solution 1434702-00-A-01

If you follow all steps described here, you will obtain the following NC program:

0 BEGIN PGM "1434702-00-A-01" MM	
1 BLK FORM 0.1 Z X0 Y0 Z-20	
2 BLK FORM 0.2 X100 Y100 Z0	
3 TOOL CALL 16 Z S6500	
4 Z+250 R0 FMAX M3	
5 X-20 FMAX	
6 Y-20 FMAX	
7 Z+5 FMAX	
8 Z-5 F3000 M8	
9 X+5 R- F700	
10 Y+95 R+	
11 X+95	

12	Y+5	
13	X+5	
14	;	
15	TOOL CALL 6 Z S6500	
16	Z+250 R0 FMAX M3	
17	CYCL DEF 200 DRILLING ~	
	Q200=+2 ;SET-UP CLEARANCE ~	
	Q201=-15 ;DEPTH ~	
	Q206=+150 ;FEED RATE FOR PLNGNG ~	
	Q202=+5 ;PLUNGING DEPTH ~	
	Q210=+0 ;DWELL TIME AT TOP ~	
	Q203=+0 ;SURFACE COORDINATE ~	
	Q204=+50 ;2ND SET-UP CLEARANCE ~	
	Q211=+0 ;DWELL TIME AT DEPTH ~	
	Q395=+0 ;TOOL TIP	
18	PATTERN DEF POS1(X+20 Y+20 Z+0) POS2(X+20 Y+80 Z+0) POS3(X+80 Y+80 Z+0) POS4 (X+80 Y+20 Z+0)	
19	CYCL CALL PAT FMAX M8	
20	Z+250 R0 FMAX	
21	M30	
22	END PGM "1434702-00-A-01" MM	

3.3.2 Simulating a workpiece

Configuring the control's user interface for simulation

In the **Editor** operating mode you can test NC programs graphically. The control simulates the active NC program in the **Program** workspace.

In order to simulate the NC program you must open the **Simulation** workspace.



For the simulation you can close the **Form** column to get a better view of the NC program and the **Simulation** workspace.

Opening the Simulation workspace

You can open additional workspaces in the **Editor** operating mode only if an NC program is open.

To open the **Simulation** workspace:

- ▶ In the application bar, select **Workspaces**
- ▶ Select **Simulation**
- > The control then additionally displays the **Simulation** workspace.



You can also open the **Simulation** workpiece with the **Test Run** operating mode key.

Configuring the Simulation workspace

You can simulate the NC program without needing to enter any special settings. However, an adjustment to the simulation speed is recommended for best viewing of the simulation.

To adjust the speed of the simulation:

- ▶ Use the slider to select the factor (e.g., **5.0 * T**)
- > The control then performs the subsequent simulation at five times the speed of the programmed feed rate.

If you use different tables, such as tool tables, for program run and the simulation, then you can define the tables in the **Simulation** workspace.

More detailed information

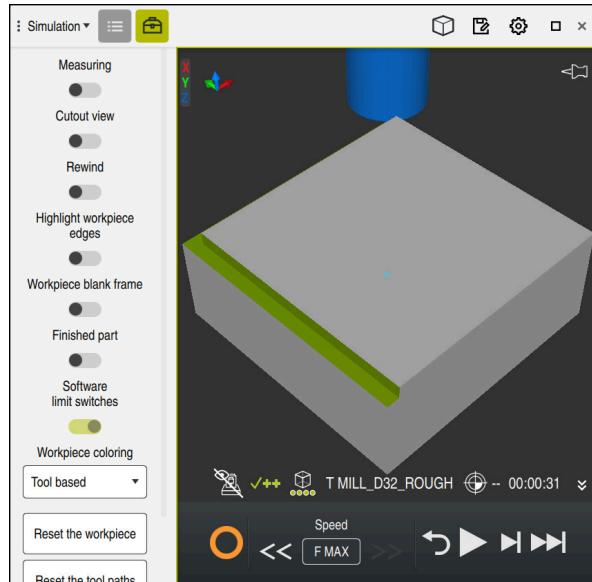
- The **Simulation** workspace

Further information: "The Simulation workspace", Page 527

Simulating an NC program

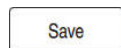
You can test the NC program in the **Simulation** workspace.

Starting the simulation



The **Simulation** workspace in the **Editor** operating mode

To start the simulation:



- ▶ Select **Start**
- The control might ask whether the file should be saved.
- ▶ Select **Save**
- The control starts the simulation.
- The control uses the **Control-in-operation** symbol to show the simulation status.

Definition

Control-in-operation:

The control uses the **Control-in-operation** icon to show the current simulation status in the action bar and on the tab of the NC program:

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

More detailed information

- The **Simulation** workspace

Further information: "The Simulation workspace", Page 527

3.4 Switching the machine off



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

NOTICE

Caution: Data may be lost!

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

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

To power-off the machine:



- ▶ Select the **Home** operating mode

Shut down

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

Shut down

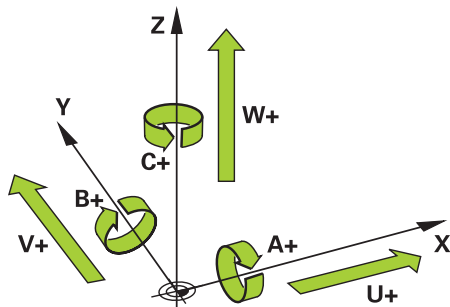
- ▶ Select **Shut down**
- If NC programs or contours contain any unsaved changes, the control displays the **Close file** window.
- ▶ If necessary, save unsaved NC programs with **Save** or **Save as**
- The control shuts down.
- After completion of the shutdown process, the control displays the text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

4

**NC and
programming
fundamentals**

4.1 NC fundamentals

4.1.1 Programmable axes



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

The programmable axes are designated as follows:

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



Refer to your machine manual.

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

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



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

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

4.1.2 Designation of the axes of milling machines

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

The axes are associated as follows:

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

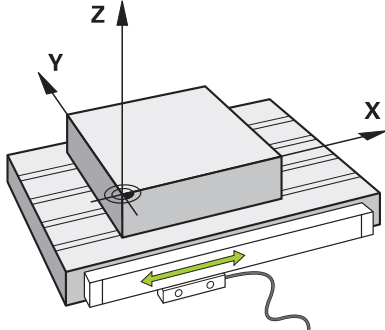


The control's full range of functions is available only if the **Z** tool axis is used (e.g., **PATTERN DEF**).

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

4.1.3 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.

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 308

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

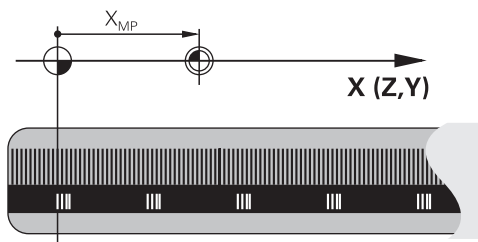
On absolute position encoders, every position on the encoder is uniquely identified. The control can thus immediately determine the association between the axis position and the coordinate system after a power interruption.

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.

Further information: User's Manual for Setup and Program Run



4.1.4 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 159

Icon	Preset
	Machine datum The machine datum is a fixed point defined in the machine configuration by the machine manufacturer. The machine datum is the origin of the machine coordinate system M-CS. Further information: "Machine coordinate system M-CS", Page 310 If you program M91 in an NC block, the defined values are referenced to the machine datum. Further information: "Traversing in the machine coordinate system M-CS with M91", Page 385
	M92 datum M92-ZP (zero point) The M92 datum is a fixed point defined relative to the machine datum by the machine manufacturer in the machine configuration. The M92 datum is the origin of the M92 coordinate system. If you program M92 in an NC block, the defined values are referenced to the M92 datum. Further information: "Traversing in the M92 coordinate system with M92", Page 387
	Tool change position The tool change position is a fixed point defined relative to the machine datum by the machine manufacturer in the tool-change macro.
	Reference point The reference point is a fixed point for initializing position encoders. Further information: "Position encoders and reference marks", Page 107 If the machine has incremental position encoders, the axes must traverse the reference point after booting. Further information: User's Manual for Setup and Program Run
	Workpiece preset With the workpiece preset you define the origin of the workpiece coordinate system W-CS. Further information: "Workpiece coordinate system W-CS", Page 312 The workpiece preset is defined in the active row of the preset table. You determine the workpiece preset with a 3D touch probe, for example. Further information: User's Manual for Setup and Program Run If no transformations are defined, the entries in the NC program refer to the workpiece preset.
	Workpiece datum You define the workpiece datum with transformations in the NC program, for example with TRANS DATUM or a datum table. The entries in the NC program refer to the workpiece datum. If no transformations are defined in the NC program, the workpiece datum corresponds to the workpiece preset.

4.2 Programming possibilities

4.2.1 Miscellaneous functions M

You can use miscellaneous functions to control the following actions:

- Program run (e.g., **M0** Program STOP)
- Machine functions (e.g., **M3** Spindle ON clockwise)

Further information: "Miscellaneous functions", Page 381

4.2.2 Subprograms and program-section repeats

Subprograms and program-section repeats enable you to program a machining sequence once and then run it as often as necessary.

Program sections that are defined in a label can be directly executed repeatedly as program-section repeats, or can be called as a subprogram at defined locations in the main program.

Within an NC program you can call a separate NC program for execution.

Further information: "Subprograms and program section repeats with the label LBL", Page 178

4.2.3 Control structures

Using control structures, you can program the NC program more clearly and with a better structure. The control indents the NC blocks within the control structures. Thus you can see right away where a control structure starts and ends.

Examples of control structures are case analyses and program loops.

You can program case analyses by using **IF** with a condition and **ELSE**, for example.

You can program loops by using **WHILE**, for example, and execute a program section several times.

Further information: "Control structures", Page 185

4.2.4 Programming with variables

In an NC program, variables are used as placeholders for numerical values or texts. A numerical value or text is assigned to a variable elsewhere.

In the **Q parameter list** window, you can see and edit the numerical values and texts of the individual variables.

Further information: "The Q parameter list window", Page 399

You can use the variables to program mathematical functions that control program execution or describe a contour.

You can also use variable programming, for example, to save and process measurement results determined by the 3D touch probe during program execution.

Further information: "Variables: Q, QL, QR, QS parameters and named parameters", Page 393

4.2.5 Possible methods for editing

Application

The editing of NC programs refers both to the insertion of NC functions as well as their modification.

Related topics

- Using the **Program** workspace
Further information: "Using the Program workspace", Page 124
- The **Insert NC function** window
Further information: "The Insert NC function window", Page 133
- Text mode
Further information: "Text mode", Page 135

Description of function

You can edit NC programs only in the **Editor** operating mode and in the **MDI** application.



In the **MDI** application you edit only the NC program **\$mdi.h** or **\$mdi_inch.h**.

Inserting NC functions

The control provides the following options to insert NC functions:

- Inserting an NC function directly with keys or buttons
Frequently required NC functions such as machining cycles can be inserted via keys.
As an alternative to the keys, the control offers both the screen keyboard as well as the **Keyboard** workspace in NC input mode.
Further information: "Virtual keyboard of the control bar", Page 493
- Inserting an NC function by selecting it
You can select all NC functions from the **Insert NC function** window.
Further information: "The Insert NC function window", Page 133
- Inserting an NC function in Text mode
In Text mode, the control offers automatic completion.
Further information: "Text mode", Page 135

Further information: "Inserting NC functions", Page 111



In the title bar of the **Program** workspace, the control displays which mode is active. You can select a different mode using the selection menu.

Editing NC functions

The control provides the following options to edit NC functions:

- Editing an NC function in the **Klartext editor** mode
By default, the control opens newly created and syntactically correct NC programs in the **Klartext editor** mode.
- Editing an NC function in the **Form** column
The **Form** column not only shows the syntax elements selected and used, but also all those that can be used for the current NC function.
- Editing an NC function in Text mode
If the control can't automatically correct syntax errors in the NC program, it activates Text mode. You must correct all errors before you can switch to the **Klartext editor** mode.

Further information: "Editing NC functions", Page 111

Inserting NC functions

Inserting an NC function directly with keys or buttons

To insert frequently needed NC functions:



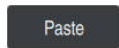
- ▶ Select **X**
- The control creates a new NC block and starts the dialog.
- ▶ Follow the instructions in the dialog

Inserting an NC function by selecting it

To insert a new NC function:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Navigate to the desired NC function
- The control highlights the selected NC function.



- ▶ Select **Paste**
- The control creates a new NC block and starts the dialog.
- ▶ Follow the instructions in the dialog

Inserting an NC function in Text mode

To insert an NC function:

- ▶ Enter any character
- The control inserts an NC block.
- Depending on the setting of the **Autocomplete in text mode** toggle switch, the control displays a selection menu with possible syntax initiators.

Further information: "Settings in the Program workspace", Page 120

- ▶ Select the desired syntax initiator
- ▶ Enter the value as needed
- Depending on the setting of the **Autocomplete in text mode** toggle switch, the control displays a selection menu with possible syntax elements.
- ▶ Select the desired syntax element

Editing NC functions

Editing an NC function in the Klartext editor mode

To edit an NC function in the **Klartext editor** mode:

- ▶ Navigate to the desired NC function
- ▶ Navigate to the desired syntax element
- The control displays alternative syntax elements in the action bar.
- ▶ Select a syntax element
- ▶ Define a value, if necessary



- ▶ Finalize your entry (e.g., by pressing **END**)

Editing an NC function in the Form column

If the **Klartext editor** mode is active, you can also use the **Form** column.

To edit an existing NC function in the **Form** column:

- ▶ Navigate to the desired NC function



- ▶ Show the **Form** column
- ▶ Select an alternative syntax element as needed (e.g., **X** instead of **Y**)
- ▶ Edit or add the value as needed
- ▶ If applicable, enter an optional syntax element or select one from a list (e.g., miscellaneous function **M8**)
- ▶ Complete your input (e.g., with the **Confirm** button)

Confirm

Editing an NC function in Text mode

To edit an incorrect NC function in Text mode:

- > The control marks the faulty syntax element with a wavy red underline and displays an information symbol in front of the NC function (e.g., if **FMX** instead of **FMAX**).
- ▶ Navigate to the desired NC function



- ▶ Select the information symbol as needed
- > The control displays the corresponding error description.
- ▶ Conclude the NC block
- > The control might open the **NC block auto-correction** window with a solution proposal.
- ▶ Apply the proposal to the NC program with **Yes** or cancel auto-correction.

Yes

Inserting an axis value into an NC block

To enter the value of an axis into an NC block:

- ▶ Program the desired axis (e.g., **X...**)



- ▶ Press **actual position capture**
- > The control opens the position display of the status overview.
- ▶ Select the desired value of an axis
- > The control inserts the value of the selected axis in the NC block.

Notes

NOTICE

Caution: Data may be lost!

When you edit NC programs outside the **Program** workspace, you have no control over whether the control will identify the changes. The changes cannot be undone on the control. This means that any such deletion or altering of data is permanent!

- Edit NC programs in the **Program** workspace only

- In Text mode, the control may not be able to suggest a solution in every case.
- When you are editing an NC function, use the arrows to navigate left and right to the syntax elements, even within cycles. The up and down arrows search for the same syntax element in the rest of the NC program.
Further information: "Searching for identical syntax elements in different NC blocks", Page 127
- If you are editing an NC block and have not yet saved it, the **Undo** and **Redo** functions will affect the changes of individual syntax elements of the NC function.
Further information: "Icons on the control's user interface", Page 73
- Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is currently being executed in the **Program Run** operating mode.
- In the **Klartext editor** mode, you can insert line breaks within comments or structuring items.

4.3 Programming fundamentals

4.3.1 Contents of an NC program

Application

You use NC programs to define the movements and behavior of your machine. NC programs consist of NC blocks that contain the syntax elements of the NC functions. With the HEIDENHAIN Klartext programming language, the control supports you by showing a dialog with information about the required content for every syntax element.

Related topics

- Creating a new NC program
Further information: "Creating a new NC program", Page 85
- Structure of an NC program for contour machining
Further information: "Structure of an NC program", Page 89

Description of function

You create NC programs in the **Editor** operating mode in the **Program** workspace.

Further information: "The Program workspace", Page 118

The first and last NC blocks of the NC program contain the following information:

- Syntax **BEGIN PGM** or **END PGM**
- Name of the NC program
- Unit of measure of the NC program (mm or inches)

The control automatically inserts the **BEGIN PGM** and **END PGM** NC blocks when creating the NC program. You cannot delete these NC blocks.

The NC blocks created after **BEGIN PGM** contain the following information:

- Workpiece blank definition
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds
- Traverse movements, cycles and other NC functions

0 BEGIN PGM EXAMPLE MM	; Start of program
1 BLK FORM 0.1 Z X-50 Y-50 Z-20	
2 BLK FORM 0.2 X+50 Y+50 Z+0	; NC function for workpiece blank definition, consisting of two NC blocks
3 TOOL CALL 5 Z S3200 F300	; NC function for tool call
4 Z+100 R0 FMAX M3	; NC function for straight-line traverse
* - ...	
11 M30	; NC function for ending the NC program
12 END PGM EXAMPLE MM	; End of program

Syntax component	Meaning
NC block	4 TOOL CALL 5 Z S3200 F300 An NC block consists of the block number and the syntax of the NC function. An NC block can consist of multiple lines, such as with cycles. The control numbers the NC blocks in ascending sequence.
NC function	TOOL CALL 5 Z S3200 F300 You use NC functions to define the behavior of the control. The block number is not a part of the NC functions.
Syntax initiator	TOOL CALL The syntax initiator clearly designates each NC function. Syntax initiators are used in the Insert NC function window. Further information: "Areas of the Insert NC function window", Page 133
Syntax element	TOOL CALL 5 Z S3200 F300 Syntax elements are all parts of the NC function, such as technology values S3200 or coordinate information. NC functions also contain optional syntax elements. The control shows certain syntax elements in color in the Program workspace. Further information: "Appearance of the NC program", Page 120
Value	3200 for spindle speed S Not every syntax element must contain a numerical value, such as tool axis Z.

If you create NC programs in a text editor or outside of the control, note the correct spelling and sequence of the syntax elements.

Notes

- NC functions can also consist of more than one NC block, such as **BLK FORM**.
 - Using the machine parameter **linebreak** (no. 105404), you can define how the control will display multi-line NC functions.
 - Miscellaneous functions **M** and comments can be both syntax elements within NC functions as well as their own NC functions.
 - Always write an NC program as if the tool were moving. This makes it irrelevant whether a head axis or a table axis performs the motion.
 - The file name extension ***.h** designates a Klartext program.
- Further information:** "Programming fundamentals", Page 114

4.3.2 The Editor operating mode

Application

In the **Editor** operating mode you can do the following:

- Create, edit and simulate NC programs

Description of function

With **Add**, you can create a new file or open an existing one. The control displays up to ten tabs.

The **Editor** operating mode presents the following workspaces if an NC program is open:

- **Document**
Further information: "The Document workspace", Page 361
- **Help**
Further information: "The Help workspace", Page 490
- **Program**
Further information: "The Program workspace", Page 118
- **Simulation**
Further information: "The Simulation workspace", Page 527
- **Simulation status**
Further information: User's Manual for Setup and Program Run
- **Keyboard**
Further information: "Virtual keyboard of the control bar", Page 493

If an NC program is selected in the **Program Run** operating mode, the controls shows the **M** status on the tab of the NC program. If the **Simulation** workspace for this NC program is open, the controls shows the **Control-in-operation** icon on the tab of the NC program.

Icons and buttons

The **Editor** operating mode contains the following icons and buttons:

Icon or button	Meaning
	The control uses this icon to show that an NC program is open.
	Execution cursor The execution cursor shows which NC block is currently being executed or is marked for execution. When simulating the opened NC program, the control displays the execution cursor.
Insert NC function	The control opens the Insert NC function window. Further information: "Possible methods for editing", Page 109
GOTO block number or GOTO record	The control opens the GOTO jump instruction window. The control selects the block number or line number that you defined. GOTO block number for Program workspace only GOTO record for Text editor workspace only Further information: "The GOTO function", Page 496
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: "The Q parameter list window", Page 399

Icon or button	Meaning
/ Skip block Off/On	Hide NC blocks with the / character. NC blocks hidden with a / character will be ignored during program run as soon as the Skip block toggle switch is active. Further information: "Hiding NC blocks", Page 499
; 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: "Adding comments", Page 498
Edit	The control opens the context menu. Further information: "Context menu", Page 511
Block scan Program run	The control opens the file in the Program Run operating mode and opens the Block scan window for the currently selected NC block. This allows you to execute the NC program directly from the selected NC block. Further information: User's Manual for Setup and Program Run
Select in Program Run	The control opens the file in the Program Run operating mode and selects the first NC block. Further information: User's Manual for Setup and Program Run
Start the simulation	The control opens the Simulation workspace and starts graphic simulation. Further information: "The Simulation workspace", Page 527

4.3.3 The Program workspace

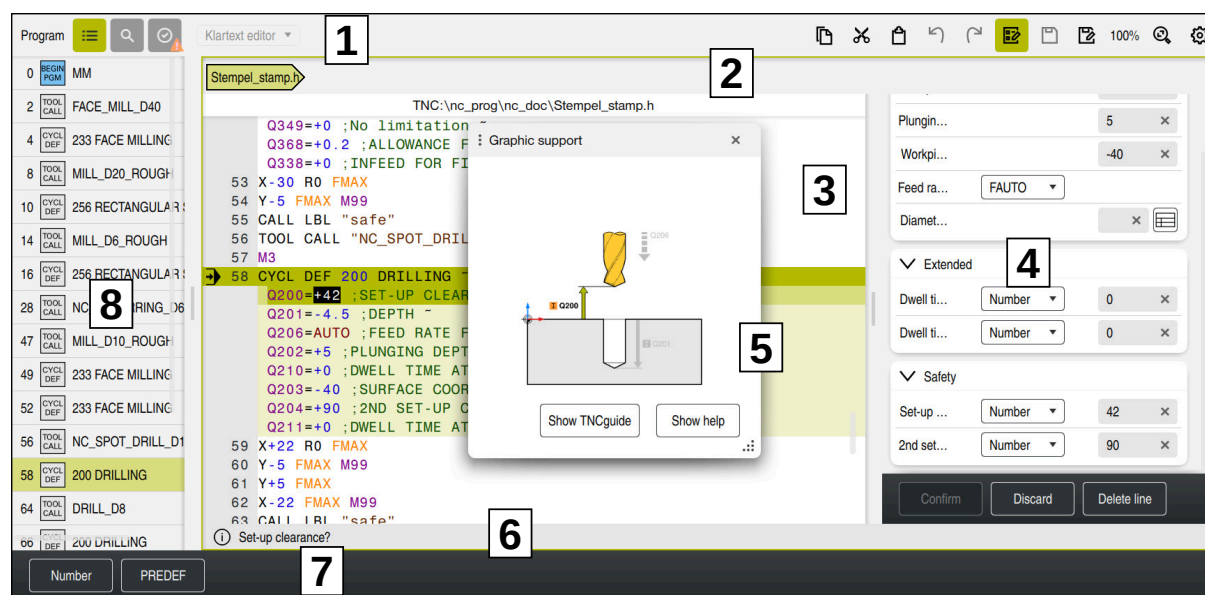
Application

The control displays the NC program in the **Program** workspace.

You can edit the NC program in the **Editor** operating mode and in the **MDI** application, but not in the **Program Run** operating mode.

Description of function

Areas of the Program workspace



The **Program** workspace with an active outline, a help graphic, and a fillable form

- 1 Title bar

Further information: "Symbols, icons, and buttons in the title bar", Page 119

- 2 File information bar

In the file information bar, the control shows the path and file name of the NC program. In the **Program Run** and **Editor** operating modes, the file information bar contains a breadcrumb navigation.

If the NC program is write-protected, then the control displays a symbol in front of the file path.

- 3 Contents of the NC program

Further information: "Appearance of the NC program", Page 120

- 4 The **Form** column

Further information: "The Form column in the Program workspace", Page 132

- 5 Help graphic of the syntax element being edited

Further information: "Help graphic", Page 120

- 6 Dialog bar

In the dialog bar, the control shows additional information or instructions for the syntax element being edited.

7 Action bar

In the action bar, the control shows selection options for the syntax element being edited.

8 **Structure**, **Search** or **Tests** column

Further information: "The Structure column in the Program workspace", Page 501

Further information: "The Search column in the Program and Text editor workspaces", Page 504

Further information: "The Tests column in the Program workspace", Page 128

Symbols, icons, and buttons in the title bar

The title bar of the **Program** workspace contains the following symbols, icons, and buttons:

Further information: "Icons on the control's user interface", Page 73

Icon, button, or shortcut	Meaning
	Open and close the Structure column Further information: "The Structure column in the Program workspace", Page 501
 [CTRL] + [F]	Open and close the Search column Further information: "The Search column in the Program and Text editor workspaces", Page 504
	Open and close the Tests column Further information: "The Tests column in the Program workspace", Page 128
	Error
	Warning If the Simulation report area contains an error or a warning, then the control displays a notification symbol. The control displays the notification symbol as part of the Tests column symbol. Further information: "The Simulation report area", Page 130
Mode selection menu	Select the mode of the Program workspace ■ Text mode Further information: "Text mode", Page 135 ■ Klartext editor Default selection ■ File comparison Further information: "Program comparison", Page 509
	Open and close the Form column Further information: "The Form column in the Program workspace", Page 132
	Open the Program settings window Further information: "Settings in the Program workspace", Page 120

Appearance of the NC program

By default, the control displays the syntax in black. Within the NC program, the control displays the syntax elements in various font colors:

Color	Syntax element
Brown	Text entries (e.g., tool name or file name)
Blue	- Numerical values - Structure items and texts
Dark green	Comments
Purple	- Variables - Miscellaneous functions M
Dark red	- Definition of spindle speed - Definition of feed rate
Orange	Rapid traverse FMAX
Gray	- Not-to-be-executed M1 miscellaneous function - Not-to-be-executed NC block hidden with a / character

Help graphic

When you are editing an NC block, the control shows for some NC functions a help graphic in a pop-up window that illustrates the current syntax element. If you change the size and position of the pop-up window, the control will save the setting separately for each tab.

Whether the control displays the help graphic depends on the **Show help graphics automatically** setting or the **stdTNC** machine parameter (no. 105405).

Further information: "Settings in the Program workspace", Page 120

The pop-up window includes the following buttons:

Button	Meaning
Show TNCguide	The control opens TNCguide at the corresponding position in the Help workspace. Further information: "User's Manual as integrated product aid: TNCguide", Page 37
Show help	The control opens the help graphic in the Help workspace. If the Help workspace is open, the control will always display the help graphic there.

Further information: "The Help workspace", Page 490

Settings in the Program workspace

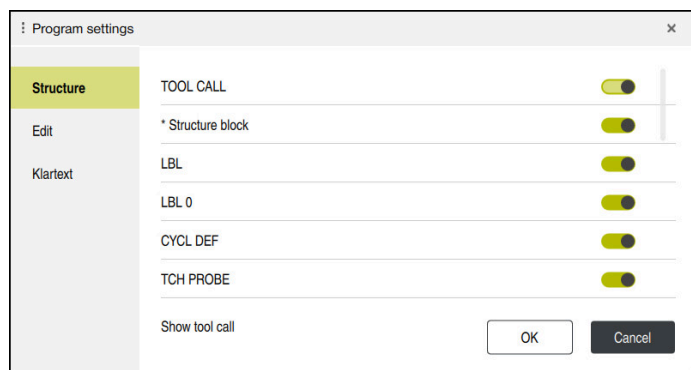
In the **Program settings** window, you can influence contents shown in the **Program** workspace as well as the control's behavior there. The selected settings are modally effective.

The settings available in the **Program settings** window depend on the operating mode or application. The **Program settings** window consists of the following areas:

Area	The Editor operating mode	The Program Run operating mode	The MDI application
Structure	✓	✓	✓
Edit	✓	-	✓
Klartext	✓	-	✓

Area	The Editor operating mode	The Program Run operating mode	The MDI application
Tables	-	✓	-
FN 16	-	✓	✓

The Structure area



The **Structure** area in the **Program settings** window

In the **Structure** area, you can use toggle switches to choose which structuring items the control should display in the **Structure** column.

Further information: "The Structure column in the Program workspace", Page 501

The following structure elements are available:

- **TOOL CALL**
- *** Structure block**
- **LBL**
- **LBL 0**
- **CYCL DEF**
- **TCH PROBE (#17 / #1-05-1)**
- **ISO cycle**
- **CALL PGM**
- **SEL PGM**
- **FUNCTION MODE**
- **M30 / M2**
- **M1**
- **M0 / STOP**

The Edit area

The **Edit** area contains the following settings:

Setting	Meaning
Automatic saving	Save changes to the NC program automatically or manually If the toggle switch is active, the control saves the NC program automatically upon the following actions: ■ Switching between tabs ■ Starting the simulation ■ Closing the NC program ■ Switching the operating mode If the toggle switch is not active, you must save manually. Upon the stated actions, the control asks whether the changes should be saved.
Autocomplete in text mode	If the toggle switch is active, the control will automatically display a selection menu with possible syntax initiators or syntax elements when you select one of the following actions: ■ Creating a new NC block ■ Entering characters ■ Press the SPACE soft key If the toggle switch is not active, you can open the selection menu by pressing [CTRL] + [SPACE]. Further information: "Text mode", Page 135
Allow syntax errors in text mode	If you activate the toggle switch, then the control can save NC blocks in Text mode, even if they contain syntax errors. If the toggle switch is not active, you must correct all syntax errors within an NC block. Otherwise you cannot save the NC block. Further information: "Editing NC functions", Page 111
Generate absolute paths	Create relative or absolute path entries If the toggle switch is active, the control uses absolute paths for called files, e.g.: TNC:\nc_prog\\$.mdi.h. If the toggle switch is not active, the control uses relative paths, e.g.: demo \reset.H. If the file is located at a higher level in the folder structure than the calling NC program, the control creates an absolute path. Further information: "Path", Page 354
Always save formatted	Format NC program while saving If an NC program has fewer than 30 000 characters, the control always formats the file when saving it, e.g.: capital letters for all syntax initiators. If the toggle switch is active, the control also formats NC programs with more than 30 000 characters each time it saves the file. This can increase the time needed for saving. If the toggle switch is not active, the control does not format NC programs with more than 30 000 characters.

Setting	Meaning
Back-up file when saving	If the toggle switch is active, the control will save a backup copy with the *.h.bak extension once you save the NC program. By removing the *.bak extension from the file name, you can restore the backup copy. The control overwrites the original file.  If you select the All Files (*.*) filter, then the control displays the file in the Open File workspace. The same setting is also available in the machine parameter createBackup (no. 105401). The control will reconcile both setting options.
Behavior of the cursor after deletion of lines	If you activate the toggle switch and delete an NC program line, the cursor will move back to the previous NC block. The same setting is also available in the machine parameter deleteBack (no. 105402). The control will reconcile both setting options.
Show help graphics automatically	If the toggle switch is active, the control will show a help graphic in a pop-up window. The same setting is also available in the optional machine parameter stdTNChelp (no. 105405). The control will reconcile both setting options. When the Help workspace is open, the control will always display the help graphic there, independently of this setting. Further information: "The Help workspace", Page 490
Confirmation request when deleting an NC block	If the toggle switch is active, the control will display a confirmation prompt in a pop-up window when you delete an NC block. The same setting is also available in the optional machine parameter warningAtDEL (no. 105407). The control will reconcile both setting options.
Comment blocks for NC sequences	If the toggle switch is active, the control adds a comment before and after each NC sequence. Each comment includes the following information: ■ Start of the NC sequence ■ Current date ■ Current time ■ Name of the NC sequence ■ End of the NC sequence Further information: "NC sequences for reuse", Page 201
Hide NC functions that aren't available	If the toggle switch is active, the control will only display currently available NC functions in the Insert NC function window. If the toggle switch is not active, the control dims unavailable NC functions (e.g., for software options that are not enabled).
Put all path information in quotation marks	If the toggle switch is active, the control will automatically enclose path information in quotation marks when you select one of the following NC functions: ■ CALL PGM ■ Cycle 12 PGM CALL ■ FN 16 F-PRINT ■ FN 26 TABOPEN The same setting is also available in the optional machine parameter quotePaths (no. 105414). The control will reconcile both setting options.

Setting	Meaning
Automatically adapt cycle comments	If the toggle switch is active, the control displays comments for some cycle parameters corresponding to the current input. This setting only affects cycle parameters where you select one of the predefined input values (e.g., Q215 MACHINING OPERATION). The comment directly displays the meaning of the selected value (e.g., Roughing and finishing in Q215 = 0). Thus you can recognize the programmed status immediately without having to edit the cycle.
Display screen keyboard for editing	If a touchscreen is used, the control will display a context-sensitive virtual keyboard. A selection menu allows you to select the position of the virtual keyboard in the workspace or to hide the virtual keyboard.

Klartext area

In the **Klartext** area, select whether the control offers certain syntax elements of an NC block during input.

The control offers the following settings as toggle switches:

Setting	Meaning
Skip comment	If you activate this toggle switch, the control skips the comment function during programming for all NC functions. Further information: "Adding comments", Page 498
Skip tool index	If you activate this toggle switch, the control skips the tool index for the following NC functions: ■ Calling a tool with TOOL CALL Further information: "Using TOOL CALL to call a tool", Page 162 ■ Preselecting a tool with TOOL DEF Further information: "Tool pre-selection using TOOL DEF", Page 169 Further information: User's Manual for Setup and Program Run

You can program the syntax elements in the form independently of the settings in the **Klartext** area.

Tables

In the **Tables** area, you can select a unique table for each of the application areas shown; this table is then active during program run.

Select the following tables using a selection window:

- **Datums**
Further information: "Datum table *.d", Page 610

FN 16

In the **FN 16** area, use the **Show pop-up window** toggle switch to select whether the control displays a window in conjunction with **FN 16**.

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 413

Using the Program workspace

The **Program** workspace can be used as follows:

- Touch operation
- Operation with keys and buttons
- Operation with a mouse

Touch operation

You use gestures to perform the following functions:

Symbol	Gesture	Meaning
	Tap	■ Select an NC block ■ Select a syntax element while editing
	Double tap	Edit an NC block or mark characters Further information: "Marking characters in an NC block", Page 127
	Long press	Open the context menu  If you are working with a mouse, click with the right mouse key. Further information: "Context menu", Page 511
	Swipe	Scroll in an NC program
	Drag	Change the area in which NC blocks are marked. Further information: "Context menu in the Program workspace", Page 514
	Spread	Increase the syntax font size
	Pinch	Reduce the syntax font size

Keys, buttons, and shortcuts

You can use keys, buttons, and key combinations for the following functions:

Key, button, or shortcut	Meaning
 	■ Navigate between NC blocks ■ Navigate within selection menus ■ During editing, search for the same syntax element in the NC program Further information: "Searching for identical syntax elements in different NC blocks", Page 127
 	■ Edit an NC block ■ During editing, navigate to previous or next syntax element ■ Arrow right: in text mode, take over a syntax element from the autocomplete feature
[CTRL] + [RIGHT] [CTRL] + [LEFT]	Navigate one position to the right or left within the value of a syntax element
	■ Use the block number to select an NC block directly Further information: "The GOTO function", Page 496 ■ Open selection menus during editing
	Open position display of the control bar in order to copy the position If you select a line in the position display, the control copies the current value of this line to an open dialog.
	Delete value of a syntax element
	Skip or remove optional syntax elements during programming
	Delete an NC block or cancel a dialog
	■ Confirm entry and conclude an NC block ■ Open the Add tab
[SHIFT] + [RETURN]	Enter a line break in text mode Insert a line break in the Form column for comments
	Cancel editing without applying changes
Insert NC function	Open the Insert NC function window Further information: "Areas of the Insert NC function window", Page 133
Edit	Open the context menu Further information: "Context menu", Page 511
[CTRL] + [H]	Insert the most recently programmed, edited, or deleted NC block

Searching for identical syntax elements in different NC blocks

If you are editing an NC block, you can search for the same syntax element in the rest of the NC program.

To search for a syntax element in the NC program:

- ▶ Select an NC block



- ▶ Edit the NC block

- ▶ Navigate to the desired syntax element



- ▶ Press the arrow up or down key

- > The control selects the next NC block that contains the syntax element. The cursor is on the same syntax element as in the previous NC block. Press the arrow up key to search backwards.



- If you also keep the **SHIFT** key held down, the control marks all NC blocks until the previous or next identical syntax element.
- You can search for identical syntax initiators in an NC program. Select the syntax initiator by double-tapping or double-clicking it.

Marking characters in an NC block

You can mark multiple characters within a single NC block.

To mark multiple characters in a single NC block:

- ▶ Double-tap or double-click the NC block
- > The control marks the selected value.
- ▶ Use the "drag" gesture to expand or reduce the marked area



In Text mode you can mark any range of characters. In the **Klartext editor** mode you can mark only the characters of a value.

Further information: "Contents of an NC program", Page 114

Notes

- When you search for the same syntax element in a very long NC program, the control displays a pop-up window. You can cancel the search at any time.
- If the NC block contains a syntax error, the control precedes the block number with a corresponding icon. Click the icon to see the associated error description.
- When you open an NC program, the control checks whether the NC program is complete and syntactically correct.
- If you open an NC program without content, you can edit the **BEGIN PGM** and **END PGM** NC blocks and change the unit of measure of the NC program.
- An NC program is incomplete without the **END PGM** NC block.
If you open an incomplete NC program in the **Editor** operating mode, the control automatically adds this NC block.
- You cannot edit an NC program in the **Editor** operating mode if this NC program is currently being executed in the **Program Run** operating mode.
- The execution cursor is always displayed in the foreground. The execution cursor may cover or hide other icons.
- If you mark characters through touch operation, the control displays two marker symbols below the cursor.
- Within the **Program** workspace, calculations aren't possible in entry fields for numerical values.
- Using the optional machine parameter **editWhileProtected** (no. 105416), you can define whether it is possible to edit a read-only file. By default, you can edit such a file, but only save the changes in a new file with **Save as**. If you set this machine parameter to **FALSE**, the control will dim any editing functions.
- While programming, you can select table values, such as tools, from a selection window. By default, the control will display the tables associated with the current program run in the selection window. You can define other tables for simulation. The control will display the simulation tables in the selection window only while the **Simulation** workspace is open.

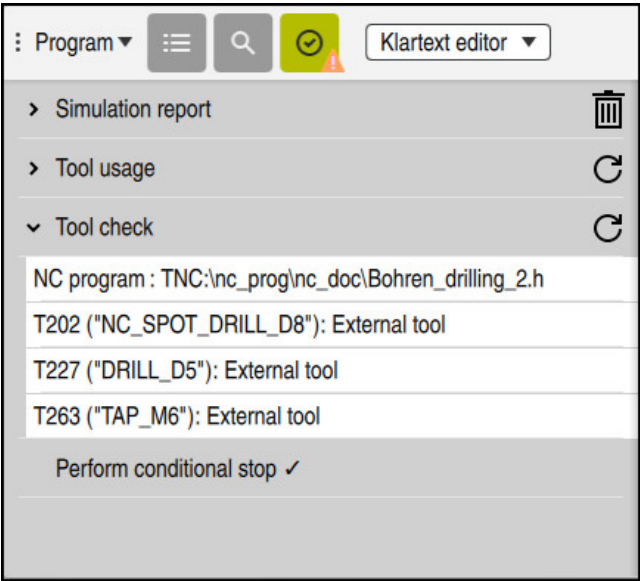
Further information: "The Simulation settings window", Page 534

The Tests column in the Program workspace

Application

In the **Tests** column of the **Program** workspace, the control offers various functions for checking your NC program (e.g., a tool check). A tool check allows you, for example, to check the remaining tool life.

Description of function



The **Tests** column in the **Program** workspace

In the **Tests** column of the **Program** workspace, the control displays the following areas:

- **Simulation report**
Only in the **Editor** operating mode and when the **Simulation report** toggle switch is active in the **Simulation settings** window
Further information: "The Simulation settings window", Page 534
Further information: "The Simulation report area", Page 130
- **Tool usage**
Further information: "The Tool usage area", Page 130
- **Tool check**
Further information: "The Tool check area", Page 131

For the tool-usage text, you need the **Tool usage** and **Tool check** areas.

Further information: User's Manual for Setup and Program Run

Icons of the **Tests** column

The control displays the following icons in the **Tests** column:

Icon	Meaning
	Open the Notifications window Only in the Simulation report area
	Refresh Create a tool-usage file or perform a tool-usage test Only in the Tool usage and Tool check areas

The Simulation report area

If the **Simulation report** toggle switch in the simulation settings is active, the control will display the **Simulation report** area. The control will then display simulation errors and messages in the simulation report rather than in the message menu.

Further information: "The Simulation settings window", Page 534

If the simulation report contains messages, the control will display a notification symbol on the icon of the **Tests** column and a message in the message menu.

Further information: "Symbols, icons, and buttons in the title bar", Page 119

If you select a message, the control will select the corresponding NC block in the NC program. You can then use the **Rewind** function to find the reason of the respective warning.

Further information: "Analyzing a simulation warning", Page 546

If the simulation report contains a message, the control will display the **Group** toggle switch. If you activate this toggle switch, the control will group all messages with the same error number. If you select the grouped message, the control will expand the group and display all message with this error number.

The Tool usage area

If no tool-usage file has been created, then the **Tool usage** area is empty.

Further information: User's Manual for Setup and Program Run

In the **Tool usage** area, the control displays the chronological sequence of all tool calls, along with the following information:

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

Select the **Refresh** icon to create a tool-usage file for your NC program.

The Tool check area

The **Tool check** area is empty until you perform a tool-usage test with the **Refresh** icon.

Further information: User's Manual for Setup and Program Run

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

- The tool is defined in the tool management.
- The tool is defined in the pocket table.
- 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.

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, then transfer the missing tool to 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: "Using TOOL CALL to call a tool", Page 162

The Form column in the Program workspace

Application

In the **Form** column of the **Program** workspace, the control shows all possible syntax elements for the currently selected NC function. In the form, you can edit all syntax elements as well as the syntax initiator, if required.

Related topics

- Editing an NC function in the **Form** column
Further information: "Editing NC functions", Page 111

Requirement

- **Klartext editor** mode must be active

Description of function

The control offers the following icons and buttons for using the **Form** column:

Icon or button	Meaning
	Show and hide the Form column
Confirm	Confirm entry and conclude an NC block
Discard	Discard entries and conclude an NC block
Delete line	Delete NC block

The control groups the syntax elements in the form depending on their functions, such as coordinates or safety.

The control indicates the required syntax elements with a red frame. Only once you have defined all of the required syntax elements can you confirm the entries and conclude the NC block. The control highlights the syntax element currently being edited.

If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

Notes

- In the following cases the control shows no contents in the form:
 - NC program is being run
 - NC blocks are being marked
 - NC block contains syntax error(s)
 - **BEGIN PGM** or **END PGM** NC blocks are selected
- If you define more than one miscellaneous function in an NC block, you can use the arrows in the form to change the sequence of the miscellaneous functions.
- If you define a label with a number, the control shows a symbol next to the input area. The control uses this symbol to assign the next available number to the label.

4.3.4 The Insert NC function window

Application

The **Insert NC function** window allows you to insert NC functions or NC sequences into an NC program.

Related topics

- Creating NC sequences

Further information: "NC sequences for reuse", Page 201

- Inserting and editing NC functions

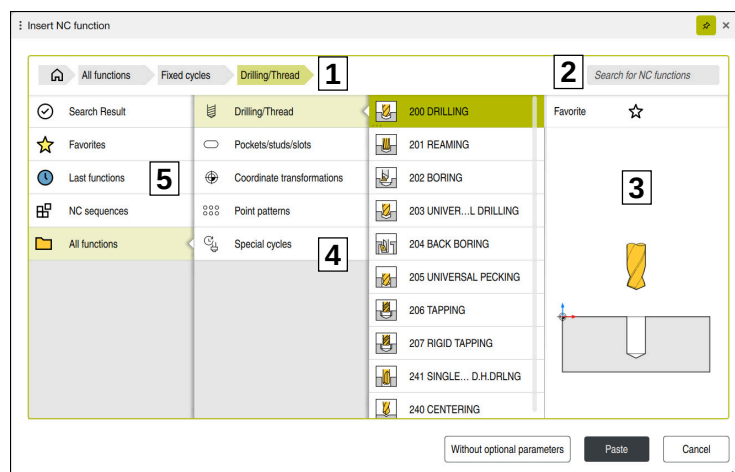
Further information: "Possible methods for editing", Page 109

Description of function

The **Insert NC function** window is available only in the **Editor** operating mode and **MDI** application.

i In the **MDI** application, you can insert NC functions into the **\$mdi.h** or **\$mdi_inch.h** NC program only.

Areas of the Insert NC function window



The **Insert NC function** window

1 Navigation path

In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level. You can change the path or use the history to open a previous path.

Further information: "File management areas", Page 351

2 Searching

Use the **Search for NC functions** feature to search for the syntax initiator of the NC function or the name of the NC sequence.

The control displays the results under **Search Result**.

i You can begin the search by entering a character as soon as the **Insert NC function** window opens.

- 3 The control shows the following information and functions:
 - Add or remove a favorite
 - Preview

The control shows a preview of the content for NC sequences and a preview image for cycles.
- 4 Content columns

The control shows NC functions or folders that contain NC functions. The control displays up to two columns.
- 5 Navigation column

The navigation column offers the following possibilities for navigation:

 - **Search Result**

The control shows the following search results:

 - NC functions or miscellaneous functions whose name includes the content being searched for (e.g., Cycle **200** in a search for "20")
 - Equivalent or alternative NC functions (e.g., **PATTERN DEF** when searching for "pattern")
 - **Favorites**

The control displays all NC functions and NC sequences that you have marked as favorites.

Further information: "Icons on the control's user interface", Page 73
 - **Last functions**

The control shows the ten most recently used NC functions and NC sequences.
 - **NC sequences**

Use the NC sequences to insert a saved sequence of NC functions.

Further information: "NC sequences for reuse", Page 201
 - **All functions**

The control shows all available NC functions in the folder structure.

You can limit the selection possibilities using the keys or buttons. When you press the **CYCL DEF** key, the control will open the groups of cycles.

Further information: "Keycaps for NC dialog", Page 69

In the **Search Result**, **Favorites** and **Last functions** areas, the control shows the path of the NC functions.

File functions in the Insert NC function window

If you drag an NC function to the right in the **Insert NC function** window, the control provides the following file functions:

- Add or remove a favorite
- Navigate to the NC function

Not available in the **All functions** area

For NC sequences, the control provides the following additional file functions:

- Edit
- Rename
- Delete
- Activate or deactivate write protection
- Open the path in the **Files** operating mode

Further information: "NC sequences for reuse", Page 201

Notes

- In order to program cycles compatible with older control systems or software versions, you can use the **Without optional parameters** button in the **Insert NC function** window. This ensures that the control will not add any parameters that have been added to the cycle at a later time. You can display the optional parameters at any time.

Further information: "Optional parameters", Page 138

- For some NC functions, the **Insert NC function** window offers the possibility of inserting the start and end of the NC function into the NC program at the same time (e.g., **IF and END IF**).

If you mark several NC blocks in the NC program and insert the combined NC functions, the control will insert the corresponding NC function before and after the marked area.

- The instructions include emphasized text strings (e.g., **200 DRILLING**). You can use these text strings for better searching in the **Insert NC function** window.
- If software options are not enabled, the control dims unavailable contents in the **Insert NC function** window.

4.3.5 Text mode

Application

The Text mode is a possibility for programming within the **Program** workspace. In Text mode you create and edit NC programs using the keyboard instead of the **Insert NC function** window.

Related topics

- Editing text files in the **Text editor** workspace
Further information: "The Text editor workspace", Page 366
- Basics and operation of the **Program** workspace
Further information: "The Program workspace", Page 118

Description of function

The title bar of the **Program** workspace contains a selection menu in which you can select the text mode. Text mode works similarly to a common text editor. For example, the arrow keys don't move the cursor from one syntax element to the next, but only to the next character.

Text mode supports all navigation possibilities of the **Program** workspace.

Further information: "Using the Program workspace", Page 124

If the control can't automatically correct syntax errors in the NC program, it activates Text mode.

Further information: "Editing NC functions", Page 111

In the **Program settings** window the control offers the following settings for Text mode:

- **Autocomplete in text mode**
- **Allow syntax errors in text mode**

Further information: "Settings in the Program workspace", Page 120

Auto-completion in Text mode

While programming in Text mode, the control opens a selection menu with all syntax elements that you can insert at the current cursor position.

If the **Autocomplete in text mode** toggle switch is set to active, the control displays the selection menu upon the following actions:

- Creating a new NC block
- Entering characters
The control adapts the proposals to the entered characters.
- Pressing the **SPACE** key

If the toggle switch is not active, you can open the selection menu by pressing **[CTRL] + [SPACE]**.

You have the following options for entering the desired syntax element:

- Tap
- Click
- The right arrow key

If you used the arrow keys to choose an element from the selection menu, or if there is only one element.

Further information: "Inserting NC functions", Page 111

Notes

- By default, the control displays only syntax elements without values when using auto-completion. If miscellaneous functions are possible, then the control displays all miscellaneous functions with numbers after the remaining possible syntax elements.
- If you are programming a cycle, the control offers the **Only downwardly-compatible cycle parameters** and **With optional cycle parameters** possibilities for auto-completion.
When you select **Only downwardly-compatible cycle parameters**, you can add optional cycle parameters later on. For this purpose, you enter a line break in the last line.
- If you press the right arrow key during auto-completion without clearly choosing an element, the control will close the selection menu.
- In the Text mode you can enter line breaks at any location. If you later edit the NC functions in the **Klartext editor** mode, the control will remove the line breaks after saving. The line breaks will be preserved in comments and structuring items even after editing.

4.4 Working with cycles

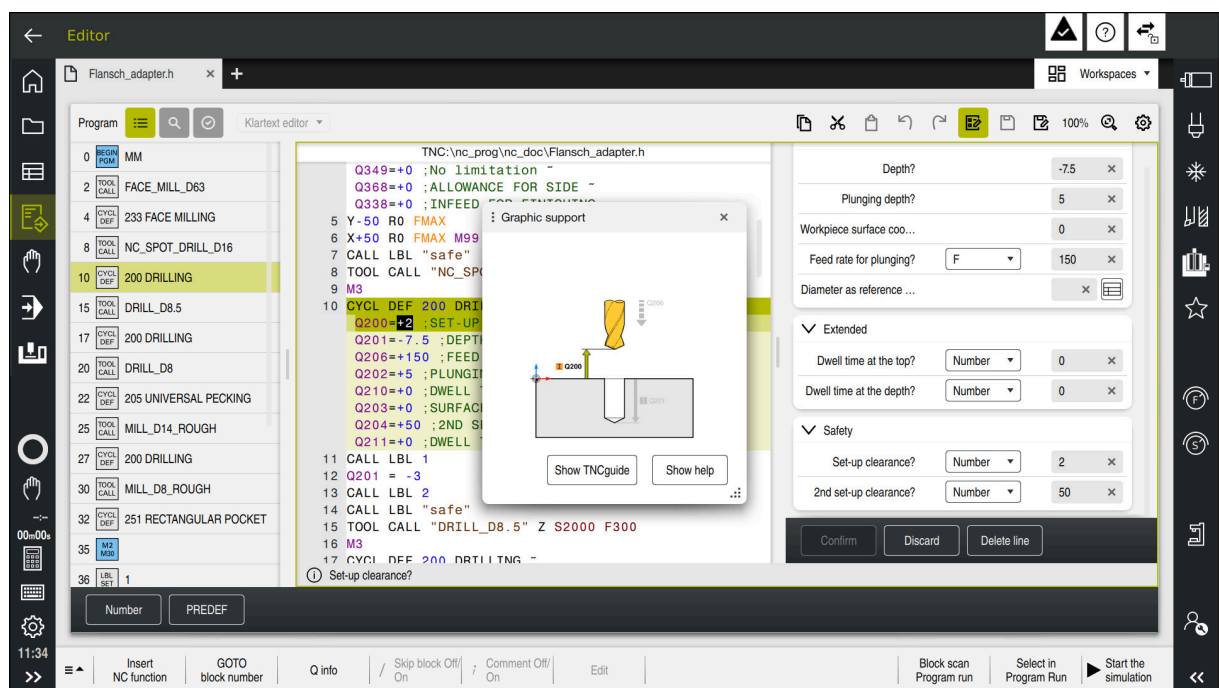
4.4.1 General information on cycles

General



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

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



Cycles are stored on the control as subprograms. The cycles can be used to execute different machining operations, thereby enabling the easy and efficient programming of recurring machining operations. Most cycles use Q parameters as transfer parameters. The control offers cycles for the following machining technologies:

- Drilling processes
- Thread machining
- Milling operations such as pockets and studs
- Cycles for coordinate transformation

NOTICE

Danger of collision!

Cycles sometimes perform complex machining movements. There is therefore a risk of collision!

- Simulate your program before executing it

NOTICE**Danger of collision!**

You can program variables as input values in HEIDENHAIN cycles. If you define cycle parameters with variables, then use only the input range of the cycle parameter. Otherwise, collisions may occur.

- ▶ Use only the input range of the cycle parameter
- ▶ Pay attention to the HEIDENHAIN documentation
- ▶ Check the machining sequence using a simulation



In inch programs, the feed rate for cycles must be defined in 0.1 inch/min.

Optional parameters

HEIDENHAIN continuously further develops the extensive package of cycles. Additional cycle parameters for cycles may become available with every new software version. These new Q parameters are **optional parameters**, some of which were not contained in previous software versions. These parameters are located at the end of the cycle definition.

You have the following alternatives when inserting a cycle:

- Insert the cycle with all optional parameters. Every optional parameter can be hidden using the **NO ENT** key.
- Insert the cycle with only its required parameters. In this case, you insert the cycle without optional parameters.



If you hide a Q parameter or insert a cycle with only its required parameters, then the values of the optional Q parameters that are not shown are their default values.

To remove optional parameters in the NC program:

Insert
NC function

Paste



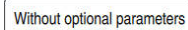
NO
ENT

- ▶ Select **Insert NC function**
 - > The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- ▶ Select **Paste**
 - > The control inserts the cycle definition with all cycle parameters into the NC program.
- ▶ Navigate to the optional cycle parameters by pressing the arrow key that points to the right
- ▶ Press the **NO ENT** key to hide the optional cycle parameter

Insert the cycle with only its required parameters:

 Insert
NC function

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle


 Without optional parameters

- ▶ Select **Without optional parameters**
- The control inserts the cycle definition without optional cycle parameters into the NC program.

If you unintentionally hide an optional parameter or need it again at a later time, then you can once again show and edit it within the cycle.

To show optional parameters:

- ▶ Edit cycle



- ▶ Navigate to the desired optional cycle parameters by pressing the arrow key that points to the right


 ENT

- ▶ Change the default value as needed
- ▶ Confirm the optional cycle parameter by pressing **ENT**


 END
□

- ▶ Save the change by pressing **END**

Compatibility

The TNC7 go can usually execute the NC programs of predecessor controls starting with the TNC 124 and later even though the cycles have been extended to include additional optional parameters.

This is possible thanks to the stored default value. To execute a current NC program on an older control or software version, remove the optional cycle parameters. This gives you a backward-compatible NC program. If NC blocks contain invalid elements, then the control indicates that these elements are ERROR blocks.

Defining cycles

Cycles can be defined in several ways.

Inserting via NC function:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- The control initiates a dialog and prompts you for all required input values.

Inserting machining cycles via the CYCL DEF key:



- ▶ Press the **CYCL DEF** key
- The control opens the **Insert NC function** window.
- ▶ Select the desired cycle
- The control initiates a dialog and prompts you for all required input values.

Navigation in the cycle

Key	Function
	Navigation within the cycle: Jump to next parameter
	Navigation within the cycle: Jump to previous parameter
	Jump to the same parameter in the next cycle
	Jump to the same parameter in the previous cycle



For some cycle parameters, the control provides selectable choices via the action bar or the form.

If an input option specifying a defined behavior is stored in certain cycle parameters, you can open a selection list with the **GOTO** key or in the form view. For example, in Cycle **200 DRILLING** the **Q395 DEPTH REFERENCE** parameter provides the following options:

- 0 | Tool tip
- 1 | Cutting edge corner

Cycle input form

The control provides a **FORM** for various functions and cycles. This **FORM** allows you to enter various syntax elements or cycle parameters.

The screenshot shows a 'Cycle input form' with a list of parameters and their values. At the top left is a downward arrow icon. The parameters and their values are:

Machining ope...	0	x	
First side lengt...	60	x	
Second side l...	20	x	
Depth?	-20	x	
Position of po...	0	x	
Plunging depth?	5	x	
Feed rate for ...	F	500	x
Feed rate for p...	F	150	x
Finishing feed ...	F	500	x
Finishing allo...	0	x	
Finishing allo...	0	x	
Infeed for finis...	0	x	
Set-up clearan...	Number	2	x

At the bottom of the form are three buttons: 'Confirm', 'Discard', and 'Delete line'.

The control allocates the cycle parameters in the **FORM** to groups based on their functions (e.g., geometry, standard, advanced, safety). The control provides selection possibilities for different cycle parameters via switches, for example. The control displays the currently edited cycle parameter in color.

After you have defined all required cycle parameters, you can confirm your input and conclude the cycle.

To open the form:



- ▶ Select the **Editor** operating mode



- ▶ Select the desired **Program**
- ▶ Select **FORM** via the title bar



If an input is invalid, the control displays an information symbol ahead of the syntax element. When you select the information symbol, the control displays information on the error.

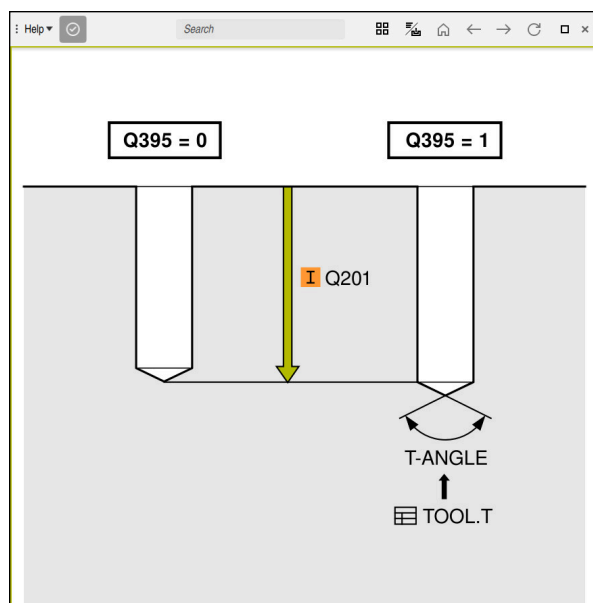
Help graphics

When you are editing a cycle, the control shows a help graphic for the current Q parameters. The size of the help graphic depends on the size of the **Program** workspace.

The control shows the help graphic at the right edge of the workspace, or at the top or bottom edge. The help graphic is positioned in the half that does not contain the cursor.

When you tap or click on the help graphic, the control maximizes the help graphic.

If the **Help** workspace is active, the control will display the help graphic in this area instead of showing it in the **Program** workspace.



The **Help** workspace with a help graphic for a cycle parameter

Calling cycles

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the machining cycle that was defined last in the NC program.

Requirements

Before calling a cycle, be sure to program:

- **BLK FORM** for graphic display (only required for simulation)
- Tool call
- Spindle direction of rotation (miscellaneous function **M3/M4**)
- Cycle definition (**CYCL DEF**)



For some cycles, additional requirements must be observed. They are detailed in the descriptions and overview tables for each cycle.

You can program the cycle call in the following ways:

Syntax	Further information
CYCL CALL	Page 143
CYCL CALL PAT	Page 143
M89/M99	Page 144

Calling a cycle with **CYCL CALL**

The **CYCL CALL** function calls the most recently defined machining cycle once. The starting point of the cycle is the position that was programmed most recently before the **CYCL CALL** block.



- ▶ Select **Insert NC function**
or
- ▶ Press the **CYCL CALL** key
- The control opens the **Insert NC function** window.
- ▶ Select **CYCL CALL M**
- ▶ Define **CYCL CALL M** and add an M function, if necessary

Calling a cycle with **CYCL CALL PAT**

The **CYCL CALL PAT** function calls the most recently defined machining cycle at all positions that you defined in a **PATTERN DEF** pattern definition or in a point table.

Further information: "Pattern definition with PATTERN DEF", Page 208

Further information: "Point tables", Page 206



- ▶ Select **Insert NC function**
or
- ▶ Press the **CYCL CALL** key
- The control opens the **Insert NC function** window.
- ▶ Select **CYCL CALL PAT**
- ▶ Define **CYCL CALL PAT** and add an M function , if necessary

Calling cycles with additional functions

M99

The **M99** miscellaneous function calls the most recently defined machining cycle once. **M99** is effective blockwise and at the end of the block (e.g., after the traverse movement)

Example

```
11 CYCL DEF 200 DRILLING
```

```
...
```

```
12 Y+50 R0 FMAX M99
```

The control traverses at **FMAX** to the position **X+50**. Then the control calls Machining Cycle **200 DRILLING** with **M99**.

M89

If the control is to execute the cycle automatically after every positioning block, program the first cycle call with **M89**.

You can cancel **M89** with the following functions:

- **M99** at the last position
- New machining cycle with **CYCL DEF**

Defining and calling an NC program as cycle

With **SEL CYCLE**, you can define any NC program as a machining cycle.

To define an NC program as a cycle:

- Insert
NC function

CYC

- ▶ Select **Insert NC function**
 - The control opens the **Insert NC function** window.
 - ▶ Select **SEL CYCLE**
 - ▶ Select file name, string parameter, or file

To call an NC program as a cycle:

- CYCL
CALL

- ▶ Press the **CYCL CALL** key
 - The control opens the **Insert NC function** window.
or
 - ▶ Program **M99**



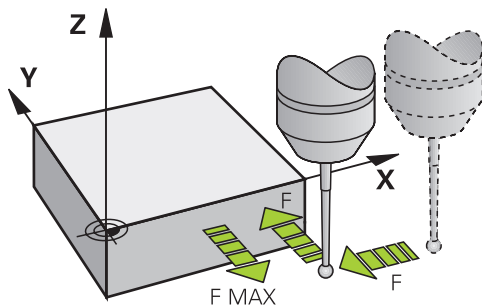
- If the called file is located in the same directory as the file you are calling it from, you can also integrate the file name without the path.
- **CYCL CALL PAT** use a positioning logic before the respective cycle is executed. With respect to the positioning logic, the **SEL CYCLE** and Cycle **12 PGM CALL** show the same behavior. In point pattern cycles, the clearance height for approaching is calculated based on:
 - The maximum Z position at the start of the pattern
 - All Z positions in the point pattern

4.4.2 General information about touch probe cycles

Description of function



- Refer to your machine manual.
- The control must be specifically prepared by the machine manufacturer for the use of a touch probe.
- HEIDENHAIN guarantees the proper operation of the touch probe cycles only in conjunction with HEIDENHAIN touch probes.
- If you are using a HEIDENHAIN touch probe with EnDat interface, then the Touch Probe Functions software option (#17 / #1-05-1) is automatically enabled.
- The control's full range of functions is available only if the **Z** tool axis is used.
- Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.



You can use the automatic touch-probe functions to calibrate the touch probe and to measure tools. In manual operation, you have the ability to set presets on the workpiece and to measure the workpiece.

When the probe stylus contacts the workpiece,

- the 3D touch probe transmits a signal to the control: the coordinates of the probed position are stored,
- the touch probe stops moving, and
- returns to its starting position at rapid traverse.

If the stylus is not deflected within a defined distance, the control displays an error message (distance: **DIST** from touch probe table).

Related topics

- Manual touch probe cycles
- Preset table
- Datum table

Further information: "Datum table *.d", Page 610

- Reference systems
- Further information:** "Reference systems", Page 308

- Preassigned variables

Further information: "Preassigned Q parameters", Page 402

Requirements

- Touch Probe Functions (#17 / #1-05-1) software option
- Calibrated workpiece touch probe

General information on the touch-probe table

In the touch probe table you define the set-up clearance, i.e., how far away from the defined touch point (or the one calculated by the cycle) the control will pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles, you can also define a set-up clearance that is added to the one from the touch probe table.

The following can be defined in the touch probe table:

- Type of tool
- Touch probe center offset
- Spindle angle during calibration
- Probing feed rate
- Rapid traverse in touch-probe cycle
- Maximum measuring range
- Set-up clearance
- Feed rate for pre-positioning
- Touch probe orientation
- Serial number
- Reaction in case of collision

Further information: User's Manual for Setup and Program Run

Touch probe cycles in the Manual Operation and Electronic Handwheel modes

In the **Setup** application, the control provides touch probe cycles in **Manual** mode that allow you to:

- Set presets
- Probe position
- Calibrate the touch probe
- Measure the tool

Further information: User's Manual for Setup and Program Run

4.4.3 Machine-specific cycles

 Refer to your machine manual for a description of the specific functionality.

Cycles are available for many machines. Your machine manufacturer can implement these cycles into the control, in addition to the HEIDENHAIN cycles. These cycles are available in a separate cycle-number range:

Cycle-number range	Description
300 to 399	Machine-specific cycles that are to be selected through the CYCL DEF key
500 to 599	Machine-specific touch probe cycles that are to be selected through the CYCL DEF key

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections, resulting in undesired behavior. Danger of collision during machining!

- ▶ Only use variable ranges recommended by HEIDENHAIN
- ▶ Do not use pre-assigned variables
- ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- ▶ Check the machining sequence using the simulation

Further information: "Calling cycles", Page 143

Further information: "Variables: Q, QL, QR, QS parameters and named parameters", Page 393

4.4.4 Available cycle groups

Machining cycles

Cycle group	Further information
Drilling/Thread	
■ Drilling, reaming	Page 232
■ Boring	Page 262
■ Counterboring, centering	
■ Tapping	Page 269
Pockets/studs/slots	
■ Pocket milling	Page 279
■ Stud milling	Page 289
■ Slot milling	
■ Face milling	Page 297
Coordinate transformations	
■ Mirroring	Page 322
■ Magnifying / Reducing	
Point patterns	
■ Bolt hole circle	Page 219
■ Linear hole pattern	
Special cycles	
■ Dwell time	Page 378
■ Oriented spindle stop	
■ Program call	Page 199

Measuring cycles

Cycle group	Further information
Measuring the tool (TT)	
■ Calibrating the TT	Page 561
■ Tool length, radius or measuring completely	Page 554
■ Calibrating the IR-TT	

5

Machining mode

5.1 Switching the operating mode with FUNCTION MODE

Application

With **FUNCTION MODE SET**, you can activate settings defined by the machine manufacturer, such as changes of the traverse range.

Related topics

- Editing kinematic models in the **Settings** application

Further information: User's Manual for Setup and Program Run

Requirement

- Control adapted by the machine manufacturer
The machine manufacturer defines which internal functions the control performs with this function. The machine manufacturer must define selection possibilities for the **FUNCTION MODE SET** function.

Description of function

When the operating modes are switched, the control executes a macro that defines the machine-specific settings for the specific operating mode.

If the machine manufacturer has enabled the selection of various kinematic models, then you can switch between them using the **FUNCTION MODE** function.

Input

11 FUNCTION MODE SET "Range1" ; Activate the machine manufacturer setting

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Machining mode (MODE)

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION MODE	Syntax initiator for the machining mode
MILL or SET	Select the machining mode or machine manufacturer setting
Name or Parameter	Name of a kinematic model or machine-manufacturer setting Text or string parameter Selection by means of a selection window Optional syntax element

Note

In the optional machine parameter **CfgModeSelect** (no. 132200), the machine manufacturer defines the settings for the **FUNCTION MODE SET** function. If the machine manufacturer does not define the machine parameter, then **FUNCTION MODE SET** is not available.

6

Workpiece blank

6.1 Defining a workpiece blank with BLK FORM

Application

You use the **BLK FORM** function to define a workpiece blank for graphic simulation of the NC program.

Related topics

- Representation of the workpiece blank in the **Simulation** workspace
Further information: "The Simulation workspace", Page 527

Description of function

You define the blank relative to the workpiece preset.

Further information: "Presets in the machine", Page 108

When you create a new NC program, the control automatically opens the **Insert NC function** window for workpiece blank definition.

Further information: "Creating a new NC program", Page 85

The control offers the following workpiece blank definitions:

Icon	Meaning	Further information
	BLK FORM QUAD Cuboid workpiece blank	Page 153
	BLK FORM CYLINDER Cylindrical workpiece blank	Page 154
	BLK FORM FILE STL file as workpiece blank and finished part	Page 155

Notes



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

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

- There are various ways to select files or subprograms:
 - Enter the file path
 - Enter the number or name of the subprogram
 - Select the file or subprogram by means of a selection window
 - Define the file path or name of the subprogram in a string parameter
 - Define the number of the subprogram in a numerical parameter

If the called file is located in the same directory as the calling NC program, it might be sufficient to enter just the file name.
- To make the control represent the workpiece blank in the simulation, the workpiece blank must have minimum dimensions. The minimum dimensions are 0.1 mm or 0.004 inches in all axes and for the radius.
- The control displays the workpiece blank in the simulation only after having processed the entire workpiece blank definition.

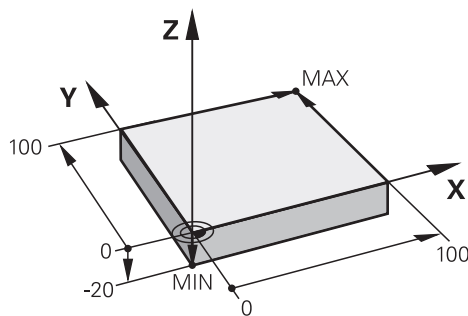
- If you have closed the **Insert NC function** window or want to add a workpiece blank definition after having created an NC program, a workpiece blank can be defined at any time in the **Insert NC function** window.
- The **Advanced checks** function in the simulation uses the information from the workpiece blank definition for workpiece monitoring. Even if several workpieces are clamped in the machine, the control can monitor only the active workpiece blank!
Further information: "Advanced checks in the simulation", Page 541
- In the **Simulation** workspace you can export the current workpiece view as an STL file. This function allows you to create missing 3D models, for example semi-finished parts if there are several machining steps.
Further information: "Exporting a simulated workpiece as STL file", Page 539

6.1.1 Cuboid workpiece blank with BLK FORM QUAD

Application

With **BLK FORM QUAD** you define a cuboid workpiece blank. You use a MIN point and a MAX point to define a spatial diagonal.

Description of function



Cuboid workpiece blank with MIN point and MAX point

The sides of the cuboid are parallel to the **X**, **Y** and **Z** axes.

You define the cuboid by entering a MIN point for the bottom front left corner and a MAX point for the top rear right corner.

You define the coordinates of the points in the **X**, **Y** and **Z** axes relative to the workpiece preset. If you define a positive value for the MAX point in the Z coordinate, the blank is given an oversize.

Further information: "Presets in the machine", Page 108

Input

1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	; Cuboid workpiece blank

To navigate to this function:

Insert NC function ► Special functions ► Program defaults ► Workpiece blank definition (BLK FORM) ► BLK FORM QUAD

The NC function includes the following syntax elements:

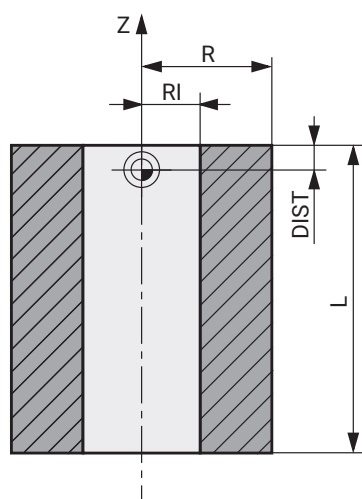
Syntax element	Meaning
BLK FORM	Syntax initiator for cuboid workpiece blank
0.1	Designation of the first NC block
Z	Tool axis Other possibilities might be available, depending on the machine.
X Y Z	Coordinate definition of the MIN point
0.2	Designation of the second NC block
X Y Z	Coordinate definition of the MAX point

6.1.2 Cylindrical workpiece blank with BLK FORM CYLINDER

Application

With **BLK FORM CYLINDER** you define a cylindrical workpiece blank. You can define a cylinder either as a solid piece or as a hollow pipe.

Description of function



Cylindrical blank

You can define the cylinder by entering the radius or diameter along with the height. The workpiece preset is in the cylinder center in the working plane. Optionally you can define an oversize and the inside radius or diameter of the blank.

Input

1 BLK FORM CYLINDER Z R50 L105 DIST ; Cylindrical blank
+5 RI10

To navigate to this function:

Insert NC function ► Special functions ► Program defaults ► Workpiece blank definition (BLK FORM) ► BLK FORM CYLINDER

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM CYLINDER	Syntax initiator for cylindrical workpiece blank
Z	Rotary axis Other possibilities might be available, depending on the machine.
R or D	Radius or diameter of the cylinder
L	Total height of the cylinder
DIST	Oversize of the cylinder relative to the workpiece preset Optional syntax element
RI or DI	Inside radius diameter of the core hole Optional syntax element

6.1.3 STL file as workpiece blank with BLK FORM FILE

Application

You can integrate 3D models in STL format as workpiece blank and optionally as finished part. This function is particularly convenient in combination with CAM programs, where the required 3D models are available in addition to the NC program.

Requirement

- Max. 20 000 triangles per STL file in ASCII format
- Max. 50 000 triangles per STL file in binary format

Description of function

The dimensions of the NC program come from the same source as the dimensions of the 3D model.

Input

```
1 BLK FORM FILE "TNC:\CAD\blank.stl" ; STL file as workpiece blank and finished
  TARGET "TNC:\CAD\finish.stl"      part
```

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Program defaults ► Workpiece blank definition (BLK FORM) ► BLK FORM FILE

The NC function includes the following syntax elements:

Syntax element	Meaning
BLK FORM FILE	Syntax initiator for an STL file as workpiece blank
File or Parameter	Path of the STL file
TARGET	STL file as finished part Optional syntax element
File or Parameter	Path of the STL file Fixed or variable path

Notes

- In the **Simulation** workspace you can export the current workpiece view as an STL file. This function allows you to create missing 3D models, for example semi-finished parts if there are several machining steps.
Further information: "Exporting a simulated workpiece as STL file", Page 539
- After integrating a workpiece blank and a finished part, you can compare the models in the simulation and easily identify residual material.
Further information: "Model comparison", Page 547
- The control loads binary-format STL files quicker than ASCII-format STL files.
- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.

7

Tools

7.1 Fundamentals

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

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

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

Further information: "Tool carrier reference point", Page 159

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

Further information: User's Manual for Setup and Program Run

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

Further information: User's Manual for Setup and Program Run

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

Further information: User's Manual for Setup and Program Run

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

Further information: "Using TOOL CALL to call a tool", Page 162

- 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: "Tool pre-selection using TOOL DEF", Page 169

- 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: User's Manual for Setup and Program Run

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

Further information: "Tool radius compensation", Page 344

7.2 Presets on the tool

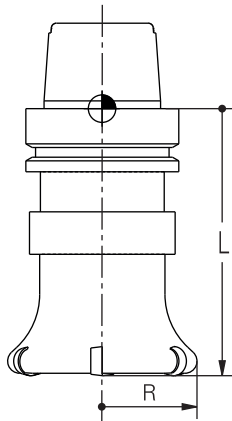
The control distinguishes the following presets on the tool for different calculations or applications.

Related topics

- Presets in the machine or on the workpiece

Further information: "Presets in the machine", Page 108

7.2.1 Tool carrier reference point



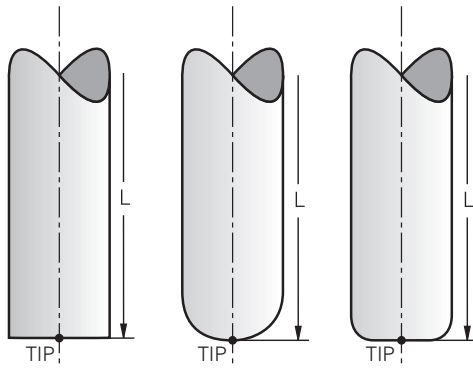
Milling cutter

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

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

Further information: User's Manual for Setup and Program Run

7.2.2 Tool tip TIP



Milling tools

Further information: "Tool coordinate system T-CS", Page 314

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

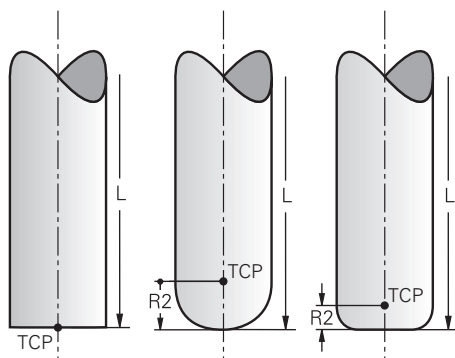
Further information: User's Manual for Setup and Program Run

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

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

Further information: "Tool location point (TLP, tool location point)", Page 161

7.2.3 Tool center point (TCP, tool center point)



Milling tools

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

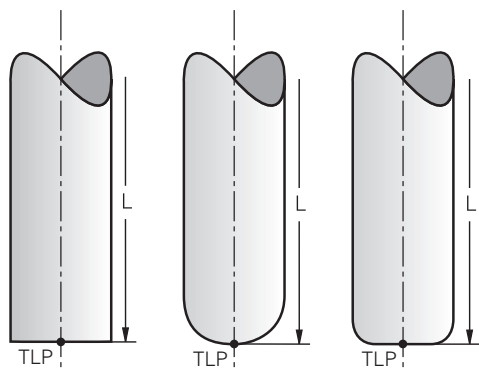
Making entries in the tool management relative to the tool carrier reference point defines the tool center point.

Further information: User's Manual for Setup and Program Run

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

Further information: "Tool location point (TLP, tool location point)", Page 161

7.2.4 Tool location point (TLP, tool location point)



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

7.3 Tool call

7.3.1 Using TOOL CALL to call a tool

Application

The **TOOL CALL** function calls a tool in the NC program. When the tool is in the tool magazine, the control inserts the tool into the spindle. When the tool is not in the magazine, you can insert it by hand.

Related topics

- Tool table **tool.t**
Further information: User's Manual for Setup and Program Run
- Pocket table **tool_p.tch**
Further information: User's Manual for Setup and Program Run

Requirement

- Tool defined
To call a tool, the tool must be defined in the tool management.
Further information: User's Manual for Setup and Program Run

Description of function

Upon calling a tool, the control reads the associated row from the tool management. The tool data is displayed on the **Tool** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run



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.

Further information: "Overview of miscellaneous functions", Page 383

Icons

The NC function **TOOL CALL** offers the following icons:

Icon	Meaning
	Open selection window for tools
	In the Tool management application, switch to the selected tool You can change the tool as needed.
	Open the Cutting data calculator Further information: "Cutting data calculator", Page 518

Input

**11 TOOL CALL 4 .1 Z S10000 F750 DL
+0.2 DR+0.2 DR2:+0.2** ; Call the tool

To navigate to this function:

Insert NC function ► All functions ► Tools ► TOOL CALL

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL CALL	Syntax initiator for a tool call
Number, Name or Parameter	Number or name of the tool Number, text, or variable
 Only the tool definition as a number is unique because the tool names of several tools may be identical!	
	Syntax element depending on technology or application Selection by means of a selection window Further information: "Technology-dependent differences when calling tools", Page 164
.1	Step index of the tool Optional syntax element Further information: User's Manual for Setup and Program Run
Z	Tool axis By default, tool axis Z . Other possibilities might be available, depending on the machine. Syntax element depending on technology or application Further information: "Technology-dependent differences when calling tools", Page 164
S or S(VC =)	Spindle speed or cutting speed Optional syntax element Selection by means of a selection window Further information: "Spindle speed S", Page 166
F, FZ or FU	Feed rate Alternative feed specifications: feed per tooth or feed per revolution Optional syntax element Selection by means of a selection window Further information: "Feed rate F", Page 167
DL	Delta value of tool length Optional syntax element Further information: "Tool compensation for tool length and tool radius", Page 340

Syntax element	Meaning
DR	Delta value of the tool radius Optional syntax element Further information: "Tool compensation for tool length and tool radius", Page 340
DR2	Delta value of the tool radius 2 Optional syntax element Further information: "Tool compensation for tool length and tool radius", Page 340

Technology-dependent differences when calling tools

Milling cutter tool call

The following tool data of a milling cutter can be defined:

- Number or name of the tool
- Step index of the tool
- Tool axis
- Spindle speed
- Feed rate
- DL
- DR
- DR2

Calling a milling cutter requires the number or the name of the tool, the tool axis and the spindle speed.

Further information: User's Manual for Setup and Program Run

Tool call for a workpiece touch probe (#17 / #1-05-1)

The following parameters of a workpiece touch probe can be defined:

- Number or name of the tool
- Step index of the tool
- Tool axis

Calling a workpiece touch probe requires the number or the name of the tool and the tool axis!

Further information: User's Manual for Setup and Program Run

Updating parameters

A **TOOL CALL** allows updating the parameters of the active tool even without tool change (e.g., change the cutting data or delta values). The parameters that can be modified depend on the technology.

In the cases below, the control updates the parameters of only the active tool:

- Without tool number or tool name and without tool axis
- Without tool number or tool name and with the same tool axis as in the previous tool call



When a tool number or a tool name or a changed tool axis is programmed in tool call, the control runs a tool change macro.

Notes



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

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

- The machine manufacturer uses the machine parameter **allowToolDefCall** (no. 118705) to specify whether a tool can be defined by its name, its number or both in the **TOOL CALL** and **TOOL DEF** functions.

Further information: "Tool pre-selection using TOOL DEF", Page 169

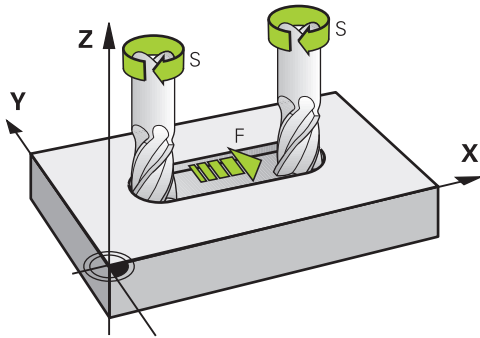
- The machine manufacturer uses the optional machine parameter **progToolCallIDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

Further information: "Tool compensation for tool length and tool radius", Page 340

7.3.2 Cutting data

Application

The cutting data consist of spindle speed **S** or alternatively constant cutting speed **VC** and feed rate **F**.



Description of function

Spindle speed S

The spindle speed **S** can be defined in the following ways:

- Tool call with **TOOL CALL**

Further information: "Using TOOL CALL to call a tool", Page 162

- **S** button in the **Manual operation** application

Further information: User's Manual for Setup and Program Run

The spindle speed **S** is defined as spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

For every tool with the **NMAX** parameter of the tool management, you can define a maximum permitted rotational speed. It is not possible to exceed this value with higher programmed values or by means of a potentiometer.

Effect

The spindle speed or the cutting speed is active until a new spindle speed or cutting speed is defined in a **TOOL CALL** NC block.

Potentiometers

The speed potentiometer allows varying the spindle speed between 0% and 150% while the program is running. The speed potentiometer setting is active only for machines with infinitely variable spindle drive. The maximum spindle speed depends on the machine.

Further information: "Potentiometers", Page 71

Status displays

The control displays the current spindle speed in the following workspaces:

- The **Positions** workspace
- The **POS** tab of the **Status** workspace

Feed rate F

The feed rate **F** can be defined in the following ways:

- Tool call with **TOOL CALL**

Further information: "Using TOOL CALL to call a tool", Page 162

- Positioning block

Further information: "Traverse movements", Page 171

- **F** button in the **Manual operation** application

Further information: User's Manual for Setup and Program Run

The feed rate for linear axes is defined in millimeters per minute (mm/min).

The feed rate for rotary axes is defined in degrees per minute (°/min).

The feed rate can be defined with an accuracy of three decimal places.

Alternatively, the feed rate can be defined in the NC program or in a tool call in the following units:

- Feed rate per tooth **FZ** in mm/tooth

FZ defines the path in millimeters that the tool covers per tooth.



When using **FZ**, the number of teeth must be defined in the **CUT** column of the tool management.

Further information: User's Manual for Setup and Program Run

- Feed rate per revolution **FU** in mm/rev

FU defines the path in millimeters that the tool covers per spindle revolution.

The feed rate defined in a **TOOL CALL** can be called up within the NC program, using **F AUTO**.

Further information: "F AUTO", Page 167

The feed rate defined in the NC program is active up to the NC block in which a new feed rate is programmed.

F MAX

If you define **F MAX**, the control moves at rapid traverse. **F MAX** is non-modal, i.e., it is active only in the block where it is called. Starting with the subsequent NC block, the last previously defined feed rate is active again. The maximum feed rate depends on the machine and may depend on the axis.

Further information: User's Manual for Setup and Program Run

F AUTO

If you defined a feed rate in a **TOOL CALL** block, this feed rate can be used in the next positioning blocks, using **F AUTO**.

F button in the Manual operation application

- If you enter $F=0$, then the feed rate that the machine manufacturer has defined as minimum feed rate is active
- If the feed rate you entered exceeds the maximum value that has been defined by the machine manufacturer, then the value defined by the machine manufacturer is active

Further information: User's Manual for Setup and Program Run

Potentiometer

The feed-rate potentiometer allows varying the feed rate between 0% and 150% while the program is running. The setting of the feed-rate potentiometer is active only for the programmed feed rate. As long as the programmed feed rate has not yet been reached, the feed-rate potentiometer has no effect.

Further information: "Potentiometers", Page 71

Status displays

The control displays the current feed rate in mm/min in the following workspaces:

- The **Positions** workspace
- The **POS** tab of the **Status** workspace



In the **Manual operation** application, the control displays the feed rate with decimal places on the **POS** tab. The control displays the feed rate with a total of six decimal places.

- The control displays the contouring feed rate as follows:

Further information: User's Manual for Setup and Program Run

Notes

- In inch programs, the feed rate must be defined in 1/10 inch/min.
- Make sure to program rapid traverse movements exclusively with the **FMAX** NC function instead of entering extremely high numerical values. This is the only way to ensure that rapid traverse is active on a block-by-block basis and that you can control rapid traverse independently of the machining feed rate.
- When positioning an axis, the control checks whether the defined speed has been reached. The control does not check the speed in positioning blocks where **FMAX** is the feed rate.

7.3.3 Tool pre-selection using TOOL DEF

Application

Using **TOOL DEF**, the control prepares a tool in the magazine, thus reducing the tool change time.



Refer to your machine manual.

The preselection of tools with **TOOL DEF** can vary depending on the individual machine tool.

Description of function

If your machine is equipped with a chaotic tool changer system and a double gripper, you can perform tool pre-selection. To do this, program the **TOOL DEF** function after a **TOOL CALL** data record and select the tool to be used next in the NC program. The control prepares the tool while the program is running.

Input

11 TOOL DEF 2 .1

; Tool pre-selection

To navigate to this function:

Insert NC function ► All functions ► Tools ► TOOL DEF

The NC function includes the following syntax elements:

Syntax element	Meaning
TOOL DEF	Syntax initiator for tool pre-selection
Number, Name or Parameter	Tool definition Number, text, or variable Selection by means of a selection window
 Only the tool definition as a number is unique because the tool names of several tools may be identical!	

.1

Step index of the tool

Optional syntax element

Further information: User's Manual for Setup and Program Run

Application example

11 TOOL CALL 5 Z S2000	; Call the tool
12 TOOL DEF 7	; Pre-select the next tool
* - ...	
21 TOOL CALL 7	; Call the pre-selected tool

8

**Traverse
movements**

8.1 Fundamentals of coordinate definitions

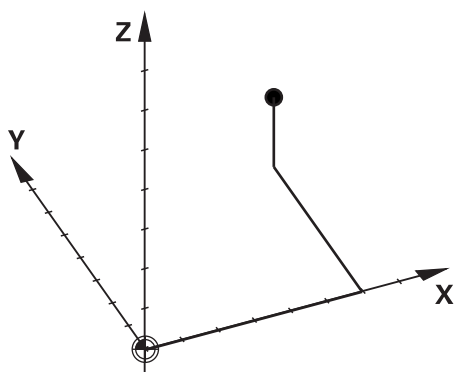
You program a workpiece by defining the target coordinates.

Depending on the dimensioning used in the technical drawing, you use Cartesian coordinates with absolute or incremental values.

8.1.1 Cartesian coordinates

Application

A Cartesian coordinate system consists of two or three axes that are all mutually perpendicular. Cartesian coordinates are relative to the datum (origin) of the coordinate system, which is at the intersection of the axes.



With Cartesian coordinates you can uniquely specify a point in space by defining the three axis values.

Description of function

In the NC program you define the values in the **X**, **Y** and **Z** linear axes.

11 X+60 RL F200
12 Y+50
13 Z+20

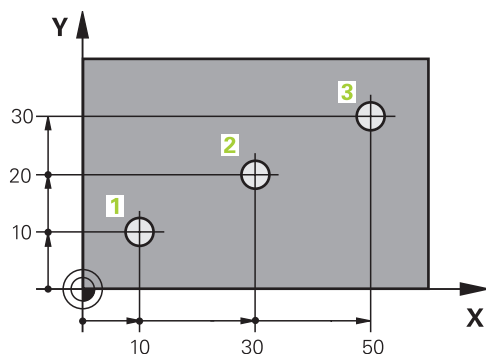
8.1.2 Absolute input

Application

Absolute input always references an origin. For Cartesian coordinates, the origin is the datum.

Description of function

Absolute values define the target point for positioning.



11 X+10 R0 F200 M3	; Position at point 1
12 Y+10 R0	; Position at point 1
13 X+30 R0	; Position at point 2
14 Y+20 R0	; Position at point 2
15 X+50 R0	; Position at point 3
16 Y+30 R0	; Position at point 3

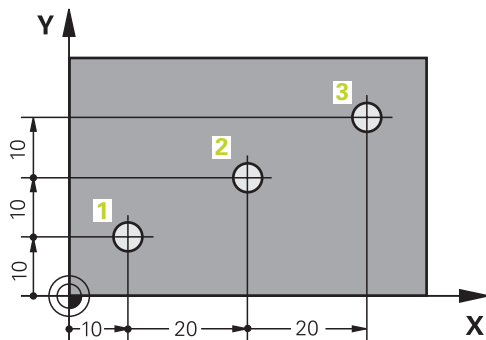
8.1.3 Incremental entries

Application

Incremental input always references the previously programmed coordinates. For Cartesian coordinates these are the values in the **X**, **Y** and **Z** axes.

Description of function

Incremental entries define the value by which the control positions. The previously programmed coordinates serve as the respective datum of the coordinate system. You define incremental coordinates with an **I** before each axis designation.



11 X+10 R0 F200 M3	; Position to point 1 absolutely
12 Y+10 R0	; Position to point 1 absolutely
13 IX+20 R0	; Position to point 2 incrementally
14 IY+10 R0	; Position to point 2 incrementally
15 IX+20 R0	; Position to point 3 incrementally
16 IY+10 R0	; Position to point 3 incrementally

8.2 Fundamentals of traverse movements

Application

When creating an NC program, you can use the orange axis keys to program the individual end points of the contour elements.

The control then uses the coordinate entries, the tool data, and the radius compensation to calculate the traverse path.

Description of function

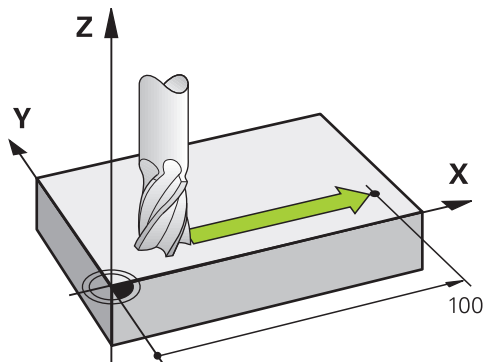
Inserting a traverse movement

The orange axis keys initiate the dialog. The control inserts the NC block in the NC program and prompts you for each piece of necessary information.



Depending on the design of the machine tool, either the tool moves or the machine table moves. When programming a path function, always assume that the tool is in motion!

Motion in one axis



If the NC block contains one coordinate, the control moves the tool paraxially to the machine axis.

Example

6 X+100 R+ F200 M3

The tool retains the Y and Z coordinates and moves to the position **X+100**.

Tool radius compensation

Tool radius compensation is defined in the NC block of the first contour element.

Further information: "Tool radius compensation", Page 344

Pre-positioning

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect pre-positioning can also lead to contour damage. There is danger of collision during the approach movement!

- ▶ Program a suitable pre-position
- ▶ Check the sequence and contour with the aid of the graphic simulation

9

**Programming
techniques**

9.1 Subprograms and program section repeats with the label LBL

Application

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as necessary. Use subprograms to insert contours or complete machining steps after the end of the program and call them in the NC program. Program section repeats repeat single or several NC blocks during the NC program. Subprograms and program section repeats can also be combined. Subprograms and program section repeats are programmed with the NC function **LBL**.

Related topics

- Executing NC programs within another NC program
Further information: "Calling an NC program with CALL PGM", Page 194
- Jumps with conditions as if-then decisions.
Further information: "The Jump commands folder", Page 410

Description of function

The label **LBL** is used for defining the machining steps for subprograms and program section repeats.

The control offers the following keys and icons in connection with labels:

Key or icon	Function
	Create LBL
	Call LBL : Jump to the label in the NC program
	In case of LBL number: Enter the next free number automatically

Defining a label with LBL SET

The **LBL SET** function defines a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by its number or name. If a number or a name exists twice in an NC program, the control shows a warning before the NC block.

LBL 0 marks the end of a subprogram. This number is the only one which may exist more than once in the NC program.

Input

11 LBL "Reset"	; Subprogram for resetting a coordinate transformation
12 TRANS DATUM RESET	
13 LBL 0	

To navigate to this function:

Insert NC function ► All functions ► Label ► LBL SET

The NC function includes the following syntax elements:

Syntax element	Meaning
LBL	Syntax initiator for a label
Number or Name	Number or name of the label Number, text, or variable Input: 0...65535 or text width 32 Use an icon to enter the next free number automatically. Further information: "Description of function", Page 178

Calling a label with CALL LBL

The **CALL LBL** function calls a label in the NC program.

When the control reads **CALL LBL**, it jumps to the defined label and continues executing the NC program from this NC block. When the control reads **LBL 0**, it jumps back to the next NC block after **CALL LBL**.

In case of program section repeats, you can optionally define that the control executes that jump several times.

Input

11 CALL LBL 1 REP2	; Call label 1 twice

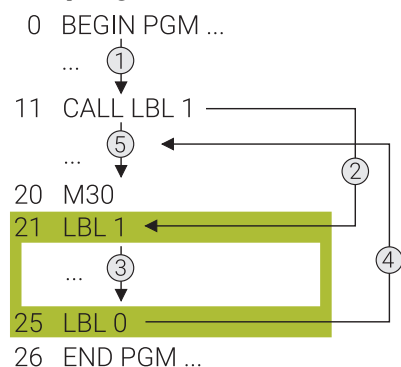
To navigate to this function:

Insert NC function ► All functions ► Label ► CALL LBL

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL LBL	Syntax initiator for calling a label
Number, Name or Parameter	Number or name of the label Number, text, or variable Input: 1...65535 or text width 32 or 0...1999 The label can be selected from a selection menu that displays all labels available in the NC program.
REP	Number of repetitions until the control executes the next NC block Optional syntax element

Subprograms



A subprogram allows calling parts of an NC program any number of times at different points of the NC program (e.g., machining positions or a contour).

A subprogram starts with a **LBL** label and ends with **LBL 0**. **CALL LBL** calls the subprogram from any point in the NC program. In this process, repetitions must not be defined with **REP**.

The control executes the NC program as follows:

- 1 The control executes the NC program up to the **CALL LBL** function.
- 2 The control jumps to the beginning of the defined subprogram **LBL**.
- 3 The control executes the subprogram up to the subprogram end **LBL 0**.
- 4 After that, the control jumps to the next NC block after **CALL LBL** and continues executing the NC program.

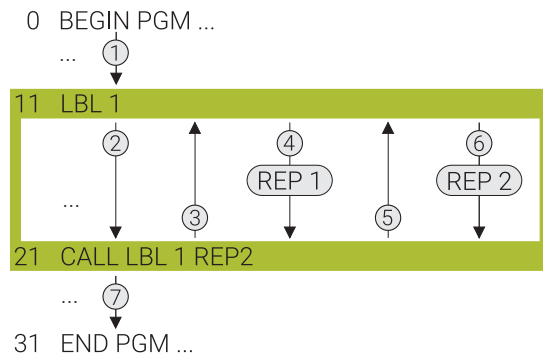
The following conditions apply to subprograms:

- A subprogram cannot call itself
- **CALL LBL 0** is not permitted (Label 0 is only used to mark the end of a subprogram).
- Write subprograms after the NC block with M2 or M30
If subprograms are located in the NC program before the NC block with M2 or M30, they will be executed at least once even if they are not called

The control displays information about the active subprogram on the **LBL** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Program-section repeats



A program section repeat allows repeating a part of an NC program any number of times (e.g., contour machining with incremental infeed).

A program section repeat starts with a **LBL** label and ends after the last programmed repetition **REP** of the label call **CALL LBL**.

The control executes the NC program as follows:

- 1 The control executes the NC program up to the **CALL LBL** function.
In this process, the control already executes the program section once because the program section to be repeated is positioned ahead of the **CALL LBL** function.
- 2 The control jumps to the beginning of the program section repeat **LBL**.
- 3 The control repeats the program section as many times as programmed under **REP**.
- 4 After that, the control continues executing the NC program.

The following conditions apply to program section repeats:

- Program the program section repeat before the end of the program with **M30** or **M2**.
- No **LBL 0** can be defined with a program section repeat.
- The total number of times the program section is executed is always one more than the programmed number of repeats, because the first repeat starts after the first machining process.

The control displays information about the active program section repeat on the **LBL** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Notes

- The control displays the NC function **LBL SET** in the structure by default.
Further information: "The Structure column in the Program workspace", Page 501
- You can repeat a program section up to 65 534 times in succession
- The following characters are allowed in the name of a label: # \$ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ a b c d e f g h i j k l m n o p q r s t u v w x y z A B C D E F G H I J K L M N O P Q R S T U V W X Y Z
- The following characters are not allowed in the name of a label: <blank> ! " ' () * + ; < = > ? [/] ^ ` { | } ~

9.2 Nesting with LBL

Application

You can combine program section repeats and subprograms with each other and also with other programming techniques, such as calling a separate NC program. If you want to return to the origin after each call, use only one nesting level. If you program another call before returning to the origin, you will get one nesting level lower.

Related topics

- Subprograms
Further information: "Subprograms", Page 180
- Program section repeats
Further information: "Program-section repeats", Page 181
- Calling a separate NC program
Further information: "Selection functions", Page 194

Description of function

Please note the maximum nesting depth:

- Maximum nesting depth for subprogram calls: 19
- Maximum nesting depth for calls of external NC programs: 19 where a **CYCL CALL** has the same effect as calling an external program
- Program-section repeats can be nested as often as desired

9.2.1 Examples

Subprogram call within a subprogram

0 BEGIN PGM UPGMS MM	
* - ...	
11 CALL LBL "UP1"	; Call subprogram "UP1"
* - ...	
21 Z+100 R0 FMAX M30	; Last program block of main program with M30
22 LBL "UP1"	; Start of subprogram "UP1"
* - ...	
31 CALL LBL 2	; Call subprogram 2
* - ...	
41 LBL 0	; End of subprogram "UP1"
42 LBL 2	; Start of subprogram 2
* - ...	
51 LBL 0	; End of subprogram 2
52 END PGM UPGMS MM	

The control executes the NC program as follows:

- 1 NC program UPGMS is executed up to NC block 11.
- 2 Subprogram UP1 is called and executed up to NC block 31.
- 3 Subprogram 2 is called and executed up to NC block 51. End of subprogram 2 and return jump to the subprogram from which it was called.
- 4 Subprogram UP1 is executed from NC block 32 up to NC block 41. End of subprogram UP1 and return jump to NC program UPGMS.
- 5 NC program UPGMS is executed from NC block 12 up to NC block 21. Program end with return jump to NC block 0.

Program-section repeat within a program section repeat

0 BEGIN PGM REPS MM	
* - ...	
11 LBL 1	; Start of program section 1
* - ...	
21 LBL 2	; Start of program section 2
* - ...	
31 CALL LBL 2 REP 2	; Call program section 2 and repeat twice
* - ...	
41 CALL LBL 1 REP 1	; Call program section 1 including program section 2 and repeat once
* - ...	
51 END PGM REPS MM	

The control executes the NC program as follows:

- 1 NC program REPS is executed up to NC block 31.
- 2 The program section between NC block 31 and NC block 21 is repeated twice, meaning that it is executed three times in total.
- 3 NC program REPS is executed from NC block 32 up to NC block 41.
- 4 The program section between NC block 41 and NC block 11 is repeated once, meaning that it is executed twice in total (including the program section repeat between NC block 21 and NC block 31).
- 5 NC program REPS is executed from NC block 42 up to NC block 51. Program end with return jump to NC block 0.

Subprogram call within a program section repeat

0 BEGIN PGM UPGREP MM	
* - ...	
11 LBL 1	; Start of program section 1
12 CALL LBL 2	; Call subprogram 2
13 CALL LBL 1 REP 2	; Call program section 1 and repeat twice
* - ...	
21 Z+100 R0 FMAX M30	; Last NC block of main program with M30
22 LBL 2	; Start of subprogram 2
* - ...	
31 LBL 0	; End of subprogram 2
32 END PGM UPGREP MM	

The control executes the NC program as follows:

- 1 NC program UPGREP is executed up to NC block 12.
- 2 Subprogram 2 is called and executed up to NC block 31.
- 3 The program section between NC block 13 and NC block 11 (including subprogram 2) is repeated twice, meaning that it is executed three times in total.
- 4 NC program UPGREP is executed from NC block 14 up to NC block 21. Program end with return jump to NC block 0.

9.3 Control structures

9.3.1 Fundamentals

The control provides the NC functions for programming the control structures.

Using control structures, you can program the NC program more clearly and with a better structure. The control indents the NC blocks within the control structures. Thus you can see right away where a control structure starts and ends.

The control provides the following NC functions:

NC function	Meaning
Case analyses: ■ IF ■ ELSE IF ■ ELSE	Case analyses allow you to define conditions. If a condition is fulfilled, the control executes the NC blocks that belong to that condition. If a condition is not fulfilled, the control skips the NC blocks. Further information: "Case analyses", Page 185
Program loops: ■ FOR ■ WHILE	Program loops enable you to repeat NC blocks several times. Further information: "Program loops", Page 188
Advanced control of program loops: ■ BREAK ■ CONTINUE	The NC functions for advanced control of program loops allow you to skip the current loop repetition or cancel the program loop. You can program these NC functions within FOR and WHILE loops. Further information: "Advanced controlling of program loops", Page 192

9.3.2 Case analyses

Application

You can use the **IF**, **ELSE IF** and **ELSE** case analyses to control the actions of the NC program. You use conditions to define whether the control executes specific program sections or skips them.

Related topics

- Jump commands with **FN**
Further information: "The Jump commands folder", Page 410

Description of function

You use the **IF** and **END IF** NC functions to define the header and footer of a case analysis. In the header you define a condition.

If the condition is fulfilled, the control executes all NC blocks located between the header and footer.

The control indents the NC blocks between the header and footer.

Within the case analysis you can use **ELSE IF** to define further conditions, and **ELSE** to define an alternative program section. The control executes the alternative program section only if none of the previous conditions are fulfilled.

You can use numerical values when programming the following conditions:

Condition	Meaning
a == b	a equal to b
a != b	a not equal to b
a < b	a less than b
a > b	a greater than b
a <= b	a less than or equal to b
a >= b	a greater than or equal to b

You can use alpha-numerical values when programming the following conditions:

Condition	Meaning
a == b	a equal to b
a != b	a not equal to b
a IN b	a contained in b The control checks whether the content to the left of IN is contained in the same sequence in the content to the right. The control also takes capitalization into account.

Input

IF

11 IF Q50 < Q60	; Header of the case analysis, with a condition
* - ...	; If this condition is fulfilled, the indented content is executed
21 END IF	; Footer of the case analysis

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Sorting tests** ► **IF and END IF**

The NC function includes the following syntax elements:

Syntax element	Meaning
IF	Syntax initiator for case analysis with IF
Q50 < Q60	Condition Number, text, or variable



You can use the **Insert NC function** window to insert a combination of header and footer or both lines.

ELSE IF

11 IF Q50 < Q60	; Header of the case analysis, with a condition
* - ...	
21 ELSE IF Q50 > Q60	; Header with additional condition
* - ...	; If this condition is fulfilled, the indented content is executed
31 END IF	; Footer of the case analysis

To navigate to this function:

Insert NC function ► All functions ► FN ► Sorting tests ► ELSE IF

The NC function includes the following syntax elements:

Syntax element	Meaning
ELSE IF	Syntax initiator for another condition with ELSE IF
Q50 > Q60	Condition Number, text, or variable

ELSE

11 IF Q50 < Q60	; Header of the case analysis, with a condition
* - ...	
21 ELSE	; Header for alternative execution
* - ...	; If the previous condition is not fulfilled, the indented content is executed
31 END IF	; Footer of the case analysis

To navigate to this function:

Insert NC function ► All functions ► FN ► Sorting tests ► ELSE

The NC function includes the following syntax elements:

Syntax element	Meaning
ELSE	Syntax initiator for alternative execution with ELSE

Notes

- If you program numerical parameters in the condition, then you must define the variables before the case analysis.
- The control always processes only one case per level. If, for example, the **IF** condition is fulfilled, the control skips all other cases on the same level.
- If multiple conditions should need to be fulfilled for a program section, then you must nest the case analysis. You can use nesting, for example, to program that a value should be within a numerical range.
- If you program jump commands (e.g., using **FN 9**) within case analyses, the control will display an error message.

Further information: "The Jump commands folder", Page 410

- Please note that the control considers digits from the tenth decimal place onward to be identical. This means, for example, that calculation results differing only after the ninth decimal place will be considered to have the same value.

9.3.3 Program loops

FOR loop

Application

You can use a **FOR** loop to program simple program-section repeats.

Related topics

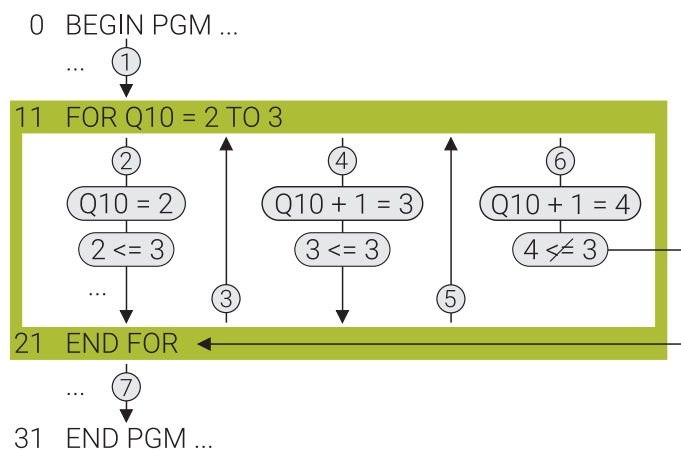
- Subprograms and program-section repeats

Further information: "Subprograms and program section repeats with the label LBL", Page 178

Description of function

A program loop consists of a header and footer and all the defined NC blocks between them. You use the **FOR** and **END FOR** NC functions to define the header and footer.

The control indents the NC blocks between the header and footer.



The control executes the NC program as follows:

- 1 The control reads the header with the counting variable, the target value, and the increment, if given.
- 2 The control writes the starting value to the counting variable.
Since the value of the counting variable is less than or equal to the target value, the control executes the loop content.
- 3 The control reads the footer and returns to the header.
- 4 The control changes the value of the counting variable by the increment.
Since the value of the counting variable is less than or equal to the target value, the control executes the loop content.
- 5 The control reads the footer and returns to the header.
- 6 The control changes the value of the counting variable by the increment.
Since the value of the counting variable is greater than the target value, the control skips the loop content and jumps directly to the footer.
- 7 The control ends the program loop and continues the NC program.

Input

11 FOR Q50 = 4 TO 10 STEP 2	; Header of the FOR loop
* - ...	; Content is executed until the value of Q50 is greater than 10
21 END FOR	; Footer of the FOR loop

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Program section repeat** ► **FOR header and footer**

The NC function includes the following syntax elements:

Syntax element	Meaning
FOR	Syntax initiator of a FOR loop
Q50	Counting variable With each repetition of the program loop the control checks the value of this variable. If the value is less than the target value, the control changes this value by the increment.
= 4	Start value At the start of the program loop, the control assigns this value to the counting variable once. Only integers smaller than the target value are possible. Number or numerical parameter
TO 10	Target value The control repeats the program loop until the value of the counting variable is greater than the target value. Only integers are possible Number or numerical parameter
STEP 2	Increment Only integers are possible If you don't define an increment, the control uses the value 1 . Number or numerical parameter Optional syntax element



You can use the **Insert NC function** window to insert a combination of header and footer or both lines.

Notes

- If you program numerical parameters in the header, then you must define the variables before the program loop.
- Do not assign a new value to the counting variable within the program loop, since that could lead to an infinite loop.
- If you program jump commands (e.g., using **FN 9**) within program loops, the control will display an error message.

Further information: "The Jump commands folder", Page 410

WHILE loop

Application

You can use a **WHILE** loop to program simple program-section repeats with various conditions. The conditions can contain numerical or alpha-numerical values.

Related topics

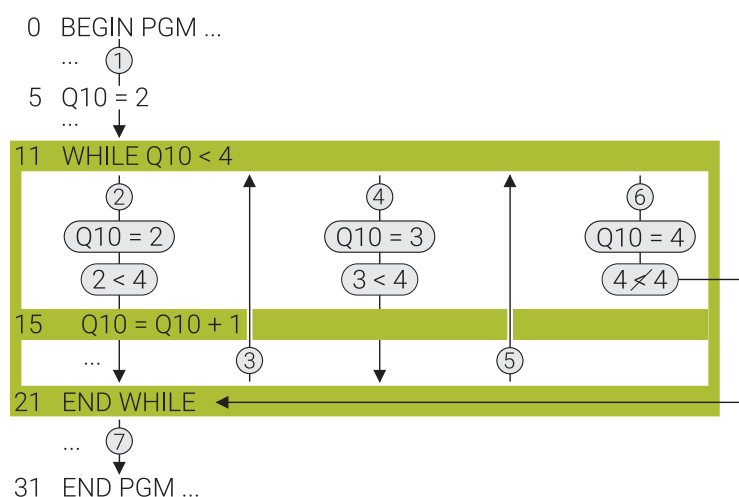
- Subprograms and program-section repeats

Further information: "Subprograms and program section repeats with the label LBL", Page 178

Description of function

A program loop consists of a header and footer and all the defined NC blocks between them. You use the **WHILE** and **END WHILE** NC functions to define the header and footer.

The control indents the NC blocks between the header and footer.



The control executes the NC program as follows:

- 1 The control assigns the value **2** to the variable **Q10** and then reads the header with the condition.
- 2 The control checks whether the condition is fulfilled.
Since the condition is fulfilled, the control executes the loop content.
Within the program loop the control increments the value of the variable **Q10**.
- 3 The control reads the footer and returns to the header.
- 4 The control checks whether the condition is fulfilled.
Since the condition is fulfilled, the control executes the loop content.
Within the program loop the control increments the value of the variable **Q10**.
- 5 The control reads the footer and returns to the header.
- 6 The control checks whether the condition is fulfilled.
Since the condition is no longer fulfilled, the control skips the loop content and jumps directly to the footer.
- 7 The control ends the program loop and continues the NC program.

You can use numerical values when programming the following conditions:

Condition	Meaning
a == b	a equal to b
a != b	a not equal to b
a < b	a less than b
a > b	a greater than b
a <= b	a less than or equal to b
a >= b	a greater than or equal to b

You can use alpha-numerical values when programming the following conditions:

Condition	Meaning
a == b	a equal to b
a != b	a not equal to b
a IN b	a contained in b The control checks whether the content to the left of IN is contained in the same sequence in the content to the right. The control also takes capitalization into account.

Input

11 Q50 = +5	; Define Q50 with the value 5
12 Q60 = +10	; Define Q60 with the value 10
13 WHILE Q50 <= Q60	; Header of the WHILE loop
14 Q50 = Q50 + +1	; Increment Q50 by the value 1
* - ...	; Content will be executed until the value in Q50 is greater than the value in Q60
21 END WHILE	; Footer of the WHILE loop

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Program section repeat** ► **WHILE header and footer**

The NC function includes the following syntax elements:

Syntax element	Meaning
WHILE	Syntax initiator of a WHILE loop
Q50 <= Q60	Condition Number, text, or variable



You can use the **Insert NC function** window to insert a combination of header and footer or both lines.

Notes

- If you program numerical parameters in the header, then you must define the variables before the program loop.
- You can use a **WHILE** loop to achieve the functionality of a **FOR** loop. To do so, define a calculation step within the program loop in order to increment or decrement a value.
- If you program jump commands (e.g., using **FN 9**) within program loops, the control will display an error message.
Further information: "The Jump commands folder", Page 410
- Please note that the control considers digits from the tenth decimal place onward to be identical. This means, for example, that calculation results differing only after the ninth decimal place will be considered to have the same value.

9.3.4 Advanced controlling of program loops

Aborting program loops with BREAK

Application

You can use the **BREAK** NC function to abort a program loop (e.g., if a certain machine or tool status occurs).

Description of function

If the control executes the **BREAK** NC function, it aborts the program loop. The control skips the NC blocks until the footer and then executes the rest of the NC program.

Input

11 FOR Q50 = 4 TO 10	; Header of the FOR loop
* - ...	
15 IF Q182 == +1	; Workpiece status: scrap after probing
16 BREAK	; If Q182 contains the value 1 , the control will execute this NC block contains and then abort the program loop
17 END IF	
* - ...	
21 END FOR	; Footer of the FOR loop

To navigate to this function:

Insert NC function ► All functions ► FN ► Program section repeat ► BREAK

The NC function includes the following syntax elements:

Syntax element	Meaning
BREAK	Syntax initiator for aborting the program loop

Note

Program a **BREAK** only in combination with a case analysis. That way the control will abort the program loop only in the defined case.

Further information: "Case analyses", Page 185

Canceling program loops with CONTINUE

Application

You can use the **CONTINUE** NC function to skip the current repetition of a program loop. The control then continues with the subsequent program loop.

Description of function

If the control executes the **CONTINUE** NC function, it cancels the current loop repetition. The control skips the NC blocks up to the footer, ignoring the rest of the program loop. The control then executes the next loop repetition.

Input

11 FOR Q50 = 4 TO 10	; Header of the FOR loop
* - ...	
15 IF Q182 == +1	; Workpiece status: scrap after probing
16 CONTINUE	; If Q182 contains the value 1 , the control will execute this NC block contains and then cancel the current loop repetition
17 END IF	
* - ...	
21 END FOR	; Footer of the FOR loop

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Program section repeat** ► **CONTINUE**

The NC function includes the following syntax elements:

Syntax element	Meaning
CONTINUE	Syntax initiator for canceling a loop repetition

9.4 Selection functions

9.4.1 Overview of selection functions

The **Selection** folder of the **Insert NC function** window contains the following functions:

Icon	Meaning	Further information
	Call an NC program with CALL PGM	Page 194
	Select a datum table with SEL TABLE	Page 321
	Select a point table with SEL PATTERN	Page 207
	Select an NC program with SEL PGM	Page 196
	Call the last selected file with CALL SELECTED PGM	Page 196
	Select any NC program with SEL CYCLE as a machining cycle	Page 143
	Open the file with OPEN FILE	Page 370

9.4.2 Calling an NC program with CALL PGM

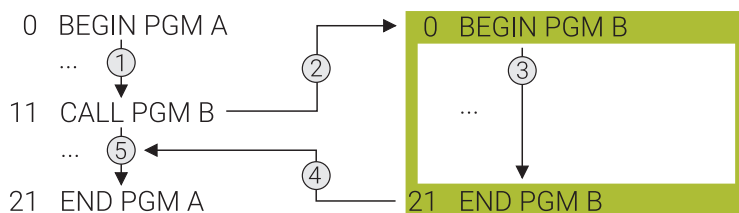
Application

With the **CALL PGM** NC function, you can call another, separate NC program from within an NC program. The control executes the called NC program at the point where you called it in the NC program. This allows a machining operation to be executed with various transformations, for example.

Related topics

- Program call with Cycle **12 PGM CALL**
Further information: "Cycle 12 PGM CALL ", Page 199
- Program call following selection
Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM ", Page 196

Description of function



The control executes the NC program as follows:

- 1 The control executes the calling NC program until you call another NC program with **CALL PGM**.
- 2 After that, the control executes the called NC program up to the last NC block.
- 3 The control then resumes the calling NC program, starting with the next NC block after **CALL PGM**.

The following conditions apply to program calls:

- The called NC program must not contain a **CALL PGM** call into the calling NC program. This creates an endless loop.
- The called NC program must not contain the miscellaneous function **M30** or **M2**. If you defined subprograms in the called NC program using labels, then you can replace **M30** or **M2** with an unconditional jump function. This keeps the control from executing a subprogram.

Further information: "Unconditional jump", Page 411

If the called NC program contains the miscellaneous functions, the control generates an error message.

- The called NC program must be complete. If the NC block **END PGM** is missing, the control outputs an error message.

Input

11 CALL PGM reset.h ; Call NC program

To navigate to this function:

Insert NC function ► **All functions** ► **Selection** ► **CALL PGM**

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL PGM	Syntax initiator for calling an NC program
File	Path of the called NC program Selection by means of a selection window

Notes

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. If you do not specifically rescind the coordinate transformations in the called NC program, these transformations will also take effect in the calling NC program. Danger of collision during machining!

- ▶ Reset used coordinate transformations in the same NC program
- ▶ Check the machining sequence using a graphic simulation if required

- The program call path including the name of the NC program may contain no more than 255 characters.
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- If you want to program variable program calls in conjunction with string parameters, use the **SEL PGM** NC function.
Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM", Page 196
- As a rule, variables are globally effective when called, such as with **CALL PGM**. Please note that changes to variables in the called NC program can also take effect on the calling NC program. If applicable, use QL or named parameters that take effect only in the active NC program.
- While the control is executing the calling NC program, editing of all called NC programs is disabled.

9.4.3 Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM

Application

The function **SEL PGM** allows selecting another separate NC program that you can call at a different position in the active NC program. The control executes the selected NC program at the position where you call it in the calling NC program using **CALL SELECTED PGM**.

Related topics

- Calling the NC program directly
Further information: "Calling an NC program with CALL PGM", Page 194

Description of function

The control executes the NC program as follows:

- 1 The control executes the NC program until another NC program is called with **CALL PGM**. When the control reads **SEL PGM**, it remembers the defined NC program.
- 2 When the control reads **CALL SELECTED PGM**, it calls the NC program previously selected at this point.
- 3 After that, the control executes the called NC program up to the last NC block.
- 4 Then the control continues executing the calling NC program with the next NC block after **CALL SELECTED PGM**.

The following conditions apply to program calls:

- The called NC program must not contain a **CALL PGM** call into the calling NC program. This creates an endless loop.
- The called NC program must not contain the miscellaneous function **M30** or **M2**. If you defined subprograms in the called NC program using labels, then you can replace **M30** or **M2** with an unconditional jump function. This keeps the control from executing a subprogram.

Further information: "Unconditional jump", Page 411

If the called NC program contains the miscellaneous functions, the control generates an error message.

- The called NC program must be complete. If the NC block **END PGM** is missing, the control outputs an error message.

Input

11 SEL PGM "reset.h"	; Select an NC program for calling
* - ...	
21 CALL SELECTED PGM	; Call the selected NC program

SEL PGM

To navigate to this function:

Insert NC function ► All functions ► Selection ► SEL PGM

The NC function includes the following syntax elements:

Syntax element	Meaning
SEL PGM	Syntax initiator for selecting an NC program to be called
Name or Parameter	Path of the NC program to be called Fixed or variable path Selection by means of a selection window

CALL SELECTED PGM

To navigate to this function:

Insert NC function ► All functions ► Selection ► CALL SELECTED PGM

The NC function includes the following syntax elements:

Syntax element	Meaning
CALL SELECTED PGM	Syntax for calling the selected NC program

Notes

- Within the **SEL PGM** NC function, the NC program can also be selected with QS parameters so that the program call can be variably controlled.
- If an NC program called by **CALL SELECTED PGM** is missing, the control interrupts the execution or simulation of the program with an error message. In order to avoid undesired interruptions during program run, you can use the **FN 18: SYSREAD (ID10 NR110 and NR111)** NC function to check all paths at program start.

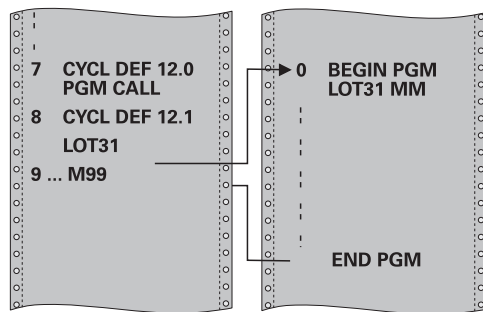
Further information: "Read system data with FN 18: SYSREAD", Page 420

- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- As a rule, variables are globally effective when called, such as with **CALL PGM**. Please note that changes to variables in the called NC program can also take effect on the calling NC program. If applicable, use QL or named parameters that take effect only in the active NC program.
- While the control is executing the calling NC program, editing of all called NC programs is disabled.

9.5 Cycle 12 PGM CALL

ISO programming
G39

Application



NC programs that you have created (such as special drilling cycles or geometrical modules) can be written as machining cycles. These NC programs can then be called like normal cycles.

Related topics

- Calling external NC programs
Further information: "Selection functions", Page 194

Note

- As a rule, Q parameters are globally effective when called with Cycle **12**. So please note that changes to Q parameters in the called NC program can also influence the calling NC program.

Notes on programming

- The NC program you are calling must be stored in the internal memory of your control.
- If the NC program you are defining to be a cycle is located in the same directory as the NC program you are calling it from, you need only enter the program name.
- If the NC program you are defining to be a cycle is not located in the same directory as the NC program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

9.5.1 Cycle parameters

Help graphic	Parameter
	Program name Enter the name of the NC program to be called and, if necessary, the path where it is located, Use the Select File Select in the action bar of the NC program to be called.

Call the NC program with:

- **CYCL CALL** (separate NC block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Declare NC program Stempel_stamp.h as a cycle and call it with M99

```
11 CYCL DEF 12.0 PGM CALL
```

```
12 CYCL DEF 12.1 PGM TNC:\nc_prog\demo\Stempel_stamp.h
```

```
13 X+20 FMAX
```

```
14 Y+50 FMAX M99
```

9.6 NC sequences for reuse

Application

You can save up to 2000 consecutive NC blocks as NC sequences and insert them during programming using the **Insert NC function** window. Unlike called NC programs, you can modify NC sequences after insertion without changing the actual sequence.

Related topics

- **Insert NC function** window
Further information: "Areas of the Insert NC function window", Page 133
- Mark and copy NC blocks with the context menu
Further information: "Context menu", Page 511
- Call NC programs unchanged
Further information: "Calling an NC program with CALL PGM", Page 194

Description of function

You can use NC sequences in the **Editor** operating mode and the **MDI** application.

The control saves the NC sequences as complete NC programs in the **TNC:\system\PGM-Templates** folder. You can also create subfolders in order to sort the NC sequences.

Here are the following possibilities for creating an NC sequence:

- Save marked NC blocks with the **Create NC sequence** button
Further information: "Context menu in the Program workspace", Page 514
- Create a new NC program in the **TNC:\system\PGM-Templates** folder
- Copy the already existing NC program to the **TNC:\system\PGM-Templates** folder



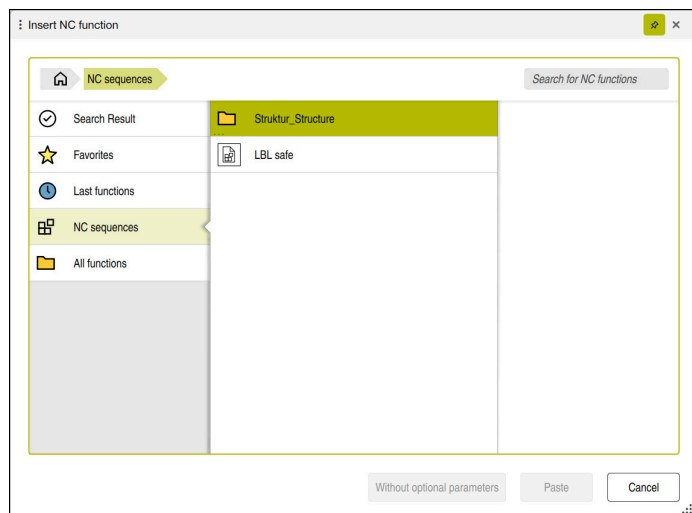
You must use the **Create NC sequence** button to create the first NC sequence; then the **TNC:\system\PGM-Templates** folder is created.

If you create an NC sequence with the **Create NC sequence** button, then the control opens the **Save NC sequence** window.

In the **Save NC sequence** window, you can enter the following information:

- Define the name of the NC sequence
- Select the storage location of the NC sequence
 If you created subfolders in the **TNC:\system\PGM-Templates** folder, the control will display a selection menu that contains all folders.

The control displays all folders and NC sequences alphabetically in the **Insert NC function** window under **NC sequences**. You can insert the desired NC sequence at the cursor position and customize it in the NC program.



Inserting NC sequences in the **Insert NC function** window

If you open an NC sequence as its own tab in the **Editor**, then you can permanently edit the contents of the NC sequence.

Adding a user-defined folder icon

You can save user-defined folder icons for the subfolders of the NC components.

To save a folder icon:

- ▶ Select the **Files** operating mode
- ▶ Open a subfolder in the **TNC:\system\PGM-Templates** folder
- ▶ Insert the desired folder icon with the name **foldericon.svg**
- ▶ The control displays the user-defined folder icon in the **NC sequences** area of the **Insert NC function** window.



You can add a different folder icon with the name **foldericon-dark.svg** for the Dark Mode.

Notes

NOTICE

Caution: Danger to the tool and workpiece!

If you insert an NC sequence with a different unit of measure despite the confirmation request, the control will not convert the values. Be aware that input values with a different unit of measure can be too small or too high, such as for feed rates. Incorrect input values can lead to increased tool wear, tool breakage, and lower surface quality.

- ▶ Always insert NC sequences only into NC programs with the same unit of measure.
- ▶ Check the input values and correct them if required

- Make sure to define an unambiguous name for each NC sequence within a folder. If you try to save an NC sequence under a name that has already been assigned, then the control opens the **Overwrite NC sequence** window. The control asks if you wish to overwrite the existing NC sequence.
- If you drag an NC sequence to the right in the **Insert NC function** window, the control will display the following file functions:
 - Edit
 - Rename
 - Delete
 - Activate or deactivate write protection
 - Open the path in the **Files** operating mode
 - Mark as favorite

You can also select some of these file functions from the context menu.

Further information: "Context menu in the Insert NC function window", Page 515

- Write-protected NC sequences cannot be renamed or deleted. It is possible to edit such an NC sequence, but you need to save it as a new file after editing. While write protection is active, the control displays a corresponding symbol next to the NC sequence.
- If you create a backup of the **TNC:** partition with the **NC/PLC Backup** function, then the backup also contains the NC sequences.

Further information: User's Manual for Setup and Program Run

- The input ranges for millimeter values in NC programs are greater than for inch values. The control will display an error message if the millimeter values of an NC sequence exceed the input ranges of an inch program.
- The control scales user-defined folder icons to max. 32x32 dp (device-independent pixels).

10

Point definitions

10.1 Point tables

Application

With a point table you can execute one or more cycles in sequence on an irregular point pattern.

Related topics

- Contents of a point table, hiding individual points

Further information: "Point table *.pnt", Page 609

Description of function

Coordinates in a point table

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting point coordinates of the respective cycle. The coordinates of the spindle axis correspond to the coordinate of the workpiece surface.

The control retracts the tool to the clearance height when traversing between the starting points. Depending on which is greater the control uses either the tool axis coordinate from the cycle call or the value from cycle parameter **Q204 2ND SET-UP CLEARANCE**.

NOTICE

Danger of collision!

If you program a clearance height for individual points in a point table, the control will ignore the value from the cycle parameter **Q204 2ND SET-UP CLEARANCE** for all points!

- ▶ Program the function **GLOBAL DEF 125 POSITIONING** so that the control will take into account the clearance height only for the respective point.

Effect with cycles

Cycles 200 to 207

The control interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table as the starting point coordinate in the tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

Cycles 210 to 215

The control interprets the points as an additional datum shift. If you want to use the points defined in the point table as the starting point coordinates, you must program the starting points and the coordinate of the workpiece upper edge (**Q203**) in the respective milling cycle as 0.



You can no longer insert these cycles on the control, but you can edit and run them in existing NC programs.

Cycles 251, 253 and 256

The control interprets the points on the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table as the starting point coordinate in the tool axis, you must define the coordinate of the workpiece upper edge (**Q203**) as 0.

10.1.1 Selecting the point table in the NC program with SEL PATTERN

To select the point table:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.



- ▶ Select **SEL PATTERN**



- ▶ Select **File selection**
- The control opens a window for the file selection.
- ▶ Select the desired point table through the file structure
- ▶ Confirm your input
- The control concludes the NC block.

If the point table is not stored in the same directory as the NC program, you must define the complete path name. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 120

Example

```
7 SEL PATTERN "TNC:\nc_prog\Positions.PNT"
```

10.1.2 Calling the cycle with a point table

If you want to call a cycle at the points that you defined in the point table, then program the cycle call with **CYCL CALL PAT**.

CYCL CALL PAT enables the control to execute the point table that you defined last.

To call a cycle in conjunction with a point table:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.



- ▶ Select **CYCL CALL PAT**
- ▶ Enter a feed rate



The control will use this feed rate to traverse between the points of the point table. If you do not enter a feed rate, the control moves the tool at the feed rate last defined.

- ▶ Define miscellaneous functions, if necessary
- ▶ Confirm your input with the **END** key

Notes

- In the **GLOBAL DEF 125** function you can use the setting **Q435=1** to force the control to always move to the 2nd set-up clearance from the cycle during the positioning between the points.
- If you want to move at reduced feed rate when pre-positioning in the tool axis, program the **M103** miscellaneous function.
- With **CYCL CALL PAT** the control runs the point table that you last defined, even if you defined the point table with an NC program that was nested with **CALL PGM**.

10.2 Pattern definition with PATTERN DEF

Application

You use the **PATTERN DEF** NC function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** NC function. Just like in cycle definitions, help graphics are available for pattern definition that clearly indicate the input parameters required.

Related topics

- Cycles for pattern definition

Further information: "Pattern definition cycles", Page 219

NOTICE

Danger of collision!

The **PATTERN DEF** function calculates the machining coordinates in the **X** and **Y** axes. For all tool axes apart from **Z** there is a danger of collision in the following operation!

- Use **PATTERN DEF** only in connection with the tool axis **Z**

To navigate to this function:

Insert NC function ► Special functions ► Contour/point machining ► Pattern ► PATTERN DEF

Possible setting	Definition	Further information
 POS and /POS	Point Definition of up to any 9 machining positions	Page 210
 ROW	Row Definition of a single row, straight or rotated	Page 211
 PAT	Pattern Definition of a single pattern, straight, rotated or distorted	Page 212
 FRAME	Frame Definition of a single frame, straight, rotated or distorted	Page 213
 CIRC	Circle Definition of a full circle	Page 215
 PITCHCIRC	Pitch circle Definition of a pitch circle	Page 216

Programming PATTERN DEF

To program the **PATTERN DEF** functions:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select the desired machining pattern (e.g., **PATTERN DEF CIRC** for a full circle)
- The control opens the dialog for entering **PATTERN DEF**.
- ▶ Enter the required definitions
- ▶ Define the machining cycle (e.g., Cycle **200 DRILLING**)
- ▶ Call cycle with **CYCL CALL PAT**



While you are programming a machining pattern, you can switch to a different machining pattern in the **Form** column.

Calling PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT NC** function.

Further information: "Calling cycles", Page 143

The control performs the most recently defined machining cycle on the machining pattern you defined.

Program structure: Machining with PATTERN DEF

0 BEGIN SL 2 MM

...

11 PATTERN DEF POS1 (X+25 Y+33.5 Z+0) POS2 (X+15 IY+6.5 Z+0)

12 CYCL DEF 200 DRILLING

...

13 CYCL CALL PAT

Notes

Programming note

- Before **CYCL CALL PAT**, you can use the **GLOBAL DEF 125** function with **Q345=1**. Then, between the holes, the control always positions the tool to the 2nd set-up clearance that was defined in the cycle.

Operating notes:

- A machining pattern remains active until you define a new one or select a point table with the **SEL PATTERN** function.
Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 207
- The control retracts the tool to the clearance height between the starting points. Depending on which is greater, the control uses either the tool axis position from the cycle call or the value from cycle parameter **Q204** as the clearance height.
- If the coordinate surface in **PATTERN DEF** is larger than in the cycle, the set-up clearance and the 2nd set-up clearance reference the coordinate surface in **PATTERN DEF**.
- You can use the mid-program startup function to select any point at which you want to start or continue machining.

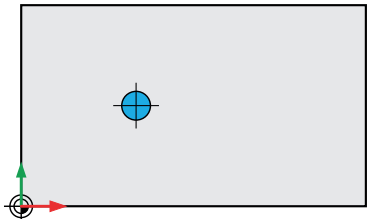
Further information: User's Manual for Setup and Program Run

10.2.1 Defining individual machining positions



Programming and operating notes:

- You can enter up to 9 machining positions. Confirm each entry with the **ENT** key.
- **POS1** must be programmed with absolute coordinates. **POS2** to **POS9** can be programmed as absolute or incremental values.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.
- You can use the **/POS** syntax element to hide positions that are already defined. The control will then skip these positions.

Help graphic	Parameter
	POS1: X coord. of machining position Enter the X coordinate as an absolute value. Input: -999999999...+999999999
	POS1: Y coord. of machining position Enter the Y coordinate as an absolute value. Input: -999999999...+999999999
	POS1: Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts. Input: -999999999...+999999999
	POS2: X coord. of machining position Enter the X coordinate as an incremental or absolute value. Input: -999999999...+999999999
	POS2: Y coord. of machining position Enter the Y coordinate as an incremental or absolute value. Input: -999999999...+999999999
	POS2: Coordinate of workpiece surface Enter the Z coordinate as an incremental or absolute value. Input: -999999999...+999999999

Example

```
11 PATTERN DEF ~
POS1( X+25 Y+33.5 Z+0 ) ~
POS2( X+15 IY+6.5 Z+0 )
```

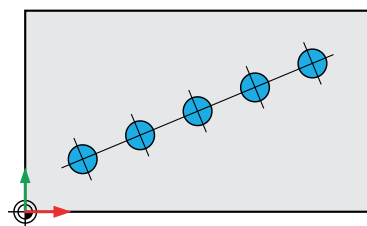
10.2.2 Defining a single row



Programming and operating note:

- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic



Parameter

Starting point in X

Coordinate of the starting point of the row in the X axis. This value has an absolute effect.

Input: **-99999.999999...+99999.999999**

Starting point in Y

Coordinate of the starting point of the row in the Y axis. This value has an absolute effect.

Input: **-99999.999999...+99999.999999**

Spacing of machining positions

Distance (incremental) between the machining positions. Enter a positive or negative value

Input: **-999999999...+999999999**

Number of operations

Total number of machining operations

Input: **0...999**

Rot. position of entire pattern

Angle of rotation around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value

Input: **-360.000...+360.000**

Coordinate of workpiece surface

Enter the Z coordinate as an absolute value at which machining starts

Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
```

```
ROW1( X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0 )
```

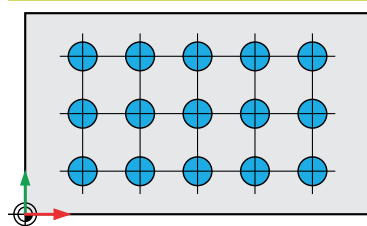
10.2.3 Defining an individual pattern



Programming and operating notes:

- The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic



Parameter

Starting point in X

Absolute coordinate of the pattern starting point in the X axis

Input: **-999999999...+999999999**

Starting point in Y

Absolute coordinate of the pattern starting point in the Y axis

Input: **-999999999...+999999999**

Spacing of machining positions X

Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value

Input: **-999999999...+999999999**

Spacing of machining positions Y

Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value

Input: **-999999999...+999999999**

Number of columns

Total number of columns in the pattern

Input: **0...999**

Number of rows

Total number of rows in the pattern

Input: **0...999**

Rot. position of entire pattern

Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value

Input: **-360.000...+360.000**

Rotary pos. ref. ax.

Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value

Input: **-360.000...+360.000**

Help graphic**Parameter****Rotary pos. minor ax.**

Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value

Input: **-360.000...+360.000**

Coordinate of workpiece surface

Enter the Z coordinate as an absolute value at which machining starts.

Input: **-999999999...+999999999**

Example

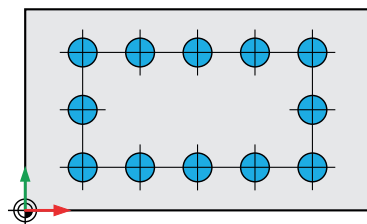
```
11 PATTERN DEF ~
```

```
PAT1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0 )
```

10.2.4 Defining an individual frame

Programming and operating notes:

- The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **Rot. position of entire pattern**.
- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic**Parameter****Starting point in X**

Absolute coordinate of the frame starting point in the X axis

Input: **-999999999...+999999999**

Starting point in Y

Absolute coordinate of the frame starting point in the Y axis

Input: **-999999999...+999999999**

Spacing of machining positions X

Distance in X direction (incremental) between the machining positions. You can enter a positive or negative value

Input: **-999999999...+999999999**

Spacing of machining positions Y

Distance in Y direction (incremental) between the machining positions. You can enter a positive or negative value

Input: **-999999999...+999999999**

Help graphic	Parameter
	Number of columns Total number of columns in the pattern Input: 0...999
	Number of rows Total number of rows in the pattern Input: 0...999
	Rot. position of entire pattern Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). Enter a positive or negative absolute value Input: -360.000...+360.000
	Rotary pos. ref. ax. Angle of rotation around which only the main axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: -360.000...+360.000
	Rotary pos. minor ax. Angle of rotation around which only the secondary axis of the working plane is distorted with respect to the entered starting point. You can enter a positive or negative value. Input: -360.000...+360.000
	Coordinate of workpiece surface Enter the Z coordinate as an absolute value at which machining starts Input: -999999999...+999999999

Example

```
11 PATTERN DEF ~
```

```
FRAME1( X+25 Y+33.5 DX+8 DY+10 NUMX5 NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0 )
```

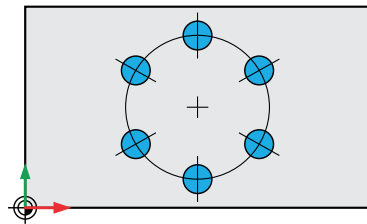
10.2.5 Defining a full circle



Programming and operating notes:

- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic



Parameter

Bolt-hole circle center X

Absolute coordinate of the circle center point in the X axis

Input: **-999999999...+999999999**

Bolt-hole circle center Y

Absolute coordinate of the circle center point in the Y axis

Input: **-999999999...+999999999**

Bolt-hole circle diameter

Diameter of the bolt hole circle

Input: **0...999999999**

Starting angle

Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). You can enter a positive or negative value

Input: **-360.000...+360.000**

Number of operations

Total number of machining positions on the circle

Input: **0...999**

Coordinate of workpiece surface

Enter the Z coordinate as an absolute value at which machining starts.

Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
```

```
CIRC1( X+25 Y+33 D80 START+45 NUM8 Z+0 )
```

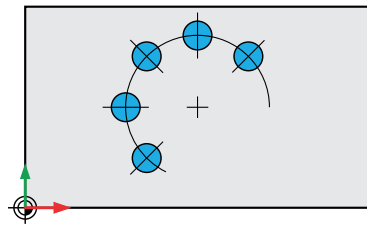
10.2.6 Defining a pitch circle



Programming and operating notes:

- If you have defined a **Workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

Help graphic



Parameter

Bolt-hole circle center X

Absolute coordinate of the circle center point in the X axis

Input: **-999999999...+999999999**

Bolt-hole circle center Y

Absolute coordinate of the circle center point in the Y axis

Input: **-999999999...+999999999**

Bolt-hole circle diameter

Diameter of the bolt hole circle

Input: **0...999999999**

Starting angle

Polar angle of the first machining position. Reference axis: Main axis of the active working plane (e.g., X for tool axis Z). You can enter a positive or negative value

Input: **-360.000...+360.000**

Stepping angle/Stopping angle

Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative you can enter the Stopping angle (switch via the selection possibility on the action bar or in the form)

Input: **-360.000...+360.000**

Number of operations

Total number of machining positions on the circle

Input: **0...999**

Coordinate of workpiece surface

Enter the Z coordinate at which machining starts.

Input: **-999999999...+999999999**

Example

```
11 PATTERN DEF ~
```

```
PITCHCIRC1( X+25 Y+33 D80 START+45 STEP+30 NUM8 Z+0 )
```

10.2.7 Example: Using cycles in conjunction with PATTERN DEF

The drill hole coordinates are stored in the PATTERN DEF POS pattern definition. The control calls the drill hole coordinates with CYCL CALL PAT.

The tool radii have been selected in such a way that all work steps can be seen in the test graphics.

Program sequence

- Centering (tool radius 4)
- **GLOBAL DEF 125 POSITIONING:** This function is used for CYCL CALL PAT and positions the tool at the 2nd set-up clearance between the points. This function remains active until M30 is executed.
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)

Further information: "Cycles for drilling, centering, and thread machining", Page 229 and "Milling cycles"

0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	; Tool call: centering tool (tool radius 4)
4 Z+50 R0 FMAX	; Move tool to clearance height
5 PATTERN DEF ~	
POS1(X+10 Y+10 Z+0) ~	
POS2(X+40 Y+30 Z+0) ~	
POS3(X+20 Y+55 Z+0) ~	
POS4(X+10 Y+90 Z+0) ~	
POS5(X+90 Y+90 Z+0) ~	
POS6(X+80 Y+65 Z+0) ~	
POS7(X+80 Y+30 Z+0) ~	
POS8(X+90 Y+10 Z+0)	
6 CYCL DEF 240 CENTERING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q343=+1 ;SELECT DIA./DEPTH ~	
Q201=-2 ;DEPTH ~	
Q344=-10 ;DIAMETER ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q342=+0 ;ROUGHING DIAMETER ~	
Q253=+750 ;F PRE-POSITIONING	
7 GLOBAL DEF 125 POSITIONING ~	
Q345=+1 ;SELECT POS. HEIGHT	
8 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
9 Z+100 R0 FMAX	; Retract the tool
10 TOOL CALL 227 Z S5000	; Tool call: drill (radius 2.4)

11 X+50 R0 F5000	; Move tool to clearance height
12 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;DEPTH REFERENCE	
13 CYCL CALL PAT F500 M3	; Cycle call in connection with the point pattern
14 Z+100 R0 FMAX	; Retract the tool
15 TOOL CALL 263 Z S200	; Tool call: tap (radius 3)
16 Z+100 R0 FMAX	; Move tool to clearance height
17 CYCL DEF 206 TAPPING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-25 ;DEPTH OF THREAD ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+10 ;2ND SET-UP CLEARANCE	
18 CYCL CALL PAT F5000 M3	; Cycle call in connection with the point pattern
19 Z+100 R0 FMAX	; Retract the tool
20 M30	; End of program run
21 END PGM 1 MM	

10.3 Pattern definition cycles

10.3.1 Overview

The control provides three cycles for machining point patterns:

Cycle	Call	Further information
220 POLAR PATTERN ■ Defining a circular pattern ■ Full circle or pitch circle ■ Input of start and end angles	DEF-active	Page 220
221 CARTESIAN PATTERN ■ Defining a linear pattern ■ Input of an angle of rotation	DEF-active	Page 223

You can combine the following cycles with point pattern cycles:

	Cycle 220	Cycle 221
200 DRILLING	✓	✓
201 REAMING	✓	✓
202 BORING	✓	✓
203 UNIVERSAL DRILLING	✓	✓
204 BACK BORING	✓	✓
205 UNIVERSAL PECKING	✓	✓
206 TAPPING	✓	✓
207 RIGID TAPPING	✓	✓
240 CENTERING	✓	✓
251 RECTANGULAR POCKET	✓	✓
253 SLOT MILLING	✓	✓
256 RECTANGULAR STUD	✓	✓



If you have to machine irregular point patterns, use **CYCL CALL PAT** to develop point tables.
More regular point patterns are available with the **PATTERN DEF** function.

Further information: "Point tables", Page 206

Further information: "Pattern definition with PATTERN DEF", Page 208

10.3.2 Cycle 220 POLAR PATTERN

ISO programming

G220

Application

This cycle enables you to define a point pattern as a full or pitch circle. It can be used for a previously defined machining cycle.



Instead of Cycle **220 POLAR PATTERN**, HEIDENHAIN recommends using the more powerful **PATTERN DEF** function.

Related topics

- Defining a full circle with **PATTERN DEF**
Further information: "Defining a full circle", Page 215
- Defining a circle segment with **PATTERN DEF**
Further information: "Defining a pitch circle", Page 216

Cycle sequence

- 1 The control moves the tool at rapid traverse from its current position to the starting point for the first machining operation.
Sequence:
 - Move to 2nd set-up clearance (spindle axis)
 - Approach the starting point in the working plane
 - Move to set-up clearance above the workpiece surface (spindle axis)
- 2 From this position, the control executes the last defined fixed machining cycle.
- 3 The tool then approaches the starting point for the next machining operation on a straight line or . The tool stops at the set-up clearance (or the 2nd set-up clearance).
- 4 This procedure (steps 1 to 3) will be repeated until all machining operations have been completed.



If you run this cycle in **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

Notes



Cycle **220 POLAR PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

- Cycle **220** is DEF-active. In addition, Cycle **220** automatically calls the last defined machining cycle.

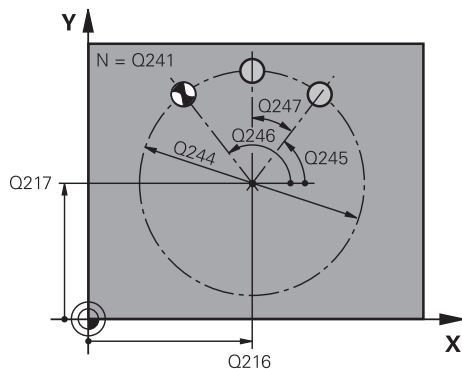
Note on programming

- If you combine one of the machining cycles **200** to **207** or **251**, **253** or **256** with Cycle **220** or Cycle **221**, the set-up clearance, the workpiece surface, and the 2nd set-up clearance from Cycle **220** or **221** are effective. This applies within the NC program until the affected parameters are overwritten again.

Example: If Cycle **200** is defined in an NC program with **Q203=0** and you then program Cycle **220** with **Q203=-5**, then the subsequent calls with **CYCL CALL** and **M99** will use **Q203=-5**. Cycles **220** and **221** overwrite the above-mentioned parameters of **CALL**-active machining cycles (if the same input parameters have been programmed in both cycles).

Cycle parameters

Help graphic



Parameter

Q216 Center in 1st axis?

Pitch circle center in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q217 Center in 2nd axis?

Pitch circle center in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q244 Pitch circle diameter?

Diameter of circle

Input: **0...99999.9999**

Q245 Starting angle?

Angle between the main axis of the working plane and the starting point for the first machining operation on the pitch circle. This value has an absolute effect.

Input: **-360.000...+360.000**

Q246 Stopping angle?

Angle between the main axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you specify a stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. This value has an absolute effect.

Input: **-360.000...+360.000**

Q247 Intermediate stepping angle?

Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the control will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the control will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). This value has an incremental effect.

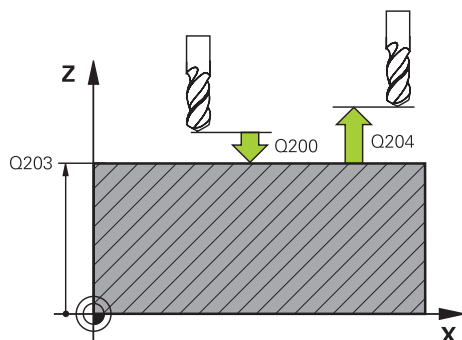
Input: **-360.000...+360.000**

Q241 Number of repetitions?

Number of machining operations on a pitch circle

Input: **1...99999**

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q301 Move to clearance height (0/1)? (optional)

Specify how the tool moves between machining processes:

0: Move to the set-up clearance between operations

1: Move to the 2nd set-up clearance between operations

Input: **0, 1**

Example

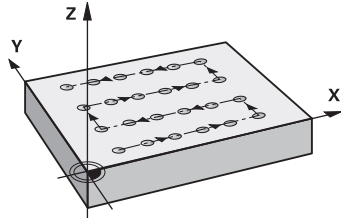
11 CYCL DEF 220 POLAR PATTERN ~	
Q216=+50	;CENTER IN 1ST AXIS ~
Q217=+50	;CENTER IN 2ND AXIS ~
Q244=+60	;PITCH CIRCLE DIAMETR ~
Q245=+0	;STARTING ANGLE ~
Q246=+360	;STOPPING ANGLE ~
Q247=+0	;STEPPING ANGLE ~
Q241=+8	;NR OF REPETITIONS ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+30	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q301=+1	;TO CLEARANCE HEIGHT
12 CYCL CALL	

10.3.3 Cycle 221 CARTESIAN PATTERN

ISO programming

G221

Application



This cycle enables you to define a point pattern as lines. It can be used for a previously defined machining cycle.



Instead of Cycle **221 CARTESIAN PATTERN**, HEIDENHAIN recommends using the more powerful **PATTERN DEF** function.

Related topics

- Defining an individual row with **PATTERN DEF**
Further information: "Defining a single row", Page 211
- Defining an individual pattern with **PATTERN DEF**
Further information: "Defining an individual pattern", Page 212

Cycle sequence

- 1 The control automatically moves the tool from its current position to the starting point for the first machining operation.
 Sequence:
 - Move to 2nd set-up clearance (spindle axis)
 - Approach the starting point in the working plane
 - Move to set-up clearance above the workpiece surface (spindle axis)
- 2 From this position, the control executes the last defined fixed machining cycle.
- 3 Then, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis. The tool stops at the set-up clearance (or the 2nd set-up clearance).
- 4 This procedure (steps 1 to 3) will be repeated until all machining operations from the first row have been completed. The tool is located above the last point of the first row.
- 5 The tool subsequently moves to the last point on the second row where it carries out the machining operation.
- 6 From this position, the tool approaches the starting point for the next machining operation in the negative direction of the reference axis.
- 7 This procedure (step 6) will be repeated until all machining operations of the second row have been completed.
- 8 The tool then moves to the starting point of the next row.
- 9 All subsequent rows are machined in a reciprocating movement.



If you run this cycle in **Program Run / Single Block** mode, the control stops between the individual points of a point pattern.

Notes



Cycle **221 CARTESIAN PATTERN** can be hidden with the optional machine parameter **hidePattern** (no. 128905).

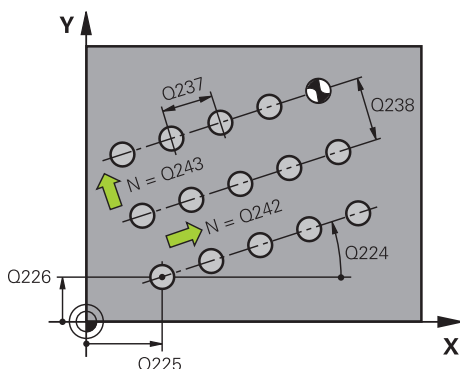
- Cycle **221** is DEF-active. In addition, Cycle **221** automatically calls the last defined machining cycle.

Notes on programming

- If you combine Cycle **221** with one of the machining cycles **200** to **207** or **251**, **253** and **256**, then the set-up clearance, the workpiece surface, the 2nd set-up clearance, and the rotary position that you defined in Cycle **221** will be effective for the selected machining cycle.

Cycle parameters

Help graphic



Parameter

Q225 Starting point in 1st axis?

Coordinate of starting point in the main axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q226 Starting point in 2nd axis?

Coordinate of starting point in the secondary axis of the working plane. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q237 Spacing in 1st axis?

Spacing between the individual points on a line. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q238 Spacing in 2nd axis?

Spacing between the individual lines. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q242 Number of columns?

Number of machining operations on a line

Input: **0...99999**

Q243 Number of lines?

Number of lines

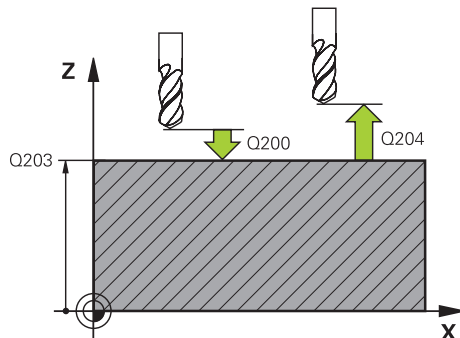
Input: **0...99999**

Q224 Angle of rotation?

Angle by which the entire pattern is rotated. The center of rotation lies in the starting point. This value has an absolute effect.

Input: **-360.000...+360.000**

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q301 Move to clearance height (0/1)? (optional)

Specify how the tool moves between machining processes:

0: Move to the set-up clearance between operations

1: Move to the 2nd set-up clearance between operations

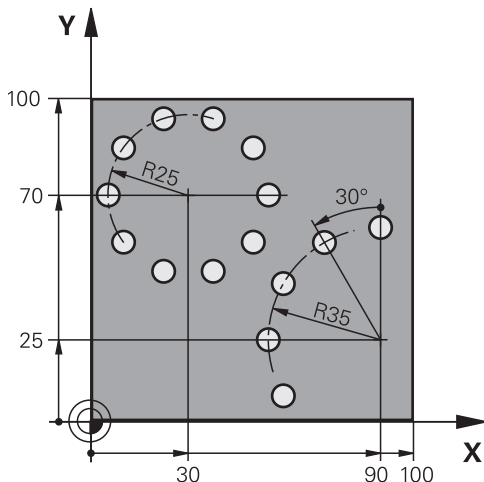
Input: **0, 1**

Example

11 CYCL DEF 221 CARTESIAN PATTERN ~	
Q225=+15	;STARTNG PNT 1ST AXIS ~
Q226=+15	;STARTNG PNT 2ND AXIS ~
Q237=+10	;SPACING IN 1ST AXIS ~
Q238=+8	;SPACING IN 2ND AXIS ~
Q242=+6	;NUMBER OF COLUMNS ~
Q243=+4	;NUMBER OF LINES ~
Q224=+15	;ANGLE OF ROTATION ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q301=+1	;TO CLEARANCE HEIGHT
12 CYCL CALL	

10.3.4 Programming examples

Example: Circular hole patterns



0 BEGIN PGM 200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 200 Z S3500	; Tool call
4 Z+100 R0 FMAX M3	; Retract the tool
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-15 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+4 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0.25 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;TOOL TIP	
6 CYCL DEF 220 POLAR PATTERN ~	
Q216=+30 ;CENTER IN 1ST AXIS ~	
Q217=+70 ;CENTER IN 2ND AXIS ~	
Q244=+50 ;PITCH CIRCLE DIAMETR ~	
Q245=+0 ;STARTING ANGLE ~	
Q246=+360 ;STOPPING ANGLE ~	
Q247=+0 ;STEPPING ANGLE ~	
Q241=+10 ;NR OF REPETITIONS ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+100 ;2ND SET-UP CLEARANCE ~	
Q301=+1 ;TO CLEARANCE HEIGHT	
7 CYCL DEF 220 POLAR PATTERN ~	

Q216=+90	;CENTER IN 1ST AXIS ~	
Q217=+25	;CENTER IN 2ND AXIS ~	
Q244=+70	;PITCH CIRCLE DIAMETR ~	
Q245=+90	;STARTING ANGLE ~	
Q246=+360	;STOPPING ANGLE ~	
Q247=+30	;STEPPING ANGLE ~	
Q241=+5	;NR OF REPETITIONS ~	
Q200=+2	;SET-UP CLEARANCE ~	
Q203=+0	;SURFACE COORDINATE ~	
Q204=+100	;2ND SET-UP CLEARANCE ~	
Q301=+1	;TO CLEARANCE HEIGHT	
4 Z+100 R0 FMAX		; Retract the tool
9 M30		; End of program run
10 END PGM 200 MM		

11

**Cycles for drilling,
centering, and
thread machining**

11.1 Overview

The control offers the following cycles for all types of drilling operations:

Drilling

Cycle		Call	Further information
200 DRILLING	- Basic hole - Input of the dwell time at top and bottom - Depth reference selectable	CALL-active	Page 232
201 REAMING	- Reaming a hole - Input of the dwell time at bottom	CALL-active	Page 236
202 BORING	- Boring a hole - Input of the retraction feed rate - Input of the dwell time at bottom - Input of the retracting movement	CALL-active	Page 238
203 UNIVERSAL DRILLING	- Degression – hole with decreasing infeed - Input of the dwell time at top and bottom - Input of chip breaking behavior - Depth reference selectable	CALL-active	Page 242
205 UNIVERSAL PECKING	- Degression – hole with decreasing infeed - Input of chip breaking behavior - Input of a deepened starting point - Input of an advanced stop distance	CALL-active	Page 247
241 SINGLE-LIP D.H.DRLNG	- Drilling with single-lip deep hole drill - Deepened starting point - Direction of rotation and rotational speed for moving into and retracting from the hole - Input of the dwell depth	CALL-active	Page 254

Countersinking and centering

Cycle		Call	Further information
204 BACK BORING	- Machining a counterbore on the underside of the workpiece - Input of the dwell time - Input of the retracting movement	CALL-active	Page 262
240 CENTERING	- Drilling a center hole - Input of the centering diameter or depth - Input of the dwell time at bottom	CALL-active	Page 266

Tapping

Cycle		Call	Further information
206	TAPPING ■ With a floating tap holder ■ Input of the dwell time at bottom	CALL -active	Page 269
207	RIGID TAPPING ■ Without a floating tap holder ■ Input of the dwell time at bottom	CALL -active	Page 272

11.2 Drilling

11.2.1 Cycle 200 DRILLING

ISO programming
G200

Application

With this cycle, you can drill basic holes. In this cycle, the depth reference is selectable.

Related topics

- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 242
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
Further information: "Cycle 205 UNIVERSAL PECKING ", Page 247
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 254

Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
- 2 The tool drills to the first plunging depth at the programmed feed rate **F**
- 3 The control retracts the tool at **FMAX** to set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to set-up clearance above the first plunging depth
- 4 The tool then drills deeper by the plunging depth at the programmed feed rate **F**.
- 5 The control repeats this procedure (steps 2 to 4) until the programmed depth is reached (the dwell time from **Q211** is effective with every infeed)
- 6 At the end, the tool path is retracted from the hole bottom at rapid traverse **FMAX** to setup clearance or to 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

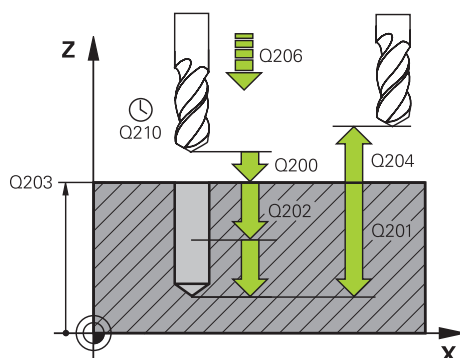
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.



If you want to drill without chip breaking, make sure to define, in the **Q202** parameter, a higher value than the depth **Q201** plus the calculated depth based on the point angle. You can enter a much higher value there.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:

- the plunging depth is equal to the depth
- the plunging depth is greater than the depth

Input: **0...99999.9999**

Q210 Dwell time at the top?

Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal.

Input: **0...3600.0000**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active preset. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q211 Dwell time at the depth? (optional)

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000**

Help graphic**Parameter****Q395 Diameter as reference (0/1)?** (optional)

Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T.

0 = Depth referenced to tool tip

1 = Depth referenced to the cylindrical part of the tool

Input: **0, 1**

Example

11 CYCL DEF 200 DRILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q202=+5	;PLUNGING DEPTH ~
Q210=+0	;DWELL TIME AT TOP ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q395=+0	;TOOL TIP
12 X+30 FMAX M3	
13 Y+20 FMAX M99	
14 X+80 FMAX	
15 Y+50 FMAX M99	

11.2.2 Cycle 201 REAMING

ISO programming

G201

Application

With this cycle, you can machine basic fits. In this cycle, you can optionally define a dwell time at the bottom of the hole.

Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface
- 2 The tool reams to the entered depth at the programmed feed rate **F**.
- 3 If programmed, the tool remains at the hole bottom for the entered dwell time.
- 4 Then, the control retracts the tool at rapid traverse **FMAX** to setup clearance or to 2nd setup clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

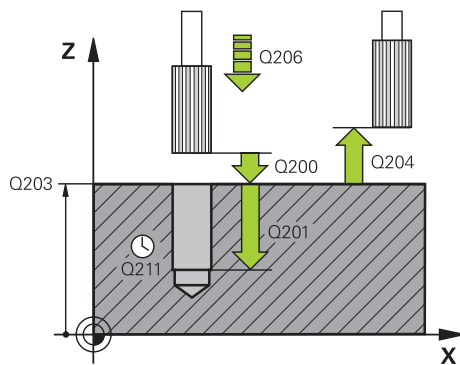
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while reaming

Input: **0...99999.999** or **FAUTO, FU**

Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000**

Q208 Feed rate for retraction?

Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208 = 0**, the feed rate for reaming applies.

Input: **0...99999.9999** or **FMAX, FAUTO**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active preset. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Example

11 CYCL DEF 201 REAMING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;RETRACTION FEED RATE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE
12 X+30 FMAX M3	
13 Y+20 FMAX M99	

11.2.3 Cycle 202 REAMING

ISO programming

G202

Application



Refer to your machine manual.

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

With this cycle, you can bore holes. In this cycle, you can optionally define a dwell time at the bottom of the hole.

Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the safety clearance **Q200** above the workpiece **Q203 SURFACE COORDINATE**
- 2 The tool drills to the programmed depth at the feed rate for plunging **Q201**
- 3 If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
- 4 The control then carries out an oriented spindle stop to the position that is defined in the **Q336** parameter
- 5 If **Q214 DISENGAGING DIRECTN** is defined, the control retracts in the programmed direction by the value in **CLEARANCE TO SIDE Q357**
- 6 Then the control moves the tool at the retraction feed rate **Q208** to the set-up clearance **Q200**
- 7 The tool is again centered in the hole
- 8 The control restores the spindle status as it was at the cycle start.
- 9 If programmed, the control moves the tool at **FMAX** to 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**. If **Q214=0** the tool tip remains on the wall of the hole

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE**Danger of collision!**

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- ▶ Check the position of the tool tip when programming an oriented spindle stop with reference to the angle entered in **Q336** (e.g., in the **MDI** application in the **Manual** operating mode). In this case, no transformations should be active.
- ▶ Select the angle so that the tool tip is parallel to the disengaging direction
- ▶ Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

NOTICE**Danger of collision!**

If you have activated **M136**, the tool will not move to the programmed set-up clearance once the machining operation is finished. The spindle rotation will stop at the bottom of the hole which, in turn, also stops the feed motion. There is a danger of collision as the tool will not be retracted!

- ▶ Use **M137** to deactivate **M136** before the cycle start

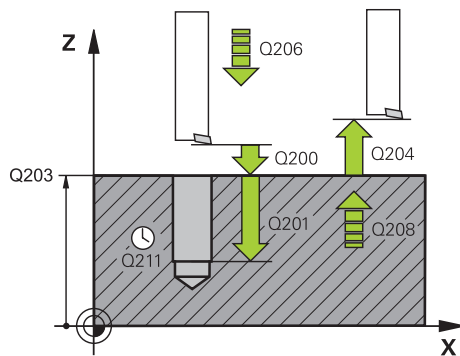
- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- If **Q214 DISENGAGING DIRECTN** is not 0, **Q357 CLEARANCE TO SIDE** is in effect.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while boring

Input: **0...99999.999** or **FAUTO, FU**

Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000**

Q208 Feed rate for retraction?

Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208=0**, the feed rate for plunging applies.

Input: **0...99999.9999** or **FMAX, FAUTO**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q214 Disengaging directn (0/1/2/3/4)?

Specify the direction in which the control retracts the tool at the hole bottom (after carrying out an oriented spindle stop)

0: Do not retract tool

1: Retract tool in negative main axis direction

2: Retract tool in negative secondary axis direction

3: Retract tool in positive main axis direction

4: Retract tool in positive secondary axis direction

Input: **0, 1, 2, 3, 4**

Q336 Angle for spindle orientation? (optional)

Angle to which the control positions the tool before retracting it. This value has an absolute effect.

Input: **0...360**

Help graphic**Parameter****Q357 Safety clearance to the side?** (optional)

Distance between tool tooth and the wall. This value has an incremental effect.

Only in effect if **Q214 DISENGAGING DIRECTN** is not 0.

Input: **0...99999.9999**

Example

11 Z+100 R0 FMAX M3	
12 CYCL DEF 202 BORING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;RETRACTION FEED RATE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q214=+0	;DO NOT RETRACT ~
Q336=+0	;ANGLE OF SPINDLE ~
Q357=+0.2	;CLEARANCE TO SIDE
13 X+30 FMAX	
14 Y+20 FMAX M99	
15 X+80 FMAX	
16 Y+50 FMAX M99	

11.2.4 Cycle 203 UNIVERSAL DRILLING

ISO programming

G203

Application

With this cycle, you can drill holes with decreasing infeed. In this cycle, you can optionally define a dwell time at the bottom of the hole. The cycle may be executed with or without chip breaking.

Related topics

- Cycle **200 DRILLING** for simple holes
Further information: "Cycle 200 DRILLING", Page 232
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
Further information: "Cycle 205 UNIVERSAL PECKING ", Page 247
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 254

Cycle sequence

Behavior without chip breaking, without decrement:

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
- 2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
- 3 Then, the control retracts the tool from the hole to **SET-UP CLEARANCE Q200**
- 4 Now, the control again plunges the tool at rapid traverse into the hole and then again drills an infeed of **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
- 5 When machining without chip breakage the control removes the tool from the hole after each infeed at **RETRACTION FEED RATE Q208** to **SET-UP CLEARANCE Q200** and, if necessary, remains there for the **DWELL TIME AT TOP Q210**
- 6 This sequence will be repeated until the **DEPTH Q201** is reached.
- 7 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking, without decrement:

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
- 2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
- 3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
- 4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** at the **FEED RATE FOR PLNGNG Q206**
- 5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
- 6 If programmed, the control will wait for the time specified in **DWELL TIME AT TOP Q210**
- 7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
- 8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
- 9 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Behavior with chip breaking, with decrement

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above the workpiece surface
- 2 The tool drills at the programmed **FEED RATE FOR PLNGNG Q206** to the first **PLUNGING DEPTH Q202**
- 3 Then, the control retracts the tool by the value in **DIST FOR CHIP BRKNG Q256**
- 4 Now, the tool is plunged again by the value in **PLUNGING DEPTH Q202** minus **DECREMENT Q212** at **FEED RATE FOR PLNGNG Q206**. The increasingly smaller difference between the updated **PLUNGING DEPTH Q202** minus **DECREMENT Q212** must never be smaller than the **MIN. PLUNGING DEPTH Q205** (example: **Q202=5, Q212=1, Q213=4, Q205= 3**: The first plunging depth is 5 mm, the second plunging depth is $5 - 1 = 4$ mm, the third plunging depth is $4 - 1 = 3$ mm, the fourth plunging depth is also 3 mm)
- 5 The control will repeat plunging until the **NR OF BREAKS Q213** is reached or until the hole has the desired **DEPTH Q201**. If the defined number of chip breaks is reached, but the hole does not have the desired **DEPTH Q201** yet, the control will retract the tool at **RETRACTION FEED RATE Q208** from the hole and set it to the **SET-UP CLEARANCE Q200**
- 6 If programmed, the control will now wait for the time specified in **DWELL TIME AT TOP Q210**
- 7 Then, the control will plunge the tool at rapid traverse speed until the value in **DIST FOR CHIP BRKNG Q256** above the last plunging depth is reached
- 8 Steps 2 to 7 will be repeated until **DEPTH Q201** is reached
- 9 If programmed, the control will now wait for the time specified in **DWELL TIME AT DEPTH Q211**
- 10 When **DEPTH Q201** is reached, the control retracts the tool at **FMAX** from the hole to the **SET-UP CLEARANCE Q200** or to the **2ND SET-UP CLEARANCE**. The **2ND SET-UP CLEARANCE Q204** will only come into effect if its value is programmed to be greater than **SET-UP CLEARANCE Q200**

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

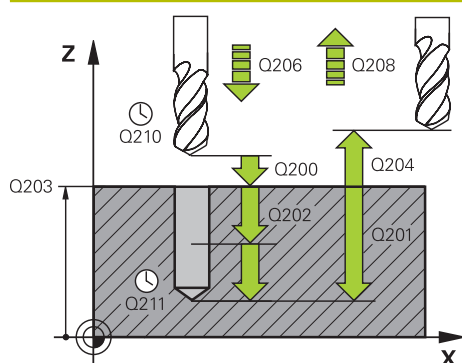
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:

- the plunging depth is equal to the depth
- the plunging depth is greater than the depth

Input: **0...99999.9999**

Q210 Dwell time at the top?

Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal.

Input: **0...3600.0000**

Help graphic	Parameter
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q204 2nd set-up clearance? Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: 0...99999.9999
	Q212 Decrement? Value by which the control decreases Q202 PLUNGING DEPTH after each infeed. This value has an incremental effect. Input: 0...99999.9999
	Q213 Nr of breaks before retracting? Number of chip breaks after which the control is to withdraw the tool from the hole for chip breaking. For chip breaking, the control retracts the tool each time by the value in Q256. Input: 0...99999
	Q205 Minimum plunging depth? If Q212 DECREMENT is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than Q205. This value has an incremental effect. Input: 0...99999.9999
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: 0...3600.0000
	Q208 Feed rate for retraction? Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the control retracts the tool at the feed rate specified in Q206. Input: 0...99999.9999 or FMAX, FAUTO
	Q256 Retract dist. for chip breaking? (optional) Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: 0...99999.999
	Q395 Diameter as reference (0/1)? (optional) Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the T-ANGLE column of the tool table TOOL.T. 0 = Depth referenced to tool tip 1 = Depth referenced to the cylindrical part of the tool Input: 0, 1

Example

11 CYCL DEF 203 UNIVERSAL DRILLING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q202=+5	;PLUNGING DEPTH ~
Q210=+0	;DWELL TIME AT TOP ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q212=+0	;DECREMENT ~
Q213=+0	;NR OF BREAKS ~
Q205=+0	;MIN. PLUNGING DEPTH ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q208=+99999	;RETRACTION FEED RATE ~
Q256=+0.2	;DIST FOR CHIP BRKNG ~
Q395=+0	;TOOL TIP
12 X+30 FMAX M3	
12 Y+20 FMAX M99	

11.2.5 Cycle 205 UNIVERSAL PECKING

ISO programming

G205

Application

With this cycle, you can drill holes with decreasing infeed. The cycle may be executed with or without chip breaking. When the plunging depth is reached the cycle performs chip removal. If there is already a pilot hole then you can enter a deepened starting point. In this cycle, you can optionally define a dwell time at the bottom of the hole. This dwell time is used for chip breaking at the bottom of the hole.

Further information: "Chip removal and chip breaking", Page 252

Related topics

- Cycle **200 DRILLING** for simple holes
Further information: "Cycle 200 DRILLING", Page 232
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 242
- Cycle **241 SINGLE-LIP D.H.DRLNG** optionally with recessed starting point, dwell depth, direction of rotation and speed when entering and leaving the hole
Further information: "Cycle 241 SINGLE-LIP D.H.DRLNG ", Page 254

Cycle sequence

- 1 The control positions the tool in the tool axis at **FMAX** to the entered **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.
- 2 If you program a recessed starting point in **Q379**, the control moves at the positioning feed rate **Q253 F PRE-POSITIONING** to the set-up clearance above the recessed starting point.
- 3 The tool drills at the programmed **Q206 FEED RATE FOR PLNGNG** to the plunging depth.
- 4 If you have programmed chip breaking, the control retracts the tool by the retraction value **Q256**.
- 5 Upon reaching the plunging depth, the control retracts the tool in the tool axis at the retraction feed rate **Q208** to the set-up clearance. The set-up clearance is above the **SURFACE COORDINATE Q203**.
- 6 The tool then moves at **FMAX** to the entered advanced stop distance above the plunging depth last reached.
- 7 The tool drills at the feed in **Q206** to the next plunging depth. If a decrement **Q212** is defined, the plunging depth is decreased after each infeed by the decrement.
- 8 The control repeats this procedure (steps 2 to 7) until the total drilling depth is reached.
- 9 If you entered a dwell time, the tool remains at the hole bottom for chip breaking. The control then retracts the tool at the retraction feed rate to the set-up clearance or the 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

i After chip removal, the depth of the next chip breaking is referenced to the last plunging depth.

Example:

- **Q202 PLUNGING DEPTH** = 10 mm
- **Q257 DEPTH FOR CHIP BRKNG** = 4 mm

The control performs chip breaking at 4 mm and 8 mm. Chip removal is performed at 10 mm. Chip breaking is next performed at 14 mm and 18 mm, etc.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

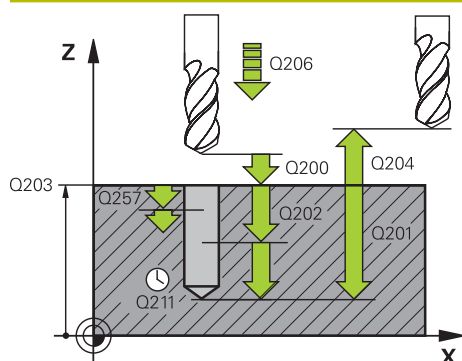
i This cycle is not suitable for overlong drills. For overlong drills, use Cycle **241 SINGLE-LIP D.H.DRLNG**.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- If you enter advance stop distances **Q258** not equal to **Q259**, the control will change the advance stop distances between the first and last plunging depths at the same rate.
- If you use **Q379** to enter a deepened starting point, the control will change the starting point of the infeed movement. Retraction movements are not changed by the control; they are always calculated with respect to the coordinate of the workpiece surface.
- If **Q257 DEPTH FOR CHIP BRKNG** is greater than **Q202 PLUNGING DEPTH**, the operation is executed without chip breaking.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of hole (depends on parameter **Q395 DEPTH REFERENCE**). This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

Q202 Plunging depth?

Tool infeed per cut. This value has an incremental effect.

The depth does not have to be a multiple of the plunging depth. The control will go to depth in one movement if:

- the plunging depth is equal to the depth
- the plunging depth is greater than the depth

Input: **0...99999.9999**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q212 Decrement?

Value by which the control decreases the plunging depth **Q202**. This value has an incremental effect.

Input: **0...99999.9999**

Q205 Minimum plunging depth?

If **Q212 DECREMENT** is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than **Q205**. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic

Parameter

Q258 Upper advanced stop distance?

Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after first chip removal. This value has an incremental effect.

Input: **0...99999.9999**

Q259 Lower advanced stop distance?

Safety clearance above the last plunging depth to which the tool returns at **Q373 FEED AFTER REMOVAL** after the last chip removal. This value has an incremental effect.

Input: **0...99999.9999**

Q257 Infeed depth for chip breaking?

Incremental depth at which the control performs chip breaking. This procedure is repeated until **DEPTH Q201** is reached. If **Q257** equals 0, the control will not perform chip breaking. This value has an incremental effect.

Input: **0...99999.9999**

Q256 Retract dist. for chip breaking?

Value by which the control retracts the tool during chip breaking. This value has an incremental effect.

Input: **0...99999.999**

Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000**

Q379 Deepened starting point? (optional)

If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect.

Input: **0...99999.9999**

Q253 Feed rate for pre-positioning? (optional)

Defines the tool traversing speed when positioning from **Q200 SET-UP CLEARANCE** to **Q379 STARTING POINT** (not equal to 0). Input in mm/min.

Input: **0...99999.9999** or **FMAX, FAUTO**

Q208 Feed rate for retraction? (optional)

Traversing speed of the tool in mm/min when retracting after the machining operation. If you enter **Q208 = 0**, the control retracts the tool at the feed rate specified in **Q206**.

Input: **0...99999.9999** or **FMAX, FAUTO**

Help graphic

Parameter

Q395 Diameter as reference (0/1)? (optional)

Select whether the entered depth is referenced to the tool tip or the cylindrical part of the tool. If the control is to reference the depth to the cylindrical part of the tool, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T.

0 = Depth referenced to tool tip

1 = Depth referenced to the cylindrical part of the tool

Input: **0, 1**

Q373 Post-chip-removal approach feed? (optional)

Traversing speed of the tool when approaching the advanced stop distance after chip removal.

0: Move at **FMAX**

>0: Feed in mm/min

Input: **0...99999** or **FAUTO, FMAX, FU, FZ**

Example

11 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-20	;DEPTH ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q202=+5	;PLUNGING DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q212=+0	;DECREMENT ~
Q205=+0	;MIN. PLUNGING DEPTH ~
Q258=+0.2	;UPPER ADV STOP DIST ~
Q259=+0.2	;LOWER ADV STOP DIST ~
Q257=+0	;DEPTH FOR CHIP BRKNG ~
Q256=+0.2	;DIST FOR CHIP BRKNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q379=+0	;STARTING POINT ~
Q253=+750	;F PRE-POSITIONING ~
Q208=+99999	;RETRACTION FEED RATE ~
Q395=+0	;TOOL TIP ~
Q373=+0	;FEED AFTER REMOVAL

Chip removal and chip breaking

Chip removal

Chip removal depends on cycle parameter **Q202 PLUNGING DEPTH**.

When the value entered in cycle parameter **Q202** is reached, the control performs chip removal. This means that the control always moves the tool to the retraction height, irrespective of the deepened starting point **Q379**. This height is calculated from **Q200 SET-UP CLEARANCE + Q203 SURFACE COORDINATE**

Example:

0 BEGIN PGM 205 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 Z+250 R0 FMAX M3	; Retract the tool; turn on spindle
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+0 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.2 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+10 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;TOOL TIP ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 X+30 R0 FMAX	; Approach drilling position in the X axis
7 Y+30 R0 FMAX	; Approach drilling position in the Y axis
8 CYCL CALL	; Cycle call
9 Z+250 R0 FMAX	; Retract the tool
10 M30	; End of program run
11 END PGM 205 MM	

Chip breaking

Chip breaking depends on cycle parameter **Q257 DEPTH FOR CHIP BRKNG**.

When the value entered in cycle parameter **Q257** is reached, the control performs chip breaking. This means that the control retracts the tool by the value defined in **Q256 DIST FOR CHIP BRKNG**. Chip removal starts once the tool reaches the **PLUNGING DEPTH**. The entire process is repeated until **Q201 DEPTH** is reached.

Example:

0 BEGIN PGM 205 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 203 Z S4500	; Tool call (tool radius 3)
4 Z+250 R0 FMAX M3	; Retract the tool; turn on spindle
5 CYCL DEF 205 UNIVERSAL PECKING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q202=+10 ;PLUNGING DEPTH ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q212=+0 ;DECREMENT ~	
Q205=+0 ;MIN. PLUNGING DEPTH ~	
Q258=+0.2 ;UPPER ADV STOP DIST ~	
Q259=+0.2 ;LOWER ADV STOP DIST ~	
Q257=+3 ;DEPTH FOR CHIP BRKNG ~	
Q256=+0.5 ;DIST FOR CHIP BRKNG ~	
Q211=+0.2 ;DWELL TIME AT DEPTH ~	
Q379=+0 ;STARTING POINT ~	
Q253=+750 ;F PRE-POSITIONING ~	
Q208=+3000 ;RETRACTION FEED RATE ~	
Q395=+0 ;DEPTH REFERENCE ~	
Q373=+0 ;FEED AFTER REMOVAL	
6 X+30 R0 FMAX	; Approach drilling position in the X axis
7 Y+30 R0 FMAX	; Approach drilling position in the Y axis
8 CYCL CALL	; Cycle call
9 Z+250 R0 FMAX	; Retract the tool
10 M30	; End of program run
11 END PGM 205 MM	

11.2.6 Cycle 241 SINGLE-LIP D.H.DRLNG

ISO programming

G241

Application

Cycle **241 SINGLE-LIP D.H.DRLNG** machines holes with a single-lip deep hole drill. It is possible to enter a recessed starting point. The control performs moving to drilling depth with **M3**. You can change the direction of rotation and the rotational speed for moving into and retracting from the hole.

Related topics

- Cycle **200 DRILLING** for simple holes
Further information: "Cycle 200 DRILLING", Page 232
- Cycle **203 UNIVERSAL DRILLING** optionally with decreasing infeed, dwell time and chip breaking
Further information: "Cycle 203 UNIVERSAL DRILLING ", Page 242
- Cycle **205 UNIVERSAL PECKING** optionally with decreasing infeed, chip breaking, recessed starting point and advanced stop distance
Further information: "Cycle 205 UNIVERSAL PECKING ", Page 247

Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered **SET-UP CLEARANCE Q200** above **SURFACE COORDINATE Q203**.
- 2 Depending on the "Positioning behavior during program run with Q379", Page 259, the control will switch on the spindle with the programmed speed either at the **Safety clearance Q200** or at a certain distance above the coordinate surface.
- 3 If you have defined a recessed starting point **Q379 > 0**, the tool moves over the recessed starting point at **Q253 F PRE-POSITIONING** by the value **Q200 SET-UP CLEARANCE Q379**.
- 4 The control executes the approach motion depending on the direction of rotation defined in **Q426 DIR. OF SPINDLE ROT.** with a spindle that rotates clockwise or counterclockwise, or is stationary.
- 5 The tool drills with **M3** and **Q206 FEED RATE FOR PLNGNG** to the drilling depth **Q201** or dwell depth **Q435** or the plunging depth **Q202**:
 - If you have entered **Q435 DWELL DEPTH**, the control reduces the feed rate by **Q401 FEED RATE FACTOR** after the dwell depth has been reached and remains there for **Q211 DWELL TIME AT DEPTH**.
 - If a smaller infeed value has been entered, the control drills to the plunging depth. With each infeed, the plunging depth is reduced by **Q212 DECREMENT**.
- 6 If programmed, the tool remains at the hole bottom for chip breaking.
- 7 After the control has reached this position, it will automatically switch off the coolant, set the speed to the value defined in **Q427 ROT.SPEED INFEEED/OUT** and, if required, change again the direction of rotation defined in **Q426**.
- 8 The control positions the tool to the retract position at the retraction feed rate. To find out the retract position value in your particular case, please refer to:
- 9 If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

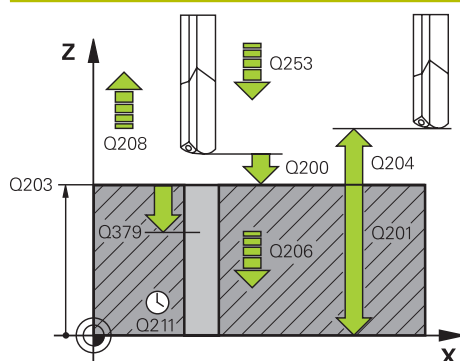
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and **Q203 SURFACE COORDINATE**. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between **Q203 SURFACE COORDINATE** and bottom of hole. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while drilling

Input: **0...99999.999** or **FAUTO, FU**

Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active preset. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Help graphic

Parameter

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q379 Deepened starting point?

If there is already a pilot hole then you can define a deepened starting point here. It is incrementally referenced to **Q203 SURFACE COORDINATE**. The control moves at **Q253 F PRE-POSITIONING** to above the deepened starting point by the value **Q200 SET-UP CLEARANCE**. This value has an incremental effect.

Input: **0...99999.9999**

Q253 Feed rate for pre-positioning?

Defines the traversing speed of the tool when re-approaching **Q201 DEPTH** after **Q256 DIST FOR CHIP BRKNG**. This feed rate is also in effect when the tool is positioned to **Q379 STARTING POINT** (not equal 0). Input in mm/min.

Input: **0...99999.9999** or **FMAX, FAUTO**

Q208 Feed rate for retraction?

Traversing speed of the tool in mm/min when retracting from the hole. If you enter **Q208=0**, the control retracts the tool at **Q206 FEED RATE FOR PLNGNG**.

Input: **0...99999.999** or **FMAX, FAUTO**

Q426 Rot. dir. of entry/exit (3/4/5)?

Rotational speed at which the tool is to rotate when moving into and retracting from the hole.

3: Spindle rotation with M3

4: Spindle rotation with M4

5: Movement with stationary spindle

Input: **3, 4, 5**

Q427 Spindle speed of entry/exit?

Rotational speed at which the tool is to rotate when moving into and retracting from the hole.

Input: **1...99999**

Q428 Spindle speed for drilling?

Desired speed for drilling.

Input: **0...99999**

Q429 M function for coolant on?

>=0: Miscellaneous function M for switching on the coolant. The control switches the coolant on when the tool has reached the set-up clearance **Q200** above the starting point **Q379**.

"...": Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically.

Further information: "User macro", Page 258

Input: **0...999**

Help graphic	Parameter
	Q430 M function for coolant off? >=0: Miscellaneous function M for switching off the coolant. The control switches the coolant off if the tool is at Q201 DEPTH. "...": Path of a user macro that is to be executed instead of an M function. All instructions in the user macro are executed automatically. Further information: "User macro", Page 258 Input: 0...999
	Q435 Dwell depth? (optional) Coordinate in the spindle axis at which the tool is to dwell. If 0 is entered, the function is not active (default setting). Application: During machining of through-holes some tools require a short dwell time before leaving the bottom of the hole in order to transport the chips to the top. Define a value smaller than Q201 DEPTH. This value has an incremental effect. Input: 0...99999.9999
	Q401 Feed rate factor in %? (optional) Factor by which the control reduces the feed rate after reaching Q435 DWELL DEPTH. Input: 0.0001...100
	Q202 Maximum plunging depth? (optional) Infeed per cut. The DEPTH Q201 does not have to be a multiple of Q202. This value has an incremental effect. Input: 0...99999.9999
	Q212 Decrement? (optional) Value by which the control decreases Q202 PLUNGING DEPTH after each infeed. This value has an incremental effect. Input: 0...99999.9999
	Q205 Minimum plunging depth? (optional) If Q212 DECREMENT is not 0, the control limits the plunging depth to this value. This means that the plunging depth cannot be less than Q205. This value has an incremental effect. Input: 0...99999.9999

User macro

User macros are separate NC programs.

A user macro contains a sequence of multiple instructions. With a macro, you can define multiple NC functions that the control executes. As a user, you create macros as NC programs.

Macros work in the same manner as NC programs that are called (e.g., with the NC function **CALL PGM**). Define a macro as an NC program with the file type *.h.

- HEIDENHAIN recommends using QL parameters in the macro. QL parameters have only a local effect for an NC program. If you use other types of variables in the macro, then changes may also have an effect on the calling NC program. In order to explicitly cause changes in the calling NC program, use Q or QS parameters with the numbers 1200 to 1399.
- Within the macro, you can read the value of the cycle parameters.

Further information: "Variables: Q, QL, QR, QS parameters and named parameters", Page 393

Example of a user macro for coolant

0 BEGIN PGM KM MM	
1 FN 18: SYSREAD QL100 = ID20 NR8	; Read the coolant level
2 FN 9: IF QL100 EQU +1 GOTO LBL "Start"	; Query the coolant level; if coolant is active, jump to the Start LBL
3 M8	; Switch coolant on
7 CYCL DEF 9.0 DWELL TIME	
8 CYCL DEF 9.1 DWELL3	
9 LBL "Start"	
10 END PGM RET MM	

Positioning behavior during program run with Q379

Especially when working with very long drills (for example, single-lip deep hole drills or overlong twist drills), there are several things to remember. The position at which the spindle is switched on is very important. If the tool is not guided properly, overlong drills might break.

It is therefore advisable to use the **STARTING POINT Q379** parameter. This parameter can be used to influence the position at which the control turns on the spindle.

Starting point for drilling

The **STARTING POINT Q379** parameter takes both **SURFACE COORDINATE Q203** and the **SET-UP CLEARANCE Q200** parameter into account. The following example illustrates the relationship between the parameters and how the starting position is calculated:

STARTING POINT Q379=0

- The control switches on the spindle at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**

STARTING POINT Q379>0

The starting point is at a certain value above the recessed starting point **Q379**. This value can be calculated as follows: $0.2 \times Q379$; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203** = 0
- **SET-UP CLEARANCE Q200** = 2
- **STARTING POINT Q379** = 2

The starting point of drilling is calculated as follows: $0.2 \times Q379 = 0.2 \times 2 = 0.4$; the starting point of drilling is 0.4 mm or inch above the recessed starting point. So if the recessed starting point is at -2, the control starts the drilling process at -1.6 mm.

The following table lists various examples of how the starting point is calculated:

Drilling start at the recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of 0.2 * Q379	Starting point for drilling
2	2	0	2	$0.2 \cdot 2 = 0.4$	-1.6
2	5	0	2	$0.2 \cdot 5 = 1$	-4
2	10	0	2	$0.2 \cdot 10 = 2$	-8
2	25	0	2	$0.2 \cdot 25 = 5$ (Q200 =2, $5 > 2$, so the value 2 is used.)	-23
2	100	0	2	$0.2 \cdot 100 = 20$ (Q200 =2, $20 > 2$, so the value 2 is used.)	-98
5	2	0	5	$0.2 \cdot 2 = 0.4$	-1.6
5	5	0	5	$0.2 \cdot 5 = 1$	-4
5	10	0	5	$0.2 \cdot 10 = 2$	-8
5	25	0	5	$0.2 \cdot 25 = 5$	-20
5	100	0	5	$0.2 \cdot 100 = 20$ (Q200 =5, $20 > 5$, so the value 5 is used.)	-95
20	2	0	20	$0.2 \cdot 2 = 0.4$	-1.6
20	5	0	20	$0.2 \cdot 5 = 1$	-4
20	10	0	20	$0.2 \cdot 10 = 2$	-8
20	25	0	20	$0.2 \cdot 25 = 5$	-20
20	100	0	20	$0.2 \cdot 100 = 20$	-80

Chip removal

The point at which the control removes chips also plays a decisive role for the work with overlong tools. The retraction position during the chip removal process does not have to be at the start position for drilling. A defined position for chip removal can ensure that the drill stays in the guide.

STARTING POINT Q379=0

- The chips are removed when the tool is positioned at the **SET-UP CLEARANCE Q200** above the **SURFACE COORDINATE Q203**.

STARTING POINT Q379>0

Chip removal is at a certain value above the recessed starting point **Q379**. This value can be calculated as follows: **$0.8 \times Q379$** ; if the result of this calculation is larger than **Q200**, the value is always **Q200**.

Example:

- **SURFACE COORDINATE Q203 =0**
- **SET-UP CLEARANCE Q200 =2**
- **STARTING POINT Q379 =2**

The position for chip removal is calculated as follows: $0.8 \times Q379 = 0.8 \times 2 = 1.6$; the position for chip removal is 1.6 mm or inches above the recessed start point. So if the recessed starting point is at -2, the control starts chip removal at -0.4.

The following table shows examples of how the position for chip removal (retraction position) is calculated:

Position for chip removal (retraction position) with recessed starting point

Q200	Q379	Q203	Position, to which you pre-position with FMAX	Factor of $0.8 \times Q379$	Return position
2	2	0	2	$0.8 \times 2 = 1.6$	-0.4
2	5	0	2	$0.8 \times 5 = 4$	-3
2	10	0	2	$0.8 \times 10 = 8$ (Q200=2 , $8 > 2$, so the value 2 is used.)	-8
2	25	0	2	$0.8 \times 25 = 20$ (Q200=2 , $20 > 2$, so the value 2 is used.)	-23
2	100	0	2	$0.8 \times 100 = 80$ (Q200=2 , $80 > 2$, so the value 2 is used.)	-98
5	2	0	5	$0.8 \times 2 = 1.6$	-0.4
5	5	0	5	$0.8 \times 5 = 4$	-1
5	10	0	5	$0.8 \times 10 = 8$ (Q200=5 , $8 > 5$, so the value 5 is used.)	-5
5	25	0	5	$0.8 \times 25 = 20$ (Q200=5 , $20 > 5$, so the value 5 is used.)	-20
5	100	0	5	$0.8 \times 100 = 80$ (Q200=5 , $80 > 5$, so the value 5 is used.)	-95
20	2	0	20	$0.8 \times 2 = 1.6$	-1.6
20	5	0	20	$0.8 \times 5 = 4$	-4
20	10	0	20	$0.8 \times 10 = 8$	-8
20	25	0	20	$0.8 \times 25 = 20$	-20
20	100	0	20	$0.8 \times 100 = 80$ (Q200=20 , $80 > 20$, so the value 20 is used.)	-80

11.3 Countersinking and centering

11.3.1 Cycle 204 BACK BORING

ISO programming

G204

Application



Refer to your machine manual.

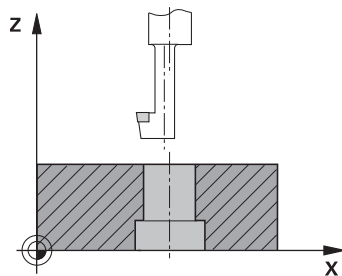
Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.



Special boring bars for upward cutting are required for this cycle.

This cycle allows counterbores to be machined from the underside of the workpiece.



Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the specified set-up clearance above the workpiece surface.
- 2 The control then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.
- 3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the cutting edge has reached the programmed set-up clearance beneath the lower workpiece edge.
- 4 The control then centers the tool again in the bore hole, switches on the spindle and, if applicable, the coolant and moves the tool at the feed rate for counterboring to the depth programmed for the counterbore.
- 5 If programmed, the tool remains at the counterbore bottom. The tool will then be retracted from the hole again. The control carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.
- 6 At the end, the tool moves at **FMAX** to set-up clearance.
- 7 The tool is again centered in the hole.
- 8 The control restores the spindle status as it was at the cycle start.
- 9 If necessary, the control moves the tool to 2nd set-up clearance. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

Notes

NOTICE

Danger of collision!

There is a risk of collision if you choose the wrong direction for retraction. Any mirroring performed in the working plane will not be taken into account for the direction of retraction. In contrast, the control will consider active transformations for retraction.

- ▶ Check the position of the tool tip when programming an oriented spindle stop with reference to the angle entered in **Q336** (e.g., in the **MDI** application in the **Manual** operating mode). In this case, no transformations should be active.
- ▶ Select the angle so that the tool tip is parallel to the disengaging direction
- ▶ Choose a disengaging direction **Q214** that moves the tool away from the wall of the hole.

- After machining, the control returns the tool to the starting point in the working plane. This way, you can continue positioning the tool incrementally.
- When calculating the starting point for boring, the control considers the cutting edge length of the boring bar and the thickness of the material.
- If the M7 or M8 function was active before calling the cycle, the control will reconstruct this previous state at the end of the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF COUNTERBORE Q249**, the control will display an error message.



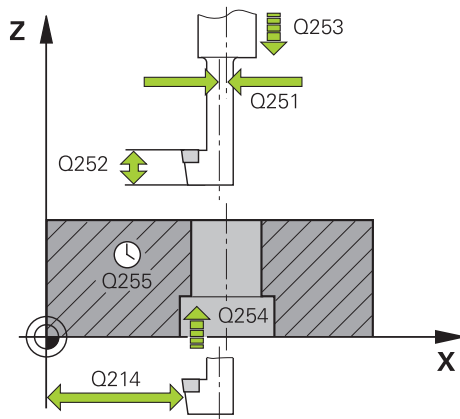
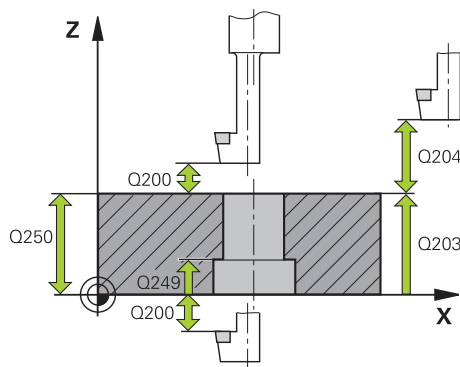
Enter the tool length measured up to the lower edge of the boring bar, not the cutting edge.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the cycle parameter depth determines the working direction. Note: If you enter a positive sign, the tool bores in the direction of the positive spindle axis.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q249 Depth of counterbore?

Distance between underside of workpiece and the top of hole. A positive sign means the hole will be bored in the positive spindle axis direction. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q250 Material thickness?

Height of the workpiece. Enter an incremental value.

Input: **0.0001...99999.9999**

Q251 Tool edge off-center distance?

Off-center distance of the boring bar. Refer to the tool data sheet. This value has an incremental effect.

Input: **0.0001...99999.9999**

Q252 Tool edge height?

Distance between underside of boring bar and main cutting tooth. Refer to the tool data sheet. This value has an incremental effect.

Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when plunging or when retracting.

Input: **0...99999.9999** or **FMAX, FAUTO**

Q254 Feed rate for counterboring?

Traversing speed of the tool in mm/min during counterboring

Input: **0...99999.999** or **FAUTO, FU**

Q255 Dwell time in secs.?

Dwell time in seconds at the bottom of the bore hole

Input: **0...99999.9999**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic**Parameter****Q214 Disengaging directn (0/1/2/3/4)?**

Specify the direction in which the control offsets the tool by the off-center distance (after orienting the spindle). Inputting 0 is not permitted

- 1: Retract tool in negative main axis direction
- 2: Retract tool in negative secondary axis direction
- 3: Retract tool in positive main axis direction
- 4: Retract tool in positive secondary axis direction

Input: **1, 2, 3, 4**

Q336 Angle for spindle orientation? (optional)

Angle at which the control positions the tool before it is plunged into or retracted from the bore hole. This value has an absolute effect.

Input: **0...360**

Example

11 CYCL DEF 204 BACK BORING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q249=+2	;DEPTH OF COUNTERBORE ~
Q250=+20	;MATERIAL THICKNESS ~
Q251=+3.5	;OFF-CENTER DISTANCE ~
Q252=+15	;TOOL EDGE HEIGHT ~
Q253=+750	;F PRE-POSITIONING ~
Q254=+200	;F COUNTERBORING ~
Q255=+0	;DWELL TIME ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q214=+0	;DO NOT RETRACT ~
Q336=+0	;ANGLE OF SPINDLE

11.3.2 Cycle 240 CENTERING

ISO programming

G240

Application

Use Cycle **240 CENTERING** to machine center holes. You can specify the centering diameter or depth and an optional dwell time at the bottom. This dwell time is used for chip breaking at the bottom of the hole. If there is already a pilot hole then you can enter a deepened starting point.

Cycle sequence

- 1 From the current position, the control positions the tool at rapid traverse **FMAX** in the working plane to the starting position.
- 2 The control positions the tool at rapid traverse **FMAX** in the tool axis to the set-up clearance **Q200** above the workpiece surface **Q203**.
- 3 If you define **Q342 ROUGHING DIAMETER** not equal to 0, the control uses this value and the point angle of the tool **T-ANGLE** to calculate a deepened starting point. The control positions the tool at the **F PRE-POSITIONING Q253** at the safety distance **Q200** above the recessed starting point.
- 4 The tool is centered at the programmed feed rate for plunging **F** to the programmed centering diameter or centering depth.
- 5 If a dwell time **Q211** is defined, the tool remains at the centering depth.
- 6 At the end, the tool is retracted to the set-up clearance or to the 2nd set-up clearance at rapid traverse **FMAX**. The 2nd set-up clearance **Q204** will only come into effect if its value is greater than the set-up clearance **Q200**.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

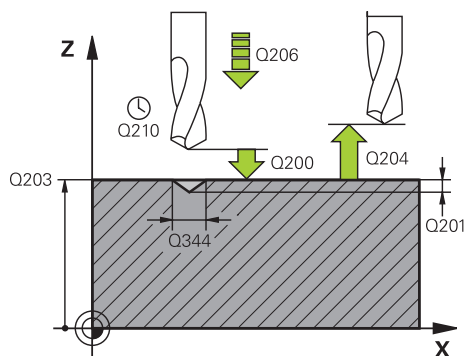
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.

Notes on programming

- Program a positioning block to position the tool at the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the **Q344** (diameter) or **Q201** (depth) cycle parameter determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q343 Select diameter/depth (1/0)

Select whether centering is based on the entered diameter or depth. If the control is to center based on the entered diameter, the point angle of the tool must be defined in the **T-ANGLE** column of the TOOL.T tool table.

0: Centering based on the entered depth

1: Centering based on the entered diameter

Input: **0, 1**

Q201 Depth?

Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if **Q343=0** is defined. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q344 Diameter of counterbore

Centering diameter. Only effective if **Q343=1** is defined.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while centering

Input: **0...99999.999** or **FAUTO, FU**

Q211 Dwell time at the depth?

Time in seconds that the tool remains at the hole bottom.

Input: **0...3600.0000**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Q342 Roughing diameter? (optional)

0: There is no hole

>0: Diameter of the pre-drilled hole

Input: **0...99999.9999**

Help graphic

Parameter

Q253 Feed rate for pre-positioning? (optional)

Traversing speed of the tool when approaching the deepened starting point. The speed is in mm/min.

Only in effect if **Q342 ROUGHING DIAMETER** is not 0.

Input: **0...99999.9999** or **FMAX, FAUTO**

Example

11 CYCL DEF 240 CENTERING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q343=+1	;DIAMETER ~
Q201=-2	;DEPTH ~
Q344=-10	;DIAMETER ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q342=+12	;ROUGHING DIAMETER ~
Q253=+500	;F PRE-POSITIONING
12 X+30 R0 FMAX M3	
13 Y+20 R0 FMAX M99	
14 X+80 R0 FMAX	
15 X+50 R0 FMAX M99	

11.4 Tapping

11.4.1 Cycle 206 TAPPING

ISO programming

G206

Application

The thread is cut in one or more passes. A floating tap holder is used.

Related topics

- Cycle **207 RIGID TAPPING** without floating tap holder
Further information: "Cycle 207 RIGID TAPPING ", Page 272

Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 The tool taps to the total hole depth in one movement.
- 3 Once the tool has reached this position, the direction of spindle rotation is reversed and the tool is retracted to set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
- 4 At the set-up clearance, the direction of spindle rotation is reversed once again.



A floating tap holder is required for tapping. It must compensate for the tolerances between feed rate and spindle speed during the tapping process.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

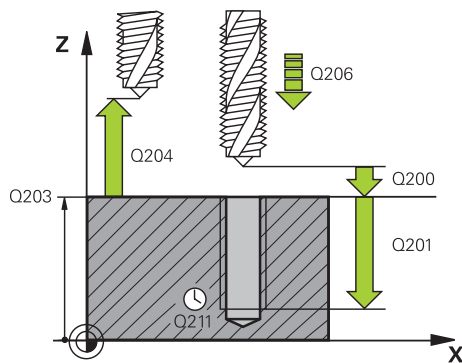
- For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.
- In Cycle **206**, the control uses the programmed rotational speed and the feed rate defined in the cycle to calculate the thread pitch.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.

Notes on programming

- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
 - **sourceOverride** (no. 113603):
FeedPotentiometer (default) (speed override is not active), the control then adjusts the speed as required
SpindlePotentiometer (feed rate override is not active)
 - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified
 - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.

Cycle parameters**Help graphic****Parameter****Q200 Set-up clearance?**

Distance between tool tip and workpiece surface. This value has an incremental effect.

Guide value: 4 times the thread pitch

Input: **0...99999.9999**

Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q206 Feed rate for plunging?

Traversing speed of the tool during tapping

Input: **0...99999.999** or **FAUTO**

Q211 Dwell time at the depth?

Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.

Input: **0...3600.0000**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Example

11 CYCL DEF 206 TAPPING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-18	;DEPTH OF THREAD ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q211=+0	;DWELL TIME AT DEPTH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE
12 CYCL CALL	

The feed rate is calculated as follows: $F = S \times p$

F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

Retraction with stopped NC program

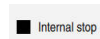
You can retract a thread-turning tool as follows in stopped state:



- ▶ Select **Tool Retract**



- ▶ Press the **NC Start** key
- The tool retracts from the hole and moves to the starting point of machining.
- The spindle is stopped automatically. The control issues an error message.



- ▶ Cancel the NC program with the **Internal stop** button or
- ▶ Acknowledge the error message and continue with **NC Start**



- **Program Run** operating mode:
When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.
- **MDI** application:
When you call a thread cycle, the **Tool Retract** button appears. The button is dimmed until you press **NC stop**.

11.4.2 Cycle 207 RIGID TAPPING

ISO programming

G207

Application



Refer to your machine manual.

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

The control cuts the thread without a floating tap holder in one or more passes.

Related topics

- Cycle **206 TAPPING** with floating tap holder

Further information: "Cycle 206 TAPPING ", Page 269

Cycle sequence

- 1 The control positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 The tool taps to the total hole depth in one movement.
- 3 The direction of spindle rotation is then reversed and the tool is retracted to set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
- 4 The control stops the spindle rotation at set-up clearance.



For tapping, the spindle and the tool axis are always synchronized with each other. The synchronization can be carried out while the spindle is rotating or while it is stationary.

Notes



Cycle **207 RIGID TAPPING** can be hidden with the optional machine parameter **hideRigidTapping** (no. 128903).

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- If you program **M3** (or **M4**) before this cycle, the spindle rotates after the end of the cycle (at the speed programmed in the **TOOL CALL** block).
- If you do not program **M3** (or **M4**) before this cycle, the spindle will stand still after the end of the cycle. In this case, you must restart the spindle with **M3** (or **M4**) before the next operation.
- If you enter the thread pitch of the tap in the **Pitch** column of the tool table, the control compares the thread pitch from the tool table with the thread pitch defined in the cycle. If the values do not match, the control displays an error message.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the **DEPTH OF THREAD Q201**, the control will display an error message.



If you do not change any dynamic parameters (e.g., set-up clearance, spindle speed, ...), it is possible to later tap the thread to a greater depth. However, make sure to select a set-up clearance **Q200** that is large enough so that the tool axis leaves the acceleration path within this distance.

Notes on programming

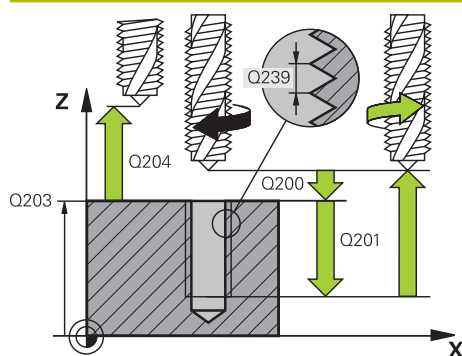
- Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.
- The algebraic sign for the **DEPTH** cycle parameter determines the working direction. If you program **DEPTH=0**, the cycle will not be executed.

Note regarding machine parameters

- Use machine parameter **CfgThreadSpindle** (no. 113600) to define the following:
 - **sourceOverride** (no. 113603): Spindle potentiometer (feed rate override is not active) and feed potentiometer (spindle speed override is not active); the control then adjusts the spindle speed as required
 - **thrdWaitingTime** (no. 113601): After the spindle stop, the tool will dwell at the bottom of the thread for the time specified.
 - **thrdPreSwitch** (no. 113602): The spindle is stopped for this period of time before reaching the bottom of the thread.
 - **limitSpindleSpeed** (no. 113604): Spindle speed limit
True: At small thread depths, spindle speed is limited so that the spindle runs with a constant speed approx. 1/3 of the time.
False: Limiting not active

Cycle parameters

Help graphic



Parameter

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth of thread?

Distance between workpiece surface and root of thread. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q239 Pitch?

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

+= right-hand thread

- = left-hand thread

Input: **-99.9999...+99.9999**

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Distance in the tool axis between tool and workpiece (fixtures) at which no collision can occur. This value has an incremental effect.

Input: **0...99999.9999**

Example

11 CYCL DEF 207 RIGID TAPPING ~	
Q200=+2	;SET-UP CLEARANCE ~
Q201=-18	;DEPTH OF THREAD ~
Q239=+1	;THREAD PITCH ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE
12 CYCL CALL	

Retraction with stopped NC program

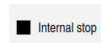
You can retract a thread-turning tool as follows in stopped state:



- ▶ Select **Tool Retract**



- ▶ Press the **NC Start** key
- The tool retracts from the hole and moves to the starting point of machining.
- The spindle is stopped automatically. The control issues an error message.



- ▶ Cancel the NC program with the **Internal stop** button or
- ▶ Acknowledge the error message and continue with **NC Start**



- **Program Run** operating mode:
When stopping the NC program with **NC stop**, the control displays the **Tool Retract** button.
- **MDI** application:
When you call a thread cycle, the **Tool Retract** button appears. The button is dimmed until you press **NC stop**.

12

Milling cycles

12.1 Overview

Pocket milling

Cycle	Call	Further information
251 RECTANGULAR POCKET ■ Roughing and finishing cycle ■ Plunging strategy: helical, reciprocating, or vertical	CALL -active	Page 279
253 SLOT MILLING ■ Roughing and finishing cycle ■ Plunging strategy: reciprocating or vertical	CALL -active	Page 284

Stud milling

Cycle	Call	Further information
256 RECTANGULAR STUD ■ Roughing and finishing cycle ■ Approach position: selectable	CALL -active	Page 289

Milling planes

Cycle	Further information
233 FACE MILLING ■ Roughing and finishing cycle ■ Roughing strategy and direction: selectable ■ Input of side walls	CALL -active Page 297

12.2 Milling pockets

12.2.1 Cycle 251 RECTANGULAR POCKET

ISO programming

G251

Application

Use Cycle **251** to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

Cycle sequence

Roughing

- 1 The tool plunges into the workpiece at the pocket center and advances to the first plunging depth.
- 2 The control roughs out the pocket from the inside out, taking the path overlap (**Q370**) and the finishing allowances (**Q368** and **Q369**) into account.
- 3 At the end of the roughing operation, the control moves the tool away from the pocket wall, then moves to set-up clearance above the current plunging depth. From there, the tool is returned at rapid traverse to the pocket center.
- 4 This process is repeated until the programmed pocket depth is reached.

Finishing

- 1 If finishing allowances have been defined, the control plunges and then approaches the contour. The control first finishes the pocket walls, with multiple infeeds, if so specified.
- 2 Then the control finishes the floor of the pocket from the inside out.

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

NOTICE

Danger of collision!

If you call the cycle with machining operation 2 (only finishing), then the tool is positioned to the first plunging depth + set-up clearance at rapid traverse. There is a danger of collision during positioning at rapid traverse.

- ▶ Conduct a roughing operation beforehand
- ▶ Ensure that the control can pre-position the tool at rapid traverse without colliding with the workpiece

- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

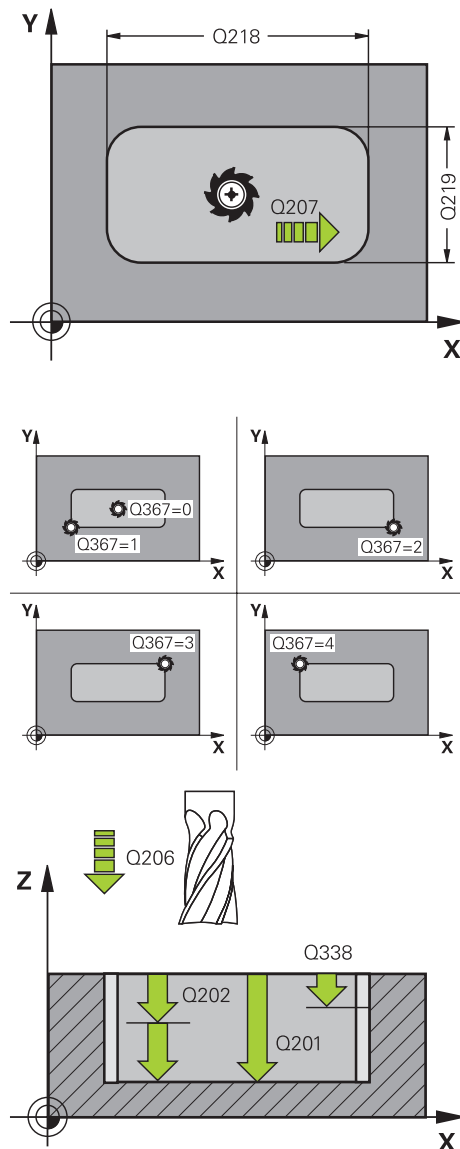
Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: 0: Roughing and finishing 1: Only roughing 2: Only finishing Side finishing and floor finishing are executed only if the respective finishing allowance (Q368, Q369) has been defined Input: 0, 1, 2

Help graphic



Parameter

Q218 First side length?

Pocket length, parallel to the main axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q219 Second side length?

Pocket length, parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of pocket. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q367 Position of pocket (0/1/2/3/4)?

Position of the pocket with respect to the tool when the cycle is called:

0: Tool position = Center of pocket

1: Tool position = Lower left corner

2: Tool position = Lower right corner

3: Tool position = Upper right corner

4: Tool position = Upper left corner

Input: **0, 1, 2, 3, 4**

Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min for moving to depth

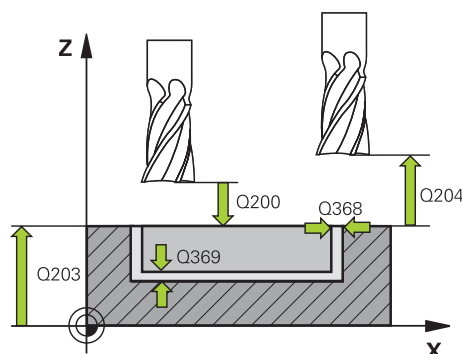
Input: **0...99999.999** or **FAUTO, FU, FZ**

Q385 Finishing feed rate? (optional)

Traversing speed of the tool in mm/min for side and floor finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

Help graphic



Parameter

Q368 Finishing allowance for side?

Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance in depth which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q338 Infeed for finishing?

Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.

0: Finishing in one infeed

Input: **0...99999.9999**

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q203 Workpiece surface coordinate?

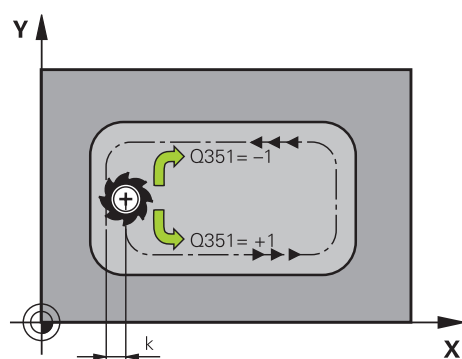
Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999**

**Q351 Direction? Climb=+1, Up-cut=-1**

Type of milling operation. The direction of spindle rotation is taken into account.

+1 = climb milling

-1 = up-cut milling

(if you enter 0, climb milling is performed)

Input: **-1, 0, +1**

Q370 Path overlap factor?

Q370 x tool radius = stepover factor k.

Input: **0.0001...1.41**

Example

11 CYCL DEF 251 RECTANGULAR POCKET ~	
Q215=+0	;MACHINING OPERATION ~
Q218=+60	;FIRST SIDE LENGTH ~
Q219=+20	;2ND SIDE LENGTH ~
Q201=-20	;DEPTH ~
Q367=+0	;POCKET POSITION ~
Q202=+5	;PLUNGING DEPTH ~
Q207=+500	;FEED RATE MILLING ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q385=+500	;FINISHING FEED RATE ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q338=+0	;INFEEED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q351=+1	;CLIMB MILLING ~
Q370=+1	;TOOL PATH OVERLAP
12 X+50 R0 FMAX	
13 Y+50 R0 FMAX M99	

12.2.2 Cycle 253 SLOT MILLING

ISO programming

G253

Application

Use Cycle **253** to completely machine a slot using straight-cut control. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, finishing
- Only roughing
- Only finishing

Cycle sequence

Roughing

- 1 The tool advances at the **FEED RATE FOR PLNGNG Q206** to the first plunging depth **Q202**. The slot created by the roughing process is exactly as wide as the diameter of the tool. When roughing, the control moves the tool only in the tool axis and along the slot length **Q218**. If the slot width is greater than the tool diameter, a subsequent finishing operation needs to be programmed.
- 2 The slot is roughed out, taking parameters **Q351** and **Q352** into account.
- 3 Depending on the setting of parameter **Q352**, the downfeed is either reciprocating (bidirectional) or always from the same side (unidirectional).
 - Bidirectional: The tool performs a cut and then advances to the next plunging depth on the side on which the tool is currently located.
 - Unidirectional: The tool performs a cut, retracts to set-up clearance **Q200** and then returns to the starting position from where it is moved to the next plunging depth. The plunging motion is always performed on the same side.
- 4 This process is repeated until the programmed slot depth is reached.
- 5 At the end, the control retracts the tool to set-up clearance **Q200**, moves it back to the center of the slot and then to 2nd set-up clearance **Q204**.

Finishing

- 6 If a finishing allowance has been defined during pre-machining, the control first finishes the slot walls, using multiple infeeds, if so specified. The slot wall is approached tangentially in the left slot arc
- 7 Then the control finishes the floor of the slot from the inside out.

Notes

NOTICE

Danger of collision!

If you define a slot position not equal to 0, then the control only positions the tool in the tool axis to the 2nd set-up clearance. This means that the position at the end of the cycle does not have to correspond to the position at cycle start! There is a danger of collision!

- ▶ Do **not** program any incremental dimensions after this cycle
- ▶ Program an absolute position in all main axes after this cycle

NOTICE**Danger of collision!**

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.
- Please note that, after roughing, the slot is as wide as the tool diameter, regardless of parameter **Q219**.

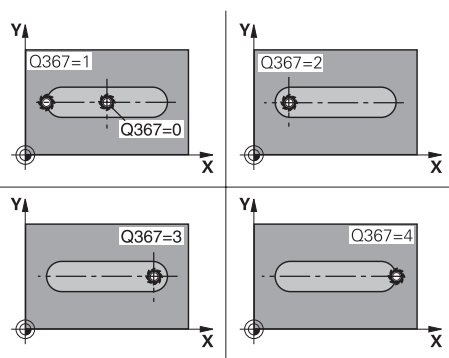
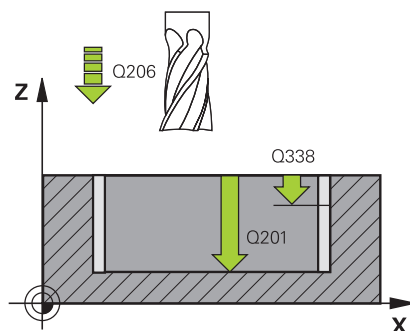
Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.
- Program a sufficient set-up clearance so that the tool cannot jam because of chips.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Specify the machining operation: 0: Roughing and finishing 1: Only roughing 2: Only finishing Input: 0, 1, 2
	Q218 Length of slot? Enter the length of the slot. It is parallel to the main axis of the working plane. This value has an incremental effect. Input: 0...99999.9999

Help graphic



Parameter

Q219 Width of slot?

Enter the width of the slot, which must be parallel to the secondary axis of the working plane. After roughing, the slot is only as wide as the tool diameter, regardless of parameter **Q219**! Maximum slot width for finishing: Twice the tool diameter. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and slot floor. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q374 Slot direction?

Enter whether the slot is rotated by 90° (vertical slot, input: 1) or whether it is not rotated (horizontal slot, input: 0). The center of rotation is at the center of the slot.

Input: **0, 1**

Q367 Position of slot (0/1/2/3/4)?

Position of the figure relative to the position of the tool when the cycle is called:

0: Tool position = Center of figure

1: Tool position = Left end of figure

2: Tool position = Center of left figure arc

3: Tool position = Center of right figure arc

4: Tool position = Right end of figure

Input: **0, 1, 2, 3, 4**

Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min for moving to depth

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q385 Finishing feed rate? (optional)

Traversing speed of the tool in mm/min for side and floor finishing

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q338 Infeed for finishing?

Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.

0: Finishing in one infeed

Input: **0...99999.9999**

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: 0...99999.9999
	Q203 Workpiece surface coordinate? Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q204 2nd set-up clearance? Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect. Input: 0...99999.9999
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. +1 = climb milling -1 = up-cut milling (If you enter 0, climb milling is performed) Input: -1, 0, +1
	Q352 Plunge position? Specify at which position along the main axis the tool should plunge: +1 : Plunging position always at right end of slot -1 : Plunging position always at left end of slot 0 : Reciprocating plunge Input: -1, 0, +1

Example

11 CYCL DEF 253 SLOT MILLING ~	
Q215=+0	;ROUGHING AND FINISHING ~
Q218=+60	;SLOT LENGTH ~
Q219=+10	;SLOT WIDTH ~
Q201=-20	;DEPTH ~
Q374=+0	;SLOT DIRECTION ~
Q367=+0	;SLOT POSITION ~
Q202=+5	;PLUNGING DEPTH ~
Q207=+500	;FEED RATE MILLING ~
Q206=+150	;FEED RATE FOR PLNGNG ~
Q385=+500	;FINISHING FEED RATE ~
Q338=+0	;INFEEED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q351=+1	;CLIMB MILLING ~
Q352=+0	;PLUNGE POSITION
12 X+50 R0 FMAX	
13 Y+50 R0 FMAX M99	

12.3 Milling studs

12.3.1 Cycle 256 RECTANGULAR STUD

ISO programming

G256

Application

Use Cycle **256** to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the control performs multiple stepovers until the finished dimension has been machined.

Cycle sequence

- 1 The tool moves from the cycle starting position (stud center) in the negative X direction to the starting position for stud machining. The starting position is to the left of the unmachined stud and is offset by the set-up clearance + tool radius.
- 2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging
- 3 The tool then moves on a straight line to the stud contour and machines one revolution
- 4 If the finished dimension cannot be machined with one revolution, the control performs a stepover with the current factor, and machines another revolution. The control takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached
- 5 If further stepovers are required, the tool is retracted from the contour and returns to the starting point of stud machining
- 6 The control then plunges the tool to the next plunging depth, and machines the stud at this depth
- 7 This process is repeated until the programmed stud depth is reached

Notes

NOTICE

Danger of collision!

If you enter the depth in a cycle as a positive value, the control reverses the calculation of the pre-positioning. The tool moves at rapid traverse in the tool axis to set-up the clearance **below** the workpiece surface! There is a danger of collision!

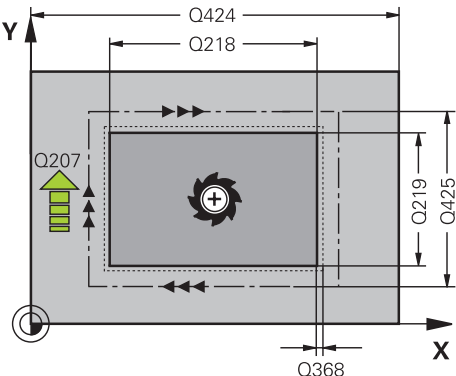
- ▶ Enter depth as negative
- ▶ Use the machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (on) or not (off) if a positive depth is entered

- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- This cycle monitors the defined usable length **LU** of the tool. If the **LU** value is less than the **DEPTH Q201**, the control will display an error message.

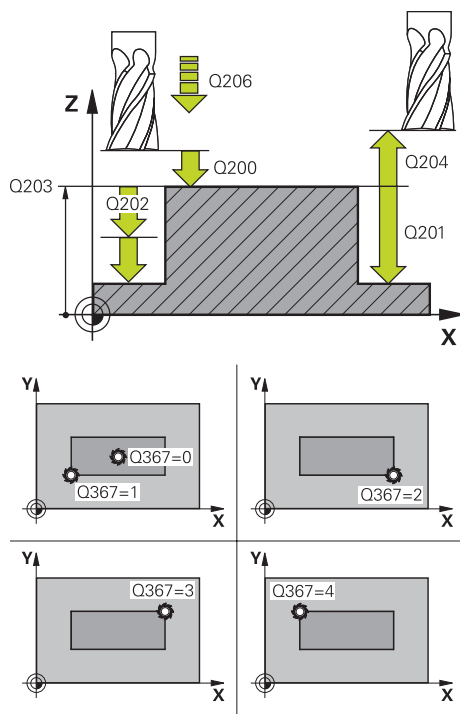
Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation **R0**. Note parameter **Q367** (position).
- The algebraic sign for the DEPTH cycle parameter determines the working direction. If you program DEPTH=0, the cycle will not be executed.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: 0: Roughing and finishing 1: Only roughing 2: Only finishing Side finishing and floor finishing are executed only if the respective finishing allowance (Q368, Q369) has been defined Input: 0, 1, 2
	Q218 First side length? Length of stud parallel to the main axis of the working plane This value has an incremental effect. Input: 0...99999.9999
	Q424 Workpiece blank side length 1? Length of stud blank parallel to the main axis of the working plane. Enter Workpiece blank side length 1 greater than First side length. The control performs multiple lateral stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap Q370). The control always calculates a constant stepover. This value has an incremental effect. Input: 0...99999.9999
	Q219 Second side length? Length of stud parallel to the secondary axis of the working plane. Enter Workpiece blank side length 2 greater than Second side length. The control performs multiple lateral stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap Q370). The control always calculates a constant stepover. This value has an incremental effect. Input: 0...99999.9999

Help graphic



Parameter

Q425 Workpiece blank side length 2?

Length of stud blank parallel to the secondary axis of the working plane. This value has an incremental effect.

Input: **0...99999.9999**

Q201 Depth?

Distance between workpiece surface and bottom of stud. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q367 Position of stud (0/1/2/3/4)?

Position of the stud with respect to the tool when the cycle is called.

0: Tool position = Center of stud

1: Tool position = Lower left corner

2: Tool position = Lower right corner

3: Tool position = Upper right corner

4: Tool position = Upper left corner

Input: **0, 1, 2, 3, 4**

Q202 Plunging depth?

Tool infeed per cut. Enter a value greater than 0. This value has an incremental effect.

Input: **0...99999.9999**

Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q206 Feed rate for plunging?

Traversing speed of the tool in mm/min while moving to depth

Input: **0...99999.999** or **FAUTO, FMAX, FU, FZ**

Q368 Finishing allowance for side?

Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance in depth which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q338 Infeed for finishing?

Infeed in the tool axis when finishing the lateral finishing allowance **Q368**. This value has an incremental effect.

0: Finishing in one infeed

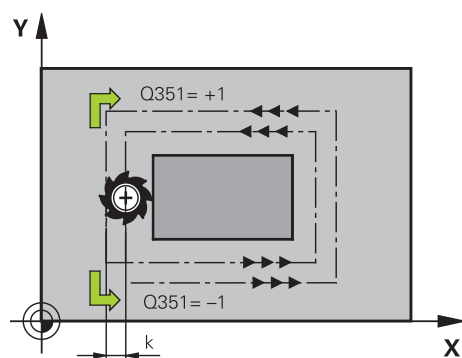
Input: **0...99999.9999**

Q200 Set-up clearance?

Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic



Parameter

Q203 Workpiece surface coordinate?

Coordinate on the workpiece surface referenced to the active datum. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q204 2nd set-up clearance?

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999**

Q351 Direction? Climb=+1, Up-cut=-1

Type of milling operation. The direction of spindle rotation is taken into account.

+1 = climb milling

-1 = up-cut milling

(if you enter 0, climb milling is performed)

Input: **-1, 0, +1**

Q370 Path overlap factor?

Q370x tool radius = stepover factor k. The overlap specified is the maximum overlap. The overlap can be reduced in order to prevent material from remaining at the corners.

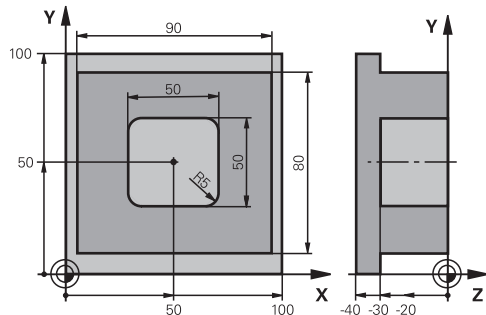
Input: **0.1...1999**

Example

11 CYCL DEF 256 RECTANGULAR STUD ~	
Q215=+1	;ROUGHING ~
Q218=+60	;FIRST SIDE LENGTH ~
Q424=+75	;WORKPC. BLANK SIDE 1 ~
Q219=+20	;2ND SIDE LENGTH ~
Q425=+60	;WORKPC. BLANK SIDE 2 ~
Q201=-20	;DEPTH ~
Q367=+0	;STUD POSITION ~
Q202=+5	;PLUNGING DEPTH ~
Q207=+500	;FEED RATE MILLING ~
Q206=+3000	;FEED RATE FOR PLNGNG ~
Q385=+500	;FINISHING FEED RATE ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q338=+0	;INFEEED FOR FINISHING ~
Q200=+2	;SET-UP CLEARANCE ~
Q203=+0	;SURFACE COORDINATE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q351=+1	;CLIMB MILLING ~
Q370=+1	;TOOL PATH OVERLAP
12 X+50 R0 FMAX	
13 Y+50 R0 FMAX M99	

12.3.2 Programming examples

Example: Milling pockets and studs



0 BEGINN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	; Workpiece blank definition
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	; Tool call: roughing/finishing
4 Z+250 R0 FMAX M3	; Retract the tool
5 CYCL DEF 256 RECTANGULAR STUD ~	; Cycle definition: outside machining
Q215=+0 ;ROUGHING AND FINISHING ~	
Q218=+90 ;FIRST SIDE LENGTH ~	
Q424=+100 ;WORKPC. BLANK SIDE 1 ~	
Q219=+80 ;2ND SIDE LENGTH ~	
Q425=+100 ;WORKPC. BLANK SIDE 2 ~	
Q201=-30 ;DEPTH ~	
Q367=+0 ;STUD POSITION ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q207=+250 ;FEED RATE MILLING ~	
Q206=+250 ;FEED RATE FOR PLNGNG ~	
Q385=+750 ;FINISHING FEED RATE ~	
Q368=+0 ;ALLOWANCE FOR SIDE ~	
Q369=+0.1 ;ALLOWANCE FOR FLOOR ~	
Q338=+5 ;INFEEED FOR FINISHING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+20 ;2ND SET-UP CLEARANCE ~	
Q351=+1 ;CLIMB MILLING ~	
Q370=+1 ;TOOL PATH OVERLAP	
6 X+50 R0	; Outside machining
7 Y+50 R0 M99	; Cycle call for outside machining
8 CYCL DEF 251 RECTANGULAR POCKET ~	; Cycle definition: rectangular pocket
Q215=+0 ;ROUGHING AND FINISHING ~	
Q218=+50 ;FIRST SIDE LENGTH ~	
Q219=+50 ;2ND SIDE LENGTH ~	

Q201=-30	;DEPTH ~	
Q367=+0	;POCKET POSITION ~	
Q202=+5	;PLUNGING DEPTH ~	
Q207=+500	;FEED RATE MILLING ~	
Q206=+150	;FEED RATE FOR PLNGNG ~	
Q385=+750	;FINISHING FEED RATE ~	
Q368=+0.2	;ALLOWANCE FOR SIDE ~	
Q369=+0.1	;ALLOWANCE FOR FLOOR ~	
Q338=+5	;INFEEED FOR FINISHING ~	
Q200=+2	;SET-UP CLEARANCE ~	
Q203=+0	;SURFACE COORDINATE ~	
Q204=+50	;2ND SET-UP CLEARANCE ~	
Q351=+1	;CLIMB MILLING ~	
Q370=+1	;TOOL PATH OVERLAP	
9 X+50 R0 FMAX		
10 Y+50 R0 FMAX M99		; Cycle call
11 Z+250 R0 FMAX M30		
12 M30		; End of program run
13 END PGM C210 MM		

12.4 Milling planes

12.4.1 Cycle 233 FACE MILLING

ISO programming

G233

Application

With Cycle **233**, you can face-mill a level surface in multiple infeeds while taking the finishing allowance into account. You can also define side walls in the cycle, which are then taken into account when machining the level surface. The cycle offers you various machining strategies:

- **Strategy Q389=0**: Meander machining, stepover outside the surface being machined
- **Strategy Q389=1**: Meander machining, stepover at the edge of the surface being machined
- **Strategy Q389=2**: The surface is machined line by line with overtravel; stepover after retracting at rapid traverse
- **Strategy Q389=3**: The surface is machined line by line without overtravel; stepover after retracting at rapid traverse
- **Strategy Q389=4**: Helical machining from the outside toward the inside

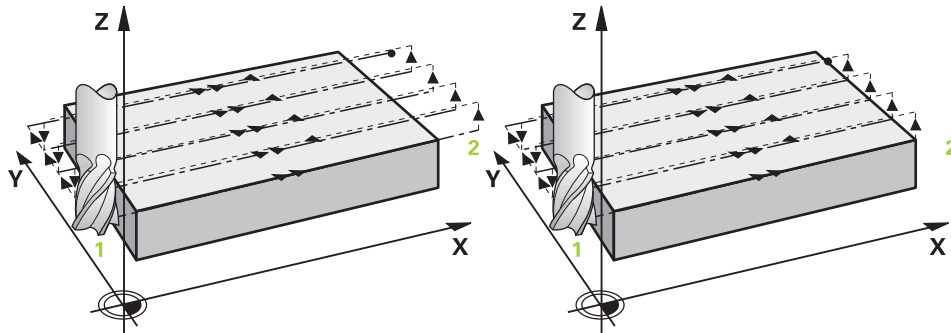
Strategies Q389=0 and Q389 =1

The strategies **Q389=0** and **Q389=1** differ in the overtravel during face milling. If **Q389=0**, the end point lies outside of the surface, with **Q389=1**, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389=0** is used, the control additionally moves the tool beyond the level surface by the tool radius.

Cycle sequence

- 1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
- 2 The control then positions the tool at rapid traverse **FMAX** to set-up clearance in the spindle axis.
- 3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
- 4 The control moves the tool to end point **2** at the programmed feed rate for milling.
- 5 The control then shifts the tool laterally to the starting point of the next line at the pre-positioning feed rate. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor and the set-up clearance to the side.
- 6 The tool then returns in the opposite direction at the feed rate for milling.
- 7 The process is repeated until the programmed surface has been machined completely.
- 8 The control then positions the tool at rapid traverse **FMAX** back to starting point **1**.
- 9 If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
- 10 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
- 11 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.

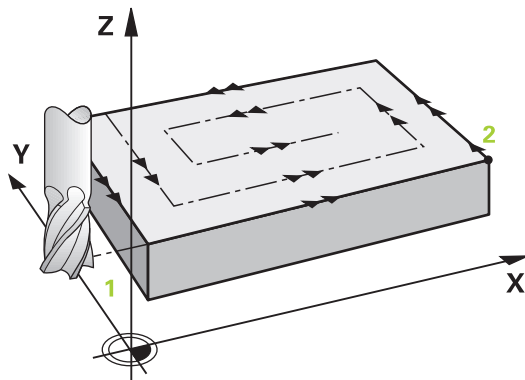
Strategies Q389=2 and Q389 =3



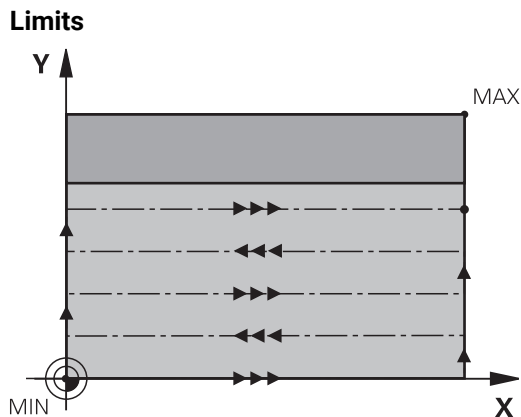
The strategies **Q389=2** and **Q389=3** differ in the overtravel during face milling. If **Q389=2**, the end point lies outside of the surface, with **Q389=3**, it lies at the edge of the surface. The control calculates end point **2** from the side length and the set-up clearance to the side. If the strategy **Q389=2** is used, the control additionally moves the tool beyond the level surface by the tool radius.

Cycle sequence

- 1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
- 2 The control then positions the tool at rapid traverse **FMAX** to set-up clearance in the spindle axis.
- 3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
- 4 The tool subsequently advances at the programmed feed rate for milling **Q207** to the end point **2**.
- 5 The control positions the tool in the tool axis to the set-up clearance above the current infeed depth, and then moves at **FMAX** paraxially back to the starting point in the next pass. The control calculates the offset from the programmed width, the tool radius, the maximum path overlap factor **Q370** and the set-up clearance to the side **Q357**.
- 6 The tool then returns to the current infeed depth and moves in the direction of the end point **2**.
- 7 The process is repeated until the programmed surface has been machined completely. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**.
- 8 If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
- 9 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
- 10 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.

Strategy Q389=4**Cycle sequence**

- 1 From the current position, the control positions the tool at rapid traverse **FMAX** to the starting point **1** in the working plane. The starting point in the working plane is offset from the edge of the workpiece by the tool radius and the set-up clearance to the side.
- 2 The control then positions the tool at rapid traverse **FMAX** to set-up clearance in the spindle axis.
- 3 The tool then moves in the spindle axis at the feed rate for milling **Q207** to the first plunging depth calculated by the control.
- 4 The tool subsequently moves to the starting point of the milling path at the programmed **Feed rate for milling** on a linear tangential approach path.
- 5 The control machines the level surface at the feed rate for milling from the outside toward the inside with ever-shorter milling paths. The constant stepover results in the tool being continuously engaged.
- 6 The process is repeated until the programmed surface has been completed. At the end of the last path, the control returns the tool at rapid traverse **FMAX** to starting point **1**.
- 7 If more than one infeed is required, the control moves the tool in the spindle axis to the next plunging depth at the positioning feed rate.
- 8 The process is repeated until all infeeds have been completed. In the last infeed, the programmed finishing allowance will be milled at the finishing feed rate.
- 9 At the end of the cycle, the tool is retracted at **FMAX** to the **2nd set-up clearance**.



The limits enable you to set limits to the machining of the level surface so that, for example, side walls or shoulders are considered during machining. A side wall that is defined by a limit is machined to the finished dimension resulting from the starting point or the side lengths of the level surface. During roughing the control takes the allowance for the side into account, whereas during finishing the allowance is used for pre-positioning the tool.

Notes

NOTICE

Risk of collision!

In Cycle **Q227**, if you enter **STARTNG PNT 3RD AXIS** deeper than **Q386 END POINT 3RD AXIS**, the control will reverse the calculation of pre-positioning. The tool moves at rapid traverse in the tool axis to a set-up clearance **below** the workpiece surface! Risk of collision!

- ▶ **Q386** Enter **END POINT 3RD AXIS** deeper than **Q227 STARTNG PNT 3RD AXIS**
- ▶ Use machine parameter **displayDepthErr** (no. 201003) to specify whether the control should display an error message (**on**) or not (**off**) if a positive depth is entered

- The control automatically pre-positions the tool in the tool axis. Make sure to program **Q204 2ND SET-UP CLEARANCE** correctly.
- The control reduces the plunging depth to the **LCUTS** cutting edge length defined in the tool table if the cutting edge length is shorter than the **Q202** plunging depth programmed in the cycle.
- Cycle **233** monitors the entries made for the tool or cutting edge length in **LCUTS** in the tool table. If the tool or cutting edge length is not sufficient for a finishing operation, the control will subdivide the process into multiple machining steps.
- This cycle monitors the defined usable length **LU** of the tool. If it is less than the machining depth, the control will display an error message.
- This cycle finishes **Q369 ALLOWANCE FOR FLOOR** with only one infeed. Parameter **Q338 INFEEED FOR FINISHING** has no effect on **Q369**. **Q338** is effective in finishing of **Q368 ALLOWANCE FOR SIDE**.

Notes on programming

- Pre-position the tool in the working plane to the starting position with radius compensation R0. Note the machining direction.
- If you enter identical values for **Q227 STARTNG PNT 3RD AXIS** and **Q386 END POINT 3RD AXIS**, the control does not run the cycle (depth = 0 has been programmed).
- If you define **Q370 TOOL PATH OVERLAP** >1, the programmed overlap factor will be taken into account right from the first machining path.

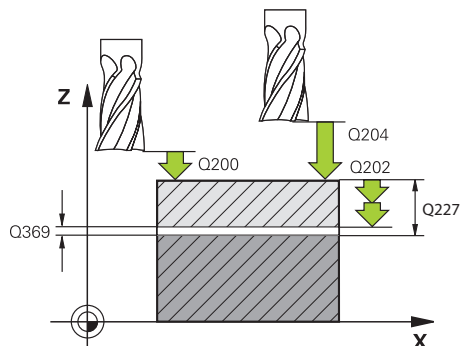


Enter **Q204 2ND SET-UP CLEARANCE** in such a way that no collision with the workpiece or the fixtures can occur.

Cycle parameters

Help graphic	Parameter
	Q215 Machining operation (0/1/2)? Define the machining operation: 0: Roughing and finishing 1: Only roughing 2: Only finishing Side finishing and floor finishing are executed only if the respective finishing allowance (Q368, Q369) has been defined Input: 0, 1, 2
	Q389 Machining strategy (0-4)? Specify how the control machines the surface: 0: Meander machining, stepover at positioning feed rate outside the surface to be machined 1: Meander machining, stepover at the feed rate for milling at the edge of the surface to be machined 2: Machining line by line, retraction and stepover at positioning feed rate outside the surface to be machined 3: Machining line by line, retraction and stepover at positioning feed rate at the edge of the surface to be machined 4: Helical machining, uniform infeed from the outside toward the inside Input: 0, 1, 2, 3, 4
	Q350 Milling direction? Axis in the working plane that defines the machining direction: 1: Main axis = Machining direction 2: Secondary axis = Machining direction Input: 1, 2

Help graphic



Parameter

Q218 First side length?

Length of the surface to be machined in the main axis of the working plane, referencing the starting point in the 1st axis. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q219 Second side length?

Length of the surface to be machined in the secondary axis of the working plane. Use algebraic signs to specify the direction of the first cross feed referenced to the **STARTNG PNT 2ND AXIS**. This value has an incremental effect.

Input: **-99999.9999...+99999.9999**

Q227 Starting point in 3rd axis?

Coordinate of the workpiece surface used to calculate the infeeds. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q386 End point in 3rd axis?

Coordinate in the spindle axis on which the surface will be face-milled. This value has an absolute effect.

Input: **-99999.9999...+99999.9999**

Q369 Finishing allowance for floor?

Finishing allowance in depth which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Q202 Maximum plunging depth?

Infeed per cut. Enter an incremental value greater than 0.

Input: **0...99999.9999**

Q370 Path overlap factor?

Maximum stepover factor k. The control calculates the actual stepover from the second side length (**Q219**) and the tool radius so that a constant stepover is used for machining.

Input: **0.0001...1.9999**

Q207 Feed rate for milling?

Traversing speed of the tool in mm/min for milling

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q385 Finishing feed rate?

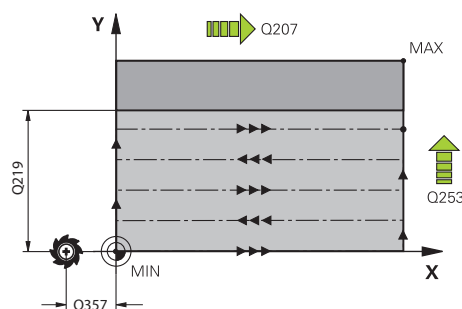
Traversing speed of the tool in mm/min while milling the last infeed

Input: **0...99999.999** or **FAUTO, FU, FZ**

Q253 Feed rate for pre-positioning?

Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely inside the material (**Q389=1**), the control uses the cross feed rate for milling **Q207**.

Input: **0...99999.9999** or **FMAX, FAUTO**



Help graphic

Parameter

Q357 Safety clearance to the side?

Parameter **Q357** influences the following situations:

Approaching the first infeed depth: Q357 is the lateral distance from the tool to the workpiece.

Roughing with the Q389 = 0 to 3 roughing strategies:

The surface to be machined is extended in **Q350 MILLING DIRECTION** by the value from **Q357** if no limit has been set in that direction.

Side finishing: The paths are extended by **Q357** in the **Q350 MILLING DIRECTION**.

This value has an incremental effect.

Input: **0...99999.9999**

Q200 Set-up clearance?

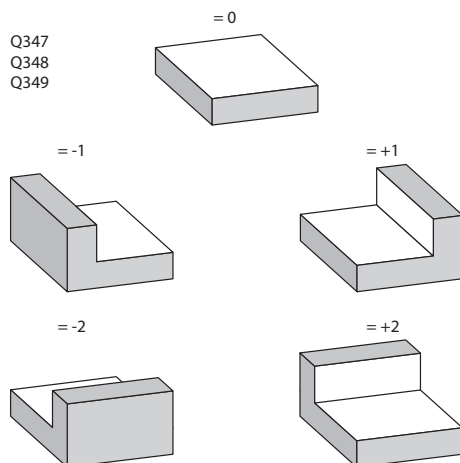
Distance between tool tip and workpiece surface. This value has an incremental effect.

Input: **0...99999.9999**

Q204 2nd set-up clearance?

Coordinate in the spindle axis at which a collision between tool and workpiece (fixtures) is impossible. This value has an incremental effect.

Input: **0...99999.9999**

**Q347 1st limit?**

Select the side of the workpiece where the plane surface is bordered by a side wall. Depending on the position of the side wall, the control limits the machining of the plane surface to the corresponding starting point coordinate or side length:

0: No limitation

-1: Limit in negative main axis

+1: Limit in positive main axis

-2: Limit in negative secondary axis

+2: Limit in positive secondary axis

Input: **-2, -1, 0, +1, +2**

Q348 2nd limit?

See parameter **Q347** 1st limit

Input: **-2, -1, 0, +1, +2**

Q349 3rd limit?

See parameter **Q347** 1st limit

Input: **-2, -1, 0, +1, +2**

Q368 Finishing allowance for side?

Finishing allowance in the machining plane which remains after roughing. This value has an incremental effect.

Input: **0...99999.9999**

Help graphic	Parameter
	Q338 Infeed for finishing? Infeed in the tool axis when finishing the lateral finishing allowance Q368 . This value has an incremental effect. 0: Finishing in one infeed Input: 0...99999.9999
	Q367 Surface position (-1/0/1/2/3/4)? (optional) Position of the surface relative to the position of the tool when the cycle is called: -1: Tool position = Current position 0: Tool position = Center of stud 1: Tool position = Lower left corner 2: Tool position = Lower right corner 3: Tool position = Upper right corner 4: Tool position = Upper left corner Input: -1, 0, +1, +2, +3, +4

Example

11 CYCL DEF 233 FACE MILLING ~	
Q215=+0	;ROUGHING AND FINISHING ~
Q389=+2	;LINE-BY-LINE WITH OVERRUN ~
Q350=+1	;MAIN AXIS ~
Q218=+60	;FIRST SIDE LENGTH ~
Q219=+20	;2ND SIDE LENGTH ~
Q227=+0	;STARTNG PNT 3RD AXIS ~
Q386=+0	;END POINT 3RD AXIS ~
Q369=+0	;ALLOWANCE FOR FLOOR ~
Q202=+5	;MAX. PLUNGING DEPTH ~
Q370=+1	;TOOL PATH OVERLAP ~
Q207=+500	;FEED RATE MILLING ~
Q385=+500	;FINISHING FEED RATE ~
Q253=+750	;F PRE-POSITIONING ~
Q357=+2	;CLEARANCE TO SIDE ~
Q200=+2	;SET-UP CLEARANCE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q347=+0	;NO LIMITATION ~
Q348=+0	;NO LIMITATION ~
Q349=+0	;NO LIMITATION ~
Q368=+0	;ALLOWANCE FOR SIDE ~
Q338=+0	;INFEEED FOR FINISHING ~
Q367=-1	;SURFACE POSITION
12 X+50 R0 FMAX	
13 Y+50 R0 FMAX M99	

13

**Coordinate
transformation**

13.1 Reference systems

13.1.1 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 310
B-CS	Basic coordinate system basic coordinate system	Page 311
W-CS	Workpiece coordinate system workpiece coordinate system	Page 312
I-CS	Input coordinate system input coordinate system	Page 313
T-CS	Tool coordinate system tool coordinate system	Page 314

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 positions of the following reference systems are determined by transformations from this basis.

Definition

Transformations

Translatory transformations each enable a shift along a number line.

13.1.2 Basics of coordinate systems

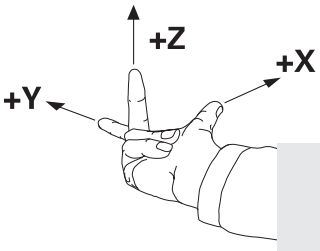
Types of coordinate systems

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.

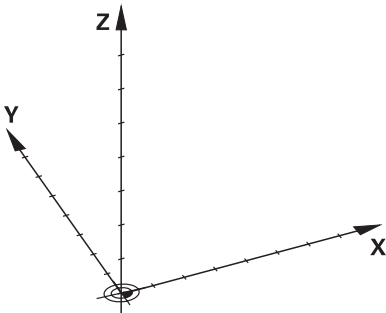
If the axes are arranged perpendicularly to each other, they create a Cartesian coordinate system.

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**.



13.1.3 Machine coordinate system M-CS

Application

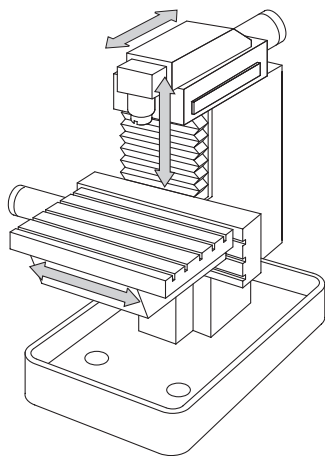
In the machine coordinate system **M-CS** you program constant positions, such as a safe position for retraction. The machine manufacturer also defines constant positions in the **M-CS**, such as the tool-change point.

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 display

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

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

The difference between the values of the **RFACTL** and **ACTL** modes of an axis results from 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: "Traversing in the machine coordinate system M-CS with M91", Page 385

13.1.4 Basic coordinate system B-CS

Application

In the basic coordinate system **B-CS** you define the position 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

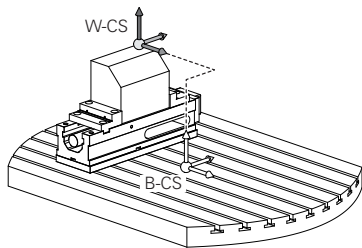
Properties of the basic coordinate system B-CS

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 control saves the determined values as basic transformations in the **B-CS** in the preset table.



Further information: User's Manual for Setup and Program Run

13.1.5 Workpiece coordinate system W-CS

Application

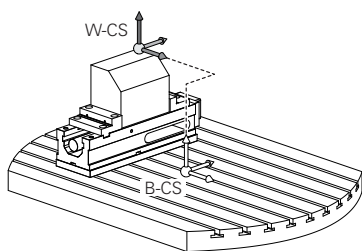
In the workpiece coordinate system **W-CS** you define the position of the working plane. Program the transformations for this purpose.

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.

Further information: User's Manual for Setup and Program Run



Transformations in the workpiece coordinate system (W-CS)

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

- Axes **X, Y, Z** of the **TRANS DATUM** function
Further information: "Datum shift with TRANS DATUM", Page 332
- The **TRANS MIRROR** function or cycle **8 MIRRORING**
Further information: "Mirroring with TRANS MIRROR", Page 334
Further information: "Cycle 8 MIRRORING", Page 323
- The **TRANS SCALE** function or cycle **11 SCALING FACTOR**
Further information: "Scaling with TRANS SCALE", Page 335
Further information: "Cycle 11 SCALING FACTOR ", Page 324
- Cycle **26 AXIS-SPECIFIC SCALING**
Further information: "Cycle 26 AXIS-SPECIFIC SCALING ", Page 326

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

Note

If you do not define any transformation, the positions of the **W-CS** and the **I-CS** are identical.

13.1.6 Input coordinate system I-CS

Application

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

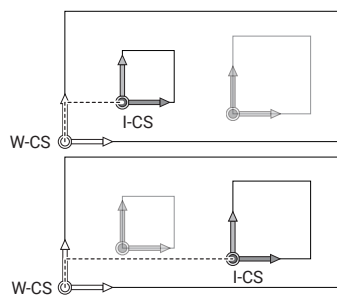
Description of function

Properties of the input coordinate system I-CS

The input coordinate system **I-CS** is a three-dimensional Cartesian coordinate system. You define the coordinate origin of the **I-CS** by means of transformations in the workpiece coordinate system **W-CS**.

Further information: "Workpiece coordinate system W-CS", Page 312

If you do not define any transformation, the positions of the **W-CS** and the **I-CS** are identical.



Position display

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

- **Nominal pos. (NOML)**
- **Actual pos. (ACT)**
- **Actual distance to go (ACTDST)**

13.1.7 Tool coordinate system T-CS

Application

The control implements tool compensations in the tool coordinate system **T-CS**.

Description of function

Properties of the tool coordinate system T-CS

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

By making entries in the tool management, you can define the tool tip relative to the tool carrier reference point. The machine manufacturer usually defines the tool carrier reference point on the spindle tip.

Further information: "Presets in the machine", Page 108

Use the following tool management parameters to define the tool tip relative to the tool carrier reference point:

- L
- DL

Further information: "Tool carrier reference point", Page 159

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

Further information: "Input coordinate system I-CS", Page 313

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

Further information: "Traversing in the machine coordinate system M-CS with M91", Page 385

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: "Tool compensation for tool length and tool radius", Page 340
- Compensation values from the tool call
Further information: "Tool compensation for tool length and tool radius", Page 340

13.2 NC functions for preset management

13.2.1 Overview

The control provides the following functions for modifying a preset directly in the NC program after it has been defined in the preset table:

- Activate the preset
- Copy the preset
- Correct the preset

13.2.2 Activating the preset with PRESET SELECT

Application

The **PRESET SELECT** function allows you to use a preset defined in the preset table and activate it as a new preset.

Requirement

- The preset table contains values
Further information: User's Manual for Setup and Program Run
- Workpiece preset has been defined
Further information: User's Manual for Setup and Program Run

Description of function

To activate the preset, use the row number or the content in the **DOC** column.

The **KEEP TRANS** syntax element allows defining that the control retains the single transformations below:

- the **TRANS DATUM** function
- Cycle **8 MIRRORING** and the **TRANS MIRROR** function
- Cycle **11 SCALING FACTOR** and the **TRANS SCALE** function
- Cycle **26 AXIS-SPECIFIC SCALING**

Input

11 PRESET SELECT #3 KEEP TRANS WP

; Activate row 3 of the table as the workpiece preset and maintain transformations

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Program defaults ► Preset management (PRESET) ► PRESET SELECT

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET SELECT	Syntax initiator for activating a preset
#, Name or QS	Select the row of the preset table Number, text, or variable Selection by means of a selection window With Name the control displays only the rows of the preset table in the selection menu for which the DOC column is defined.
KEEP TRANS	Retain simple transformations Optional syntax element

Notes

NOTICE

Caution: Significant property damage!

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

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

- If you program **PRESET SELECT** without optional parameters, then the behavior is identical to Cycle **247 PRESETTING**.

Further information: "Cycle 247 PRESETTING ", Page 327

- With the optional machine parameter **CfgColumnDescription** (no. 105607), the machine manufacturer specifies whether you can define the same content multiple times in the **DOC** column of the preset table. In this case, if you activate the preset using the **DOC** column, the control cannot clearly identify the desired preset. The control will display the error message **Table access failed**.

13.2.3 Copying the preset with PRESET COPY

Application

The function **PRESET COPY** allows you to copy a preset defined in the preset table and activate the copy.

Requirement

- The preset table contains values
Further information: User's Manual for Setup and Program Run
- Workpiece preset has been defined
Further information: User's Manual for Setup and Program Run

Description of function

To select the preset to be copied, use the row number or the entry in the **DOC** column.

Input

**11 PRESET COPY #1 TO #3 SELECT
TARGET KEEP TRANS**

; Copy row 1 of the preset table to row 3,
activate row 3 as the workpiece preset and
maintain transformations

To navigate to this function:

**Insert NC function ► All functions ► Special functions ► Program defaults ►
Preset management (PRESET) ► PRESET COPY**

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET COPY	Syntax initiator for copying and activating a workpiece preset
#, Name or QS	Select the row of the preset table to be copied Number, text, or variable The row can be chosen from a selection menu. With Name the control displays only the rows of the preset table in the selection menu for which the DOC column is defined.
TO #, Name or QS	Select the new row of the preset table Number, text, or variable Selection by means of a selection window With Name the control displays only the rows of the preset table in the selection menu for which the DOC column is defined.
SELECT TARGET	Activate the copied row of the preset table as the workpiece preset Optional syntax element
KEEP TRANS	Retain simple transformations Optional syntax element

NOTICE**Danger of collision!**

With the optional machine parameter **CfgColumnDescription** (no. 105607), the machine manufacturer specifies whether you can define the same content multiple times in the **DOC** column of the preset table. This means that if you copy a preset using the **DOC** column, the control cannot clearly identify that preset. The control copies the preset with the lowest row number. So if another preset is copied instead of the desired one, there is a risk of collision in subsequent machining operations.

- ▶ Uniquely define the content of the **DOC** column
- ▶ Only copy the preset by indicating its row number

13.2.4 Correcting the preset with PRESET CORR**Application**

The function **PRESET CORR** allows you to correct the active preset.

Requirement

- The preset table contains values
Further information: User's Manual for Setup and Program Run
- Workpiece preset has been defined
Further information: User's Manual for Setup and Program Run

Description of function

The control offsets the entered value against the actual table value. You can enter either a positive or a negative value.

If both the basic rotation and a translation are corrected in an NC block, the control will first correct the translation and then the basic rotation.

The compensation values are given with respect to the input coordinate system **I-CS**. When correcting the OFFS values, the values are based on the machine coordinate system **M-CS**.

Further information: "Reference systems", Page 308

Input

11 PRESET CORR X+10 SPC+45

; Correct the workpiece preset in **X** by +10 mm and in **SPC** by +45°

To navigate to this function:

Insert NC function ▶ All functions ▶ Special functions ▶ Program defaults ▶ Preset management (PRESET) ▶ PRESET CORR

The NC function includes the following syntax elements:

Syntax element	Meaning
PRESET CORR	Syntax initiator for correcting the workpiece preset
X, Y, Z	Compensation values in the main axes Optional syntax element
X_OFFS, Y_OFFS, Z_OFFS, U_OFFS, V_OFFS, W_OFFS	Compensation value for the offsets, referenced to the machine datum Optional syntax element

Note

If you use **PRESET CORR** to correct the offset of a modulo axis, the control always writes a value in the module range -360° to $+360^\circ$ in the preset table.

If a rotary axis already has an offset outside of the modulo range, you can reduce the value with **PRESET CORR** and the entry **0** in the modulo range.

Definitions**Modulo axis**

Modulo axes are axes whose encoder only returns values between 0° and 359.9999° . If an axis is used as a spindle, then the machine manufacturer must configure this axis as a modulo axis.

Modulo counting method

The position display of a rotary axis with the modulo counting method is between 0° and 359.9999° . If the value exceeds 359.9999° , the display starts over at 0° .

13.3 Datum table

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. The datums can be called from within an NC program, for example in order to execute machining processes on several workpieces at the same position. The active row of the datum table serves as the workpiece datum in the NC program.

Related topics

- Contents and creation of a datum table
Further information: "Datum table *.d", Page 610
- Editing a datum table during a program run
Further information: User's Manual for Setup and Program Run
- Preset table
Further information: User's Manual for Setup and Program Run

Description of function

The datums from a datum table are referenced to the current workpiece preset. The coordinate values from datum tables are only effective as absolute coordinate values.

Datum tables can be used in the following situations:

- Frequent use of the same datum shift
- Recurring machining sequences on different workpieces
- Recurring machining sequences at different positions on the workpiece

Activating the datum table manually

A datum table can be activated manually for the **Program Run** operating mode.

In the **Program Run** operating mode, the **Program settings** window contains the **Tables** area. In this area, a datum table and both compensation tables can be selected in one selection window for running the program.

When activating a table, the control will highlight this table with the status **M**.

13.3.1 Activating a datum table in the NC program

To activate a datum table in the NC program:



- ▶ Select **Insert NC function**
- > The control opens the **Insert NC function window**.



- ▶ Select **SEL TABLE**
- > The control opens the action bar.



- ▶ Select **Selection**
- > A file selection window opens.



- ▶ Select datum table
- ▶ Select **Select**

If the datum table is not stored in the same directory as the NC program, the complete path name must be defined. In the **Program settings** window you can define whether the control creates absolute or relative paths.

Further information: "Settings in the Program workspace", Page 120



If you enter the datum table name manually, please note the following:

- If the datum table is stored in the same directory as the NC program, enter the file name only.
- If the datum table is not stored in the same directory as the NC program, enter the complete path.

Definition

File format	Definition
.d	Datum table

13.4 Coordinate transformation cycles

13.4.1 Fundamentals

Once a contour has been programmed, the control can execute it on the workpiece at various locations and in different sizes by using cycles for coordinate transformation.

Effectiveness of coordinate transformations

Beginning of effect: A coordinate transformation takes effect as soon as it is defined –it is not called separately. It remains in effect until it is changed or canceled.

Reset coordinate transformation:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM NC block (these M functions depend on the machine parameters)
- Select a new NC program

13.4.2 Cycle 8 MIRRORING

ISO programming

G28

Application

The control can machine the mirror image of a contour in the working plane.

Mirroring takes effect as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed.
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element "jumps" accordingly.

Reset

Program Cycle **8 MIRRORING** again with **NO ENT**.

Related topics

- Mirroring with **TRANS MIRROR**

Further information: "Mirroring with TRANS MIRROR", Page 334

Cycle parameters

Help graphic	Parameter
	Mirror image axis?
	Enter the axes to be mirrored. You cannot mirror the spindle axis.
	Entry: X, Y, Z

Example

```
11 CYCL DEF 8.0 MIRRORING
```

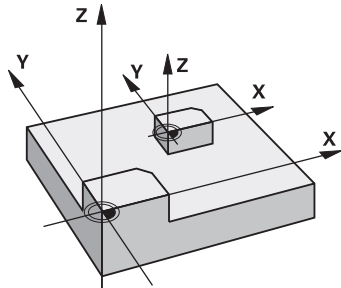
```
12 CYCL DEF 8.1 X Y Z
```

13.4.3 Cycle 11 SCALING FACTOR

ISO programming

G72

Application



The control can increase or reduce the size of contours within an NC program. This enables you to program shrinkage and oversize allowances.

The scaling factor takes effect as soon as it has been defined in the NC program. It is also in effect in the **Manual** operating mode in the **MDI** application. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- all three coordinate axes at the same time
- dimensions in cycles

Requirement

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction: SCL less than 1 (down to 0.000 001)

Reset

Program Cycle **11 SCALING FACTOR** again and specify a scaling factor of 1.

Related topics

- Scaling with **TRANS SCALE**

Further information: "Scaling with TRANS SCALE", Page 335

Cycle parameters

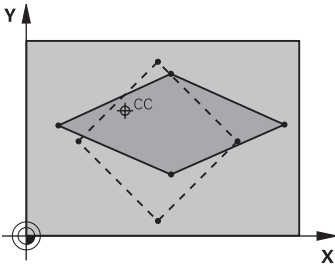
Help graphic	Parameter
	Factor? Enter the scaling factor SCL. The control multiplies the coordinates with SCL. Input: 0.000001...99.999999
Example 11 CYCL DEF 11.0 SCALING FACTOR 12 CYCL DEF 11.1 SCL 0.75	

13.4.4 Cycle 26 AXIS-SPECIFIC SCALING

ISO programming

NC syntax is available only in Klartext programming.

Application



Use Cycle **26** to account for shrinkage and allowance factors for each axis.
 The scaling factor takes effect as soon as it has been defined in the NC program.
 It is also in effect in the **Manual** operating mode in the **MDI** application. The active scaling factor is shown in the additional status display.

Reset

Program Cycle **11 SCALING FACTOR** again and enter a scaling factor of 1 for the corresponding axis.

Notes

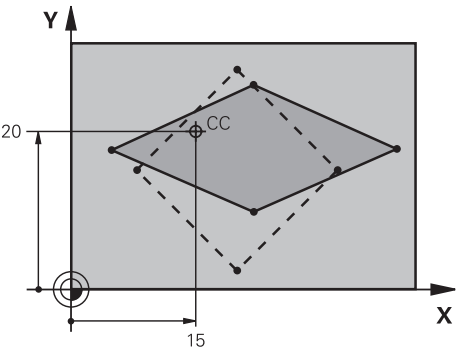
- The contour is enlarged or reduced relative to the center, and not necessarily (as in Cycle **11 SCALING FACTOR**) relative to the active datum.

Notes on programming

- You can program each coordinate axis with its own axis-specific scaling factor.
- In addition, you can enter the coordinates of a center for all scaling factors.

Cycle parameters

Help graphic



Parameter

Axis and factor?

Select the coordinate axis/axes via the action bar. Enter the factor(s) for axis-specific enlargement or reduction.

Input: **0.000001...99.999999**

Centerpoint coord. of extension?

Center of the axis-specific enlargement or reduction.

Input: **-999999999...+999999999**

Example

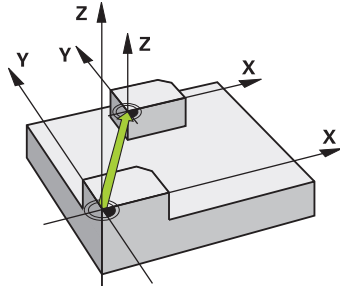
```
11 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
12 CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20
```

13.4.5 Cycle 247 PRESETTING

ISO programming

G247

Application



Use Cycle **247 PRESETTING** to activate a preset defined in the preset table as the new preset.

After cycle definition, all coordinate input and datum shifts (absolute or incremental) reference the new preset.

Status display

In **Program Run** the control shows the active preset number behind the preset symbol in the **Positions** workspace.

Related topics

- Activate the preset
Further information: "Activating the preset with PRESET SELECT", Page 315
- Copy the preset
Further information: "Copying the preset with PRESET COPY", Page 317
- Correct the preset
Further information: "Correcting the preset with PRESET CORR", Page 318
- Setting and activating presets
Further information: User's Manual for Setup and Program Run

Notes

NOTICE

Caution: Significant property damage!

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

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

- When activating a preset from the preset table, the control resets the datum shift, mirroring,, scaling factor, and axis-specific scaling factor.
- If you activate preset number 0 (line 0), then you activate the preset that you last set in the **Manual operation** operating mode.
- Cycle **247** is also in effect in the simulation.

Cycle parameters

Help graphic	Parameter
	Number for preset? Enter the number of the desired preset from the preset table. Alternatively, you can use the button with the preset symbol in the action bar to directly select the desired preset from the preset table. Input: 0...65535

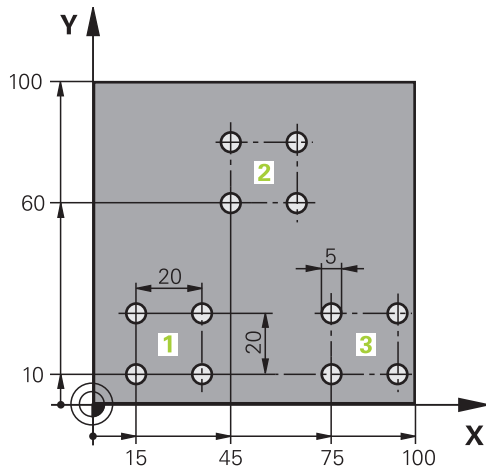
Example

11 CYCL DEF 247 PRESETTING ~	
Q339=+4	;PRESET NUMBER

13.4.6 Example: Coordinate conversion cycles

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1) in the main program
- Program the group of holes only once in subprogram 1



0 BEGIN PGM UP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3000	; Tool call
4 Z+250 R0 FMAX M3	
5 CYCL DEF 200 DRILLING ~	
Q200=+2 ;SET-UP CLEARANCE ~	
Q201=-20 ;DEPTH ~	
Q206=+150 ;FEED RATE FOR PLNGNG ~	
Q202=+5 ;PLUNGING DEPTH ~	
Q210=+0 ;DWELL TIME AT TOP ~	
Q203=+0 ;SURFACE COORDINATE ~	
Q204=+50 ;2ND SET-UP CLEARANCE ~	
Q211=+0 ;DWELL TIME AT DEPTH ~	
Q395=+0 ;TOOL TIP	
6 TRANS DATUM AXIS X+15 Y+10	; Datum shift
7 CALL LBL 1	
8 TRANS DATUM AXIS X+75 Y+10	; Datum shift
9 CALL LBL 1	
10 TRANS DATUM AXIS X+45 Y+60	; Datum shift
11 CALL LBL 1	
12 TRANS DATUM AXIS X+0 Y+0	; Datum shift
13 Z+100 R0 FMAX M30	
14 LBL 1	

15 X+0 R0 FMAX	
16 Y+0 R0 FMAX M99	; Move to 1st hole, call cycle
17 X+20 R0 FMAX M99	; Move to 2nd hole, call cycle
18 Y+20 R0 FMAX M99	; Move to 3rd hole, call cycle
19 X-20 R0 FMAX M99	; Move to 4th hole, call cycle
20 LBL 0	
21 END PGM UP2 MM	

13.5 NC functions for coordinate transformation

13.5.1 Overview

The control provides the following **TRANS** functions:

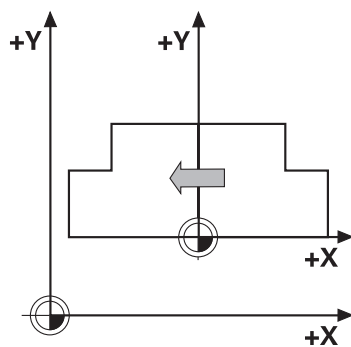
Syntax	Meaning	Further information
TRANS DATUM	Shift the workpiece datum	Page 332
TRANS MIRROR	Mirror an axis	Page 334
TRANS SCALE	Scale contours and positions	Page 335
TRANS RESET	Reset the coordinate transformation	Page 337

Define the functions in the sequence in which they are listed in the table and reset them in reverse order. The sequence of programming will have an impact on the result.

For example, if you first shift the workpiece datum and then mirror the contour and then reverse the sequence, the contour will be mirrored at the original workpiece datum.

All **TRANS** functions reference the workpiece datum. The workpiece datum is the origin of the input coordinate system (**I-CS**).

Further information: "Input coordinate system I-CS", Page 313



Related topics

- Coordinate transformation cycles
Further information: "Coordinate transformation cycles", Page 322
- Reference systems
Further information: "Reference systems", Page 308

13.5.2 Datum shift with TRANS DATUM

Application

The **TRANS DATUM** function allows you to shift the workpiece datum by either entering fixed or variable coordinates or by specifying a table row in the datum table. Use the **TRANS DATUM RESET** function to reset the datum shift.

Related topics

- Contents of the datum table
Further information: "Datum table *.d", Page 610
- Activating the datum table
Further information: "Activating a datum table in the NC program", Page 321
- Machine presets
Further information: "Presets in the machine", Page 108

Description of function

TRANS DATUM AXIS

You can define a datum shift by entering values in the respective axis with the **TRANS DATUM AXIS** function. You can define up to three coordinates in one NC block; incremental entries are possible.

The control displays the result of the datum shift in the **Positions** workspace.

Further information: User's Manual for Setup and Program Run

TRANS DATUM TABLE

You can use the **TRANS DATUM TABLE** function to define a datum shift by selecting a row from a datum table.

Optionally, you can set the path to a datum table. If you do not define a path, the control will use the datum table that has been activated with **SEL TABLE**.

Further information: "Activating a datum table in the NC program", Page 321

The control displays the datum shift and the path to the datum table on the **TRANS** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

TRANS DATUM RESET

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant.

Input

11 TRANS DATUM AXIS X+10 Y+25 Z+42 ; Shift the workpiece datum in the **X, Y** and **Z** axes

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Functions ► Coordinate transformations TRANS ► TRANS DATUM

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS DATUM	Syntax initiator for a datum shift
AXIS, TABLE or RESET	Datum shift with coordinate input, with a datum table or reset of the datum shift
X, Y or Z	Possible axes for coordinate input Fixed or variable number Only if AXIS has been selected
TABLINE	Row in the datum table Fixed or variable number Only if TABLE has been selected
Name or Parameter	Path to the datum table Fixed or variable path Selection by means of a selection window Optional syntax element Only if TABLE has been selected

Notes

- The **TRANS DATUM** function replaces Cycle **7 DATUM SHIFT**. If you import an NC program from an older control, then, during editing, the control turns Cycle **7** into the **TRANS DATUM** NC function.
- If you execute an absolute datum shift with **TRANS DATUM** or Cycle **7 DATUM SHIFT**, then the control overwrites the values of the current datum shift. The control adds the incremental values to the values of the current datum shift.
- Absolute values reference the workpiece preset. Incremental values reference the workpiece datum.

Further information: "Presets in the machine", Page 108

- In machine parameter **transDatumCoordSys** (no. 127501), the machine manufacturer defines the reference system referred to by the values in the position display.

Further information: "Reference systems", Page 308

13.5.3 Mirroring with TRANS MIRROR

Application

Use the **TRANS MIRROR** function to mirror contours or positions about one or more axes.

The **TRANS MIRROR RESET** function allows you to reset mirroring.

Related topics

- Cycle **8 MIRRORING**

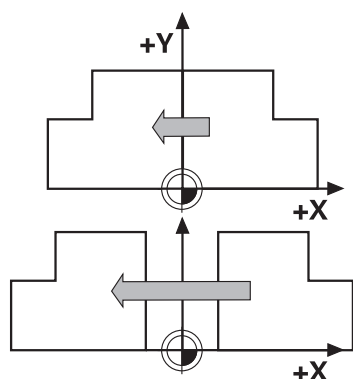
Further information: "Cycle 8 MIRRORING", Page 323

Description of function

Mirroring is a modal function that is effective as soon as it has been defined in the NC program.

The control mirrors contours or positions about the active workpiece datum. If the datum is outside the contour, the control will also mirror the distance to the datum.

Further information: "Presets in the machine", Page 108



If you mirror only one axis, the machining direction of the tool is reversed. If the machining direction is defined in the cycle, it is retained.

Depending on the selected **AXIS** axis values, the control will mirror the following working planes:

- **X**: The control mirrors the **YZ** working plane
- **Y**: The control mirrors the **ZX** working plane
- **Z**: The control mirrors the **XY** working plane

Further information: "Designation of the axes of milling machines", Page 106

You can select up to three axis values.

If mirroring is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Input**11 TRANS MIRROR AXIS X**

; Mirror X coordinates about the Y axis

To navigate to this function:

Insert NC function ► **All functions** ► **Special functions** ► **Functions** ►
Coordinate transformations TRANS ► **TRANS MIRROR**

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS MIRROR	Syntax initiator for mirroring
AXIS or RESET	Enter mirroring of axis values or reset mirroring
X, Y or Z	Axis values to be mirrored Only if AXIS has been selected

Notes

- If you execute mirroring with **TRANS MIRROR** or Cycle **8 MIRRORING**, then the control overwrites the current mirroring.

Further information: "Cycle 8 MIRRORING", Page 323

13.5.4 Scaling with TRANS SCALE**Application**

The **TRANS SCALE** function lets you change the scale of the contours or distances to the datum, thereby evenly enlarging or shrinking them. This enables you to program shrinkage and oversize allowances, for example.

Use the **TRANS SCALE RESET** function to reset scaling.

Related topics

- Cycle **11 SCALING FACTOR**

Further information: "Cycle 11 SCALING FACTOR ", Page 324

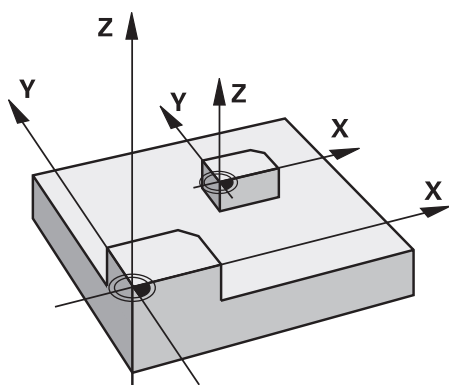
Description of function

Scaling is a modal function that is in effect as soon as it has been defined in the NC program.

Depending on the position of the workpiece datum, scaling is carried out as follows:

- Workpiece datum at the center of the contour:
The contour is scaled uniformly in all directions.
- Workpiece datum at the bottom left of the contour:
The contour is scaled in the positive X and Y axis directions.
- Workpiece datum at the top right of the contour:
The contour is scaled in the negative X and Y axis directions.

Further information: "Presets in the machine", Page 108



If you enter a scaling factor **SCL** less than 1, the contour will be reduced in size. If you enter a scaling factor **SCL** greater than 1, the contour will be enlarged.

When scaling, the control takes the coordinate input and dimensions from all cycles into account.

If Scaling is active, the control displays it on the **TRANS** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Input

11 TRANS SCALE SCL1.5

; Enlarge the contour by the factor 1.5

To navigate to this function:

**Insert NC function ► All functions ► Special functions ► Functions ►
Coordinate transformations TRANS ► TRANS DATUM**

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS SCALE	Syntax initiator for scaling
SCL or RESET	Enter the scaling factor or reset scaling Number or numerical parameter

Note

If you execute a change of scale with **TRANS SCALE** or Cycle **11 SCALING FACTOR**, then the control overwrites the current scaling factor.

Further information: "Cycle 11 SCALING FACTOR ", Page 324

13.5.5 Resetting with TRANS RESET

Application

Use the NC function **TRANS RESET** to reset all simple coordinate transformations simultaneously.

Related topics

- NC functions for coordinate transformation
Further information: "NC functions for coordinate transformation", Page 331
- Coordinate transformation cycles
Further information: "Coordinate transformation cycles", Page 322

Description of function

The control resets the following simple coordinate transformations:

Coordinate transformation	Syntax	Further information
Datum shift	TRANS DATUM	Page 332
Mirroring	TRANS MIRROR	Page 334
	Cycle 8 MIRRORING	Page 323
Scaling	TRANS SCALE	Page 335
	Cycle 11 SCALING FACTOR	Page 324
	Cycle 26 AXIS-SPECIFIC SCALING	Page 326



The control also resets simple coordinate transformations defined by the machine manufacturer.

Input

11 TRANS RESET

; Reset simple coordinate transformations

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Functions ► Coordinate transformations TRANS ► TRANS RESET

The NC function includes the following syntax elements:

Syntax element	Meaning
TRANS RESET	Syntax initiator for resetting simple coordinate transformations

14

Compensations

14.1 Tool compensation for tool length and tool radius

Application

Delta values allow implementing tool compensation of the tool length and the tool radius. Delta values influence the calculated and therefore the active tool dimensions.

The tool length delta value **DL** is active in the tool axis. The tool radius delta value **DR** is active exclusively for radius-compensated traverses with the path functions and cycles.

Further information: "Traverse movements", Page 171

Related topics

- Tool radius compensation

Further information: "Tool radius compensation", Page 344

Description of function

The control distinguishes between two types of delta values:

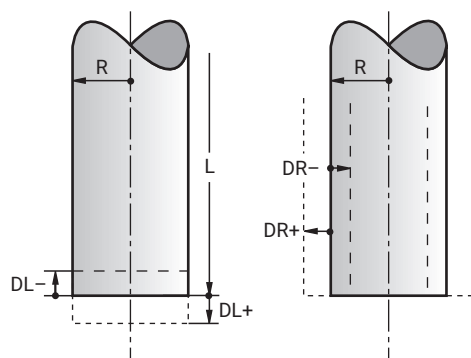
- Delta values within the tool table serve for permanent tool compensation that is required (e.g., due to wear).

These delta values can be determined, for example, by using a tool touch probe. The control automatically enters the delta values in the tool management.

Further information: User's Manual for Setup and Program Run

- Delta values within a tool call serve for a tool compensation that is active exclusively in the current NC program (e.g., a workpiece oversize).

Further information: "Using TOOL CALL to call a tool", Page 162



Delta values represent deviations from the length and radius of a tool.

A positive delta value enlarges the current tool length or the tool radius. The tool then cuts less material during machining (e.g., for a workpiece oversize).

A negative delta value reduces the current tool length or the tool radius. The tool then cuts more material during machining.

For programming delta values in an NC program, define the value within a tool call or by using a compensation table.

Further information: "Using TOOL CALL to call a tool", Page 162

Delta values within a tool call can also be defined by using variables.

Further information: "Tool data within variables", Page 342

Tool length compensation

The control takes the tool length compensation into account as soon as a tool is called. The control performs tool length compensation only on tools of length $L > 0$.

In tool length compensation, the control takes delta values from the tool table and the NC program into account.

Active tool length = $L + DL_{TAB} + DL_{Prog}$

L: Tool length **L** from the tool table

DL_{TAB}: Tool length delta value **DL** from the tool table

DL_{Prog}: Tool length delta value **DL** from the tool call or the compensation table

The most recently programmed value becomes active.

Further information: "Using TOOL CALL to call a tool", Page 162

NOTICE

Danger of collision!

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

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

Tool radius compensation

The control takes the tool radius compensation into account in the following cases:

- If tool radius compensation **R+** or **R-** is active

Further information: "Tool radius compensation", Page 344

- Within machining cycles

Further information: "Working with cycles", Page 137

In tool radius compensation, the control takes the delta values from the tool table and the NC program into account.

Active tool radius = **R** + **DR**_{TAB} + **DR**_{Prog}

R: Tool radius **R** from the tool table

Further information: User's Manual for Setup and Program Run

DR_{TAB}: Tool radius delta value **DR** from the tool table

Further information: User's Manual for Setup and Program Run

DR_{Prog}: Tool radius delta value **DR** from the tool call or the compensation table

The most recently programmed value becomes active.

Further information: "Using TOOL CALL to call a tool", Page 162

Tool data within variables

When executing a tool call, the control calculates all tool-specific values and saves them within variables.

Further information: "Preassigned Q parameters", Page 402

Active tool length and tool radius:

Q parameters	Function
Q108	ACTIVE TOOL RADIUS
Q114	ACTIVE TOOL LENGTH

After the control has saved the current values within variables, the variables can be used in the NC program.

Application example

You can use the Q parameter **Q108 ACTIVE TOOL RADIUS** in order to shift the tool center point of the ball-nose cutter to the sphere center using the delta value for the tool length.

```
11 TOOL CALL "BALL_MILL_D4" Z S10000
```

```
12 TOOL CALL DL-Q108
```

This allows the control to monitor the complete tool for collisions and the dimensions used in the NC program can still be programmed with reference to the ball center.

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 changes only the position of the tool in the simulation.

Further information: "Simulation of tools", Page 537

- The machine manufacturer uses the optional machine parameter **progToolCallDL** (no. 124501) to define whether the control will consider delta values from a tool call in the **Positions** workspace.

Further information: "Tool call", Page 162

Further information: User's Manual for Setup and Program Run

14.2 Tool radius compensation

Application

When tool radius compensation is active, the control will no longer reference the positions in the NC program to the tool center point, but to the cutting edge.

Use tool radius compensation to program drawing dimensions without having to consider the tool radius. This lets you use a tool with deviating dimensions without having to modify the program after a tool has broken.

Related topics

- Presets on the tool

Further information: "Presets on the tool", Page 159

Requirements

- Parameters have been defined in tool management

Further information: User's Manual for Setup and Program Run

Description of function

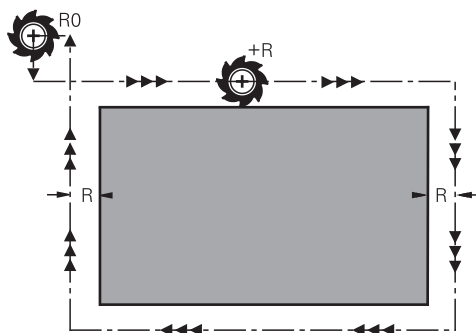
The control takes the active tool radius into account during tool radius compensation. The active tool radius results from the tool radius R and the delta values DR from the tool management and the **NC program**.

Active tool radius = $R + DR_{TAB} + DR_{Prog}$

Further information: "Tool compensation for tool length and tool radius", Page 340

To correct paraxial positioning blocks:

- **R+**: lengthens a paraxial positioning block by the tool radius
- **R-**: shortens a paraxial positioning block by the tool radius
- **R0** positioning using the tool center



Radius-compensated traverse with paraxial movements

Effect

Tool radius compensation is active starting from the NC block in which tool radius compensation is programmed. Tool radius compensation is effective modally and at the end of the block.



Program tool radius compensation only once, allowing for quicker implementation of changes, for example.

The control resets tool radius compensation in the following cases:

- Positioning block with **R0**
- Selection of a new NC program

Notes

NOTICE

Danger of collision!

The control needs safe positions for contour approach and departure. These positions must enable the control to perform compensating movements when radius compensation is activated and deactivated. Incorrect positions can lead to contour damage. Danger of collision during machining!

- ▶ Program safe approach and departure positions at a sufficient distance from the contour
- ▶ Consider the tool radius

- When tool radius compensation is active, the control displays an icon in the **Positions** workspace.
Further information: User's Manual for Setup and Program Run
- Between two NC blocks with different tool radius compensations **R+** and **R-**, there must be at least one traversing block in the working plane without tool radius compensation **R0**.
- If radius compensation is active and you execute the following functions, the control aborts program run and displays an error message:
 - **CALL PGM**
 - Cycle **12 PGM CALL**

15

Files

15.1 File management

15.1.1 Basic information

Application

In the file manager, the control displays drives, folders, and files. You can, for example, create or delete folders or files and can also connect drives.

The file management function encompasses the **Files** operating mode as well as the **Open File** workspace and the windows with the same name.

Related topics

- Data backup
- Connecting network drives

Further information: User's Manual for Setup and Program Run

Description of function

Icons, buttons and shortcuts

The file management provides the following icons, buttons, and shortcuts:

Icon, button, or shortcut	Meaning
	Sort in ascending order
	Sort in descending order
	Activate custom filter Further information: "Creating or changing a user-defined filter", Page 357 Only in the Files operating mode
	Back
	Open or close the History The control opens a selection menu with up to 20 previous paths since switch-on.
 [ALT] + [O]	Edit The control displays a navigation path as editable text.
 ENT	Finish editing The control saves the changes to the path and calls the new path.
	Refresh
	Favorite If you add a favorite, then the control displays this icon next to the file or the folder.
	Eject Eject USB device
	With end of file , the control indicates that the complete file is visible in the preview area.

Icon, button, or shortcut	Meaning
	The control only displays a part of the file in the preview area.
New folder	Create new folder
New file	Create new file
	 You create a new table in the Tables operating mode. Further information: "The Tables operating mode", Page 580
File functions	The control opens the context menu. Further information: "Context menu", Page 511 Only in the Files operating mode
Mark [CTRL] + [SPACE]	The control marks the file and opens the action bar. Only in the Files operating mode
 [CTRL] + [Z]	Undo
 [CTRL] + [Y]	Redo
Open in text editor	The control opens the file in the Text editor workspace. Further information: "The Text editor workspace", Page 366 Only in the Files operating mode
Show as document	The control opens the file in the Document workspace. Further information: "The Document workspace", Page 361 Only in the Files operating mode
Open	The control opens the file in the appropriate operating mode or application.
Select in Program Run	The control opens the file in the Program Run operating mode and selects the first NC block. Only in the Files operating mode
Mount network share	Further information: User's Manual for Setup and Program Run Only in the Files operating mode
Update TAB / PGM	Adapt the format and content of files from the iTNC 530 Modify faulty files Further information: "Adapting files", Page 363 Only in the Files operating mode

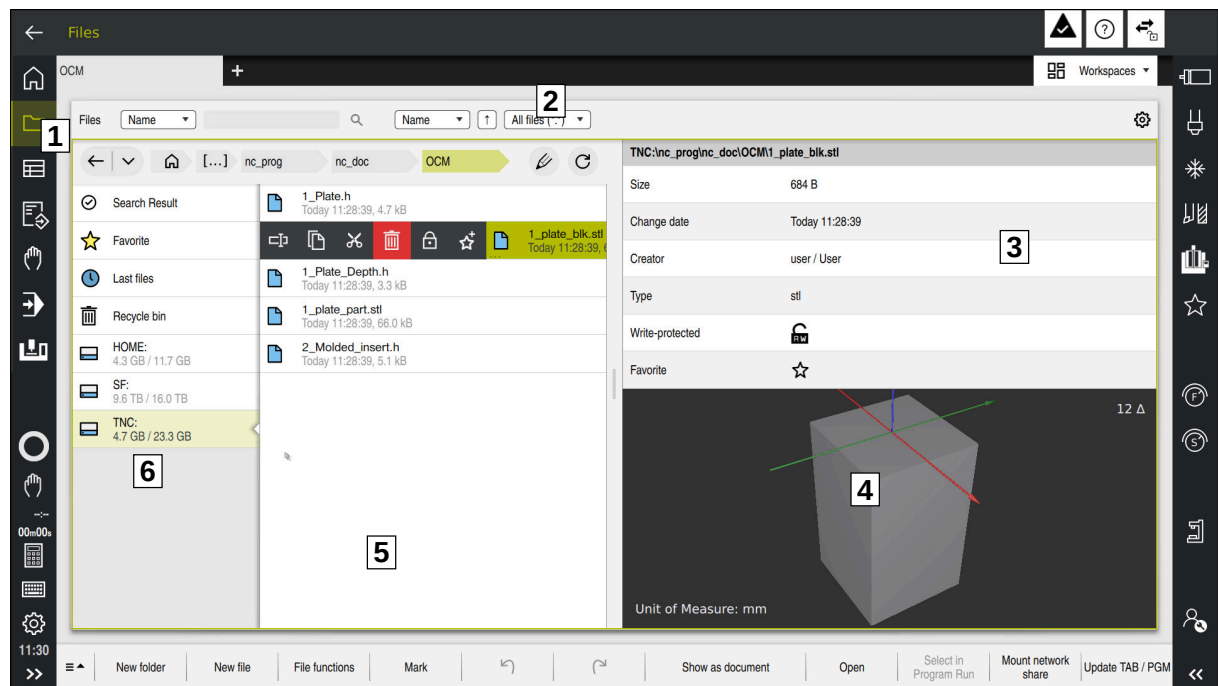
If you swipe a file or folder to the right in the file management, the control displays the following file functions:

Icon	Meaning
	Rename
	Copy
	Cut If you cut a file or a folder, then the control dims the icon of the file or the folder.
	Delete
	Activate write-protection If write-protection is active, then the control displays this icon next to the file or the folder. The optional machine parameter editWhileProtected (no. 105416) allows you to define how the control interprets the write protection in the Editor operating mode.
	Deactivate write-protection
	Add favorite
	Remove favorite

You can also select some of these file functions from the context menu.

Further information: "Context menu", Page 511

File management areas



The **Files** operating mode

- 1 Navigation path
In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level. You can change the path or use the history to open a previous path.
- 2 Title bar
 - Full-text search
Further information: "Full-text search in the title bar", Page 352
 - Sorting
Further information: "Sorting in the title bar", Page 352
 - Filters
Further information: "Filters in the title bar", Page 352
 - Settings
Further information: "Title bar settings", Page 352
- 3 Information area
Further information: "Information area", Page 353
- 4 Preview area
In the preview area the control shows a preview of the selected file; for example, an excerpt from an NC program.
- 5 Content column
In the content column the control shows all folders and files for selection using the navigation column.
The control displays the following status for a file, if applicable:
 - **M**: the file is active in the **Program Run** operating mode
 - **S**: the file is active in the **Simulation** workspace
 - **E**: the file is active in the **Editor** operating mode
- 6 Navigation column
Further information: "Navigation column", Page 353

Full-text search in the title bar

Use the full-text search to find any strings in the names or contents of files. Use the selection menu to choose whether the control searches the names or contents of the files.

Before a search, you first need to choose the path in which the control is to conduct the search. Based on the chosen path, the control only searches within the subordinate structure. In order to refine a search, you can search again within an existing search result.

You can use the ***** character as a placeholder. This placeholder can stand for any single characters or even an entire word. You can also use the placeholder to search for specific file types (e.g., ***.pdf**).

Sorting in the title bar

You can sort folders and files by the following criteria in ascending or descending order:

- **Name**
- **Type**
- **Size**
- **Change date**

If you sort by name or type, the control lists the files alphabetically.

Filters in the title bar

The control provides standard filters and a user-defined filter that can be used to filter by the desired file types.

Further information: "Creating or changing a user-defined filter", Page 357

Title bar settings

The control provides the following settings in the **Settings** window:

- **Show hidden files**
If the toggle switch is active, the control displays hidden files. Names of hidden files start with a period.
- **Show dependent files**
If the toggle switch is active, the control displays dependent files. Dependent files have ***.dep** or ***.t.csv** extensions.
- **Waiting time for network drives (s)**
You can enter a timeout period in seconds. If a network drive cannot be accessed before this period has elapsed, the control will show the network drive and the paths or files on this network drive dimmed. As soon as the network drive can be accessed again, all functions will be available again.
- **Default sorting**
You can define criteria based on which the control will sort the files in file management by default after the next restart.

Information area

In the information area the control shows the path of the file or folder.

Further information: "Path", Page 354

Depending on which element is selected, the control displays the following additional information:

- **Size**
- **Change date**
- **Creator**
- **Type**

You can select the following functions in the information area:

- **Activate and deactivate write-protection**
The optional machine parameter **editWhileProtected** (no. 105416) allows you to define how the control interprets the write protection in the **Editor** operating mode.
- **Add or remove favorites**

Navigation column

The navigation column offers the following possibilities for navigation:

- **Search Result**
The control displays the results of the full-text search. If there was no search, or if nothing was found, then this area is empty.
- **Favorite**
The control displays all folders and files that you have marked as favorites.
- **Last files**
The control displays the 15 most recently opened files.
- **Recycle bin**
The control moves deleted folders and files to the recycle bin. You can use the context menu to restore these files or empty the recycle bin.
Further information: "Context menu", Page 511
When the recycle bin is full, the control will display a confirmation prompt. If you still delete a file, the control will permanently delete this file. You cannot undo this action.
- **Drives (e.g., TNC:)**
The control displays internal and external drives (e.g., a USB device).
The control displays the occupied and total memory space under each drive.

Permitted characters

You can use the following characters for the names of drives, folders, files, etc.:

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 _ -

Only use characters that are shown here; otherwise problems might occur (for example, during data transmission).

The following characters have specific functions, and must therefore not be used in a name:

Character	Function
.	Separates the file name from the file type
\ /	Separates between drive, folder, and file in the path
:	Separates the drive names

Name

When you create a file, you first define its name. The file name is followed by the file name extension, consisting of a period and the file type.

Path

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

Absolute path

An absolute path specifies the exact position of a file. The path begins with the drive and then goes through the folder structure in sequence all the way to the file (e.g., **TNC:\nc_prog\mdi.h**). If the file being called has been moved, then a new absolute path must be entered.

Relative path

A relative path specifies the position of a file in relation to the file that is calling it. The path goes through the folder structure in sequence all the way to the file, starting from the file that is calling it (e.g., **demo\reset.H**). If a file has been moved, then a new relative path must be entered.

File types

You can use uppercase or lowercase letters to define the file type.

HEIDENHAIN-specific file types

The control can open the following HEIDENHAIN-specific file types:

File type	Application
H	NC program written in HEIDENHAIN Klartext Further information: "Contents of an NC program", Page 114
D	Table with workpiece datums Further information: "Datum table *.d", Page 610
DEP	Automatically generated table with data that depend on the NC program (e.g., the tool usage file) Further information: User's Manual for Setup and Program Run
PNT	Table with machining positions (e.g., for the machining of irregular point patterns) Further information: "Point table *.pnt", Page 609
PR	Table with workpiece presets Further information: User's Manual for Setup and Program Run
TAB	Freely definable table (e.g., for protocol files or as WMAT and TMAT tables for automatic calculation of cutting data) Further information: "Freely definable tables *.tab", Page 606 Further information: "Cutting data calculator", Page 518
TCH	Table with the assignment of the tool magazine Further information: User's Manual for Setup and Program Run
T	Table with tools for all technologies Further information: User's Manual for Setup and Program Run
TP	Table with touch probes (#17 / #1-05-1) Further information: User's Manual for Setup and Program Run
TNCBCK	File for data backup and restoration Further information: User's Manual for Setup and Program Run
EXP	Configuration file for saving and importing configurations of the control interface Further information: User's Manual for Setup and Program Run

The control opens these file types with an internal application or with a HEROS tool.

Standardized file types

The control can open the following standardized file types:

File type	Application
CSV	Text file for saving or exchanging simple structured data Further information: "Importing and exporting of table contents", Page 597
XLSX (XLS)	File type for various spreadsheet programs (e.g., Microsoft Excel)
DXF	2D CAD files
IGS/IGES STP/STEP	3D CAD files Further information: User's Manual for Setup and Program Run
CHM	Help files in compiled or compressed format
CFG	Configuration files of the control Further information: User's Manual for Setup and Program Run
CFT	3D data of a parameterizable tool-carrier template Further information: User's Manual for Setup and Program Run
CFX	3D data of a geometrically determined tool carrier Further information: User's Manual for Setup and Program Run
HTM/HTML	Text file with structured content of a website that can be opened in a browser (e.g., the integrated product aid) Further information: "User's Manual as integrated product aid: TNCguide", Page 37
XML	Text file with hierarchically-structured data
PDF	Document format that visually reproduces the original file identically, regardless of the source application
BAK	Data-backup file Further information: User's Manual for Setup and Program Run
INI	Initialization file (e.g., can contain program settings)
A	Format file (e.g., for defining the screen output format in connection with FN 16)
TXT	Text file (e.g., for saving the results of measurement cycles in connection with FN 16)
SVG	Picture format for vector graphics
BMP	Picture formats for pixel graphics
GIF	By default, the control uses the PNG format for screenshots
JPG/JPEG PNG	Further information: User's Manual for Setup and Program Run
OGG	Container file format for the OGA, OGV, and OGX media types
ZIP	Container file format that collects multiple compressed files.

The control opens some of these file types with the HEROS tools.

Further information: User's Manual for Setup and Program Run

Creating or changing a user-defined filter

To create or edit a user-defined filter in file management:

- ▶ Select the **Files** operating mode, if required
- ▶ Open the standard filters selection menu
- ▶ Select **User-defined**
- ▶ The control displays an input field next to the selection menu.
- ▶ Enter the desired file types in the input field (e.g., ***.h, *.txt**)
- 
 - ▶ Confirm your input and activate the filter
 - ▶ The control displays all files of the specified file types and takes over the text to the selection menu.



The user-defined filter will remain active only until you shut down the control.

Further information: "Filters in the title bar", Page 352

Notes

- The control has 7 GB of disk space. The maximum size of any file is limited to 2 GB.
- When you open an NC program, the control requires free disk space that is three times the file size of the NC program.
- When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode.

In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.

Further information: "The Tables operating mode", Page 580

- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). These characters can cause problems when inputting or reading data in conjunction with SQL commands.

Further information: "Table access with SQL statements", Page 451

- If the cursor is within the content column, you can start inputting through the keyboard. The control opens a separate input field and automatically searches for the entered string. If it finds a file or folder with that string, then the control moves the cursor to it.
- If you exit an NC program by pressing the **END BLK** key, the control opens the **Add** tab. The cursor is on the NC program that was just closed.
If you press the **END BLK** key again, the control opens the NC program again with the cursor on the last selected line. With large files, this behavior can cause a delay.
If you press the **ENT** key, the control always opens an NC program with the cursor on line 0.
- The control creates dependency files with the ***.dep** extension for the tool-usage file (e.g., in order to perform a tool usage test).
- In the machine parameter **createBackup** (no. 105401) the machine manufacturer defines whether the control creates a backup file when saving an NC program. Please note that these backup files will take up disk space.

- Even if the inch unit of measure is active in the control or NC program, the control will interpret dimensions of 3D files in mm.
- In the following situations, the control will display a message that an opened NC program was deleted:
 - File renamed, moved, or deleted
 - Folder of the file renamed, moved, or deleted

Hints about copied files

- If you copy a file and then paste it to the same folder, the control adds the suffix **_1** to the file name. The control increments the number sequentially for each consecutive copy.
- If you paste a file to another folder and that folder contains a file with the same name, the control opens the **Insert file** window. The control displays the path of the two files and provides the following options:
 - Replace existing file
 - Skip copied file
 - Add suffix to file name

You can also apply the selected option to all such cases.

15.1.2 The Open File workspace

Application

In the **Open File** workspace you select or create files, for example.

Description of function

The **Open File** workspace can be opened by the icons below, depending on the active operating mode:

Icon	Meaning
	Add in the Tables and Editor operating modes
	Open File in the Program Run operating mode

Icons and buttons

The **Open File** workspace provides the following icons and buttons:

Icon or button	Meaning
	Show or hide the File preview
New folder	Create new folder Only in the Tables and Editor operating modes
New file	Create new file Only in the Tables and Editor operating modes
Use in simulation	Use the table for simulation Only in the Editor operating mode if tool tables, preset tables, or datum tables have been selected
Open	Open the selected file

15.1.3 The Quick selection workspaces

Application

In the **Quick selection new table** and **Quick selection new file** workspaces, you can create files or open existing files, depending on the active operating mode.

Description of function

You can open the workspaces by using the **Add** function in the operating modes below:

- **Tables**

Further information: "The Quick selection new table workspace", Page 360

- **Editor**

Further information: "The Quick selection new file workspace", Page 360

Further information: "Icons on the control's user interface", Page 73

The Quick selection new table workspace

The **Quick selection new table** workspace makes the following buttons available:

- **Create new table**

Further information: "The Create new table window", Page 584

- **Tool management**

- **Pocket table**

- **Presets**

- **Touch probes**

- **Datums**

The **Quick selection new table** workspace contains the following areas:

- **Active tables for machining**

- **Active tables for simulation**

In both areas, the control displays the **Tool management**, **Datums**, and **Presets** buttons. They will open the table that is active in program run or in the simulation. The control displays the **M** or **S** status on the application tab.

The Quick selection new file workspace

The **Quick selection new file** workspace offers the following buttons:

Area	Button
New NC program	■ NC program mm
	■ NC program inch
	■ ISO program mm
	■ ISO program inch
	Further information: "Programming fundamentals", Page 114
New text file	■ Text file with the *.txt extension
	■ Format file with the *.a extension
	Further information: "The Text editor workspace", Page 366

15.1.4 The Document workspace

Application

You can open files for viewing in the **Document** workspace, for example a technical drawing.

Related topics

- Supported file types
Further information: "File types", Page 355
- The **Show as document** button in the **Files** operating mode
Further information: "Icons, buttons and shortcuts", Page 348

Description of function

The **Document** workspace is available in every operating mode and application. If you open a file, then the control displays the same file in all operating modes.

Further information: "Overview of the operating modes", Page 59

The control shows the file path in the file information bar.

You can open the following file types in the **Document** workspace:

- PDF files
The **Document** workspace makes a search function available for PDF files.
- HTML files
- Text files, such as *.txt
- Image files, such as *.png
- Video files (e.g., *.webm)

Further information: "File types", Page 355

You can, for example, transfer dimensions from a technical drawing using the clipboard in the NC program.

Icons in the Document workspace

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

Icon	Meaning
	Open File Further information: "Open file", Page 362
	Open or close the Internet window The Internet window allows entering and calling a URL. You may also bookmark the URL.
	Navigate Navigate between the last opened files
	Refresh (e.g., log file or a touch probe cycle)

When a PDF file is open, the **Document** workspace additionally displays the following icons:

Icon	Meaning
	Activate or deactivate Move If this icon is active, highlighting text with the mouse is not possible. Instead, the visible area can be shifted in any direction with the mouse.
	Navigate Select the previous or the next element Depending on the position of icons, you either navigate between the file pages or the search results.
Page X/X	Current page number and total number of pages
100%	Current size of content Open or close the Scale select menu
	Reset scaling Scaling the content to the full width
	Rotate Rotate the content by 90° anti-clockwise or clockwise

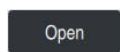
Open file

To open the file in the **Document** workspace:

- ▶ If applicable, open the **Document** workspace



- ▶ Select **Open File**
- ▶ The control opens a selection window with the file manager.
- ▶ Select the desired file



- ▶ Select **Open**
- ▶ The control displays the file in the **Document** workspace.

15.1.5 Adapting files

Application

There are certain files that the TNC7 go cannot open or display correctly (e.g., files from the iTNC 530 or files edited in a text editor). These files can be adapted (i.e., converted with the **Update TAB / PGM** function to suit the control's requirements).

Description of function

NC programs

The control uses the **Update TAB / PGM** function to remove umlauts and checks whether the NC block **END PGM** exists. Without **END PGM**, the NC program is not complete.

Tables

Adaptations may be required for tables of control predecessors or previous software versions TNC7 go.

When adapting tables with the **Update TAB / PGM** function, the control makes the following changes as needed:

- Changing the decimal separator from a comma to a point
- Adding missing table columns
- Taking over all supported tool types and defining all unknown tool types as **Undefined**
- Changing the character encoding of text columns to UTF-8

The following characters are permitted in the **NAME** column of the tool table:

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

Adapting a file

Prepare a backup of the original file before adapting

To adapt (convert) a file (format and contents):



- ▶ Select the **Files** operating mode

Update TAB / PGM

- ▶ Select the desired file
- ▶ Select **Update TAB / PGM**
- The control adapts the file format and content.



The control saves the changes and overwrites the original file.

- ▶ Check the content after adapting

Notes

NOTICE

Caution: Data may be lost!

If you use the **Update TAB / PGM** function, then data may be irrevocably deleted or altered!

- ▶ Create a backup copy prior to converting the file

- Using the **Update TAB / PGM** function, you can adapt all files in an entire folder at the same time.
- The machine manufacturer uses import and update rules to define which adaptations the control is to execute, such as umlaut removal.
- The machine manufacturer uses the optional machine parameter **importFromExternal** (no. 102909) to define for each file type if automatic adaptation is carried out upon copying to the control.

15.1.6 USB devices

Application

A USB device allows transmitting data and saving data externally.

Requirements

- USB 2.0 or 3.0
- USB device with supported file system
The control supports USB devices with the following file systems:
 - FAT
 - VFAT
 - exFAT
 - ISO9660



The control does not support USB devices with other file systems, such as NTFS.

- A ready data interface
Further information: User's Manual for Setup and Program Run
- USB devices are permitted in **SELinux**
Further information: User's Manual for Setup and Program Run
- If user administration is active, the HEROS.NormalUser role
Further information: User's Manual for Setup and Program Run

Description of function

The control displays a USB device as a drive in the navigation column of the **Files** operating mode or of the **Open File** workspace.

The control automatically detects USB devices. If you connect a USB device with a file system that is not supported, the control generates an error message.

Before executing an NC program saved on the USB device, the file must be transferred to the control hard disk.

When transmitting large files, the control displays the data transmission progress at the bottom of the navigation and content column.

Removing a USB device

To remove a USB device:



- ▶ Select **Eject**
- The control opens a pop-up window and asks whether you want to eject the USB device.



- ▶ Select **OK**
- The control shows the message **The USB device can be removed now.**

Notes

NOTICE

Caution: Danger due to manipulated data!

If you execute NC programs directly from a network drive or a USB device, you have no control over whether the NC program has been changed or manipulated. In addition, the network speed can slow down the execution of the NC program. Undesirable machine movements or collisions may result.

- ▶ Copy the NC program and all called files to the **TNC:** drive

NOTICE

Caution: Data may be lost!

Always remove a connected USB device properly, otherwise data may be damaged or deleted!

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

- If an error message is displayed when connecting a USB device, check the setting in the **SELinux** security software.

Further information: User's Manual for Setup and Program Run

- If the control displays an error message when using a USB hub, ignore and acknowledge the message with the **CE** key.
- Prepare a backup of the files on the control at regular intervals.

Further information: User's Manual for Setup and Program Run

15.2 The Text editor workspace

Application

Use the **Text editor** workspace to create and edit text files.

Related topics

- File types
Further information: "File types", Page 355
- Displaying text files in the **Document** workspace
Further information: "The Document workspace", Page 361

Description of function

The **Text editor** workspace is available in the **Editor** operating mode.

By default, the control opens the following file types in the **Text editor** workspace:

- Text files, such as ***.txt**
Example: measuring logs output with **FN 16**
- Format files, such as ***.a**
Example: format file for **FN 16**

Further information: "Outputting text formatted with FN 16: F-PRINT", Page 413



Refer to your machine manual.

In the machine parameter **standardEditor** (no. 102902), the machine manufacturer defines the default editor to be used for the file types (e.g., the **Text editor**).

Select the **Open in text editor** function in the context menu of the **Files** operating mode to open any file type in the **Text editor** workspace.

Further information: "Context menu in the Files operating mode", Page 512

Further information: "File types", Page 355

Icons in the Text editor workspace

The following icons are shown in the **Text editor** workspace:

Icon or shortcut	Meaning
 [CTRL] + [F]	Open and close the Search column Further information: "The Search column in the Program and Text editor workspaces", Page 504
	Show or hide the Line number
	Activate or deactivate the Line break When activating the Line break , the control will automatically add line breaks in the text. The control adapts the text length to the workspace size. The text is not truncated, but wraps to the next line, as indicated by the missing line number in the first line.
	Show or hide the Formatting characters If you show formatting characters, the control will replace blanks with symbols, such as dots, for the space characters.

If you show formatting characters, the control will display the following icons:

Icon	Meaning
	Blank space
	Tabulator When you press the TAB key during text entry, the control will insert a blank up to the next tab stop (the maximum is a whole tab width). The tab width can be defined in the editor settings.
	End of line The control indicates the line end by an LF symbol. If Line break is active, the text contents of a text line may be distributed to several lines, depending on the workspace width. This symbol indicates the actual end of the text line.

 The control also supports the **CR LF** or **CR** line ends, which might be included in transmitted files.
The line break behavior is always identical, independent of which symbol is displayed.

Representation of *.a files

The control emphasizes, for example, formatting characters within ***.a** files as follows:

Display mode	Text
Italics	Texts without function (e.g., while making entries)
Bold	Keywords (e.g., M_CLOSE)
Green	Text between quotation marks
Dark green	Comments after *
Blue	Formatting characters (e.g., %F)
Purple	Variables (e.g., QS parameters)
Wavy red underlines	For faulty FN 16 texts (e.g., texts without quotation marks)

NC editor settings

In the **NC editor settings** window, you can influence contents shown in the **Text editor** workspace as well as the control's behavior there. The selected settings are modally effective.

The General information area

The **General information** area contains the following settings:

Setting	Meaning
Automatic saving	Save changes made to the file automatically or manually If this toggle switch is active, the control will save the file automatically when you do the following: ■ Close the file ■ Navigate to another tab ■ Activate another operating mode If the toggle switch is not active, you must save manually. Upon the stated actions, the control asks whether the changes should be saved.
Replace tabs with blanks	If this toggle switch is active, the control will insert a predefined number of blanks instead of a tab character.
Tab width	Number of characters or number of blanks for a tab until the tab stop



If you change the width, the control will also change the width of the existing tabs.

If you inserted blanks instead of a tab character, the control will not update their number after a change of the width.

Notes

- If you mark characters through touch operation, the control displays two marker symbols below the cursor.
- Using the optional machine parameter **editWhileProtected** (no. 105416), you can define whether it is possible to edit a read-only file. By default, you can edit such a file, but only save the changes in a new file with **Save as**. If you set this machine parameter to **FALSE**, the control will dim any editing functions.
- In order to avoid undesired edits to a file, open it in the **Document** workspace.

Further information: "The Document workspace", Page 361

15.3 Programmable file functions

Application

Programmable file functions enable management of files from within the NC program. Files can be opened, copied, relocated, and deleted.

Description of function

Opening a file with OPEN FILE

The **OPEN FILE** function allows you to open a file from within an NC program.

If you define **OPEN FILE**, the control continues the dialog and you can program a **STOP**.

Using this function, the control can open all file types that you can open manually.

Further information: "File types", Page 355

The control opens the file in the HEROS tool last used for this file type. If you have never opened a file of a certain file type and multiple HEROS tools are available, the control will interrupt program run and open the **Application?** window. In the **Application?** window, you can select the HEROS tool the control should use to open the file. The control saves this selection.

Multiple HEROS tools are available for opening the following file types:

- CFG
- SVG
- BMP
- GIF
- JPG/JPEG
- PNG



In order to avoid program run interruptions or having to select an alternative HEROS tool, open a file of the corresponding file type once in the file manager. If the files of a certain file type can be opened in multiple HEROS tools, you can use the file manager to select the HEROS tool to be used for opening files of this file type.

Further information: "File management", Page 348

Input

11 OPEN FILE "FILE1.PDF" STOP

To navigate to this function:

Insert NC function ► All functions ► Selection ► OPEN FILE

The NC function includes the following syntax elements:

Syntax element	Meaning
OPEN FILE	Syntax initiator for the OPEN FILE function
File or Parameter	Path of the file to be opened Fixed or variable path Selection by means of a selection window
STOP	Interrupts the program run or simulation Optional syntax element

Copying, moving and deleting files with FUNCTION FILE

The control offers the functions below for copying, moving and deleting files from an NC program:

NC function	Description
FUNCTION FILE COPY	This function copies a file into a target file. The control substitutes the content of the target file. This function requires specifying the path to both files.
FUNCTION FILE MOVE	This function moves a file to a target file. The control substitutes the content of the target file and deletes the file to be moved. This function requires specifying the path to both files.
FUNCTION FILE DELETE	This function deletes the selected file. This function requires specifying the path to the file to be deleted.

Input

Copying a file

```
11 FUNCTION FILE COPY "FILE1.PDF" TO ; Copy the file from the NC program
   "FILE2.PDF"
```

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Functions ► File functions FILE ► FUNCTION FILE COPY

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE COPY	Syntax initiator for the Open file function
File or Parameter	Path of the file to be copied Fixed or variable path Selection by means of a selection window
TO File or Parameter	Path of the file to be substituted Fixed or variable path Selection by means of a selection window

Moving a file

**11 FUNCTION FILE MOVE "FILE1.PDF"
TO "FILE2.PDF"**

; Move the file from the NC program

To navigate to this function:

**Insert NC function ► All functions ► Special functions ► Functions ► File
functions FILE ► FUNCTION FILE MOVE**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE MOVE	Syntax initiator for the Move file function
File or Parame- ter	Path of the file to be relocated Fixed or variable path Selection by means of a selection window
TO File or Parameter	Path of the file to be substituted Fixed or variable path Selection by means of a selection window

Deleting a file

11 FUNCTION FILE DELETE "FILE1.PDF"

; Delete the file from the NC program

To navigate to this function:

**Insert NC function ► All functions ► Special functions ► Functions ► File
functions FILE ► FUNCTION FILE DELETE**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FILE DELETE	Syntax initiator for the Delete file function
File or Parame- ter	Path of the file to be deleted Fixed or variable path Selection by means of a selection window

Notes

NOTICE

Caution: Data may be lost!

When deleting a file with the **FUNCTION FILE DELETE** function, the control will not put this file into the recycle bin. The control deletes the file once and for all!

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

- There are various ways to select files:
 - Enter the file path
 - Select the file in a selection window
 - Define the file path or name of the subprogram in a QS parameter
If the called file is located in the same directory as the calling file, you may also enter just the file name.
- When applying file functions relating to the calling NC program in a called NC program, the control will display an error message.
- When intending to copy or move a non-existent file, the control displays an error message.
- If the file to be deleted does not exist, the control does not display an error message.

16

Control functions

16.1 Functions for controlling program run

16.1.1 Overview

The control provides the following NC functions for program control:

Syntax	Function	Further information
FUNCTION DWELL	Program singular dwell time	Page 376
FUNCTION FEED DWELL	Program cyclic dwell time	Page 377

16.1.2 Programmed dwell time with FUNCTION DWELL

Application

The **FUNCTION DWELL** function allows you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Related topics

- Cycle **9 DWELL TIME**
Further information: "Cycle 9 DWELL TIME ", Page 379
- Program recurring dwell time
Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 377

Input

11 FUNCTION DWELL TIME10	; Dwell time for 10 seconds
12 FUNCTION DWELL REV5.8	; Dwell time for 5.8 spindle revolutions

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION DWELL	Syntax initiator for singular dwell time
TIME or REV	Duration of dwell time in seconds or spindle revolutions

16.1.3 Cyclic dwell time with FUNCTION FEED DWELL

Application

FUNCTION FEED DWELL allows you to program a cyclic dwell time in seconds, such as for forcing chip breaking.

Related topics

- Program a one-time dwell time

Further information: "Programmed dwell time with FUNCTION DWELL",
Page 376

Description of function

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motions.

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

The control automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

Program **FUNCTION FEED DWELL** immediately prior to the operation you wish to run with chip breaking. Reset the dwell time immediately following the machining with chip breaking.

Input

11 FUNCTION FEED DWELL D-TIME0.5 F-TIME5

; Activate cyclic dwell time: Machine for 5 seconds, dwell for 0.5 seconds

To navigate to this function:

Insert NC function ► Special functions ► Functions ► Dwell time FEED/DWELL ► FUNCTION FEED DWELL

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION FEED DWELL	Syntax initiator for cyclic dwell time
D-TIME or RESET	Define dwell time duration in seconds or reset recurring dwell time
F-TIME	Duration of machining time until the next dwell time in seconds Only if D-TIME is selected

Notes

NOTICE

Caution: Danger to the tool and workpiece!

When the **FUNCTION FEED DWELL** function is active, the control will repeatedly interrupt the feed movement. While the feed movement is interrupted, the tool remains at its current position, and the spindle continues to turn. During thread cutting, this behavior will cause the workpiece to become scrap. There is also a risk of tool breakage during execution!

- Deactivate the **FUNCTION FEED DWELL** function before cutting threads

- You can also reset the dwell time by entering **D-TIME 0**.

16.2 Cycles with control function

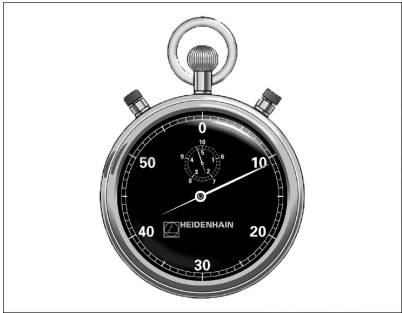
16.2.1 Overview

Cycle		Further information	
9	DWELL TIME ■ Delay execution by the programmed dwell time	DEF-active	Page 379
13	ORIENTATION ■ Rotate spindle to a specific angle	DEF-active	Page 380

16.2.2 Cycle 9 DWELL TIME

ISO programming
G4

Application



Execution of the program run is delayed by the programmed **DWELL TIME**. A dwell time can be used for purposes such as chip breaking.
The cycle takes effect as soon as it has been defined in the NC program. Modal conditions such as spindle rotation are not affected.

Related topics

- Dwell time with **FUNCTION FEED DWELL**
Further information: "Cyclic dwell time with FUNCTION FEED DWELL", Page 377
- Dwell time with **FUNCTION DWELL**
Further information: "Programmed dwell time with FUNCTION DWELL", Page 376

Cycle parameters

Help graphic	Parameter
	Dwell time in secs.? Enter the dwell time in seconds. Input: 0...3600 s (1 hour) in steps of 0.001 seconds

Example

89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5

16.2.3 Cycle 13 ORIENTATION

ISO programming

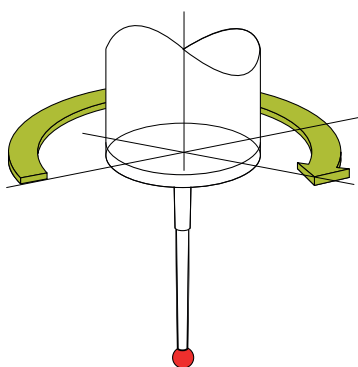
G36

Application



Refer to your machine manual.

Machine and control must be specially prepared by the machine manufacturer for use of this cycle.



The numerical control can control the main machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for purposes such as:

- Tool changing systems with a defined tool change position
- Orientation of the transceiver window of HEIDENHAIN 3D touch probes with infrared transmission

With **M19** or **M20**, the control positions the spindle at the angle of orientation defined in the cycle (depending on the machine).

If you program **M19** or **M20** without having defined Cycle **13** beforehand, the control positions the main spindle at an angle that has been set by the machine manufacturer.

Notes

- Cycle **13** is used internally for Cycles **202** and **204**. Please note that, if required, you must program Cycle **13** again in your NC program after one of the machining cycles mentioned above.

Cycle parameters

Help graphic

Parameter

Orientation angle

Enter the angle relative to the angle reference axis of the working plane.

Input: **0...360**

Example

```
11 CYCL DEF 13.0 ORIENTATION
```

```
12 CYCL DEF 13.1 ANGLE180
```


17

**Miscellaneous
functions**

17.1 Miscellaneous functions M

Application

Use miscellaneous functions to activate or deactivate functions of the control and to influence the behavior of the control.

Description of function

You can define up to four miscellaneous functions **M** at the end of an NC block or in a separate NC block. Once you confirm the entry of a miscellaneous function, the control continues with the dialog and you can define additional parameters, such as **M140 MB MAX**.

In the **Manual operation** application, use the **M** button to activate a miscellaneous function.

Further information: User's Manual for Setup and Program Run

Effects of the miscellaneous functions M

Miscellaneous functions **M** are in effect blockwise or modally. Miscellaneous functions take effect from their point of definition. Other functions or the end of the NC program reset modally effective miscellaneous functions.

Some miscellaneous functions take effect at the start of the NC block and others at the end, regardless of the sequence in which they were programmed.

If you program more than one miscellaneous function in an NC block, the execution sequence is as follows:

- Miscellaneous functions taking effect at the start of the block are executed before those taking effect at the end of the block.
- If more than one miscellaneous function takes effect at the start or end of the block, they are executed in the same sequence as programmed.

STOP function

The **STOP** function interrupts the program run or simulation (e.g., for tool inspection). You can also enter up to four miscellaneous functions **M** in a **STOP** block.

Notes

- Using the optional machine parameter **hideFromUser** (no. 103920), the machine manufacturer can specify the miscellaneous functions that will not be displayed in the selection menus. Thus, the machine manufacturer can hide miscellaneous functions, such as those required for service, in order to keep the selection menus well organized.
- Using the optional machine parameter **forbidManual** (no. 103917), the machine manufacturer defines the miscellaneous functions that are allowed in the "Manual operation" application and are displayed in the selection menu.

Further information: User's Manual for Setup and Program Run

Refer to your machine manual.

17.2 Overview of miscellaneous functions



Refer to your machine manual.

The machine manufacturer can influence the behavior of the miscellaneous functions described below.

M0 to **M30** are standardized miscellaneous functions.

This table shows at what point the miscellaneous functions take effect:

- At the start of the block
- At the end of the block

Function	Effect	Further information
M0 Stop program run and the spindle, switch coolant supply off	■	
M1 Optionally stop program run, optionally stop the spindle, optionally switch the coolant supply off Function depends on the machine manufacturer The function is only effective if the Pause at M1 toggle switch is active.	■	See the User's Manual for Setup and Program Run
M2 Stop program run and the spindle, switch coolant supply off, return to beginning of the program, optionally reset the program information The function depends on the setting by the machine manufacturer in the machine parameter resetAt (no. 100901)	■	
M3 Switch spindle on clockwise	□	
M4 Switch spindle on counterclockwise	□	
M5 Stop the spindle	■	
M8 Switch coolant supply on	□	
M9 Switch coolant supply off	■	
M13 Switch spindle on clockwise, switch coolant supply on	□	
M14 Switch spindle on counterclockwise, switch coolant supply on	□	
M30 Function is identical to M2	■	

Function	Effect	Further information
M89 Call the cycle modally	■	Page 143
M91 Traverse in the machine coordinate system M-CS	□	Page 385
M92 Traverse in the M92 coordinate system	□	Page 387
M99 Call a cycle once per block	■	Page 143
M103 Reduce feed rate for infeed movements	□	Page 388
M136 Interpret feed rate as mm/rev	□	Page 388
M137 Reset M136	■	
M140 Retract in the tool axis	□	Page 389

17.3 Miscellaneous functions for coordinate entries

17.3.1 Traversing in the machine coordinate system M-CS with M91

Application

You can use **M91** to program machine-based positions, such as for moving to safe positions. The coordinates of positioning blocks with **M91** are in effect in the machine coordinate system **M-CS**.

Further information: "Machine coordinate system M-CS", Page 310

Description of function

Effect

M91 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 Z+250 R0 FMAX M91	; Approach a safe position in the tool axis
13 X-200 R0 FMAX M91	; Approach a safe position in the X axis
14 Y-200 R0 FMAX M91	; Approach a safe position in the Y axis
15 LBL 0	

Here **M91** is in a subprogram in which the control moves the tool to a safe position by first moving in the tool axis and then in the X axis and Y axis.

Since the coordinates refer to the machine datum, the tool always moves to the same position. That way, regardless of the workpiece preset, the subprogram can be repeatedly called in the NC program.

Without **M91** the control references the programmed coordinates to the workpiece preset.

Further information: "Presets in the machine", Page 108



The coordinates for a safe position depend on the machine!
The machine manufacturer defines the position of the machine datum.

Notes

- If you program incremental coordinates in an NC block with the miscellaneous function **M91**, then these coordinates are relative to the last position programmed with **M91**. For the first position programmed with **M91**, the incremental coordinates are relative to the current tool position.
- The control considers any active tool radius compensation when positioning with **M91**.

Further information: User's Manual for Setup and Program Run

- The control uses the tool carrier reference point when positioning in the tool axis.

Further information: "Presets in the machine", Page 108

- The following position displays refer to the machine coordinate system **M-CS** and show the values defined with **M91**:

- **Nominal reference position (RFNOML)**

- **Actual reference position (RFACTL)**

Further information: User's Manual for Setup and Program Run

- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset to the simulation. In this constellation you can simulate traverse movements with **M91**.

Further information: "The Visualization options column", Page 530

- In the machine parameter **refPosition** (no. 400403) the machine manufacturer defines the position of the machine datum.

17.3.2 Traversing in the M92 coordinate system with M92

Application

You can use **M92** to program machine-based positions, such as for moving to safe positions. The coordinates of positioning blocks with **M92** are relative to the **M92** datum and are in effect in the **M92** coordinate system.

Further information: "Presets in the machine", Page 108

Description of function

Effect

M92 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 Z+250 R0 FMAX M92	; Approach a safe position in the tool axis
13 X-200 R0 FMAX M92	; Approach a safe position in the X axis
14 Y-200 R0 FMAX M92	; Approach a safe position in the Y axis
15 LBL 0	

Here **M92** is in a subprogram in which the tool moves to a safe position by first moving in the tool axis and then in the plane.

Since the coordinates refer to the **M92** datum, the tool always moves to the same position. That way, regardless of the workpiece preset, the subprogram can be repeatedly called in the NC program.

Without **M92** the control references the programmed coordinates to the workpiece preset.

Further information: "Presets in the machine", Page 108



The coordinates for a safe position depend on the machine!
The machine manufacturer defines the position of the **M92** datum.

Notes

- The control considers any active tool radius compensation when positioning with **M92**.

Further information: "Tool radius compensation", Page 344

- The control uses the tool carrier reference point when positioning in the tool axis.

Further information: "Presets in the machine", Page 108

- In the **Editor** operating mode, use the **Workpiece position** window to apply the current workpiece preset to the simulation. In this constellation you can simulate traverse movements with **M92**.

Further information: "The Visualization options column", Page 530

- In the optional machine parameter **distFromMachDatum** (no. 300501) the machine manufacturer defines the position of the **M92** datum.

17.4 Miscellaneous functions for path behavior

17.4.1 Reducing the feed rate for infeed movements with M103

Application

With **M103** the control performs infeed movements at a lower feed rate, for example when plunging. You use a percentage factor to define the feed-rate value.

Description of function

Effect

M103 is in effect for straight lines in the tool axis at the start of the block.
In order to reset **M103**, program **M103** without a defined factor.

Application example

10 X+20 R0 F1000	; Move in the X axis
11 Y+20 R0	; Move in the Y axis
12 Z-2.5 R0 M103 F20	; Activate feed rate reduction and move at reduced feed rate

In the first NC block the control positions the tool in the working plane.

In NC block **12** the control activates **M103** with the percent factor 20 and then performs the infeed movement in the Z axis at a reduced feed rate of 200 mm/min.
Without **M103** the infeed movement is performed at the programmed feed rate.

Input

If you define **M103**, the control continues the dialog and prompts you for the factor **F**.

Note

The infeed rate F_Z is calculated from the last programmed feed rate F_{Prog} and the percent factor **F**.

$$F_Z = F_{Prog} \times F$$

17.4.2 Interpreting the feed rate as mm/rev with M136

Application

With **M136**, the control interprets the feed rate as millimeters per spindle revolution. The feed rate depends on the spindle speed.

Description of function

Effect

M136 takes effect at the start of the block.
In order to reset **M136**, program **M137**.

Application example

11 M136	; Switch interpretation of the feed rate to mm/rev

With **M136** the control interprets the feed rate as millimeters per revolution.
Without **M136** the control interprets the feed rate as millimeters per minute.

Notes

- In NC programs based on inch units, **M136** is not allowed in combination with **FU** or **FZ**.
- When you move the axes while **M136** is active, the control will display the feed rate in mm/rev in the **Positions** workspace and on the **POS** tab of the **Status** workspace.
Further information: User's Manual for Setup and Program Run
- **M136** is not possible in combination with an oriented spindle stop. The control cannot calculate the feed rate because the spindle does not rotate during an oriented spindle stop, such as when tapping.

17.4.3 Retracting in the tool axis with M140

Application

With **M140** the control retracts the tool in the tool axis.

Description of function

Effect

M140 is in effect blockwise and takes effect at the start of the block.

Application example

11 LBL "SAFE"	
12 M140 MB MAX	; Retract by the maximum distance in the tool axis
13 X+350 R0 FMAX M91	; Approach a safe position in the X axis
14 Y+400 R0 FMAX M91	; Approach a safe position in the Y axis
15 LBL 0	

Here **M140** is in a subprogram in which the control moves the tool to a safe position. With **M140 MB MAX** the control retracts the tool by the maximum distance in the positive direction in the tool axis. The control stops the tool before reaching a limit switch or a collision object.

In the next NC blocks the control moves the tool to a safe position in the X axis and Y axis.

Without **M140** the control does not execute a retraction.

Input

If you define **M140**, the control continues the dialog and prompts you for the retraction distance **MB**. You can program the retraction distance as a positive or negative incremental value. With **MB MAX** the control retracts the tool in the positive direction in the tool axis before reaching a limit switch or a collision object.

After **MB** you can define a feed rate for the retraction movement. If you do not define a feed rate, the control retracts the tool at rapid traverse.

Notes

- With **M140 MB MAX** the control retracts the tool only in the positive direction in the tool axis.
- If you define a negative value for **MB**, the control retracts the tool in the negative direction in the tool axis.
- The control gleans the necessary information about the tool axis for **M140** from the tool call.
- In the optional machine parameter **moveBack** (no. 200903) the machine manufacturer defines the distance to a limit switch or a collision object upon a maximum retraction with **MB MAX**.

Definition

Abbreviation	Definition
MB (move back)	Tool axis retraction

18

**Programming with
variables**

18.1 Overview of variable programming

Variables are placeholders for numbers and texts which can take on different values. You can use the variables to, for example, carry out calculations or create variable logs in the NC program.

The control provides the following options for variable programming:

Topic	Further information
Variables	Page 393
String functions	Page 432
Format strings	Page 439
Defining the counter with FUNCTION COUNT	Page 443
Program defaults for cycles	
Table access with SQL statements	Page 451

18.2 Variables: Q, QL, QR, QS parameters and named parameters

18.2.1 Basics

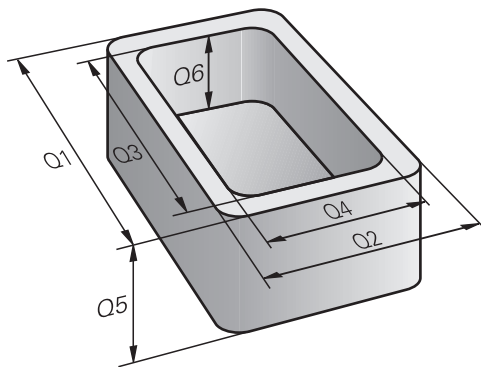
Application

You can use the Q, QL, QR, QS and named parameters of the control, also referred to as variables, to take measurement results into account dynamically within calculations while machining.

For instance, you can program the following syntax elements variably:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

This means that the same NC program can be used for different workpieces and values have to be changed in only one central place.



Description of function

Variables are composed of the variable name and the variable value.

The control provides the following types of variables:

Variable type	Category	Example	Further information
Q parameters	Numerical parameter	Q10 = +10	Page 395
QL parameters	Numerical parameter	QL10 = +10	Page 395
QR parameters	Numerical parameter	QR10 = +10	Page 395
QS parameters	String parameters	QS10 = "123"	Page 395
Named parameters	Numerical parameter	{DEPTH} = -10	Page 396
	or String parameters	or {TOOL} = "MILL_D8"	

Variable name

The control displays the variable name to the left of the equal sign.

For the different variable types, the variable name is as follows:

Variable	Contents
Q, QL, QR, or QS parameters	The variable name of these parameters consists of letters and numbers (e.g., Q10 or QS10). The control defines the letters for the variable type.
Named parameter	The variable name of named parameters consists of a freely chosen designation enclosed in curly brackets (e.g., {DEPTH_1}). The variable name can include letters, numbers and under-scores, but it must start with a letter. For named parameters you can define a variable name with up to 31 characters.

Variable value

The control shows the variable value to the right of the equal sign.

The possible variable value differs as follows, depending on the category:

Category	Contents
Numerical parameter	Numerical parameters can be assigned a variable value between -999 999 999 and +999 999 999. The input range is limited to no more than 16 characters; up to nine of these characters can precede the comma. The control can calculate numbers up to a magnitude of 10^{10}.
String parameters	String parameters can be assigned a variable value of up to 255 characters. The control displays the variable values of string parameters in quotation marks (e.g., "TOOL_3"). The following characters are allowed in the variable value of string parameters: A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 ; ! # \$ % & ' () + , - . / : < = > ? @ [] ^ _ ` *



The control provides the **FMT** syntax element for QS parameters and named parameters to define format strings. Using format strings, you no longer have to convert numerical values or concatenate strings.

Further information: "Format strings", Page 439

Q parameters

Q parameters affect all NC programs in the control's memory.

Q and QS parameters between 0 and 99 have a local effect in macros and cycles.

This means that the control will not return changes to the NC program.

The control provides the following Q parameters:

Variable range	Meaning
0 to 99	User-defined Q parameters, if there are no overlaps with the HEIDENHAIN SL cycles
100 to 199	Q parameters for special functions on the control that can be read by user-defined NC programs or by cycles
200 to 1199	Q parameters for functions defined by HEIDENHAIN (e.g., cycles)
1200 to 1599	Q parameters for functions defined by the machine manufacturer (e.g., cycles)
1600 to 1999	User-defined Q parameters

QL parameters

QL parameters are active locally within an NC program.

The control provides the following QL parameters:

Variable range	Meaning
0 to 499	User-defined QL parameters

QR parameters

QR parameters affect all NC programs in the control's memory; they are retained even after a restart of the control.

The control provides the following QR parameters:

Variable range	Meaning
0 to 99	User-defined QR parameters
100 to 199	QR parameters for functions defined by HEIDENHAIN (e.g., cycles)
200 to 499	QR parameters for functions defined by the machine manufacturer (e.g., cycles)

QS parameters

QS parameters affect all NC programs in the control's memory.

QS parameters between 0 and 99 have a local effect within macros and cycles. This means that the control will not return changes to the NC program.

The control provides the following QS parameters:

Variable range	Meaning
0 to 99	User-defined QS parameters, if there are no overlaps with the HEIDENHAIN cycles
100 to 199	QS parameters for special functions on the control that can be read by user-defined NC programs or by cycles
200 to 1199	QS parameters for functions defined by HEIDENHAIN (e.g., cycles)
1200 to 1399	QS parameters for functions defined by the machine manufacturer (e.g., cycles)
1400 to 1999	User-defined QS parameters

Named parameter

Named parameters, just like QL parameters, take effect locally in the NC program. You can define named parameters as numerical parameters or as string parameters.

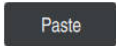
Defining the variables

The following instructions show how to define the various variables using the **Form** column.

Defining Q, QL, or QR parameters

To define a Q, QL, or QR parameter:

- 





 - ▶ Select **Insert NC function**
 - The control opens the **Insert NC function** window.
 - ▶ Select **Formula**
 - ▶ Select **Paste**
 - The control inserts the NC function.
 - ▶ Tap or click in the input field **Parameter number for result?**
 - ▶ If necessary, define the variable type (e.g., **QL**)
 - ▶ Define the variable name (e.g., **21**)
 - ▶ Tap or click in the input field **Formula expression for assignment**
 - ▶ Open **Keyboard** for formula input
 - ▶ Define the variable value, if necessary with arithmetic operations and string functions (e.g., **100 - Q23**)
 - ▶ End the NC block
 - ▶ Run the NC block if necessary
 - After running the NC block, the control saves the entered value in the target parameter.



The control provides various arithmetic operations and string functions in the NC function **Formula**.

Further information: "Formulas in the NC program", Page 427

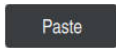
Further information: "String functions", Page 432

Defining QS parameters

To define a QS parameter:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.



- ▶ Select **String formula**
- ▶ Select **Paste**
- The control inserts the NC function.
- ▶ Tap or click in the input field **Form a text string and assign**
- ▶ Define the variable name (e.g., **21**)
- ▶ In the input field **Formula expression for assignment**, tap or click between the quotation marks



- ▶ Open **Keyboard** for formula input
- ▶ Define the variable value, if necessary with string functions (e.g., **"diameter =" || QS23**)
- ▶ End the NC block
- ▶ Run the NC block if necessary
- After running the NC block, the control saves the entered value in the target parameter.



The control provides various string functions in the NC function **String formula**.

Further information: "String functions", Page 432

If you enter **FMT** before the variable value, you define a format string.

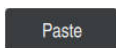
Further information: "Format strings", Page 439

Defining named parameters

To define a named parameter:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.



- ▶ Select **Variable**
- ▶ Select **Paste**
- The control inserts the NC function.
- ▶ Tap or click in the input field **Variable**
- ▶ Define the variable name (e.g., **{measured_value}**)



- ▶ Open **Keyboard** for formula input
- ▶ Define the variable value, if necessary with arithmetic operations and string functions (e.g., **100 - Q23**)
- ▶ End the NC block
- ▶ Run the NC block if necessary
- After running the NC block, the control saves the entered value in the target parameter.



The control provides various arithmetic operations and string functions in the NC function **Variable**. The options depend on whether you define a string parameter or a numerical parameter.

Further information: "Formulas in the NC program", Page 427

Further information: "String functions", Page 432

If, with a string parameter, you enter **FMT** before the variable value, you define a format string.

Further information: "Format strings", Page 439

Notes

NOTICE

Danger of collision!

HEIDENHAIN cycles, machine manufacturer cycles and third-party functions use variables. You can also program variables within NC programs. Using variables outside the recommended ranges can lead to intersections, resulting in undesired behavior. Danger of collision during machining!

- ▶ Only use variable ranges recommended by HEIDENHAIN
- ▶ Do not use pre-assigned variables
- ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer and third-party providers
- ▶ Check the machining sequence using the simulation

NOTICE

Caution: Significant property damage!

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

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

Further information: "Preassigned Q parameters", Page 402

- You can enter fixed and variable values mixed in the NC program.
- You can use the **Q** key to create an NC block to assign a value to a variable. If you press the key again, the control changes the variable type in the order **Q QL QR**.
On the virtual keyboard, this procedure only works with the **Q** key in the NC functions area.

Further information: "Virtual keyboard of the control bar", Page 493

- With the **SET UNDEFINED** syntax element you assign the **undefined** status to the variables.

For example, if you program a position using an undefined Q parameter, the control will ignore this movement.

If you use an undefined variable in arithmetic operations in the NC program, the control displays an error message and stops program run.

Further information: "Assigning the Undefined status to a variable", Page 406

- The control saves numerical values internally in a binary number format (standard IEEE 754). Due to the standardized format used, some decimal numbers cannot be represented with a binary value that is 100% exact (rounding error).

If you use calculated variable values for jump commands or positioning moves, you must keep this in mind.

- For each variable type, you can define the variable range or variables that the control will display on the **QPARA** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Notes on QR parameters and backup

The control saves QR parameters within a backup.

If the machine manufacturer did not define a specific path, the control saves the QR parameters in the following path: **SYS:\runtime\sys.cfg**. The **SYS:** partition will only be backed up in full backups.

Machine manufacturers can use the following optional machine parameters to specify the paths:

- **pathNcQR** (no. 131201)
- **pathSimQR** (no. 131202)

If the machine manufacturer used the optional machine parameters to specify a path on the **TNC:** partition, you can perform a backup with the **NC/PLC Backup** functions without entering a code number.

Further information: User's Manual for Setup and Program Run

18.2.2 The Q parameter list window

Application

In the **Q parameter list** window, you can view and edit all values of all variables, if required.

Related topics

- Fundamentals of variables

Further information: "Basics", Page 393

- The **QPARA** tab in the **Status** workspace

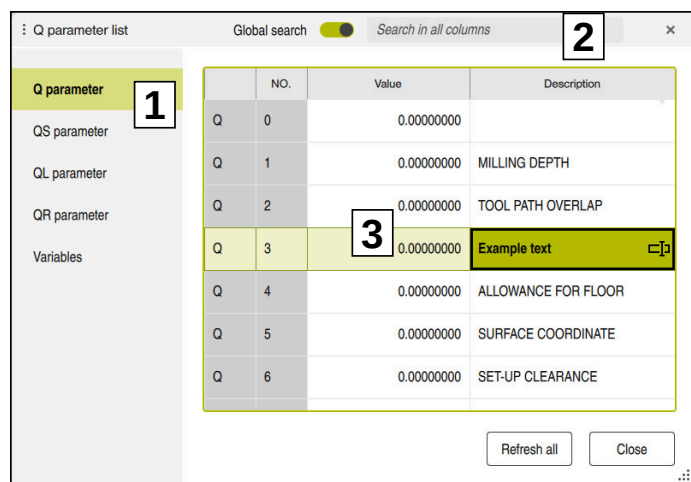
Further information: User's Manual for Setup and Program Run

Description of function

You can open the **Q parameter list** window with the **Q info** button in the following operating modes:

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

In the **Manual** operating mode, you can also use the **Q** key to open the window.



The **Q parameter list** window with the Q parameter values

The **Q parameter list** window shows the following areas:

- 1 Variable type
You can select the variable type shown by the control such as Q parameters.
- 2 Search
By default, the control searches in all columns. If you deactivate the **Global search** toggle switch, the search will be restricted to the currently selected column.
You can begin the search by entering a character as soon as the **Q parameter list** window opens.
- 3 Contents
Depending on the variable type, the control shows the following information:
 - Variable type
 - Number or name
 - Value
If the cell in the **Value** column is highlighted in white, you can edit its value.
 - Description
Texts that are preassigned by HEIDENHAIN, defined by the machine manufacturer or entered by the operator
Further information: "Options for descriptive texts", Page 401

Options for descriptive texts

Upon a long-press gesture or a right-click the control offers the following options for descriptive texts:

- **Default configuration**
The control shows the descriptive text preassigned by HEIDENHAIN.
- **Machine manufacturer configuration**
The control shows the language-sensitive descriptive text defined by the machine manufacturer.
This selection is available only if the machine manufacturer has stored a descriptive text for this variable.
- **Edit**
The control opens the input field and you can enter a descriptive text.
You can also activate the input field with a double-tap or double-click.

If the machine manufacturer has stored descriptive texts, the control shows the **Machine manufacturer configuration** by default.

The control displays the selected text until you select a different option.

Icons

The **Q parameter list** window contains the following icons:

Icon	Meaning
	Descriptive text of machine manufacturer is selected
	User-defined descriptive text is selected

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- While the control is executing an NC program, you cannot edit the variables using the **Q parameter list** window. Changes are only possible while program run has been interrupted or aborted.
The control reaches this required status after, for example, an NC block has been executed in **Single Block** mode.
Further information: User's Manual for Setup and Program Run
- The following Q and QS parameters cannot be edited in the **Q parameter list** window:
 - Variable range from 100 to 199, because there might be interferences with special functions in the control.
 - Variable range from 1200 to 1399, because there might be interferences with machine manufacturer-specific functions.
- If you select or enter a different descriptive text, the control does not store the present user-defined descriptive texts.

18.2.3 Preassigned Q parameters

For example, the control assigns the following values to the Q parameters **Q100** to **Q199**:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Measurement results from touch-probe cycles

The control saves the values of the Q parameters **Q108** and **Q114** to **Q117** in the unit of measure used by the active NC program.

Values from the PLC: Q100 to Q107

The control assigns values from the PLC to the Q parameters **Q100** to **Q107**.

Active tool radius: Q108

The control assigns the value of the active tool radius to the Q parameter **Q108**.

The active tool radius is calculated from the following values:

- Tool radius **R** from the tool table
- Delta value **DR** from the tool table
- Delta value **DR** from the NC program, if a compensation table or tool call is used



The control will remember the active tool radius even after a restart of the control.

Further information: User's Manual for Setup and Program Run

Tool axis: Q109

The value of the Q parameter **Q109** depends on the current tool axis:

Q parameters	Tool axis
Q109 = -1	No tool axis defined
Q109 = 0	X axis
Q109 = 1	Y axis
Q109 = 2	Z axis
Q109 = 6	U axis
Q109 = 7	V axis
Q109 = 8	W axis

Further information: "Designation of the axes of milling machines", Page 106

Spindle status: Q110

The value of the Q parameter **Q110** depends on the M function last activated for the spindle:

Q parameters	M function
Q110 = -1	No spindle status defined
Q110 = 0	M3 Switch spindle on clockwise
Q110 = 1	M4 Switch spindle on counterclockwise
Q110 = 2	M5 after M3 Stop the spindle
Q110 = 3	M5 after M4 Stop the spindle

Further information: "Miscellaneous functions", Page 381

Coolant on/off: Q111

The value of the Q parameter **Q111** depends on the M function for the coolant on/off function that was last activated:

Q parameters	M function
Q111 = 1	M8 Switch coolant supply on
Q111 = 0	M9 Switch coolant supply off

Overlap factor: Q112

The control assigns the overlap factor for pocket milling to the Q parameter **Q112**.

Further information: "Milling cycles", Page 277

Unit of measure in the NC program Q113

The value of the Q parameter **Q113** depends on the unit of measure selected in the NC program. In case of program nesting (e.g., with **CALL PGM**), the control will use the unit of measure defined for the main program:

Q parameters	Unit of measure of the main program
Q113 = 0	Metric system (mm)
Q113 = 1	Imperial system (inch)

Tool length: Q114

The control assigns the value of the active tool length to the Q parameter **Q114**.

The active tool length is calculated from the following values:

- Tool length **L** from the tool table
- Delta value **DL** from the tool table
- Delta value **DL** from the NC program, if a compensation table or tool call is used



The control remembers the active tool length even after a restart of the control.

Further information: User's Manual for Setup and Program Run

18.2.4 The Basic arithmetic folder**Application**

In the **Basic arithmetic** folder of the **Insert NC function** window, the control offers the functions **FN 0** to **FN 5**.

You can assign numerical values to variables using the **FN 0** function. You then use a variable instead of the fixed number in the NC program. You can also use preassigned variables (e.g., the active tool radius **Q108**). Using the functions **FN 1** to **FN 5**, you can make calculations with the variable values in your NC program.

Related topics

- Preassigned variables
Further information: "Preassigned Q parameters", Page 402
- Calculations using formulas
Further information: "Formulas in the NC program", Page 427

Description of function

The **Basic arithmetic** folder contains the following functions:

Icon	Function
	FN 0: Assignment Example: FN 0: Q5 = +60 $Q5 = 60$ Assign a value or the Undefined status
	FN 1: Addition Example: FN 1: Q1 = -Q2 + -5 $Q1 = -Q2 + (-5)$ Calculate and assign the sum of two values
	FN 2: Subtraction Example: FN 2: Q1 = +10 - +5 $Q1 = +10 - (+5)$ Calculate and assign the difference of two values.
	FN 3: Multiplication Example: FN 3: Q2 = +3 * +3 $Q2 = 3 * 3$ Calculate and assign the product of two values.
	FN 4: Division Example: FN 4: Q4 = +8 DIV +Q2 $Q4 = 8 / Q2$ Calculate and assign the quotient of two values Restriction: You cannot divide by 0
	FN 5: Square root Example: FN 5: Q20 = SQRT 4 $Q20 = \sqrt{4}$ Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

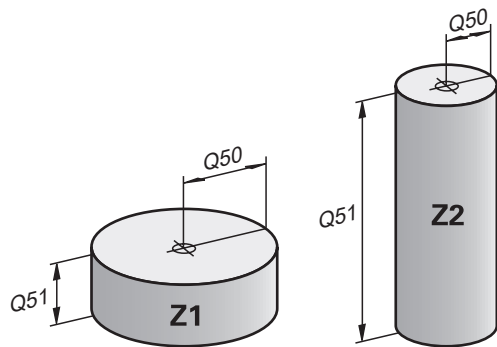
To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, you can use fixed or variable values. The variables and numbers in the equations can be entered with an algebraic sign.

Part families

For part families, for example, you can program the characteristic workpiece dimensions as variables. When machining the individual workpieces, assign a numerical value to each variable.

11 LBL "Z1"	
12 FN 0: Q50 = +30	; Assign the value 30 to the cylinder radius Q50
13 FN 0: Q51 = +10	; Assign the value 10 to the cylinder height Q51
* - ...	
21 X +Q50	; Result corresponds to X +30

Example: Cylinder with Q parameters

Cylinder radius:	$R = Q50$
Cylinder height:	$H = Q51$
Cylinder Z1:	$Q50 = +30$
	$Q51 = +10$
Cylinder Z2:	$Q50 = +10$
	$Q51 = +50$

Assigning the Undefined status to a variable

To assign the **Undefined** status to a variable:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select **FN 0**
- ▶ Enter the number of the variable (e.g., **Q5**)
- ▶ Select **SET UNDEFINED**
- ▶ Confirm your input
- The control assigns the **Undefined** status to the variable.

Notes

- The control distinguishes between undefined variables and variables with the value 0.
- You cannot divide by 0 (**FN 4**).
- You cannot extract a square root from a negative value (**FN 5**).

18.2.5 The Trigonometric functions folder

Application

In the **Trigonometric functions** folder of the **Insert NC function** window, the control provides the functions **FN 6** to **FN 8** and **FN 13**.

You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

Description of function

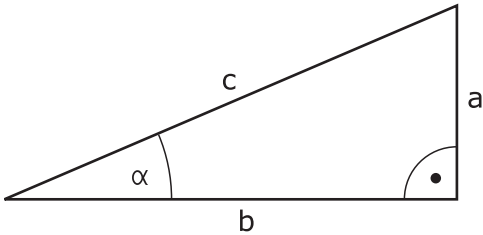
The **Trigonometric functions** folder contains the following functions:

Icon	Function
	FN 6: Sine Example: FN 6: Q20 = SIN -Q5 $Q20 = \sin(-Q5)$ Calculate and assign the sine of an angle in degrees
	FN 7: Cosine Example: FN 7: Q21 = COS -Q5 $Q21 = \cos(-Q5)$ Calculate and assign the cosine of an angle in degrees
	FN 8: Root of the sum of squares Example: FN 8: Q10 = +5 LEN +4 $Q10 = \sqrt{5^2 + 4^2}$ Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
	FN 13: angle Example: FN 13: Q20 = +25 ANG -Q1 $Q20 = \arctan(25/-Q1)$ Calculate and assign the angle from the opposite side and the adjacent side using arctan or from the sine and cosine of the angle ($0 < \text{angle} < 360^\circ$)

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, you can use fixed or variable values. The variables and numbers in the equations can be entered with an algebraic sign.

Definition



Side or trigono- metric function	Meaning
a	Opposite side The side opposite to angle α
b	Adjacent side The side adjacent to angle α
c	Hypotenuse The longest side of the triangle, opposite to the right angle
Sine	$\sin \alpha = \text{opposite side/hypotenuse}$ $\sin \alpha = a/c$
Cosine	$\cos \alpha = \text{adjacent side/hypotenuse}$ $\cos \alpha = b/c$
Tangent	$\tan \alpha = \text{opposite side/adjacent side}$ $\tan \alpha = a/b$ or $\tan \alpha = \sin \alpha / \cos \alpha$
Arc tangent	$\alpha = \arctan(a/b)$ or $\alpha = \arctan(\sin \alpha / \cos \alpha)$

Example

a = 25 mm

b = 50 mm

$\alpha = \arctan(a/b) = \arctan 0.5 = 26.57^\circ$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \cdot a$)

$c = \sqrt{(a^2 + b^2)}$

11 Q50 = ATAN (+25 / +50)	; Calculate angle α
12 FN 8: Q51 = +25 LEN +50	; Calculate side length c

18.2.6 The Circle calculation folder

Application

In the **Circle calculation** folder of the **Insert NC function** window, the control provides the functions **FN 23** and **FN 24**.

These functions allow you to calculate the center of a circle and the radius of the circle based on the coordinates of three or four points on the circle (e.g., the position and size of a circle segment).

Description of function

The **Circle calculation** folder contains the following functions:

Icon	Function
	FN 23: Circle data from three points on the circle Example: FN 23: Q20 = CDATA Q30 The control saves the determined values in the Q parameters Q20 to Q22 .
	FN 24: Circle data from four points on the circle Example: FN 24: Q20 = CDATA Q30 The control saves the determined values in the Q parameters Q20 to Q22 .

To the left of the equal sign, define the variable to which the result should be assigned.

To the right of the equal sign, define the variable starting from which the control is to determine the circle data from the next variables.

The coordinates of the circle data are stored in successive variables. These coordinates must be in the working plane. You must save the coordinates of the main axis before the coordinates of the secondary axis (e.g., **X** before **Y** for tool axis **Z**).

Further information: "Designation of the axes of milling machines", Page 106

Application example

11 FN 23: Q20 = CDATA Q30

; Circle calculation with three points on the circle

The control checks the values in the Q parameters **Q30** to **Q35** and determines the circle data.

The control saves the results in the following Q parameters:

- Circle center on the main axis in the Q parameter **Q20**
For the tool axis **Z**, the main axis is **X**
- Circle center on the secondary axis in the Q parameter **Q21**
For the tool axis **Z**, the secondary axis is **Y**
- Circle radius in the Q parameter **Q22**



NC function **FN 24** uses four pairs of coordinate values and thus eight successive Q parameters.

Note

FN 23 and **FN 24** not only assign a value to the results variable to the left of the equal sign, but also to the subsequent variables.

18.2.7 The Jump commands folder

Application

In the **Jump commands** folder of the **Insert NC function** window, the control provides the functions **FN 9** to **FN 12** for jumps with if-then decisions.

In if-then decisions, the control compares a variable or fixed value with another variable or fixed value. If the condition is fulfilled, the control jumps to the label programmed for the condition.

If the condition is not fulfilled, the control continues with the next NC block.

Related topics

- Jumps without condition with **CALL LBL** label call

Further information: "Subprograms and program section repeats with the label LBL", Page 178

Description of function

The **Jump commands** folder contains the following functions for if-then decisions:

Icon	Function
	FN 9: jump if equal Example: FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25" If both values are equal, the control jumps to the defined label. FN 9: jump if undefined Example: FN 9: IF +Q1 IS UNDEFINED GOTO LBL "UPCAN25" If the variable is undefined, the control jumps to the defined label. FN 9: jump if defined Example: FN 9: IF +Q1 IS DEFINED GOTO LBL "UPCAN25" If the variable is defined, the control jumps to the defined label.
	FN 10: jump if not equal Example: FN 10: IF +10 NE -Q5 GOTO LBL 10 If both values are not equal, the control jumps to the defined label.
	FN 11: jump if greater than Example: FN 11: IF+Q1 GT+10 GOTO LBL QS5 If the first value is greater than the second value, the control jumps to the defined label.
	FN 12: jump if less than Example: FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME" If the first value is less than the second value, the control jumps to the defined label.

You can enter fixed or variable values for if-then decisions.

Unconditional jump

Unconditional jumps are jumps whose condition is always fulfilled.

11 FN 9: IF+0 EQU+0 GOTO LBL1

; Unconditional jump with **FN 9** whose condition is always fulfilled

You can use such jumps, for example, in a called NC program in which you work with subprograms. In an NC program without **M30** or **M2**, you can prevent the control from executing subprograms without a call with **LBL CALL**. As the jump address, program a label that is located directly before the program end.

Further information: "Subprograms", Page 180

Definitions

Abbreviation	Definition
IF	If
EQU (equal)	Equal to
NE (not equal)	Not equal to
GT (greater than)	Greater than
LT (less than)	Less than
GOTO (go to)	Go to
UNDEFINED	Undefined
DEFINED	Defined

18.2.8 Special functions for programming with variables

Output error messages with FN 14: ERROR

Application

With the **FN 14: ERROR** function, you can output error messages under program control. The messages are pre-defined by the machine manufacturer or by HEIDENHAIN.

Related topics

- Error messages output with **FUNCTION REPORT**
Further information: "Issuing notifications with FUNCTION REPORT", Page 522
- Error messages in the notification menu
Further information: User's Manual for Setup and Program Run



Overview of the Machine Parameters, Error Numbers and System Data

The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:

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

ID 1445456-xx

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

TNCguide

Description of function

If, during program run or during simulation, the control executes the **FN 14: ERROR** function, it will interrupt program run and display the defined message. You must then restart the NC program.

You define the error number for the desired error message.

The error numbers are grouped as follows:

Error number range	Error message
0 ... 999	Machine-dependent dialog
1000 ... 2999	Control-dependent dialog
3000 ... 9999	Machine-dependent dialog
10 000 and higher	Control-dependent dialog



Refer to your machine manual.

The error numbers from 0 to 999 as well as those from 3000 to 9999 are defined by the machine manufacturer.

Input

11 FN 14: ERROR=1000

; Output error message with **FN 14**

To navigate to this function:

Insert NC function ► All functions ► FN ► Special functions ► FN 14 ERROR

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 14: ERROR	Syntax initiator for error message output
Number	Number of the error message Number or numerical parameter

Note

Please be aware that not all error messages might be available, depending on the control and the software version.

Outputting text formatted with FN 16: F-PRINT

Application

With the **FN 16: F-PRINT** function you can output formatted fixed and variable texts (e.g., in order to save measuring logs).

You can output the values as follows:

- Save them to a file on the control
- Display them in a window on the screen
- Save them to a file on an external drive or USB device
- Print them to a connected printer

Related topics

- Outputting texts with **FUNCTION PRINT**
Further information: "Issuing text with FUNCTION PRINT", Page 524
- Printing using the function **FN 16: F-PRINT**
Further information: User's Manual for Setup and Program Run

Description of function

In order to output fixed or variable numbers and texts, the following is required:

- Format file
The format file determines the contents and formatting.
- NC function **FN 16: F-PRINT**
The control creates the output file using the NC function **FN 16**.
The maximum size of the output file is 20 kB.

Further information: "Format file for contents and formatting", Page 413

The control creates the output file in the following cases:

- End of program **END PGM**
- Cancellation of program with the **NC STOP** key
- **M_CLOSE** keyword in the format file
Further information: "Keywords", Page 415

Format file for contents and formatting

Define the formatting and the contents of the output file in a format file with the extension ***.a**.

Further information: "The Text editor workspace", Page 366

Formatting

The formatting of the source file can be defined with the following formatting characters:



Please note that the input is case-sensitive.

Formatting characters	Meaning
"..."	Identify the formatting of the contents to be output For text output, you can use the UTF-8 character set.
%F, %D or %I	Initiate the formatted output of numerical parameters ■ F: Float (32-bit floating-point number) ■ D: Double (64-bit floating-point number) ■ I: Integer (32-bit integer)
9.3	Define the number of digits for the output of numerical values ■ 9: Total number of digits, including decimal separator ■ 3: Number of decimal places
%S or %RS	Initiate the formatted or unformatted output of a string parameter ■ S: String ■ RS: Raw String The control takes over the following text without any changes and formatting.
,	Separate the input within a format-file line (e.g., data type and variable)
;	End of the format-file line
*	Initiate a comment line within the format file Comments are not included in the output file
%"	Output quotation marks in the output file
%%	Output a percentage sign in the output file
\\	Output a backslash in the output file
\n	Output a line break in the output file
+	Output the variable value right-aligned in the output file
-	Output the variable value left-aligned in the output file

Keywords

You can define the contents of the output file with the following keywords:

Keyword	Meaning
CALL_PATH	Output the path name of the NC program that contains the FN 16 function (e.g., " TouchProbe: %S ", CALL_PATH ;))
M_CLOSE	Close the file written to with FN 16
M_APPEND	Upon renewed output, append the contents of the output file to the existing output file
M_APPEND_MAX	Upon renewed output, append the contents of the output file to the existing output file until the maximum file size of 20 kB is reached (e.g., M_APPEND_MAX20 ;))
M_TRUNCATE	Upon renewed output, overwrite the output file
M_EMPTY_HIDE	Do not output blank lines for undefined or empty QS parameters in the output file
M_EMPTY_SHOW	Output blank lines for undefined or empty QS parameters and reset M_EMPTY_HIDE
L_ENGLISH	Outputs text only for English conversational language
L_GERMAN	Outputs text only for German conversational language
L_CZECH	Outputs text only for Czech conversational language
L_FRENCH	Outputs text only for French conversational language
L_ITALIAN	Outputs text only for Italian conversational language
L_SPANISH	Outputs text only for Spanish conversational language
L_PORTUGUE	Outputs text only for Portuguese conversational language
L_SWEDISH	Outputs text only for Swedish conversational language
L_DANISH	Outputs text only for Danish conversational language
L_FINNISH	Outputs text only for Finnish conversational language
L_DUTCH	Outputs text only for Dutch conversational language
L_POLISH	Outputs text only for Polish conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_JAPANESE	Output text only for Japanese conversational language
L_RUSSIAN	Outputs text only for Russian conversational language
L_CHINESE	Outputs text only for Chinese conversational language
L_CHINESE_TRAD	Outputs text only for Chinese (traditional) conversational language
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_KOREAN	Outputs text only for Korean conversational language
L_NORWEGIAN	Outputs text only for Norwegian conversational language
L_ROMANIAN	Outputs text only for Romanian conversational language

Keyword	Meaning
L_SLOVAK	Outputs text only for Slovakian conversational language
L_TURKISH	Outputs text only for Turkish conversational language
L_ALL	Display text independently of the conversational language
HOURL	Output the hours of the current time
MIN	Output the minutes of the current time
SEC	Output the seconds of the current time
DAY	Output the day of the current date
MONTH	Output the month of the current date
STR_MONTH	Output the month of the current date in short form
YEAR2	Output the year of the current date in two-digit format
YEAR4	Output the year of the current date in four-digit format

Input

11 FN 16: F-PRINT TNC:\mask.a / TNC: \Prot1.txt ; Output file **Prot1.txt** with the source from **Mask.a**

To navigate to this function:

Insert NC function ► FN ► Special functions ► FN 16 F-PRINT

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 16: F-PRINT	Syntax initiator for formatted output of contents
File	Path of the format file for the output format Text or named parameter Selection by means of a selection window
/	Separator between the two paths
File	Path under which the control saves the output file Text or named parameter Selection by means of a selection window The file name extension of the log file determines the file type of the output (e.g., TXT, A, XLS, HTML).

If you want to define variable paths, use the following syntax to enter the QS parameters:

Syntax element	Meaning
: 'QS1'	Enter QS parameters with a preceding colon and between single quotation marks
: 'QL3'.txt	Specify the file name extension of the target file, if required

Output options

Screen output

You can use the **FN 16** function to display messages in a window on the control screen. This allows you to display explanatory texts in such a way that the user cannot continue without reacting to them. The contents of the output text and the position in the NC program can be chosen freely. You can also output variable values.

In order to display the message on the control screen, enter **SCREEN:** as the output path.

The message is also displayed on the **FN 16** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

Example

```
11 FN 16: F-PRINT TNC:MASKE -
    MASKE1.A / SCREEN:
```

; Display the output file with **FN 16** on the control screen



If you want to replace the content of the window for multiple screen outputs in the NC program, define the **M_CLOSE** or **M_TRUNCATE** keyword.

The control opens the **FN 16 output** window for screen output. The window remains open until you close it. While the window is open, you can operate the control in the background and change to another operating mode.

You can close the window in the following ways:

- Define the **SCLR:** output path (Screen Clear)
- Select the **OK** button
- Select the **Reset program** button
- Select a new NC program

Saving the output file

With the **FN 16** function, you can save the output files to a drive or a USB device. To save the output file, define the path including the drive in the **FN 16** function.

Example

```
11 FN 16: F-PRINT TNC:MSKMSK1.A /
    PC325:\LOG\PRO1.TXT
```

; Save output file with **FN 16**

If you program the same output multiple times in the NC program, the control appends the current output to the end of the contents already output within the target file.

Printing the output file

You can use the **FN 16** function to print output files to a connected printer.

Further information: User's Manual for Setup and Program Run

The control will only print the output file if the format file ends with the **M_CLOSE** keyword.

To use the default printer, enter **Printer:** as the target path and a file name.

If you do not use the default printer, enter the path to the respective printer (e.g., **Printer:\PR0739**) and a file name.

The control saves the file using the defined file name and the defined path. The control will not print the file name.

The control saves the file temporarily until printing is complete.

Notes

- Use the optional machine parameters **fn16DefaultPath** (no. 102202) and **fn16DefaultPathSim** (no. 102203) to define a path under which the control saves the output files.
If you define a path both in the machine parameters and in the **FN 16** function, the path in the **FN 16** function has priority.
- If you only define the file name as the target path of the output file in the FN function, the control saves the output file in the folder of the NC program.
- If the called file is located in the same directory as the file you are calling it from, you can also enter just the file name without the path. If you select the file using the selection menu, the control automatically proceeds in this manner.
- If you specify the **%RS** function in the format file, the control takes over the defined content without formatting. This allows you to output a path specification with QS parameters, for example.
- In the settings of the **Program** workspace, you can specify whether the control displays a screen output in a window.
If you deactivate the screen output, the control will not display a window.
The control will display the contents anyway on the **FN 16** tab of the **Status** workspace.

Further information: "Settings in the Program workspace", Page 120

Further information: User's Manual for Setup and Program Run

Example

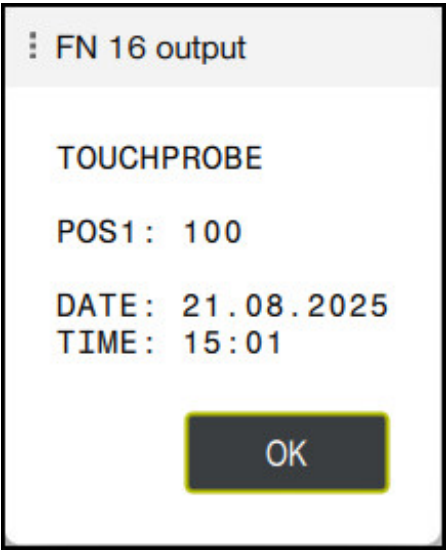
Example of a format file that generates an output file with variable contents:

```
"TOUCHPROBE";
"%S",QS1;
M_EMPTY_HIDE;
"%S",QS2;
"%S",QS3;
M_EMPTY_SHOW;
"%S",QS4;
"DATE: %02d.%02d.%04d",DAY,MONTH,YEAR4;
"TIME: %02d:%02d",HOUR,MIN;
M_CLOSE;
```

Example of an NC program that defines only **QS3**:

11 Q1 = 100	; Assign the value 100 to Q1
12 QS3 = "Pos 1: " TOCHAR(DAT +Q1)	; Convert the numerical value of Q1 to a text and assign it to the defined character string
13 FN 16: F-PRINT TNC:\fn16.a / SCREEN:	; Display the output file with FN 16 on the control screen

Example of a screen output with two empty lines resulting from **QS1** and **QS4**:



The **FN 16 output** window

Read system data with FN 18: SYSREAD

Application

With the **FN 18: SYSREAD** function you can read numerical system data and store this data in variables.

Related topics

- Reading system data using string parameters

Further information: "Read system data with SYSSTR", Page 433



Overview of the Machine Parameters, Error Numbers and System Data

The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:

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

ID 1445456-xx

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

TNCguide

Description of function

The control always outputs system data in the metric system with **FN 18: SYSREAD**, regardless of the unit of the NC program.

Input

**11 FN 18: SYSREAD Q25 = ID210 NR4
IDX3**

; Save the active dimension factor of the Z
axis in **Q25**

To navigate to this function:

Insert NC function ► FN ► Special functions ► FN 18 SYSREAD

The NC function includes the following syntax elements:

Syntax element	Meaning
FN18: SYSREAD	Syntax initiator for reading system data
Parameter	Numerical parameter in which the control stores the information
ID	Group number of the system datum Number or numerical parameter
NR	System data number Number or numerical parameter Optional syntax element
IDX	Index Number or numerical parameter Optional syntax element
.	Sub-index for system data for tools Number or numerical parameter Optional syntax element

Note

As an alternative, you can use **TABDATA READ** to read out data from the active tool table. In this case, the control will automatically convert the table values to the unit of measure used in the NC program.

Further information: "Reading table values with TABDATA READ", Page 603

Sending information from the NC program with FN 38: SEND

Application

The function **FN 38: SEND** enables you to retrieve fixed or variable values from the NC program and write them to the log or send them to an external application (e.g., StateMonitor).

Description of function

Data is transferred via a TCP/IP connection.



For more detailed information, consult the RemoTools SDK manual.

Input

**11 FN 38: SEND /"Q-Parameter Q1: %F
Q23: %F" / +Q1 / +Q23**

; Write values from **Q1** and **Q23** to the logbook

To navigate to this function:

Insert NC function ► FN ► Special functions ► FN 38 SEND

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 38: SEND	Syntax initiator for sending information
Name or Parameter	Format of the text to be transmitted Text or string parameter Output text with up to seven placeholders for the values of the variables (e.g., %F) Further information: "Format file for contents and formatting", Page 413
/	Contents of the up to seven placeholders in the output text Number or numerical parameter Optional syntax element

Notes

- Both fixed and variable numbers and texts are case-sensitive, so enter them correctly.
- To obtain % in the output text, enter %% at the desired position.

18.2.9 NC functions for accessing tables with variables

Opening a table with FN 26: TABOPEN

Application

With the NC function **FN 26: TABOPEN**, you open any table to be written to with **FN 27: TABWRITE** or to be read from with **FN 28: TABREAD**.

Related topics

- Content and creation of freely definable tables
Further information: "Freely definable tables *.tab", Page 606
- Access to table values in case of low computing power
Further information: "Table access with SQL statements", Page 451

Description of function

Select the table to be opened by entering its path.

If you select the path using the selection window, the control by default only displays freely definable tables ***.tab**. You can change the filter to **All Files (*.*)**.

Input

11 FN 26: TABOPEN TNC:\table TAB1.TAB	; Open table with FN 26

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Special functions** ► **FN 26 TABOPEN**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 26: TABOPEN	Syntax initiator for opening a table
File or Variable	Path of the table to be opened Text or named parameter Selection by means of a selection window

Note

Only one table can be opened in an NC program at any one time. A new NC block with **FN 26: TABOPEN** automatically closes the last opened table.

Writing to a table with FN 27: TABWRITE

Application

With the NC function **FN 27: TABWRITE**, you write to the table that you previously opened with **FN 26: TABOPEN**.

Related topics

- Open table
Further information: "Opening a table with FN 26: TABOPEN", Page 423

Description of function

Use the NC function **FN 27** to define the table columns to be written to by the control. Within an NC block, you can specify multiple table columns, but only one table row. Previously define the contents to be written to the columns in variables or directly in the NC function **FN 27**.

Input

11 FN 27: TABWRITE 2 / "Length,Radius" ; Write to table with **FN 27**
= Q2

To navigate to this function:

Insert NC function ► All functions ► FN ► Special functions ► FN 27 TABWRITE

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 27: TABWRITE	Syntax initiator for writing to a table
Number	Row number of the table to be written to Number or numerical parameter
Name or Parameter	Column names in the table to be written to Text or string parameter Use commas to separate multiple column names.
= or SET UNDEFINED	Write the table value or assign the status undefined
Number, Name or Parameter	Table value Number, text, or variable Only if = has been selected

Notes

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

The **FN 27: TABWRITE** function is also effective on currently active table rows and during the simulation. If you change process-relevant data such as tool or preset data, unexpected traverse movements or collisions may occur.

- ▶ Change process-relevant data only intentionally
- ▶ Hide **FN 27: TABWRITE** in the simulation
- ▶ Create backup copies of tables

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

If you use the **TABDATA WRITE** or **FN 27** functions to write to a table cell that contains existing values, the control overwrites these values without confirmation prompting.

- ▶ Write only to empty or no longer needed table cells
- ▶ Create a backup copy of the table

- If you write to multiple columns within one NC block, you need to define the values to be written to the columns in consecutive variables.
- If you try to write to a locked or a non-existing table cell, the control displays an error message.
- When you write to several columns, the control can only write either numbers or texts.
- If you define a fixed value in the **FN 27** NC function, the control will write the same value to each defined column.
- With the **SET UNDEFINED** syntax element you assign the **undefined** status to the variables.

For example, if you program a position using an undefined Q parameter, the control will ignore this movement.

If you use an undefined variable in arithmetic operations in the NC program, the control displays an error message and stops program run.

Further information: "Assigning the Undefined status to a variable", Page 406

Example

11 Q5 = 3.75	; Define the value for the Radius column
12 Q6 = -5	; Define the value for the Depth column
13 Q7 = 7.5	; Define the value for the D column
14 FN 27: TABWRITE 5 / "Radius,Depth,D" = Q5	; Write defined values to the table

The control writes to the columns **Radius**, **Depth**, and **D** of row **5** of the currently open table. The control writes the values from the Q parameters **Q5**, **Q6**, and **Q7** to the table.

Reading table content with FN 28: TABREAD

Application

With the NC function **FN 28: TABREAD**, you read from the table that you previously opened with **FN 26: TABOPEN**.

Related topics

- Opening a table

Further information: "Opening a table with FN 26: TABOPEN", Page 423

- Writing to a table

Further information: "Writing to a table with FN 27: TABWRITE", Page 423

Description of function

Use the **FN 28** NC function to define the table columns that the control is to read from. Within an NC block, you can specify multiple table columns, but only one table row.

Input

```
11 FN 28: TABREAD Q1 = 2 / "Length" ; Read table with FN 28
```

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Special functions** ► **FN 28 TABREAD**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 28: TABREAD	Syntax initiator for reading from a table
Parameter	Variable for the source text The control uses this variable to save the contents from the table cells to be read.
Number	Row number in the table to be read Number or numerical parameter
Name or Parameter	Column name in the table to be read Text or string parameter Use commas to separate multiple column names.

Note

If you specify multiple columns in an NC block, the control saves the read values in consecutive variables of the same type (e.g., **QL1**, **QL2**, and **QL3**).

Example

```
11 FN 28: TABREAD Q10 = 6 / "X,Y,D" ; Read numeric values from columns X, Y and D
12 FN 28: TABREAD QS1 = 6 / "DOC" ; Read the text from column DOC
```

The control reads the values of columns **X**, **Y**, and **D** from row **6** of the currently open table. The control saves the values to the Q parameters **Q10**, **Q11**, and **Q12**.

The content from the **DOC** column of the same row is saved to the **QS1** QS parameter.

18.2.10 Formulas in the NC program

Application

The NC functions **Formula** or **Variable** allow you to define several arithmetic operations in one NC block. You can also assign a single value to a variable.

Related topics

- String formula for texts
Further information: "String functions", Page 432
- Defining a single calculation in an NC block
Further information: "The Basic arithmetic folder", Page 404

Description of function

The **Formula** NC function is used to define the **Q**, **QL** and **QR** parameters.

Use the **Variable** NC function to define a named parameter.

As the first entry, you define the variable to which you assign the result.

To the right of the equal sign, define the arithmetic operations or a value that the control assigns to the variable.

The control provides the following options to enter formulas:

- Auto-complete
Further information: "Entering a formula using the auto-complete function", Page 430
- Pop-up keyboard for formula input from the action bar or from within the form
- Formula input mode of the virtual keyboard
Further information: "Virtual keyboard of the control bar", Page 493

Rules for formulas

Evaluation order for different operators

If a formula includes arithmetic operations involving a combination of different operators, the control evaluates the operations in a certain order. A familiar example of this is the rule that multiplication/division takes precedence over addition/subtraction (higher-level operations are performed first).

Further information: "Examples", Page 430

The control evaluates the arithmetic operations in the following order:

Order	Arithmetic operation	Operator	Arithmetic operator
1	Perform operations in parentheses first	Parentheses	()
2	Note the algebraic sign	Algebraic sign	-
3	Calculate functions	Function	SIN, COS, LN, etc.
4	Exponentiation	Power	^
5	Multiplication and division	Point	*, /
6	Addition and subtraction	Line	+, -

Further information: "Arithmetic operations", Page 428

Order in the evaluation of equivalent operators

The control evaluates arithmetic operations with equivalent operators from left to right.

Example: $2 + 3 - 2 = (2 + 3) - 2 = 3$

Exception: Concatenated powers are evaluated from right to left.

Example: $2 ^ 3 ^ 2 = 2 ^ (3 ^ 2) = 2 ^ 9 = 512$

Arithmetic operations

The virtual keyboard for formula input allows you to perform the following arithmetic operations:

Button	Arithmetic operation	Operator
 +	Addition Example: $Q10 = Q1 + Q5$	Line
 -	Subtraction Example: $Q25 = Q7 - Q108$	Line
 *	Multiplication Example: $Q12 = 5 * Q5$	Point
 /	Division Example: $Q25 = Q1 / Q2$	Point
 ()	Parenthesize Example: $Q12 = Q1 * (Q2 + Q3)$	Expression in parentheses
 SQ	Square (square) Example: $Q15 = SQ 5$	Function
 SQRT	Calculate square root (square root) Example: $Q22 = SQRT 25$	Function
 SIN	Calculate sine Example: $Q44 = SIN 45$	Function
 COS	Calculate cosine Example: $Q45 = COS 45$	Function
 TAN	Calculate tangent Example: $Q46 = TAN 45$	Function
 ASIN	Calculate arcsine Inverse function of sine The control determines the angle from the ratio of the opposite side to the hypotenuse. Example: $Q10 = ASIN (Q40 / Q20)$	Function

Button	Arithmetic operation	Operator
ACOS ACOS	Calculate arccosine Inverse function of cosine The control determines the angle from the ratio of the adjacent side to the hypotenuse. Example: Q11 = ACOS Q40	Function
ATAN ATAN	Calculate arctangent Inverse function of tangent The control determines the angle from the ratio of the opposite side to the adjacent side. Example: Q12 = ATAN Q50	Function
^ ^	Exponentiation Example: Q15 = 3 ^ 3	Power
PI PI	Use the “pi” constant $\pi = 3.14159$ Example: Q15 = PI	
LN LN	Calculate the natural logarithm (LN) Base = e = 2.7183 Example: Q15 = LN Q11	Function
LOG LOG	Calculate the logarithm Base = 10 Example: Q33 = LOG Q22	Function
EXP EXP	Use the exponential function (e ^ n) Base = e = 2.7183 Example: Q1 = EXP Q12	Function
NEG NEG	Negate Multiply by –1 Example: Q2 = NEG Q1	Function
INT INT	Calculate an integer Truncate decimal places Example: Q3 = INT Q42	Function
 The INT function does not round off—it simply truncates the decimal places.		
Input: 0...999999999		
ABS ABS	Calculate the absolute value Example: Q4 = ABS Q22	Function
FRAC FRAC	Calculate a fraction Truncate the digits before the decimal point Example: Q5 = FRAC Q23	Function

Button	Arithmetic operation	Operator
 SGN	Check the algebraic sign Example: $Q12 = \text{SGN } Q50$ If $Q50 = 0$, then $\text{SGN } Q50 = 0$ If $Q50 < 0$, then $\text{SGN } Q50 = -1$ If $Q50 > 0$, then $\text{SGN } Q50 = 1$	Function
 %	Calculate the modulo value (division remainder) Example: $Q12 = 400 \% 360$ Result: $Q12 = 40$	Function

Further information: "The Basic arithmetic folder", Page 404

Further information: "The Trigonometric functions folder", Page 407

Entering a formula using the auto-complete function

To enter a formula using the auto-complete function:

- 

 - ▶ Select **Insert NC function**
 - The control opens the **Insert NC function** window.
 - ▶ Select **Formula** or **Variable**
 - ▶ Define a variable name for the result
 - ▶ Confirm your input
 - ▶ Select the arithmetic operation (e.g., **SIN**)
 - ▶ Enter the desired value
 - ▶ Press the spacebar
 - The control displays the currently available arithmetic operations.
 - ▶ Select the desired arithmetic operation
 - ▶ Enter the desired value
 - ▶ If required, press the spacebar again
 - ▶ If required, select the desired arithmetic operation
 - ▶ Complete the NC block once all required data has been entered

Examples

Multiplication and division before addition and subtraction

11 $Q1 = 5 * 3 + 2 * 10$; Result = 35

11 {a} = 5 * 3 + 2 * 10 ; Result = 35

- 1st calculation: $5 * 3 = 15$
- 2nd calculation: $2 * 10 = 20$
- 3rd calculation: $15 + 20 = 35$

Power before addition and subtraction

11 Q2 = SQ 10 - 3^3	; Result = 73

11 {b} = SQ 10 - 3^3	; Result = 73

- 1st calculation: 10 squared = 100
- 2nd calculation: 3 to the power of 3 = 27
- 3rd calculation: 100 – 27 = 73

Function before power

11 Q4 = SIN 30 ^ 2	; Result = 0.25

11 {c} = SIN 30 ^ 2	; Result = 0.25

- 1st calculation: Calculate sine of 30 = 0.5
- 2nd calculation: 0.5 squared = 0.25

Brackets before function

11 Q5 = SIN (50 - 20)	; Result = 0.5

11 {d} = SIN (50 - 20)	; Result = 0.5

- 1st calculation: Perform operations in parentheses first: 50 – 20 = 30
- 2nd calculation: Calculate sine of 30 = 0.5

18.3 String functions

Application

The string functions allow you to define and process texts using string parameters, for example, in order to create variable logs with **FN 16: F-PRINT**. In computing, a string designates a text.

Related topics

- Ranges of variables
Further information: "Basics", Page 393
- Programming of format strings
Further information: "Format strings", Page 439

Description of function

Within the **Formula**, **Variable** or **String formula** NC functions you can use the following string functions:

Syntax element	Meaning	NC function
DECLARE STRING	Assign a text to a QS parameter Further information: "Assigning text to a string parameter", Page 434	DECLARE STRING
 	Concatenate contents of string parameters and assign them to a string parameter Further information: "Concatenating values of string parameters", Page 435	■ String formula ■ Variable
TONUMB	Convert the value of a QS parameter to a numerical value and assign it to a numerical parameter Further information: "Converting the value of a string parameter to number ", Page 435	■ Formula ■ Variable
TOCHAR	Convert numerical value to a text and assign it to a string parameter Further information: "Converting numerical values to texts", Page 436	■ String formula ■ Variable
SUBSTR	Copy a substring from a QS parameter and assign it to a string parameter Further information: "Copying a substring from a string parameter", Page 436	■ String formula ■ Variable
SYSSTR	Read system data and assign the contents to a string parameter Further information: "Read system data with SYSSTR", Page 433	■ String formula ■ Variable
INSTR	Search for a substring in a QS parameter and assign the retrieved characters to a numerical parameter Further information: "Searching for a substring within QS parameter contents", Page 436	■ Formula ■ Variable
STRLEN	Determine the string length of a QS parameter and assign it to a numerical parameter Further information: "Determining the number of characters in QS parameter contents", Page 437	■ Formula ■ Variable

Syntax element	Meaning	NC function
STRCOMP	Compare QS parameters in ascending lexical order and assign the result to a numerical parameter Further information: "Comparing the lexical order of two texts", Page 437	■ Formula ■ Variable
CFGREAD	Read the content of a machine parameter and assign it to a variable Further information: "Applying the contents of a machine parameter", Page 438	■ String formula ■ Formula

The control provides the following options to enter formulas:

- Auto-complete
Further information: "Entering a formula using the auto-complete function", Page 430
- Pop-up keyboard for formula input from the action bar or from within the form
- Formula input mode of the virtual keyboard
Further information: "Virtual keyboard of the control bar", Page 493

Read system data with SYSSTR

With the **SYSSTR** NC function you can read system data and save the contents in string parameters. Select the system datum by means of a group number **ID** and a number **NR**. Optionally, you can enter **IDX** and **DAT**.

SYSSTR only allows you to read alphanumeric values.

For numerical values, the control offers the **FN 18: SYSREAD** NC function. **SYSSTR** is programmed identically to **FN 18: SYSREAD**.

Further information: "Read system data with FN 18: SYSREAD", Page 420



Overview of the Machine Parameters, Error Numbers and System Data

The additional documentation **Overview of the Machine Parameters, Error Numbers and System Data** provides an overview of the following functions:

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

ID 1445456-xx

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

TNCguide

18.3.1 Assigning text to a string parameter

Before you can use and process texts, you must assign characters to the string parameters.

To assign a text to a string parameter:



- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select **String formula** or **Variable**
- ▶ Define a variable name for the result
- ▶ Move cursor to the right
- ▶ If required, enter ""
- ▶ Enter the desired value between the quotation marks
- ▶ End the NC block
- ▶ Execute the NC block
- The control saves the entered value in the target parameter.



Alternatively, you can assign characters to a QS parameter using the **DECLARE STRING** NC function.

In these examples, the control assigns a text to a string parameter.

11 QS10 = "workpiece"

; Assign text to the QS parameter **QS10**

11 {a} = "workpiece"

; Assign text to the named parameter **{a}**

18.3.2 Concatenating values of string parameters

With the `||` concatenation operator, you can concatenate the contents of multiple string parameters. This allows you, for example, to combine fixed and variable texts.

To concatenate the contents of multiple string parameters:



- ▶ Select **Insert NC function**
 - The control opens the **Insert NC function** window.
- ▶ Select **String formula** or **Variable**
 - ▶ Define a string parameter for the result
 - ▶ Confirm your input
- ▶ Select the backspace key, if required
 - The control deletes the quotation marks.
- ▶ Select **QS** or **Variable**
 - ▶ Enter the variable name
 - ▶ Press the spacebar
 - The control displays the currently available syntax elements.
- ▶ Select concatenation operator `||`
 - ▶ Select **QS** or **Variable**
 - ▶ Enter the variable name
 - ▶ End NC block
 - After their execution, the control saves the substrings in succession as text in the target parameter.

In these examples, the control concatenates the contents of two string parameters and assigns the result to a third string parameter.

Parameter contents:

- **QS12** and **{b}**: **Status**:
- **QS13** and **{c}**: **Scrap**
- **QS10** and **{a}**: **Status: Scrap**

```
11 QS10 = QS12 || QS13
```

; Concatenate contents of **QS12** and **QS13** and assign them to the QS parameter **QS10**

```
11 {a} = {b} || {c}
```

; Concatenate contents from **{b}** and **{c}** and assign them to named parameter **{a}**

18.3.3 Converting the value of a string parameter to number

With the **TONUMB** NC function, you save exclusively numeric characters from a QS parameter to a different variable type. Then, you can use these values in calculations.

In these examples, the control converts the variable value of a QS parameter to a numerical value. The control assigns this value to a numerical parameter.

```
11 Q82 = TONUMB ( SRC_QS11 )
```

; Convert value from **QS11** to a numerical value and assign it to **Q82**

```
11 {a} = TONUMB ( SRC_QS11 )
```

; Convert value from **QS11** to a numerical value and assign it to **{a}**

18.3.4 Converting numerical values to texts

With the **TOCHAR** NC function, you can save the content of a numerical parameter in a string parameter. You can concatenate the saved content with other string parameters, for example.

In these examples, the control converts the numerical value of a Q parameter to a text. The control assigns this text to a string parameter.

11 QS11 = TOCHAR (DAT+Q50 DECIMALS3)	; Convert numerical value from Q50 to a text and assign it to the QS parameter QS11

11 {a} = TOCHAR (DAT+Q50 DECIMALS3)	; Convert numerical value from Q50 to a text and assign it to the named parameter {a}

18.3.5 Copying a substring from a string parameter

With the **SUBSTR** NC function, you can save a defined substring from a QS parameter in another string parameter. For example, you can use this NC function to extract the file name from an absolute file path.

In these examples, the control saves the substring of a QS parameter in another string parameter. Using the **BEG2** syntax element, you define that the control ignores the first two characters and starts copying from the third character. With the **LEN4** syntax element, you define that the control copies the next four characters.

11 QS13 = SUBSTR (SRC_QS10 BEG2 LEN4)	; Assign substring from QS10 to the QS parameter QS13

11 {a} = SUBSTR (SRC_QS10 BEG2 LEN4)	; Assign substring from QS10 to the named parameter {a}

18.3.6 Searching for a substring within QS parameter contents

With the **INSTR** NC function, you can check whether a particular substring is contained within a QS parameter. This allows you to determine, for example, whether the concatenation of multiple QS parameters was successful. For the check, you must indicate two QS parameters. The control searches the first QS parameter for the content of the second QS parameter.

If the substring is found, the control saves the number of characters until it reaches the occurrence of the substring in the result parameter. If multiple occurrences are found, the result is identical because the control saves the first one.

If the substring searched for is not found, the control saves the total number of characters in the result parameter.

In this example, the control searches a QS parameter for the text of a second QS parameter. When counting the characters, the control starts from zero. The control assigns the occurrence to the numerical parameter as a number.

37 Q50 = INSTR (SRC_QS10 SEA_QS13 BEG2)	; Search QS10 for substring from QS13

18.3.7 Determining the number of characters in QS parameter contents

The **STRLEN** NC function determines the number of characters in QS parameter contents. With this NC function, you can, for example, determine the length of a file path.

If the selected QS parameter has not been defined, the control returns the value **-1**.

In this example, the control determines the number of characters in a QS parameter. The control assigns the determined number to a numerical parameter.

```
11 Q52 = STRLEN ( SRC_QS15 )
```

; Determine the number of characters in
QS15 and assign it to **Q52**

18.3.8 Comparing the lexical order of two texts

With the **STRCOMP** NC function, you can compare the lexical order of the content of two QS parameters.

The control returns the following results:

- **0**: The content of the two parameters is identical
- **-1**: In the lexical order, the content of the first QS parameter comes **before** the content of the second QS parameter
- **+1**: In the lexical order, the content of the first QS parameter comes **after** the content of the second QS parameter

The lexical order is as follows:

- 1 Special characters (e.g., ?_)
- 2 Numerals (e.g., 123)
- 3 Uppercase letters (e.g., ABC)
- 4 Lowercase letters (e.g., abc)



Starting from the first character, the control proceeds until the contents of the QS parameters differ from each other. If the contents differ starting from, for example, the fourth digit, the control aborts the check at this point. Shorter contents with identical strings are displayed first in the order (e.g., abc before abcd).

In this example, the control compares the lexical order of the values of two QS parameters. The control assigns the result to a numerical parameter as a number.

```
11 Q52 = STRCOMP ( SRC_QS12  
SEA_QS14 )
```

; Compare the lexical order of the values of
QS12 and **QS14**

18.3.9 Applying the contents of a machine parameter

The NC function **CFGREAD** allows you to read out the values from machine parameters.

Before using **CFGREAD**, you must program one string parameter each with key, entity, and attribute. Pay attention to enter the correct upper- and lowercase letters.

The NC function includes the following syntax elements:

Syntax element	Meaning
KEY_	String parameter with the group name of the machine parameter (key) If no key is available, define the string parameter without content.
TAG_	String parameter with the object name of the machine parameter (entity)
ATR_	String parameter with the name of the machine parameter (attribute)
IDX	Index of the machine parameter Number or numerical parameter Optional syntax element



You determine the required values using the table view of the configuration editor.

Further information: User's Manual for Setup and Program Run

If the machine parameter contains numerical values, you can use the values in Q, QL, or QR parameters. The control always outputs numerical values in the metric system. Strings can be transferred in QS parameters.

Examples

In this example you transfer the overlap factor from the machine parameter **pocketOverlap** (no. 201001):

11 QS11 = "CH_NC"	; Assign the key to the QS parameter QS11
12 QS12 = "CfgGeoCycle"	; Assign the entity to the QS parameter QS12
13 QS13 = "pocketOverlap"	; Assign the attribute to the QS parameter QS13
14 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	; Read out the contents of the machine parameter

In this example you transfer the position of the tool touch probe in the Y axis (index 1) from the machine parameter **centerPos** (no. 114313):

11 {a} = "TT140_2"	; Assign key to the named parameter {a}
12 {b} = "CfgTTRectStylus"	; Assign entity to the named parameter {b}
13 {c} = "centerPos"	; Assign attribute to the named parameter {c}
14 Q50 = CFGREAD(KEY_{a} TAG_{b} ATR_{c} IDX1)	; Read out the contents of the machine parameter

Note

If you use the **String formula** NC function, the result will always be a text. If you use the **Formula** NC function, the result will always be a numerical value.

18.4 Format strings

Application

The control provides the **FMT** syntax element for QS parameters and named parameters to define format strings. Using format strings, you no longer have to convert numerical values or concatenate strings.

You can use format strings in the following NC functions, for example:

- **String formula**
- **SQL SELECT**
- **TEXT** within **FUNCTION REPORT**

Related topics

- Types of variables
Further information: "Basics", Page 393
- Assign text to a string parameter
Further information: "Assigning text to a string parameter", Page 434
- Concatenate values of string parameters
Further information: "Concatenating values of string parameters", Page 435
- Outputting texts with **FUNCTION PRINT**
Further information: "Issuing text with FUNCTION PRINT", Page 524

Requirements

- Code number 555343
- Variables defined beforehand that are programmed within a format string

Description of function

The **FMT** syntax element allows you to define format strings. Program format strings in the text mode of the **Program** workspace.

Further information: "Text mode", Page 135

The syntax of the format strings is based on the f strings in Python. After the syntax initiator follows the string between double quotation marks. You can embed variables in the string using curly brackets. Once the NC block has been executed, the string contains the values of the embedded variables.

11 Q1 = +7	; Numerical value
12 QS1 = "TNC"	; Alphanumeric value
13 QS2 = FMT"ENJOY {QS1}{Q1}"	; Result after execution: ENJOY TNC7

Influencing the formatting of the string is also possible (e.g., by defining whether or not and how the control outputs algebraic signs). You define the formatting inside the curly brackets after a colon.

11 Q1 = +7	; Numerical value
12 QS1 = FMT"{Q1:+"}	; Result including algebraic sign after execution: +7

Formatting

A variety of formatting options is available, depending on the variable type and the values.

When you program formatting options, you must observe the following sequence:
[[fill character] alignment] [sign] ['0'] [field width] ['.' accuracy] [type]



- [] are not programmed and only serve as separators within the sequence.
- ' ' are not programmed, however they contain the character to be programmed or an example input.
- '0' between the sign and the field width can be used as a fill character for numerical values.

11 QS2 = FMT"{QS1:X>10}" ; [[Fill character] alignment] [field width]

12 QS3 = FMT"{Q1:+.2f}" ; [Sign] ['.' accuracy] [type]

Fill character, alignment and field width

Formatting character or example	Meaning
[Fill character] (e.g., 'X')	In connection with alignment and field width you can optionally define fill characters that replace any existing blank spaces.
<	Left-aligns the text within the available field width Default setting that is also effective without input
>	Right-aligns the text within the available field width
^	Centers the text within the available field width
[Field width] (e.g., '10')	You can define the field width as the number of places. If you define a field width that is smaller than the number of characters of the string, the value will have no function. If you do not define a value, the content will determine the field width.

11 QS1 = "LEFT" ; Alphanumeric value

12 QS2 = "RIGHT" ; Alphanumeric value

13 QS3 = FMT"{QS1:<4}{QS2:>6}" ; Result including algebraic sign after execution:
LEFT RIGHT

14 QS4 = FMT"{QS1:X>10}" ; Result including algebraic sign after execution:
XXXXXXLEFT

Algebraic sign

Formatting characters	Meaning
+	Outputs the sign for negative values as well as positive values
-	Outputs the sign for negative values only Default setting that is also effective without input
Spaces	If a space character is programmed as a formatting character, positive values will induce the control to output a leading space character.

11 Q1 = +7 ; Numerical value

```
12 Q$1 = FMT"{Q1}{Q1:+}"
```

; Result after execution:
7+7

Integers

Formatting characters	Meaning
d	Outputs a decimal integer If you program this formatting for a floating-point number, the control will abort program run with an error message. If you do not define a type, the control will use the default setting for decimal numbers.

```
11 Q1 = +1
```

; Numerical value is integer

```
12 Q$1 = FMT"{Q1:d}"
```

; Result after execution:
1

```
13 Q1 = +1.23
```

; Numerical value is decimal number

```
14 Q$1 = FMT"{Q1:d}"
```

; Program cancellation with error message

Decimal numbers

Formatting characters	Meaning
[Accuracy] (e.g., '.2')	You can define the accuracy of the output (e.g., by the number of decimal places for fixed-point numbers f). The effect of the accuracy varies, depending on the combination with the following formatting signs.
e	Outputs the exponent representation with e as a separator
E	Outputs the exponent representation with E as a separator
f	Outputs a decimal fixed-point number Define the number of decimal places using the accuracy. If you do not define the accuracy, the control will use the default value '6'.
g	Depending on the value, the control automatically decides on the type of representation, for example, the exponent representation using e as separator. An additional accuracy allows you to define the number of significant digits to which the value is rounded by the control. If you do not define the accuracy, the control will use the default value '6'.
G	Like type g , but using E as separator for the exponent representation
Default [type]	If you program neither type nor accuracy, the control will use type g with an unlimited accuracy. The default type '6' does not apply in this case.

```
11 Q1 = +1.23
```

; Numerical value

```
12 Q$1 = FMT"{Q1:e}"
```

; Result after execution:
1.230000e+00

```
13 Q$1 = FMT"{Q1:E}"
```

; Result after execution:
1.230000E+00

14 <code>Q\$1 = FMT"{Q1:f}"</code>	; Result after execution: 1.23
15 <code>Q1 = +0.0000123</code>	; Numerical value
16 <code>Q\$1 = FMT"{Q1:f}"</code>	; Result after execution: 0.0000123
17 <code>Q\$1 = FMT"{Q1:g}"</code>	; Result after execution: 1.23e-05
18 <code>Q\$1 = FMT"{Q1:G}"</code>	; Result after execution: 1.23E-05
19 <code>Q1 = +123.456</code>	; Numerical value
20 <code>Q\$1 = FMT"{Q1:.2}"</code>	; Result after execution: 1.2e+02
21 <code>Q\$1 = FMT"{Q1:.2f}"</code>	; Result after execution: 123.46
22 <code>Q\$1 = FMT"{Q1:.2g}"</code>	; Result after execution: 1.2e+02
23 <code>Q\$1 = FMT"{Q1:.3}"</code>	; Result after execution: 123

Definition

Significant digits

Significant digits are the places of a number which contain valuable information. These digits start from the first number after the leading zeros, if applicable, and end with the last still meaningful number. For example, you can restrict a probing result to a number of significant digits by programming the accuracy.

Notes

- The control exclusively supports the described formatting options.
- If you, for example, apply a formatting for decimal numbers to texts, the control will abort program run with an error message.

18.5 Defining counters with FUNCTION COUNT

Application

With the **FUNCTION COUNT** NC function, you control a counter from within the NC program. This counter allows you, for example, to define a target count of workpieces up to which the control repeats the NC program.

Description of function

The control takes the **FUNCTION COUNT** function into consideration during program run and in the simulation.

The control uses a separate counter each for program run and for the simulation.

The control shows the current counter reading and the defined target count of machining operations on the **PGM** tab of the **Status** workspace.

Further information: User's Manual for Setup and Program Run

The counter reading remains the same after a restart of the control.

Input

11 FUNCTION COUNT TARGET5 ; Set the target count of the counter to **5**

Insert NC function ► All functions ► FN ► **FUNCTION COUNT**

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION COUNT	Syntax initiator for the counter
INC, RESET, ADD, SET, TARGET or REPEAT	Define counting function Further information: "Counting functions", Page 443

Counting functions

The **FUNCTION COUNT** NC function provides the following counter functions:

Syntax	Function
INC	Increase the counter by 1
RESET	Reset the counter
ADD	Increase the counter by a defined value Number, text, or variable Input: 0...9999
SET	Assign a defined value to the counter Number, text, or variable Input: 0...9999
TARGET	Define the target count to be reached Number, text, or variable Input: 0...9999
REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet Number, text, or variable

Notes

NOTICE

Caution: Data may be lost!

The control manages the same counter for the **Program Run** operating mode and for the **MDI** application. The counter applies to all programs. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted, if applicable.

- Please check prior to machining whether a counter is active.

- On the **PGM** tab of the **Status** workspace you also find the **Counter settings** window, which allows you to define the counter.

When the control executes **FUNCTION COUNT** in the NC program, it overwrites the values defined in the **Counter settings** window.

Further information: User's Manual for Setup and Program Run

- The machine manufacturer uses the optional machine parameter **CfgNcCounter** (no. 129100) to define whether you can edit the counter.
- You can save the current counter reading in a variable using the **FN 18: SYSREAD ID920 NR1** function.

Further information: "Read system data with FN 18: SYSREAD", Page 420

- Client applications can change the counter reading using OPC UA and the NC.RemoteOperator role (#56-61 / #3-02-1*).

18.5.1 Example

11 FUNCTION COUNT RESET	; Reset counter value
12 FUNCTION COUNT TARGET10	; Define the target count of machining operations
13 LBL 11	; Set a jump label
* - ...	; Execute the machining operation
21 FUNCTION COUNT INC	; Increase the counter reading by 1
22 FUNCTION COUNT REPEAT LBL 11	; Repeat the machining operation until the target count has been reached

18.6 Program defaults for cycles

18.6.1 Overview

Some cycles always use identical cycle parameters, such as the set-up clearance **Q200**, which you must enter for each cycle definition. With the **GLOBAL DEF** function you can define these cycle parameters at the beginning of the program, so that they are globally effective for all cycles used in the NC program. In the respective cycle you then use **PREDEF** to simply reference the value defined at the beginning of the program.

The following **GLOBAL DEF** functions are available:

Cycle	Call	Further information
100 GENERAL Definition of generally valid cycle parameters ■ Q200 SET-UP CLEARANCE ■ Q204 2ND SET-UP CLEARANCE ■ Q253 F PRE-POSITIONING ■ Q208 RETRACTION FEED RATE	DEF-active	Page 447
105 DRILLING Definition of specific drilling cycle parameters ■ Q256 DIST FOR CHIP BRKNG ■ Q210 DWELL TIME AT TOP ■ Q211 DWELL TIME AT DEPTH	DEF-active	Page 448
110 POCKET MILLING Definition of specific pocket-milling cycle parameters ■ Q370 TOOL PATH OVERLAP ■ Q351 CLIMB OR UP-CUT ■ Q366 PLUNGE	DEF-active	Page 449
111 CONTOUR MILLING In connection with a TNC7 go, the definition of specific contour-milling cycle parameters is not available to you.	DEF-active	Page 449
125 POSITIONING Definition of the positioning behavior with CYCL CALL PAT ■ Q345 SELECT POS. HEIGHT	DEF-active	Page 450
120 PROBING Definition of specific touch probe cycle parameters ■ Q320 SET-UP CLEARANCE ■ Q260 CLEARANCE HEIGHT ■ Q301 MOVE TO CLEARANCE	DEF-active	Page 450

18.6.2 Entering GLOBAL DEF definitions

Insert
NC function

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select **GLOBAL DEF**
- ▶ Select the desired **GLOBAL DEF** function (e.g., **100 GENERAL**)
- ▶ Enter the required definitions

18.6.3 Using GLOBAL DEF information

If you entered the corresponding **GLOBAL DEF** functions at program start, you can reference these globally valid values for the definition of any cycle.

Proceed as follows:

Insert
NC function

- ▶ Select **Insert NC function**
- The control opens the **Insert NC function** window.
- ▶ Select and define **GLOBAL DEF**
- ▶ Select **Insert NC function** again
- ▶ Select the desired cycle (e.g., **200 DRILLING**)
- If the cycle includes global cycle parameters, the control superimposes the selection possibility **PREDEF** in the action bar or in the form as a selection menu.

PREDEF

- ▶ Select **PREDEF**
- The control then enters the word **PREDEF** in the cycle definition. This creates a link to the corresponding **GLOBAL DEF** parameter that you defined at the beginning of the program.

NOTICE

Danger of collision!

If you later edit the program settings with **GLOBAL DEF**, these changes will affect the entire NC program. This may change the machining sequence significantly. There is a danger of collision!

- ▶ Make sure to use **GLOBAL DEF** carefully. Simulate your program before executing it
- ▶ If you enter fixed values in the cycles, they will not be changed by **GLOBAL DEF**.

18.6.4 Global data valid everywhere

Parameters valid for all machining cycles **2xx**

Help graphic	Parameter
	Q200 Set-up clearance? Distance between tool tip and workpiece surface. This value has an incremental effect. Input: 0...99999.9999
	Q204 2nd set-up clearance? Distance in the tool axis between the tool and the workpiece (fixtures) at which no collision can occur. This value has an incremental effect. Input: 0...99999.9999
	Q253 Feed rate for pre-positioning? Feed rate at which the control moves the tool within a cycle. Input: 0...99999.999 or FMAX, FAUTO
	Q208 Feed rate for retraction? Feed rate at which the control retracts the tool. Input: 0...99999.999 or FMAX, FAUTO

Example

11 GLOBAL DEF 100 GENERAL ~	
Q200=+2	;SET-UP CLEARANCE ~
Q204=+50	;2ND SET-UP CLEARANCE ~
Q253=+750	;F PRE-POSITIONING ~
Q208=+999	;RETRACTION FEED RATE

18.6.5 Global data for drilling operations

The parameters apply to the drilling, tapping, and thread milling cycles **200** to **207**, **240**, and **241**.

Help graphic	Parameter
	Q256 Retract dist. for chip breaking? Value by which the control retracts the tool during chip breaking. This value has an incremental effect. Input: 0.1...99999.9999
	Q210 Dwell time at the top? Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input: 0...3600.0000
	Q211 Dwell time at the depth? Time in seconds that the tool remains at the hole bottom. Input: 0...3600.0000

Example

11 GLOBAL DEF 105 DRILLING ~	
Q256=+0.2	;DIST FOR CHIP BRKNG ~
Q210=+0	;DWELL TIME AT TOP ~
Q211=+0	;DWELL TIME AT DEPTH

18.6.6 Global data for milling operations with pocket cycles

The parameters apply to the cycles **233**, **251**, **253**, and **256**

Help graphic	Parameter
	Q370 Path overlap factor? Q370 x tool radius = stepover factor k. Input: 0.1...1999
	Q351 Direction? Climb=+1, Up-cut=-1 Type of milling operation. The direction of spindle rotation is taken into account. +1 = climb milling -1 = up-cut milling (If you enter 0, climb milling is performed.) Input: -1, 0, +1
	Q366 Plunging strategy (0/1/2)? Type of plunging strategy: 0 : Vertical plunging. The control plunges perpendicularly, regardless of the plunging angle ANGLE defined in the tool table. 1 : Helical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message 2 : Reciprocating plunge. In the tool table, the plunging angle ANGLE for the active tool must be defined as not equal to 0. Otherwise, the control will display an error message. The reciprocation length depends on the plunging angle. As a minimum value the control uses twice the tool diameter. Input: 0, 1, 2

Example

11 GLOBAL DEF 110 POCKET MILLING ~	
Q370=+1	;TOOL PATH OVERLAP ~
Q351=+1	;CLIMB OR UP-CUT ~
Q366=+1	;PLUNGE

18.6.7 Global data for milling operations with contour cycles



For the TNC7 go straight-cut control, the **GLOBAL DEF CNTR MLLNG** function has no functionality. This soft key was added for reasons of compatibility.

18.6.8 Global data for positioning behavior

The parameters apply to each fixed cycle that you call with the **CYCL CALL PAT** function.

Help graphic	Parameter
	Q345 Select positioning height (0/1) Retraction in the tool axis at the end of a machining step, return to the 2nd set-up clearance or to the position at the beginning of the unit. Input: 0, 1

Example

```
11 GLOBAL DEF 125 POSITIONING ~
```

```
Q345=+1 ;SELECT POS. HEIGHT
```

18.6.9 Global data for probing functions

The parameters are valid for all touch probe cycles **4xx**

Help graphic	Parameter
	Q320 Set-up clearance? Additional distance between touch point and ball tip. Q320 is active in addition to the SET_UP column in the touch probe table. This value has an incremental effect. Input: 0...99999.9999
	Q260 Clearance height? Coordinate in the tool axis at which no collision between touch probe and workpiece (fixtures) can occur. This value has an absolute effect. Input: -99999.9999...+99999.9999
	Q301 Move to clearance height (0/1)? Define how the touch probe will move between the measuring points: 0: Move to measuring height between measuring points 1: Move to clearance height between measuring points Input: 0, 1

Example

```
11 GLOBAL DEF 120 PROBING ~
```

```
Q320=+0 ;SET-UP CLEARANCE ~
```

```
Q260=+100 ;CLEARANCE HEIGHT ~
```

```
Q301=+1 ;MOVE TO CLEARANCE
```

18.7 Table access with SQL statements

18.7.1 Fundamentals

Application

If you would like to access content in a table or manipulate the tables (e.g., rename columns or rows), then use the available SQL commands.

The syntax of the SQL commands available on the control is strongly influenced by the SQL programming language but does not conform with it entirely. In addition, the control does not support the full scope of the SQL language.

Related topics

- Opening, reading and writing to freely definable tables

Further information: "NC functions for accessing tables with variables", Page 423

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function

In the NC software, table accesses occur through an SQL server. This server is controlled via the available SQL commands. The SQL commands can be defined directly in an NC program.

The server is based on a transaction model. A **transaction** consists of multiple steps that are executed together, thereby ensuring that the table entries are processed in an orderly and well-defined manner.

The SQL commands take effect in the **Program Run** operating mode and the **MDI** application.

Example of transaction:

- Assign variables to table columns for read- or write-access using **SQL BIND**
- Select data using **SQL EXECUTE** with the instruction **SELECT**
- Read, change, or add data using **SQL FETCH**, **SQL UPDATE**, or **SQL INSERT**
- Confirm or discard interaction using **SQL COMMIT** or **SQL ROLLBACK**
- Approve bindings between table columns and variables using **SQL BIND**



You must conclude all transactions that have been started—even exclusively reading accesses. Concluding the transaction is the only way to ensure that changes and additions are transferred, that locks are removed, and that used resources are released.

The **result set** contains a subset of a table file. It results from a **SELECT** query performed on the table.

The **result set** is created when a query is executed in the SQL server, thereby occupying resources there.

This query has the same effect as applying a filter to the table, so that only a subset of the data records becomes visible. To perform this query, the table file must be read at this point.

The SQL server assigns a **handle** to the **result set**, which enables you to identify the result set for reading or editing data and completing the transaction. The **handle** is the result of the query, which is visible in the NC program. The value 0 indicates an **invalid handle**, i.e. it was not possible to create a **result set** for that query. If no rows are found that satisfy the specified condition, an empty **result set** is created and assigned a valid **handle**.

Overview of SQL commands

The control provides the following SQL commands:

Syntax	Function	Further information
SQL BIND	SQL BIND establishes or removes connections between table columns and variables	Page 454
SQL SELECT	SQL SELECT reads out a single value from a table and does not open any transaction	Page 455
SQL EXECUTE	SQL EXECUTE opens a transaction for selected table columns and table rows or enables the use of other SQL instructions (miscellaneous functions).	Page 457
SQL FETCH	SQL FETCH transfers the values to the bound variables	Page 461
SQL ROLLBACK	SQL ROLLBACK discards all changes and concludes the transaction	Page 462
SQL COMMIT	SQL COMMIT saves all changes and concludes the transaction	Page 464
SQL UPDATE	SQL UPDATE expands the transaction to include the change of an existing row	Page 465
SQL INSERT	SQL INSERT creates a new table row	Page 467

Notes

NOTICE

Danger of collision!

Read and write accesses performed with the help of SQL commands always occur in metric units, regardless of the unit of measure selected for the table or the NC program.

If, for example, you save a length from a table to a Q parameter, then the value is thereafter always in metric units. If this value is then used for the purpose of positioning in an inch program (**L X+Q1800**), then an incorrect position will result.

- ▶ In inch programs, convert the read value prior to use

NOTICE

Danger of collision!

Even during simulation of an NC program, the control executes all SQL commands. Here, SQL commands may overwrite table values, for example, that also take effect in the **Program Run** operating mode. The overwritten values can lead to unexpected behavior or incorrect positioning during subsequent machining operations. There is a danger of collision.

- ▶ Skip SQL commands during simulations by using conditional jumps, for example
- ▶ Use **FN 18: SYSREAD ID992 NR16** to check whether the NC program is active in a different operating mode or in the **Simulation**.

- HEIDENHAIN recommends that you use SQL functions instead of **FN 26**, **FN 27**, or **FN 28** in order to achieve maximum HDR hard-disk speeds for table applications and to reduce the amount of computing power used.

18.7.2 Binding a variable to a table column with SQL BIND

Application

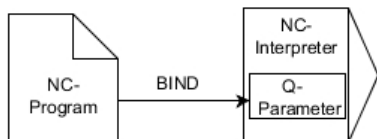
SQL BIND links a variable to a table column. The SQL commands **FETCH**, **UPDATE**, and **INSERT** evaluate this binding (assignment) during data transfer between the **result set** and the NC program.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Program any number of bindings with **SQL BIND...**, before using the **FETCH**, **UPDATE**, or **INSERT** commands.

An **SQL BIND** command without a table name or column name cancels the binding. At the latest, the binding is terminated at the end of the NC program or subprogram.

Input

11 SQL BIND Q881
"Tab_example.Position_Nr"

; Bind **Q881** to the "Position_No" column of the "Tab_Example" table

To navigate to this function:

Insert NC function ► All functions ► FN ► SQL table access ► SQL BIND

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL BIND	Syntax initiator for the BIND SQL command
Parameter	Variable to be bound
Name or Parameter	Table name and table column, separated by . or string parameter with definition Text or string parameter Only when binding a variable

Notes

- Enter the path of the table or a synonym as the table name.
Further information: "Executing SQL statements with SQL EXECUTE", Page 457
- During the read and write operations, the control considers only those columns that you have specified by means of the **SELECT** command. If you specify columns without a binding in the **SELECT** command, then the control interrupts the read or write operation with an error message.

18.7.3 Reading out a table value with SQL SELECT

Application

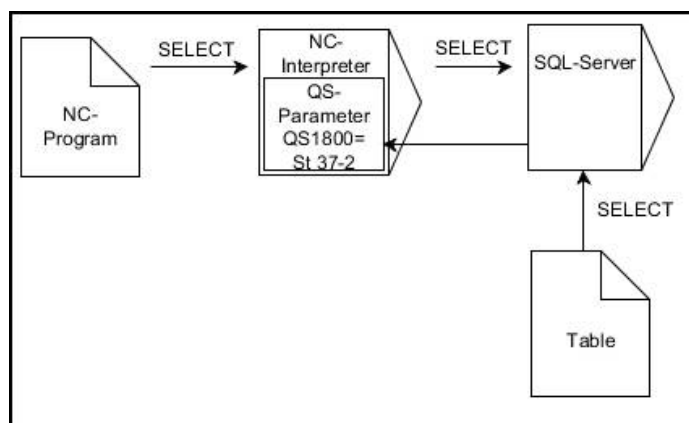
SQL SELECT reads a single value from a table and saves the result in the defined variable.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Black arrows and associated syntax show internal processes of **SQL SELECT**

With **SQL SELECT**, there is neither a transaction nor a binding between the table column and variable. The control does not consider any bindings that may exist to the specified column. The control copies the read value only into the parameter specified for the result.

Input

11 SQL SELECT Q5 "SELECT Mess_X
FROM Tab_Example WHERE
Position_NR==3"

; Save the value of the "Position_No" column
of the "Tab_Example" table in **Q5**

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **SQL table access** ► **SQL SELECT**

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL SELECT	Syntax initiator for the SELECT SQL command
Parameter	Variable in which the control stores the result
Name or Parameter	SQL statement or string parameter with the definition containing: ■ SELECT: Table column of the value to be transferred ■ FROM: Synonym or absolute path of the table (path in single quotation marks) ■ WHERE: Column designation, condition, and comparison value (variable after : in single quotation marks) Text, string parameter, or format string

Notes

- You can select multiple values or multiple columns using the SQL command **SQL EXECUTE** and the **SELECT** statement.
- After the **WHERE** syntax element, you can define the comparison value as a variable as well. If you use numerical parameters for the comparison, the control rounds the defined value to an integer. If you use string parameters, the control uses the defined value.
- For the instructions within the SQL command, you can likewise use single or combined string parameters.
Further information: "Concatenating values of string parameters", Page 435
- If you check the content of a string parameter on the **QPARA** tab of the **Status** workspace, you possibly do not see the complete content.
Further information: User's Manual for Setup and Program Run

Example

The result of the following NC programs is identical.

0 BEGIN PGM SQL_READ_WMAT MM	
1 SQL Q1800 "CREATE SYNONYM my_table FOR 'TNC:\table \WMAT.TAB'"	; Create synonym
2 SQL BIND QS1800 "my_table.WMAT"	; Bind QS parameters
3 SQL QL1 "SELECT WMAT FROM my_table WHERE NR==3"	; Define search
* - ...	
* - ...	
3 SQL SELECT QS1800 "SELECT WMAT FROM my_table WHERE NR==3"	; Read and save value
* - ...	
* - ...	
3 DECLARE STRING QS1 = "SELECT "	
4 DECLARE STRING QS2 = "WMAT "	
5 DECLARE STRING QS3 = "FROM "	
6 DECLARE STRING QS4 = "my_table "	
7 DECLARE STRING QS5 = "WHERE "	
8 DECLARE STRING QS6 = "NR==3"	
9 QS7 = QS1 QS2 QS3 QS4 QS5 QS6	
10 SQL SELECT QL1 QS7	
* - ...	

18.7.4 Executing SQL statements with SQL EXECUTE

Application

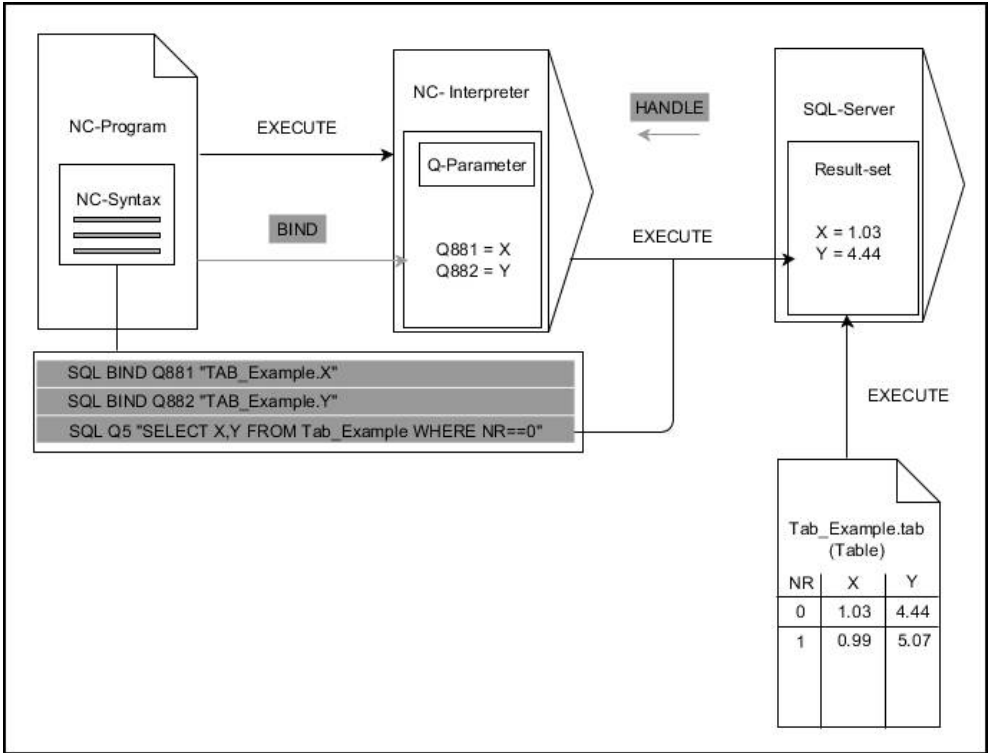
SQL EXECUTE can be used in conjunction with various SQL instructions.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL EXECUTE**. The gray arrows and associated syntax do not directly belong to the **SQL EXECUTE** command.

The control provides the following SQL statements in the **SQL EXECUTE** command:

Instruction	Function
SELECT	Select data
CREATE SYNONYM	Create synonym (replace long path names with short names)
DROP SYNONYM	Delete synonym
CREATE TABLE	Generate table
COPY TABLE	Copy table
RENAME TABLE	Rename table
DROP TABLE	Delete table
INSERT	Insert table rows
UPDATE	Update table rows
DELETE	Delete table rows
ALTER TABLE	■ Add table columns using ADD ■ Delete table columns using DROP
RENAME COLUMN	Rename table columns

SQL EXECUTE with the SQL SELECT instruction

The SQL server places the data in the **result set** row-by-row. The rows are numbered in ascending order, starting with 0. The SQL commands **FETCH** and **UPDATE** use these row numbers (the **INDEX**).

SQL EXECUTE, in conjunction with the SQL instruction **SELECT**, selects the table values, transfers them to the **result set**, and always opens a transaction in the process. Unlike the SQL command **SQL SELECT**, the combination of **SQL EXECUTE** and the **SELECT** instruction allows multiple columns and rows to be selected at the same time.

Enter the search criteria in the **SQL ... "SELECT...WHERE..."** function. You thereby restrict the number of rows to be transferred. If you do not use this option, then all of the rows in the table are loaded.

Enter the ordering criteria in the **SQL ... "SELECT...ORDER BY..."** function. This entry consists of the column designation and the keyword **ASC** for ascending or **DESC** for descending order. If you do not use this option, then rows will be stored in a random order.

With the function **SQL ... "SELECT...FOR UPDATE"**, you can lock the selected rows for other applications. Other applications can continue to read these rows but are unable to change them. If you make changes to the table entries, then it is absolutely necessary to use this option.

Empty result set: If no rows meet the search criterion, then the SQL server returns a valid **HANDLE** without table entries.

Conditions for WHERE entries

Condition	Programming
Equals	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Empty	IS NULL
Not empty	IS NOT NULL

Linking multiple conditions:

Logical AND	AND
Logical OR	OR

Notes

- If you use the **SQL EXECUTE** NC function, the control will insert the **SQL** syntax element into the NC program only.
- You can also define synonyms for tables that have not yet been generated.
- The sequence of the columns in the created file corresponds to the sequence within the **AS SELECT** instruction.
- For the instructions within the SQL command, you can likewise use single or combined string parameters.
Further information: "Concatenating values of string parameters", Page 435
- After the **WHERE** syntax element, you can define the comparison value as a variable as well. If you use numerical parameters for the comparison, the control rounds the defined value to an integer. If you use string parameters, the control uses the defined value.
- If you check the content of a string parameter on the **QPARA** tab of the **Status** workspace, you possibly do not see the complete content.
Further information: User's Manual for Setup and Program Run

Example

Example: selecting table rows

11 SQL BIND Q881 "Tab_Example.Position_Nr"	
12 SQL BIND Q882 "Tab_Example.Measure_X"	
13 SQL BIND Q883 "Tab_Example.Measure_Y"	
14 SQL BIND Q884 "Tab_Example.Measure_Z"	
* - ...	
20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"	

Example: selecting table rows with the WHERE function

20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example WHERE Position_Nr<20"	

Example: selecting table rows with the WHERE function and Q parameter

20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example WHERE Position_Nr==:'Q11'"	

Example: defining the table name with absolute path information

20 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM 'V:\table\Tab_Example' WHERE Position_Nr<20"	

Example: generating a table with CREATE TABLE

0 BEGIN PGM SQL_CREATE_TAB MM	
1 SQL Q10 "CREATE SYNONYM NEW FOR 'TNC: \table\NewTab.TAB'"	; Create synonym
2 SQL Q10 "CREATE TABLE NEW AS SELECT X,Y,Z FROM 'TNC:\prototype_for_NewTab.tab'"	; Create columns X, Y and Z of an existing table as a new table
3 END PGM SQL_CREATE_TAB MM	

0	BEGIN PGM SQL_CREATE_TABLE_QS MM	
1	DECLARE STRING QS1 = "CREATE TABLE "	
2	DECLARE STRING QS2 = "'TNC:\nc_prog\demo \Doku\NewTab.t' "	
3	DECLARE STRING QS3 = "AS SELECT "	
4	DECLARE STRING QS4 = "DL,R,DR,L "	
5	DECLARE STRING QS5 = "FROM "	
6	DECLARE STRING QS6 = "'TNC:\table\tool.t'"	
7	QS7 = QS1 QS2 QS3 QS4 QS5 QS6	
8	SQL Q1800 QS7	
9	END PGM SQL_CREATE_TABLE_QS MM	

18.7.5 Reading a line from a result set with SQL FETCH

Application

SQL FETCH reads a row from the **result set**. The values of the individual cells are stored by the control in the bound variables. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**.

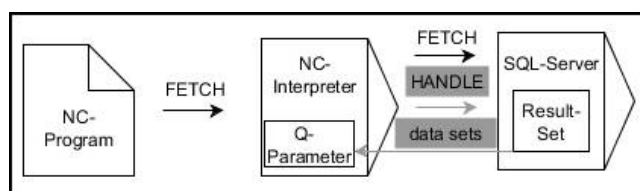
SQL FETCH takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL FETCH**. The gray arrows and associated syntax do not directly belong to the **SQL FETCH** command.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

```
11 SQL FETCH Q1 HANDLE Q5 INDEX
5 IGNORE UNBOUND UNDEFINE
MISSING ; Read out result of transaction Q5 line 5
```

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL FETCH	Syntax initiator for the FETCH SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction
INDEX	Row number within the Result-set as a number or variable If not specified, the control accesses line 0. Optional syntax element
IGNORE UNBOUND	For the machine manufacturer only Optional syntax element
UNDEFINE MISSING	For the machine manufacturer only Optional syntax element

Example

Transfer line number in the Q parameter

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
12 SQL BIND Q882 "Tab_Example.Measure_X"
13 SQL BIND Q883 "Tab_Example.Measure_Y"
14 SQL BIND Q884 "Tab_Example.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM
Tab_Example"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

18.7.6 Discarding changes to a transaction using SQL ROLLBACK

Application

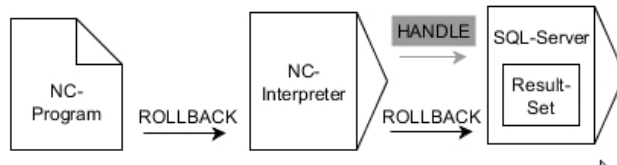
SQL ROLLBACK discards all of the changes and additions of a transaction. The transaction is defined via the **HANDLE** to be specified.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL ROLLBACK**. The gray arrows and associated syntax do not directly belong to the **SQL ROLLBACK** command.

The function of the SQL command **SQL ROLLBACK** depends on the **INDEX**:

- Without **INDEX**:
 - The control discards all changes and additions of the transaction
 - The control resets a lock set with **SELECT...FOR UPDATE**
 - The control completes the transaction (the **HANDLE** loses its validity)
- With **INDEX**:
 - Only the indexed row remains in the **result set** (the control removes all of the other rows)
 - The control discards any changes and additions that may have been made in the non-specified rows
 - The control locks only those rows indexed with **SELECT...FOR UPDATE** (the control resets all of the other locks)
 - The specified (indexed) row is then the new Row 0 of the **result set**
 - The control does **not** complete the transaction (the **HANDLE** keeps its validity)
 - The transaction must be completed manually with **SQL ROLLBACK** or **SQL COMMIT** at a later time

Input

11 SQL ROLLBACK Q1 HANDLE Q5 INDEX 5	; Delete all rows of transaction Q5 except row 5

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL ROLLBACK	Syntax initiator for the ROLLBACK SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction
INDEX	Row number within the Result-set as a number or variable that is retained Optional syntax element If not specified, the control discards all changes and additions to the transaction

Example

11	SQL BIND Q881 "Tab_Example.Position_Nr"
12	SQL BIND Q882 "Tab_Example.Measure_X"
13	SQL BIND Q883 "Tab_Example.Measure_Y"
14	SQL BIND Q884 "Tab_Example.Measure_Z"
* - ...	
21	SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM Tab_Example"
* - ...	
31	SQL FETCH Q1 HANDLE Q5 INDEX+Q2
* - ...	
41	SQL ROLLBACK Q1 HANDLE Q5

18.7.7 Completing a transaction with SQL COMMIT

Application

SQL COMMIT simultaneously transfers all of the rows that have been changed and added in a transaction back into the table. The transaction is defined via the **HANDLE** to be specified. In this context, a lock that has been set with **SELECT...FOR UPDATE** resets the control.

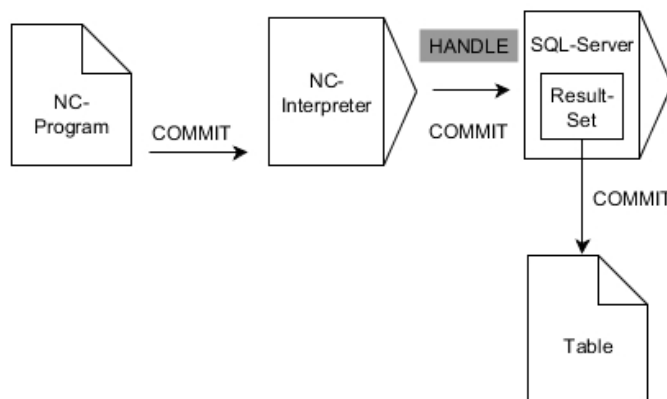
Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function

The assigned **HANDLE** (operation) loses its validity.



Black arrows and associated syntax indicate internal processes of **SQL COMMIT**.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

```
11 SQL COMMIT Q1 HANDLE Q5
```

```
; Complete all rows of transaction Q5 and  
update table
```

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL COMMIT	Syntax initiator for the COMMIT SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction

Example

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
```

```
12 SQL BIND Q882 "Tab_Example.Measure_X"
```

```
13 SQL BIND Q883 "Tab_Example.Measure_Y"
```

```
14 SQL BIND Q884 "Tab_Example.Measure_Z"
```

```
* - ...
```

```
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM  
Tab_Example"
```

```
* - ...
```

```
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

```
* - ...
```

```
41 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
```

```
* - ...
```

```
51 SQL COMMIT Q1 HANDLE Q5
```

18.7.8 Changing the row of a result set with SQL UPDATE**Application**

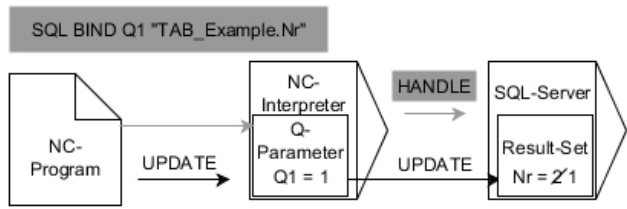
SQL UPDATE changes a row in the **result set**. The new values of the individual cells are copied by the control from the bound variables. The transaction is defined through the **HANDLE** to be specified, and the row is defined by the **INDEX**. The control completely overwrites the already existing rows in the **result set**.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Black arrows and the associated syntax show internal **SQL UPDATE** processes. Gray arrows and the associated syntax are not directly associated with the **SQL UPDATE** command.

SQL UPDATE takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**).

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

```
11 SQL UPDATE Q1 HANDLE Q5 index5 ; Complete all rows of transaction Q5 and
   RESET UNBOUND                  update table
```

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL UPDATE	Syntax initiator for the UPDATE SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction
INDEX	Row number within the Result-set as a number or variable Optional syntax element If not specified, the control accesses line 0.
RESET UNBOUND	For the machine manufacturer only Optional syntax element

Note

When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example

Transfer line number in the Q parameter

```

11 SQL BIND Q881 "TAB_EXAMPLE.Position_Nr"
12 SQL BIND Q882 "TAB_EXAMPLE.Measure_X"
13 SQL BIND Q883 "TAB_EXAMPLE.Measure_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.Measure_Z"
* - ...
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y,Measure_Z FROM
    TAB_EXAMPLE"
* - ...
31 SQL FETCH Q1 HANDLE Q5 INDEX+Q2

```

Program the row number directly

```

31 SQL UPDATE Q1 HANDLE Q5 INDEX5

```

18.7.9 Creating a new row in the result set with SQL INSERT

Application

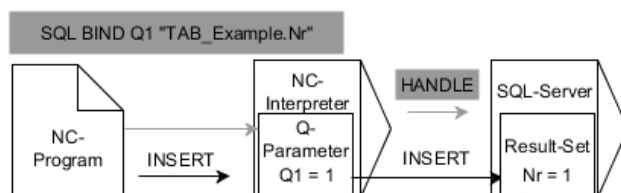
SQL INSERT creates a new row in the **result set**. The values of the individual cells are copied by the control from the bound variables. The transaction is defined via the **HANDLE** to be specified.

Requirements

- Code number 555343
- Table exists
- Appropriate table name

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.

Description of function



Black arrows and associated syntax indicate internal processes of **SQL INSERT**. The gray arrows and associated syntax do not directly belong to the **SQL INSERT** command.

SQL INSERT takes all of the columns into consideration that contain the **SELECT** instruction (SQL command **SQL EXECUTE**). Table columns without a corresponding **SELECT** instruction (not contained in the query result) are described by the control with default values.

The control shows in the defined variable whether the read operation was successful (0) or incorrect (1).

Input

```
11 SQL INSERT Q1 HANDLE Q5
```

```
; Create a new row in transaction Q5
```

The NC function includes the following syntax elements:

Syntax element	Meaning
SQL INSERT	Syntax initiator for the INSERT SQL command
Parameter	Variable in which the control stores the result
HANDLE	Variable with identification of the transaction

Note

When writing to tables, the control checks the lengths of the string parameters. If the entries exceed the length of the columns to be described, then the control outputs an error message.

Example

```
11 SQL BIND Q881 "Tab_Example.Position_Nr"
```

```
12 SQL BIND Q882 "Tab_Example.Measure_X"
```

```
13 SQL BIND Q883 "Tab_Example.Measure_Y"
```

```
14 SQL BIND Q884 "Tab_Example.Measure_Z"
```

```
* - ...
```

```
21 SQL Q5 "SELECT Position_Nr,Measure_X,Measure_Y, Measure_Z FROM  
Tab_Example"
```

```
* - ...
```

```
31SQL INSERT Q1 HANDLE Q5
```


19

ISO

19.1 Fundamentals

Application

The ISO 6983 standard defines a universal NC syntax.

On the TNC7 go, you can program and execute NC programs using the supported ISO syntax elements.

Description of function

In connection with ISO programs, the TNC7 go provides the following possibilities:

- Transferring files to the control
Further information: User's Manual for Setup and Program Run
- Programming ISO programs on the control
Further information: "ISO syntax", Page 472
 - In addition to the standardized ISO syntax, you can program HEIDENHAIN-specific cycles as G functions.
Further information: "Cycles", Page 486
 - Coding in Klartext syntax allows you to use some NC functions in ISO programs.
Further information: "Klartext functions in ISO programming", Page 487
- Testing of NC programs using Simulation mode
Further information: "The Simulation workspace", Page 527
- Running NC programs
Further information: User's Manual for Setup and Program Run

Contents of an ISO program

An ISO program is structured as follows:

ISO syntax	Function
I	File type ISO programs have an *.i file name extension.
%NAME G71	Start and end of the program
G71	Unit of measure: mm
G70	Unit of measure: Inch
N10	NC block numbers In the optional machine parameter blockIncrement (no. 105409), you define the increment between the block numbers.
N20	
N30	
...	
N99999999	NC block number for the end of the program An NC program is incomplete without this NC block number. The control adds and updates the NC block numbers within the file automatically. The Program workspace exclusively shows successive numbers without taking the defined increment into account.
G07 X+0 ...	NC functions

Further information: "Contents of an NC program", Page 114

Contents of an NC block

N110 G07 G90 X+10 G43 F3000 M3

An NC block contains the following syntax elements:

ISO syntax	Function
G07	Syntax initiator
G90	Absolute or incremental input Further information: "Absolute and incremental input", Page 472
X+10	Coordinates Further information: "Fundamentals of coordinate definitions", Page 172
G43	Tool radius compensation Further information: "Tool radius compensation", Page 477
F3000	Feed rate Further information: "Feed rate", Page 473
M3	Miscellaneous functions (M functions) Further information: "Miscellaneous functions", Page 381

Notes

- The **Insert NC function** window allows you add ISO syntax, too.
Further information: "The Insert NC function window", Page 133
- You can call a Klartext program within an ISO program.
Further information: "Calling an NC program", Page 475
- You can call a Klartext program within an ISO program (e.g., to use NC functions that are available only for Klartext programming).
- Use the up and down cursor keys to search for identical syntax elements in different NC blocks.
Further information: "Searching for identical syntax elements in different NC blocks", Page 127

19.2 ISO syntax

Keys

You can use the following keys to insert ISO syntax:

Key	ISO syntax	Further information
TOOL CALL	Tool call T	Page 473
LBL SET	Label G98	Page 474
LBL CALL	Subprogram call and program-section repeat L	Page 475 Page 475
STOP	Stop in the NC program G38	Page 477

Absolute and incremental input

The control provides the following possibilities to enter dimensions:

Syntax	Meaning
G90	Absolute input always references an origin. For Cartesian coordinates, the origin is the datum.
G91 corresponds to the I Klartext syntax	Incremental input always references the previously programmed coordinates.

Tool axis

In some NC functions, you can select a tool axis in order, for example, to define the working plane.



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

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

The control differentiates between the following tool axes:

Syntax	Working plane
G17 corresponds to the Z tool axis	XY
G18 corresponds to the Y tool axis	ZX
G19 corresponds to the X tool axis	YZ

Workpiece blank

Use the **G30** and **G31** NC functions to define a cuboid workpiece blank for simulation in the NC program.

You define the cuboid by entering a MIN point for the bottom front left corner and a MAX point for the top rear right corner.

N10 G30 G17 X+0 Y+0 Z-40	; Define MIN point
N20 G31 X+100 Y+100 Z+0	; Define MAX point

G30 and **G31** correspond to the Klartext syntax **BLK FORM 0.1** and **BLK FORM 0.2**.

Further information: "Defining a workpiece blank with BLK FORM", Page 152

With **G17**, **G18**, and **G19** you define the tool axis.

Further information: "Tool axis", Page 472

With the Klartext syntax, you can additionally define cylindrical workpiece blanks with **BLK FORM CYLINDER**.

Further information: "Cylindrical workpiece blank with BLK FORM CYLINDER", Page 154

Tools

Tool call

With the **T** NC function, you call a tool in the NC program.

T corresponds to the **TOOL CALL** Klartext syntax.

Further information: "Using TOOL CALL to call a tool", Page 162

With **G17**, **G18**, and **G19**, you define the tool axis.

Further information: "Tool axis", Page 472

Cutting data

Spindle speed

The spindle speed **S** is defined as spindle revolutions per minute (rpm).

Alternatively, the constant cutting speed **VC** in meters per minute (m/min) can be defined.

N110 T1 G17 S(VC = 200)	; Tool call with constant cutting speed

Further information: "Spindle speed S", Page 166

Feed rate

The feed rate for linear axes is defined in millimeters per minute (mm/min).

In inch programs, the feed rate must be defined in 1/10 inch/min.

The feed rate for rotary axes is defined in degrees per minute (°/min).

The feed rate can be defined with an accuracy of three decimal places.

Further information: "Feed rate F", Page 167

Tool pre-selection

When you use the **G51** NC function, the control prepares a tool in the magazine, thus reducing the tool-change time.



Refer to your machine manual.

A tool pre-selection defined with **G99** is a machine-dependent function.

N110 G51 T3

; Tool pre-selection

G51 corresponds to the **TOOL DEF** Klartext syntax.

Further information: "Tool pre-selection using TOOL DEF", Page 169

Tool movements

The NC function **G07** allows you to program a paraxial positioning block in any direction.

N110 G07 Z+100 M3

; Paraxial positioning block in Z direction

Further information: "Fundamentals of traverse movements", Page 175

Programming techniques

Subprograms and program-section repeats

Programming techniques are useful in structuring your NC program and avoiding unnecessary repeats. By using subprograms, you need to define machining positions for multiple tools only once, for example. Program-section repeats, on the other hand, help you avoid multiple programming of identical, successive NC blocks or program sequences. By combining and nesting these two programming techniques, you can keep your NC programs rather short and restrict changes to a few central program locations.

Further information: "Subprograms and program section repeats with the label LBL", Page 178

Defining labels

With the **G98** NC function, you define a new label in the NC program.

Each label must be unambiguously identifiable in the NC program by its number or name. If a number or a name exists twice in an NC program, the control shows a warning before the NC block.

If you define a label after **M30** or **M2**, it corresponds to a subprogram. Subprograms must always be concluded with a **G98 L0**. This number is the only one which may exist any number of times in the NC program.

N110 G98 L1

; Start of subprogram defined by a number

N120 G07 Z+100

; Retract in Z direction

N130 G98 L0

; End of subprogram

N110 G98 L "UP"

; Start of subprogram defined by a name

G98 L corresponds to the **LBL** Klartext syntax.

Further information: "Defining a label with LBL SET", Page 178

Calling a subprogram

With the **L** NC function, you call a subprogram programmed after **M30** or **M2**.

When the control reads the **L** NC function, it will jump to the defined label and continue execution of the NC program from this NC block. When the control reads **G98 L0**, it will jump back to the next NC block after the call with **L**.

N110 L1 ; Call subprogram

L without **G98** corresponds to the **CALL LBL** Klartext syntax.

Further information: "Calling a label with CALL LBL", Page 179



In order to define a certain number of desired repetitions (e.g., **L1.3**), program a program-section repeat.

Further information: "Program-section repeat", Page 475

Program-section repeat

Program-section repeats allow you to have a particular program section executed any number of times. The program section must start with a **G98 L** label definition and end with **L**. With the numeral after the decimal point, you can define optionally how often you want the control to repeat this program section.

N110 L1.2 ; Call label 1 twice

L without **98** and the numeral after the decimal point correspond to the **CALL LBL REP** Klartext syntax.

Further information: "Program-section repeats", Page 181

Selection functions

Further information: "Selection functions", Page 194

Calling an NC program

With the **%** NC function, you can call another, separate NC program from within an NC program.

N110 %TNC:\nc_prog\reset.i ; Call NC program

% corresponds to the **CALL PGM** Klartext syntax.

Further information: "Calling an NC program with CALL PGM", Page 194

Activating a datum table in the NC program

With the **:%TAB:** NC function, you can activate a datum table from within an NC program.

N110 %:TAB: "TNC:\table\zeroshift.d" ; Activate datum table

:%TAB: corresponds to the **SEL TABLE** Klartext syntax.

Further information: "Activating a datum table in the NC program", Page 321

Selecting a point table

With the **:%PAT:** NC function, you can activate a point table from within an NC program.

N110 %:PAT: "TNC:\nc_prog\positions.pnt"	; Activate point table

:%PAT corresponds to the **SEL PATTERN** Klartext syntax.

Further information: "Selecting the point table in the NC program with SEL PATTERN", Page 207

Selecting and calling an NC program

With the **:%PGM:** NC function, you can select another, separate NC program. With the **%<>%** NC function, you call the selected NC program at a different location in the active NC program.

N110 %:PGM: "TNC:\nc_prog\reset.i"	; Select NC program
* - ...	
N210 %<>%	; Call the selected NC program

:%PGM: and **%<>%** correspond to the **SEL PGM** and **CALL SELECTED PGM** Klartext syntax.

Further information: "Calling an NC program with CALL PGM", Page 194

Further information: "Selecting an NC program and calling it with SEL PGM and CALL SELECTED PGM", Page 196

Defining an NC program as a cycle

With the **G: :** NC function, you can define another NC program as a machining cycle from within an NC program.

N110 G: : "TNC:\nc_prog\cycle.i"	; Define NC program as a machining cycle

G: : corresponds to the **SEL CYCLE** Klartext syntax.

Further information: "Defining and calling an NC program as cycle", Page 144

Cycle call

For cycles that remove material, you have to enter not only the cycle definition, but also the cycle call in the NC program. The call always refers to the machining cycle that was defined last in the NC program.

The control provides the following options for calling a cycle:

Syntax	Meaning
G79 corresponds to the CYCL CALL Klartext syntax	The control calls the most recently programmed machining cycle at the last programmed position.
G79 PAT corresponds to the CYCL CALL PAT Klartext syntax	The control calls the most recently programmed machining cycle at all positions you have defined in a point table.

Syntax	Meaning
M89 and M99	With M99 , the control executes the most recently programmed machining cycle at the most recently programmed position. With M89 , the control executes the most recently programmed machining cycle after each positioning block until it reads M99 .
N110 G79 M3	; Call cycle
N110 G79 PAT F200 M3	; Call cycle at all positions in the point table
N110 G07 X+0 M89	; Call cycle at the defined position and for each new positioning block
N120 G07 X+25	
N130 G07 Y+25 M99	; Call cycle for the last time at the defined position

Further information: "Calling cycles", Page 143

Tool radius compensation

When tool radius compensation is active, the control will no longer reference the positions in the NC program to the tool center point, but to the cutting edge.

An NC block can contain the following tool radius compensations:

Syntax	Meaning
G40 corresponds to the R0 Klartext syntax	Reset an active tool radius compensation, positioning based on the tool center point
G43 corresponds to the R+ Klartext syntax	Tool radius compensation, extends the traverse
G44 corresponds to the R- Klartext syntax	Tool radius compensation, shortens the traverse

Further information: "Tool radius compensation", Page 344

Miscellaneous functions (M functions)

Use miscellaneous functions to activate or deactivate functions of the control and to influence the behavior of the control.

Further information: "Miscellaneous functions", Page 381

G38 corresponds to the **STOP** Klartext syntax.

Further information: "Miscellaneous functions M", Page 382

Programming variables

The control provides the following options for programming variables in ISO programs:

Function group	Further information
Basic arithmetic operations	Page 479
Trigonometric functions	Page 480
Circle calculations	Page 481
Jump commands	Page 482
Special functions	Page 484
String functions	Corresponds to the Klartext syntax Page 432
Counters	Corresponds to the Klartext syntax Page 443
Calculations using formulas	Corresponds to the Klartext syntax Page 427

The control distinguishes between the **Q**, **QL**, **QR**, and **QS** variable types (parameter types).

Further information: "Programming with variables", Page 391



Not all NC functions for programming variables are available in ISO programs (e.g., accessing tables with SQL statements).

Further information: "Table access with SQL statements", Page 451

Basic arithmetic operations

With the **D01** through **D05** functions, you can calculate values within your NC program. If you want to calculate with variables, you need to assign an initial value to each variable by means of the **D00** function.

The control provides the following functions:

Syntax	Meaning
D00	Assignment Assign a value or the Undefined status
D01	Addition Calculate and assign the sum of two values
D02	Subtraction Calculate and assign the difference of two values.
D03	Multiplication Calculate and assign the product of two values.
D04	Division Calculate and assign the quotient of two values Restriction: You cannot divide by 0
D05	Square root Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

N110 D00 Q5 P01 +60 ; Assignment Q5 = 60

N110 D01 Q1 P01 -Q2 P02 -5 ; Addition Q1 = -Q2+(-5)

N110 D02 Q1 P01 +10 P02 +5 ; Subtraction Q1 = +10- (+5)

N110 D03 Q2 P01 +3 P02 +3 ; Multiplication Q2 = 3*3

N110 D04 Q4 P01 +8 P02 +Q2 ; Division Q4 = 8/Q2

N110 D05 Q20 P01 4 ; Square root Q20 = $\sqrt{4}$

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Basic arithmetic folder", Page 404



HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.

Further information: "Formulas in the NC program", Page 427

Trigonometric functions

You can use these functions to calculate trigonometric functions for purposes such as programming variable triangular contours.

The control provides the following functions:

Syntax	Meaning
D06	Sine Calculate and assign the sine of an angle in degrees
D07	Cosine Calculate and assign the cosine of an angle in degrees
D08	Root of the sum of squares Calculate and assign the length based on two values (e.g., to calculate the third side of a triangle).
D13	Angle Calculate and assign the angle from the opposite side and the adjacent side using arctan or from the sine and cosine of the angle ($0 < \text{angle} < 360^\circ$)

N110 D06 Q20 P01 -Q5 ; Sine, $Q20 = \sin(-Q5)$

N110 D07 Q21 P01 -Q5 ; Cosine, $Q21 = \cos(-Q5)$

N110 D08 Q10 P01 +5 P02 +4 ; Root of the sum of squares, $Q10 = \sqrt{5^2+4^2}$

N110 D13 Q20 P01 +10 P02 -Q1 ; Angle, $Q20 = \arctan(25/-Q1)$

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, P02 etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Trigonometric functions folder", Page 407



HEIDENHAIN recommends direct formula input, as this allows you to program multiple arithmetic operations in one NC block.

Further information: "Formulas in the NC program", Page 427

Circle calculation

These functions allow you to calculate the center of a circle and the radius of the circle based on the coordinates of three or four points on the circle (e.g., the position and size of a circle segment).

The control provides the following functions:

Syntax	Meaning
D23	Circle data from three points on the circle The control saves the determined values in three successive Q parameters so that you only need to program the number of the first variable.
D24	Circle data from four points on the circle The control saves the determined values in three successive Q parameters so that you only need to program the number of the first variable.

N110 D23 Q20 P01 Q30

; Circle data from three points on the circle

N110 D24 Q20 P01 Q30

; Circle data from four points on the circle

D corresponds to the **FN** Klartext syntax.
The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.
P01, P02 etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Circle calculation folder", Page 408

Jump commands

In if-then decisions, the control compares a variable or fixed value with another variable or fixed value. If the condition is fulfilled, the control jumps to the label programmed for the condition.

If the condition is not fulfilled, the control continues with the next NC block.

The control provides the following functions:

Syntax	Meaning
D09	Jump if equal If both values are equal, the control jumps to the defined label.
	Jump if undefined If the variable is undefined, the control jumps to the defined label.
	Jump if defined If the variable is defined, the control jumps to the defined label.
D10	Jump if not equal If both values are not equal, the control jumps to the defined label.
D11	Jump if greater than If the first value is greater than the second one, the control jumps to the defined label.
D12	Jump if less than If the first value is less than the second one, the control jumps to the defined label.

N110 D09 P01 Q1 P02 Q3 P03 10 ; Jump if equal

N110 D09 P01 Q1 IS UNDEFINED P03 Q5 ; Jump if undefined

N110 D09 P01 Q1 IS DEFINED P03 QS5 ; Jump if defined

N110 D10 P01 +10 P02 -Q5 P03 10 ; Jump if not equal

N110 D11 P01 Q1 P02 +10 P03 QS5 ; Jump if greater than

N110 D12 P01 Q5 P02 +0 P03 Q5 ; Jump if less than

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, P02 etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Further information: "The Jump commands folder", Page 410

Functions for accessing tables with variables

You can open any table and subsequently write to it or read from it.

If you select the path using the selection window, the control by default only displays freely definable tables ***.tab**. You can change the filter to **All Files (*.*)**.

The control provides the following functions:

Syntax	Meaning
D26	Opening a table Further information: "Opening a table with FN 26: TABOPEN", Page 423
D27	Writing to a table Further information: "Writing to a table with FN 27: TABWRITE", Page 423
D28	Read table content Further information: "Reading table content with FN 28: TABREAD", Page 426

N110 D26 TNC:\DIR1\TAB1.TAB	; Open table
N110 Q5 = 3.75	; Define the value for the Radius column
N120 Q6 = -5	; Define the value for the Depth column
N130 Q7 = 7.5	; Define the value for the D column
N140 D27 P01 5 / "Radius,Depth,D" = Q5	; Write defined values to the table
N110 D28 Q10 = 6 / "X,Y,D"	; Read numerical values from the X , Y , and D columns
N120 D28 QS1 = 6 / "DOC"	; Read the alphanumeric value from the DOC column

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

Special functions

The control provides the following functions:

Syntax	Meaning
D14	Output error messages Further information: "Output error messages with FN 14: ERROR", Page 412
D16	Output formatted texts Further information: "Outputting text formatted with FN 16: F-PRINT", Page 413
D18	Read system data Further information: "Read system data with FN 18: SYSREAD", Page 420
D19	Transfer values to the PLC Further information: "Special functions defining the machine behavior", Page 616
D20	Synchronize NC and PLC Further information: "Special functions defining the machine behavior", Page 616
D29	Transfer values to the PLC Further information: "Special functions defining the machine behavior", Page 616
D37	Create user-defined cycles Further information: "Special functions defining the machine behavior", Page 616
D38	Send information from the NC program Further information: "Sending information from the NC program with FN 38: SEND", Page 422
N110 D14 P01 1000	; Output error message no. 1000
N110 D16 P01 TNC:\mask.a / TNC:\Prot1.txt	; Display the output file with D16 on the control screen
N110 D18 Q25 ID210 NR4 IDX3	; Save the active dimension factor of the Z axis in Q25
N110 D38 /"Q-Parameter Q1: %F Q23: %F" P02 +Q1 P02 +Q23	; Write the values of Q1 and Q23 to the log

D corresponds to the **FN** Klartext syntax.

The numbers of the ISO syntax correspond to the numbers of the Klartext syntax.

P01, **P02** etc. are considered as placeholders (e.g., for arithmetic operators included in the Klartext syntax).

NOTICE**Danger of collision!**

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control might become inoperable). For this reason, access to the PLC is password-protected. The functions **D19**, **D20**, **D29**, and **D37** enable HEIDENHAIN, the machine manufacturer, and suppliers to communicate with the PLC from within an NC program. It is not recommended that machine operators or NC programmers use this function. There is a danger of collision during the execution of these functions and during the subsequent machining operations!

- ▶ Only use the function in consultation after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider.
- ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

19.3 Cycles

Fundamentals

In ISO programs, you can use selected cycles with Klartext syntax in addition to the NC functions with ISO syntax. Programming is identical to Klartext programming.

The numbers of the Klartext cycles correspond to the numbers of the G functions. There are exceptions for earlier cycles that have numbers below **200**. In these cases, the corresponding G function number is mentioned in the cycle description.

Further information: "Available cycle groups", Page 148

Datum shift

With the **G53** or **G54** NC functions, you can program datum shifts. **G54** shifts the workpiece datum to the coordinates you define directly within this function. **G53** uses coordinate values from a datum table. A datum shift allows machining operations to be repeated at any locations on the workpiece.

N110 G54 X+0 Y+50	; Shift the workpiece datum to the defined coordinates
N110 G53 P01 10	; Shift the workpiece datum to the coordinates of table row 10

To reset a datum shift:

- Define the value **0** for each axis in function **G54**
- In function **G53**, select a table row where all columns have the value **0**

The control displays the following information in the **Status** workspace:

- Name and path of the active datum table
- Active datum number
- Comment from the **DOC** column of the active datum number

Notes



In the machine parameter **CfgDisplayCoordSys** (no. 127501) the machine manufacturer defines the coordinate system in which the status display shows an active datum shift.

- Datums from a datum table always reference the current workpiece preset.
- Before shifting the workpiece datum by means of a datum table, you need to activate the datum table with **%;TAB;**:

Further information: "Activating a datum table in the NC program", Page 475

- If you do not use **%;TAB;**, you have to activate the datum table manually.

Further information: "Activating the datum table manually", Page 320

19.4 Klartext functions in ISO programming

Fundamentals

In ISO programs, you can use selected NC functions with Klartext syntax in addition to the NC functions with ISO syntax. Programming is identical to Klartext programming.

For more information about programming, refer to the respective chapters describing the individual NC functions.

The following NC functions are available only in Klartext programs:

- Pattern definitions with **PATTERN DEF**
Further information: "Pattern definition with PATTERN DEF", Page 208
- Program defaults for cycles with **GLOBAL DEF**
Further information: "Program defaults for cycles", Page 445
- Table access with SQL statements
Further information: "Table access with SQL statements", Page 451

20

User aids

20.1 The Help workspace

Application

In the **Help** workspace, the control displays a help graphic for the current syntax element of an NC function or the integrated product aid **TNCguide**.

Related topics

- The **Help** application
Further information: "The Help application", Page 38
- User's Manual as the **TNCguide** integrated product aid
Further information: "User's Manual as integrated product aid: TNCguide", Page 37

Description of function

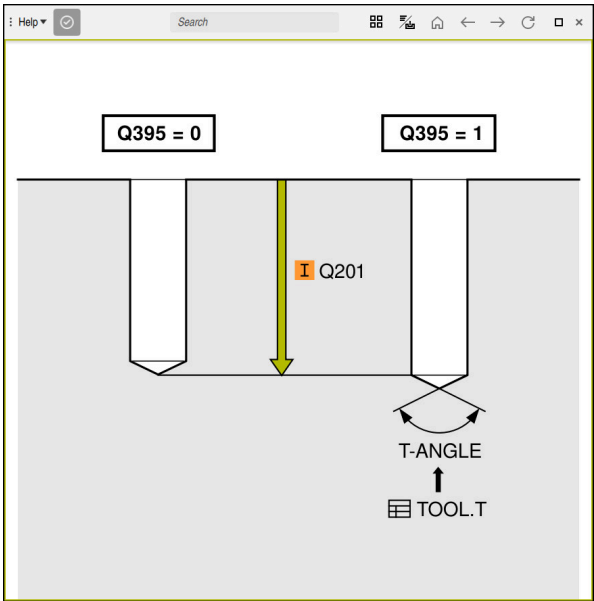
The **Help** workspace can be selected in the **Editor** operating mode and in the **MDI** application.

Further information: "The Editor operating mode", Page 116

Further information: User's Manual for Setup and Program Run

While the **Help** workspace is active, the control displays the help graphic there and not in a pop-up window.

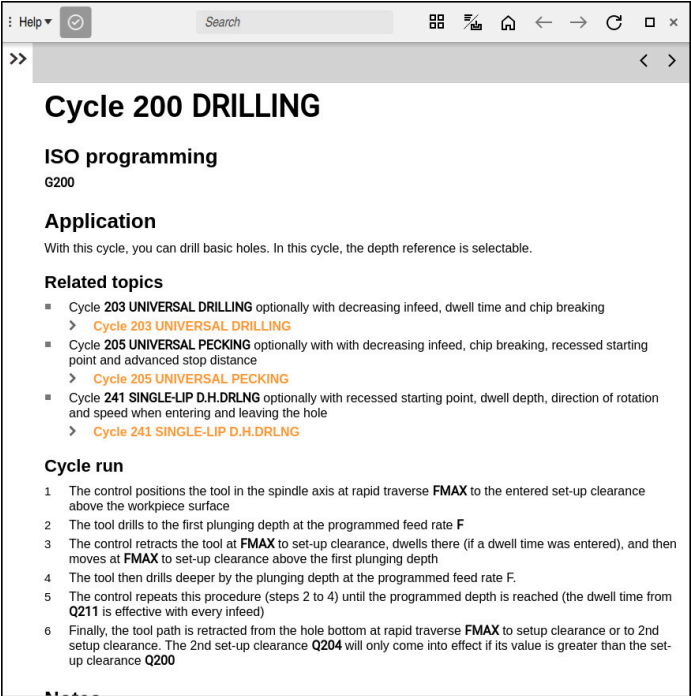
Further information: "Help graphic", Page 120



The **Help** workspace with a help graphic of a cycle parameter

When the **Help** workspace is active, the control can display the integrated **TNCguide** product aid.

Further information: "User's Manual as integrated product aid: TNCguide", Page 37



The **Help** workspace with **TNCguide** open

Icons

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

Icon	Meaning
	Open or close the Search results column Further information: "Searching in TNCguide", Page 40
	Open Home page The start page displays all available documentation. Select the desired documentation using navigation tiles (e.g., TNCguide). If only one piece of documentation is available, the control opens the content directly. When a documentation is open, you can use the search function. Further information: "Icons", Page 39
	Open TNCguide or the Help Graphic The control toggles between TNCguide and the Help Graphic . The control will only display a Help Graphic if you edit an NC block for which an associated Help Graphic exists.
	Open TNCguide in the Help application The control opens TNCguide at the current position. Further information: "The Help application", Page 38

Icon	Meaning
← →	Navigate Navigate between the contents opened recently
↺	Refresh

TNCguide has additional icons.
Further information: "User's Manual as integrated product aid: TNCguide", Page 37

20.2 Virtual keyboard of the control bar

Application

You can use the virtual keyboard for entering NC functions, letters, and numbers, and for navigation.

The virtual keyboard offers the following modes:

- NC input
- Text input
- Formula entry

Description of function

The control opens NC input mode by default after the start procedure.

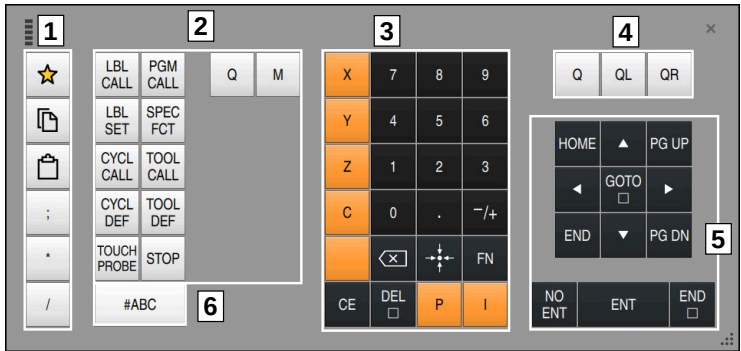
You can move the keyboard on the screen. The keyboard remains active, even when the operating mode is switched, until the keyboard is closed.

The control remembers the position and mode of the virtual keyboard until it is shut down.

The **Keyboard** workspace provides the same functions as the virtual keyboard.

The **+**, **-**, *****, **/**, **(** and **)** keys permit calculations concerning numerical values in input fields and table rows.

NC input areas



Virtual keyboard in NC input mode

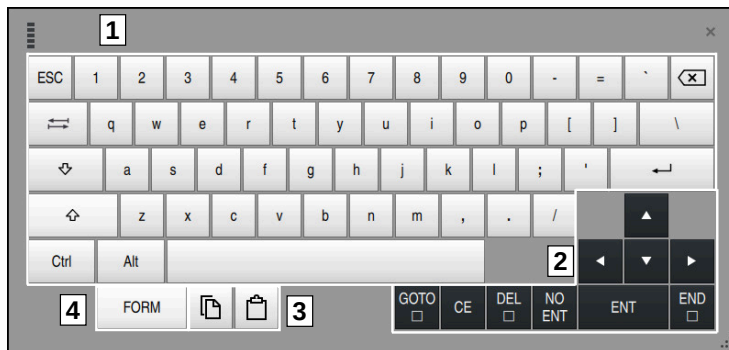
NC input mode contains the following areas:

- 1 File functions
 - Define favorites
 - Copy
 - Paste
 - Add comment
 - Add structure item
 - Hide NC block
- 2 NC functions
- 3 Axis keys and numerical input
- 4 Q parameters
- 5 Navigation and dialog keys
- 6 Switch to text input

If you press the **Q** button in the NC functions area repeatedly, the control cycles through the syntax in the following sequence:

- **Q**
- **QL**
- **QR**

Text input areas

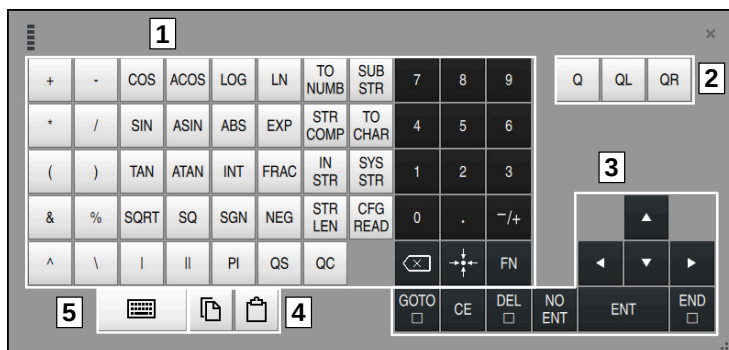


Virtual keyboard in text input mode

Text input mode contains the following areas:

- 1 Input
- 2 Navigation and dialog keys
- 3 Copying and pasting
- 4 Switch to formula input

Formula input areas



Virtual keyboard in formula input mode

Formula input mode contains the following areas:

- 1 Input
- 2 Q parameters
- 3 Navigation and dialog keys
- 4 Copying and pasting
- 5 Switch to NC input

20.2.1 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.

20.3 The GOTO function

Application

Use the **GOTO** function for navigating rapidly and purposefully within NC programs, text files or tables.

Related topics

- Selecting an NC block for program run with **Block scan**
Further information: User's Manual for Setup and Program Run

Description of function

The control offers the **GOTO** function in the following areas of the control:

- The **Program** workspace
Further information: "The Program workspace", Page 118
- The **Text editor** workspace
Further information: "The Text editor workspace", Page 366
- The **Table** workspace
Further information: "The Table workspace", Page 586

GOTO block number

Click the **GOTO block number** button to open the **GOTO jump instruction** window in the **Program** workspace. In this window you define an NC block selected by the control. If an NC program is open for simulation or execution, the control additionally positions the execution cursor in front of the NC block. The control then starts program run or the simulation beginning from the defined NC block without considering the preceding lines of the NC program.

In the **GOTO jump instruction** window, the control displays the following icons or buttons:

Icon or button	Meaning
	Search If, for example, the required block number is not known, use this icon to open the Search column.
Select last	The control applies the number of the previously run NC block in the Block number row. Only after an aborted program run
GOTO 0	The control selects the start of the NC program and closes the window.

GOTO record

Click the **GOTO record** button to open the **GOTO jump instruction** window in the **Table** and **Text editor** workspaces. In this window you define a row selected by the control.

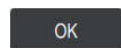
The control also provides the **Search** icon in the **Text editor** workspace.

20.3.1 Selecting an NC block or row with GOTO

To select an NC block or a row:



- ▶ Select **GOTO**
- The control opens the **GOTO jump instruction** window.
- ▶ Enter the block or row number



- ▶ Press **OK**
- The control selects the defined NC block or the defined row.

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

Further information: User's Manual for Setup and Program Run

Notes

- You can also use the **GOTO** function with the **[CTRL] + [G]** shortcut.
- If the control in the action bar shows an icon for selection, you can open the selection window with **GOTO**.
- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

20.4 Adding comments

Application

You can add comments to an NC program in order to explain program steps or make general notes.

Description of function

You have the following possibilities for adding comments:

- Comment within an NC block
- Comment as a separate NC block
- Define existing NC block as comment

The control marks comments with a preceding ; character. The control does not execute comments during simulation or program run.

A comment may contain up to 255 characters.

Comments that include line breaks can be edited only in the Text editor mode or in the **Form** column.

Further information: "Using the Program workspace", Page 124

20.4.1 Adding a comment as an NC block

To add a comment as a separate NC block:

- ▶ Select the NC block after which the comment is to be added



- ▶ Select ;
- ▶ After the selected NC block, the control adds a comment as a new NC block.
- ▶ Define the comment

20.4.2 Adding a comment in an NC block

To add a comment within an NC block:

- ▶ Edit the desired NC block



- ▶ Select ;
- ▶ The control inserts a ; character at the end of the block.
- ▶ Define the comment

20.4.3 Commenting an NC block out or in

Use the **Comment out/in** button to define an existing NC block as a comment or to change a comment back to an NC block.

To comment an existing NC block in or out:

- ▶ Select the desired NC block



- ▶ Select **Comment Off/On**
- ▶ The control inserts a ; character at the beginning of the block.
- ▶ If the NC block is already defined as a comment, the control removes the ; character.



You can also comment or uncomment NC blocks with the shortcuts **[SHIFT] + [.]** or **[SHIFT] + [,]**.

20.5 Hiding NC blocks

Application

Use / or the **Skip block Off/On** button to hide NC blocks.

By hiding NC blocks, you can skip the corresponding NC blocks during program run.

Related topics

- The **Program Run** operating mode

Further information: User's Manual for Setup and Program Run

Description of function

If you mark an NC block with a / character, then the NC block is hidden. If you activate the **Skip block** toggle switch in the **Program Run** operating mode or in the **MDI** application, the control skips this NC block during program run.

If the toggle switch is active, then the control dims the NC blocks to be skipped.

Further information: User's Manual for Setup and Program Run

20.5.1 Hiding or showing NC blocks

To hide or show an NC block:

- ▶ Select the desired NC block



- ▶ Select **Skip block Off/On**
- > The control adds a / character before the NC block.
- > If the NC block is already hidden, the control removes the / character.



Instead of the button, you can also use the following keys:

- Hiding or showing with the / key
- Showing with the **BACKSPACE** key

Further information: "Keycaps for alphabetic keyboard", Page 66

20.6 Structuring of NC programs

Application

You can use structure items to make long and complex NC programs clearer and more legible, and also to navigate more quickly through an NC program.

Related topics

- The **Structure** column of the **Program** workspace

Further information: "The Structure column in the Program workspace",
Page 501

Description of function

You can use structure items to arrange your NC programs. Structure items are texts that you can use as comments or headlines for the subsequent program lines.

A structure item may contain up to 255 characters.

The control displays the structuring items in the **Structure** column.

Further information: "The Structure column in the Program workspace", Page 501

20.6.1 Adding a structure item

To insert a structure item:

- ▶ Select the NC block after which you want to add the structure item



- ▶ Select *
- After the selected NC block, the control adds a structure item as a new NC block.
- ▶ Define the structure text

20.7 The Structure column in the Program workspace

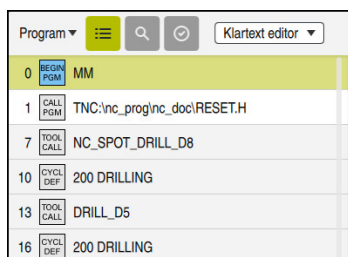
Application

When you open an NC program, the control searches the NC program for structuring items and displays these structure elements in the **Structure** column. The structuring items act like links and thus allow fast navigation in the NC program.

Related topics

- The **Program** workspace, defining contents of the **Structure** column
Further information: "Settings in the Program workspace", Page 120
- Inserting structure items manually
Further information: "Structuring of NC programs", Page 500

Description of function



The **Structure** column with automatically created structuring items

When you open an NC program, the control automatically creates the structure.

In the **Program settings** window you define which structuring items the control displays in the structure. The **PGM BEGIN** and **PGM END** structuring items cannot be hidden.

Further information: "Settings in the Program workspace", Page 120

The **Structure** column shows the following information:

- NC block number
- Icon of the NC function
- Function-dependent information

The control displays the following icons within the structure:

Icon	Syntax	Information
	BEGIN PGM	Unit of measurement of the NC program MM or INCH
	TOOL CALL	■ Name or number of the tool, if applicable ■ Index of the tool, if applicable ■ Comment, if applicable
	* Structure block	■ Entered string, if applicable ■ Comment, if applicable
	LBL SET	■ Name or number of the label ■ Comment, if applicable
	LBL 0	■ Number of the label ■ Comment, if applicable

Icon	Syntax	Information
	CYCL DEF	Number and name of the defined cycle
	TCH PROBE	Number and name of the defined cycle
	■ CALL PGM ■ CALL SELECTED PGM	■ Path of the called NC program (e.g., TNC:\Safe.h), if applicable ■ Comment, if applicable
	■ Cycle 12.1 PGM ■ SEL PGM	■ Path of the NC program (e.g., TNC:\Safe.h) ■ Comment, if applicable
	FUNCTION MODE	■ Selected machining mode (possibilities: MILL, and SET) ■ Selected kinematics, if applicable ■ Comment, if applicable
	M2 or M30	Comment, if applicable
	M1	Comment, if applicable
	STOP or M0	Comment, if applicable
	END PGM	No additional information

In the **Program Run** operating mode, the **Structure** column contains all structuring items, even those of the called NC programs. The control indents the structure of the called NC programs.



The control displays comments as separate NC blocks, rather than including them in the structure. These NC blocks start with the semicolon character (;).

Further information: "Adding comments", Page 498

20.7.1 Editing an NC block using the structure

To edit an NC block using the structure:

- ▶ Open an NC program
 -  ▶ Open the **Structure** column
 - ▶ Select structure element
 - ▶ The control positions the cursor on the corresponding NC block in the NC program. The focus of the cursor remains in the **Structure** column.
 -  ▶ Select the right arrow
 - ▶ The focus of the cursor changes to the NC block.
 -  ▶ Select the right arrow
 - ▶ The control edits the NC block.

20.7.2 Marking NC blocks using the structure

To mark NC blocks using the structure:

- ▶ Open an NC program



- ▶ Open the **Structure** column
- ▶ Hold or right-click the structuring item
- ▶ The control positions the cursor on the corresponding NC block in the NC program.
- ▶ The control opens the context menu.
Further information: "Context menu", Page 511
- ▶ Select **Mark**
- ▶ The control displays check boxes next to the structuring items in the **Structure** column.
- ▶ The control marks the NC block in the NC program.
- ▶ Enable additional check boxes, if required
- ▶ The control marks all structuring items between the two selected structuring items as well as the associated NC blocks.



Instead of the context menu, you can use the **[CTRL] + [SPACE]** shortcut.

Notes

- In the case of long NC programs, generating the structure view may take longer than loading the NC program itself. Even if the structure view has not been fully generated, you can already work in the loaded NC program.
- You can navigate within the **Structure** column using the up and down arrow keys.
- The control shows called NC programs in the structure with a white background. If you double-tap or click on such a structure element, the control opens the NC program if necessary in a new tab. If the NC program is open, the control switches to the corresponding tab.

20.8 The Search column in the Program and Text editor workspaces

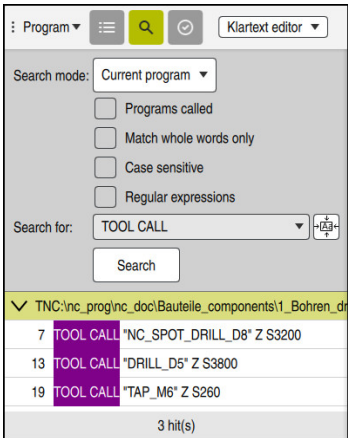
Application

In the **Search** column, you can search files for any character strings, such as individual syntax elements in the NC program. The control lists all the results found.

Related topics

- Search for the same syntax element in the NC program with the arrow keys
Further information: "Searching for identical syntax elements in different NC blocks", Page 127
- The **Search** column in the **Table** workspace
Further information: "The Search column in the Table workspace", Page 590
- The **Program** workspace
Further information: "The Program workspace", Page 118
- The **Text editor** workspace
Further information: "The Text editor workspace", Page 366

Description of function



The **Search** column in the **Program** workspace

The **Search** column can be selected in the **Program** and **Text editor** workspaces. In the **MDI** application, you can search only in the active NC program. The **Search and replace** mode is not available in the **Program Run** operating mode.

The control provides the following icons and buttons in the **Search** column:

Icon or button	Meaning
Search mode:	■ Current program or Current file Search current file ■ Opened programs or Opened files Search all opened files of the same type (e.g., *.h or *.txt) ■ Search and replace Search for strings and replace them with new strings, such as syntax elements Further information: "Search and replace mode", Page 506
Programs called	Search all called NC programs of the current NC program Only in the Current program mode
Match whole words only	If you select the check box, the control only displays exact matches. This means that if you search for Z+10 , for example, the control ignores Z+100 .
Case sensitive	If you select this checkbox, the control will do a case-sensitive search. If you, for example, search for tool , the control will ignore TOOL .
Regular expressions	If you select this checkbox, you can use regular expressions to define specific search term criteria. Further information: "Regular expressions", Page 507
Search for:	In the input area, you define the search term. If you have not yet entered any characters, the control suggests the last six search terms for selection. The search is not case-sensitive.
	The control loads the word or syntax element at the current cursor position into the input area. If the selected NC block is not edited, the control accepts the syntax initiator.
Search	Start search

The control shows the following information about the results:

- File paths
- Row numbers or NC block numbers
- Complete row contents
- Number of results

The control groups the results according to files. If you select a result, the control positions the cursor in the corresponding row.

Search and replace mode

In **Search and replace** mode, you can search for strings and replace the results found with other strings, such as individual syntax elements.

The control performs a syntax check in the NC programs before replacing a syntax element. With the syntax check, the control ensures that the new content results in correct syntax. If the result produces a syntax error, the control does not replace the content and displays a message.

In **Search and replace** mode, the control provides the following checkboxes and buttons:

Buttons	Meaning
Search backward	The control searches the file from bottom to top.
Wrap around	The control searches the entire file, wrapping around the start and end.
Replace with:	In the input area, you define the new term. If you have not yet entered any characters, the control suggests the last six terms for selection.
Find next	The control searches the file for the search term. The control marks the next result in the file.
Replace	The control performs a syntax check, if required, and automatically replaces the marked content with the contents of the Replace with: field.
Replace and find next	If a search has not yet been performed, the control only marks the first result. When a result is highlighted, the control performs a syntax check, if required, and automatically replaces the found content with the contents of the Replace with: field. The control then marks the next result.
Replace all	The control performs a syntax check, if required, and automatically replaces all found results with the contents of the Replace with: field.

Regular expressions

You can use regular expressions to include criteria in the search term. This allows you to search for a specific character string, as well as, for example, to search for all rows meeting the defined criteria.

The following table shows a list of the most common regular expressions:

Syntax	Meaning
.	Each . is a placeholder for any single character. If you, for example, define Q... , the control will search for all Q parameters with a 3-digit number.
^	The control searches for all rows that start with the defined criterion. If you, for example, define ^TOOL , the control will search for all rows that start with TOOL .
\$	The control searches for all rows that end with the defined criterion. If you, for example, define F500\$, the control will search for all rows that end with F500 .
?	A ? indicates that the character preceding it is not included or is included once in the search term. If, for example, you define D_?3 , the control searches for all terms with no or with one _ . Some possible search results are D3 or D_3 .
*	A * indicates that the character preceding it is not included in the search term, or is included once, or several times in succession. If, for example, you define D_*3 , the control searches for all terms with no, with one or with several _ . Some possible search results are D3 , D_3 , and D___3 .
+	A + indicates that the character preceding it is included at least once in the search term. If, for example, you define D_+3 , the control searches for all terms with at least one _ . Some possible search results are D_3 and D___3 .



You can combine multiple search criteria in one search.

If, for example, you define **D_?3\$**, the control will search for all lines ending with **D3** or **D_3**.

20.8.1 Search for and replace syntax elements

To search for and replace syntax elements in the NC program:



- ▶ Select an operating mode (e.g., **Editor**)
- ▶ Select the desired NC program
- > The control opens the selected NC program in the **Program** workspace.



- ▶ Open the **Search** column
- ▶ In the **Search mode:** field, select the **Search and replace** function
- > The control displays the **Search for:** and **Replace with:** fields.
- ▶ In the **Search for:** field, enter the search content (e.g., **M4**)
- ▶ In the **Replace with:** field, enter the desired content (e.g., **M3**)
- ▶ Select **Find next**
- > The control closes previously called NC programs, if any had been called, and highlights the first result in the main program in purple.



- ▶ Select **Replace**
- > The control performs a syntax check and replaces the content if the check is successful.

Notes

- The search results are retained until you shut down the control or search again.
- If you double-tap or click on a search result in a called NC program, the control opens the NC program (on a new tab if not already open). If the NC program is already open, the control switches to the corresponding tab.
- If you have not entered a value for **Replace with:**, the control deletes the search value.

20.9 Program comparison

Application

Use the **Program comparison** function to determine differences between two NC programs. You can transfer the differences to the active NC program. If there are unsaved changes in the active NC program, you can compare the NC program with the last saved version.

Requirements

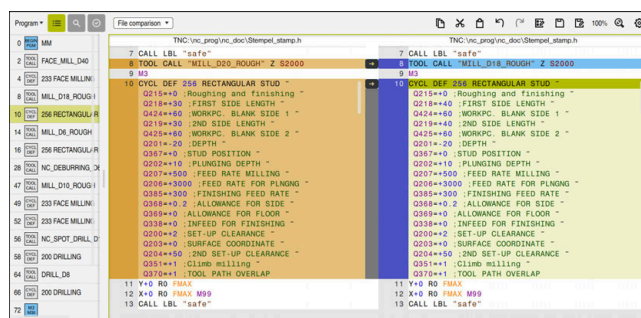
- Max. 30,000 lines per NC program

The control takes into account the actual lines, not the number of NC blocks.

Some NC blocks, particularly those consisting of cycles, can contain several lines within one block number.

Further information: "Contents of an NC program", Page 114

Description of function



Program comparison of two NC programs

The title bar of the **Program** workspace contains a selection menu for selecting the **File comparison** mode.

You can use the program comparison only in the **Editor** operating mode in the **Program** workspace.

The control shows the active NC program on the right and the comparison program on the left.

The control marks differences with the following colors:

Color	Syntax element
Gray	Missing NC block or missing line for NC functions of different length
Orange	NC block with difference in comparison program
Blue	NC block with difference in the active NC program

During the program comparison, you can edit the active NC program, but not the comparison program.

If NC blocks differ, you can use an arrow symbol to transfer the NC blocks of the comparison program to the active NC program.

20.9.1 Applying differences to the active NC program

To transfer differences to the active NC program:



- ▶ Select the **Editor** operating mode
- ▶ Open an NC program
- ▶ Select the selection menu in the title bar
- ▶ Select **File comparison**
 - > The control opens a pop-up window for file selection.
- ▶ Select comparison program
- ▶ Select **Select**
 - > The control shows both NC programs in the comparison view and marks all differing NC blocks.
- ▶ Select the arrow symbol for the desired NC block
 - > The control transfers the NC block to the active NC program.



Notes

- If the compared NC programs contain more than 1000 differences, the control cancels the comparison.
- If an NC program contains unsaved changes, the control displays an asterisk in front of the name of the NC program in the tab of the application bar.
- If you mark multiple NC blocks in the comparison program, you can apply those NC blocks simultaneously. If you mark multiple NC blocks in the active NC program, you can overwrite those NC blocks simultaneously.

Further information: "Context menu", Page 511

20.10 Context menu

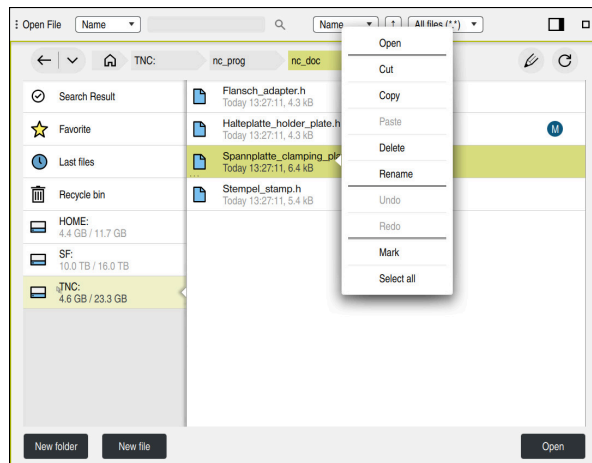
Application

With a long-press gesture or by right-clicking with the mouse, the control opens a context menu for the selected element, such as an NC block or file. Use the various functions of the context menu to run commands that affect the currently selected element(s).

Description of function

The functions available in the context menu depend on the selected element as well as the selected operating mode.

General



Context menu in the **Open File** workspace

Depending on the selected workspace and operating mode, the context menu provides the following functions:

- **Cut**
- **Copy**
- **Paste**
- **Delete**
- **Undo**
- **Redo**
- **Mark**
- **Select all**



If you select the **Mark** or **Select all** functions, the control opens the action bar. The action bar displays all functions that are currently available for selection from the context menu.

As an alternative to the context menu, you can use keyboard shortcuts:

Further information: "Icons on the control's user interface", Page 73

Key or keyboard shortcut	Meaning
[CTRL] + [SPACE]	Mark the selected line
[SHIFT] + [UP]	■ Mark the selected line ■ Additionally mark the line directly above ■ During the editing, mark all NC blocks up to the preceding same syntax element
[SHIFT] + [DOWN]	■ Mark the selected line ■ Additionally mark the line directly below ■ During the editing, mark all NC blocks up to the next same syntax element
[SHIFT] + [PG UP]	Mark from the cursor position to the beginning of the page Not available in the Tables operating mode
[SHIFT] + [PG DN]	Mark from the cursor position to the end of the page Not available in the Tables operating mode
[SHIFT] + [HOME]	Mark from the cursor position to the first row Not available in the Tables operating mode
[SHIFT] + [END]	Mark from the cursor position to the last row Not available in the Tables operating mode
ESC	Cancel marking

Context menu in the Files operating mode

In the **Files** operating mode the context menu offers the following additional functions:

- **Open**
- **Open in text editor**
- **Select in Program Run**
- **Show as document**
- **Rename**
- **Open and convert**

For the navigation functions, the context menu offers the respectively relevant functions, such as **Discard search results**.

Further information: "Context menu", Page 511

Context menu in the Tables operating mode

In the **Tables** operating mode the context menu offers the following additional functions:

- **Cancel**

The control aborts marking.

- **Enter as text**

Using this function, you can enter a value directly when editing a table cell. This is an alternative to opening a selection window automatically.

In the **Tables** operating mode, the context menu provides some functions applicable both for cells and rows.

To cut or copy an entire table row, the control provides the following functions in the action bar:

- **Overwrite**

The control inserts the row instead of the currently selected table row.

- **Append**

The control appends the row at the end of the table.

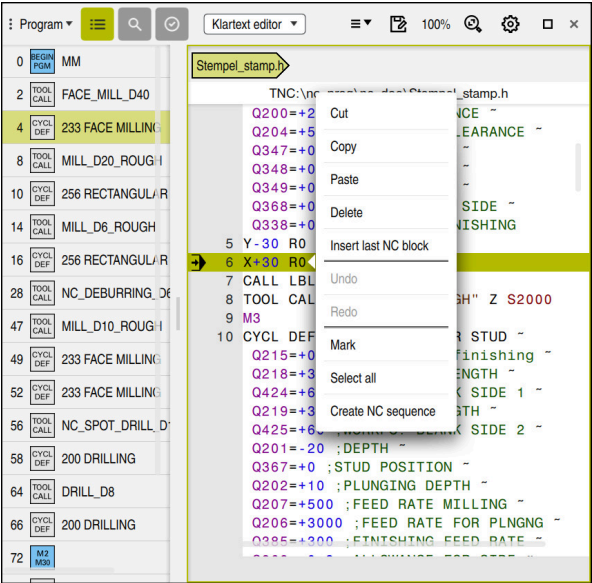


If the clipboard of the **Tool management** application contains indexed tools only, the control will create the rows as indexes of the currently selected tool.

- **Cancel**

Further information: "The Tables operating mode", Page 580

Context menu in the Program workspace



Context menu in the **Program** workspace of the **Editor** operating mode

In the **Program** workspace, the context menu offers the following additional functions:

- **Insert last NC block**
This function allows you to insert the most recently deleted or edited NC block. You can insert this NC block in any desired NC program.
Only in the **Editor** operating mode and the **MDI** application
 - **Create NC sequence**
Only in the **Editor** operating mode and the **MDI** application
Further information: "NC sequences for reuse", Page 201
- The following functions in the context menu of the **Program** workspace are available only if you select a value of an NC block:
- **Select value**
 - **Replace value**
Add values from the clipboard in the NC block such as values from the calculator
- Further information:** "The Program workspace", Page 118

The **Select value** and **Replace value** functions are only available in the **Editor** operating mode and in the **MDI** application.

The **Replace value** function also works if no value has been defined yet for the syntax element. In this case the otherwise necessary marking of the value to be replaced is omitted.

Further information: "Calculator", Page 516

Context menu in the Insert NC function window

In the **Insert NC function** window, the context menu offers the following functions:

- **Open path**
Open the NC function in the **All functions** area
- **Edit**
Open the NC sequence in a separate tab
- **Organize**
Open the path of the NC sequence in the **Files** operating mode
- **Delete**
Delete the NC sequence
- **Rename**
Rename the NC sequence

Further information: "The Insert NC function window", Page 133

Context menu in the configuration editor

In the configuration editor, the context menu also provides the following functions:

- **Direct entry of values**
- **Create copy**
- **Restore copy**
- **Change key name**
- **Move**
- **Open element**
- **Remove element**

The context menu also offers general functions (e.g., **Paste**).

Further information: "General", Page 511

Further information: User's Manual for Setup and Program Run

20.11 Calculator

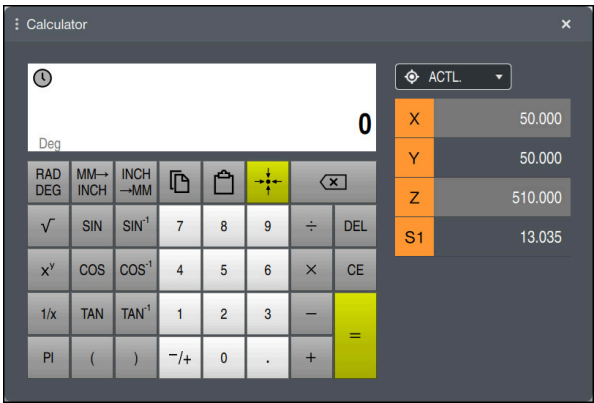
Application

The control offers a calculator on the control bar. You can copy the result to the clipboard and also paste values from the clipboard.

Description of function

The calculator provides arithmetic functions such as:

- Basic mathematical operations
- Basic trigonometric functions
- Square root
- Exponential calculation
- Reciprocal value
- Conversion between the mm and inch units of measure



Calculator

You can switch between the radian **RAD** or degrees **DEG** modes.

You can copy the result to the clipboard as well as paste the last stored value from the clipboard to the calculator and load current axis positions.

The calculator saves the last ten calculations in the history. You can use these saved results for further calculations. You can clear the history manually.

The control additionally offers the following functions of the calculator:

- The **P** key corresponds to **PI**
- The **RETURN** or **ENT** key corresponds to **=**

Further information: "Operating elements of the keyboard unit", Page 65

20.11.1 Opening and closing the calculator

To open the calculator:

-
- ▶ Select the **calculator** on the control bar
 - > The control opens the calculator.

To close the calculator:

-
- ▶ Select the **calculator** when the calculator is open
 - > The control closes the calculator.

20.11.2 Actual position capture

To load the actual position of an axis into the calculator:



- ▶ Select **actual position capture**
- > The control opens the position display in the calculator.
- ▶ If required, select the mode for the position display such as **Actual pos. (ACT)**
- ▶ Select the desired value
- > The control transfers the value to the calculator.



- ▶ Select **actual position capture**
- > The control closes the position display.



The calculator shows the same mode of the position display as the status overview of the TNC bar. If you change the mode, the control synchronizes the two position displays.

The position display mode can be selected independently of the **Positions** workspace (e.g., **Actual pos. (ACT)**).

Further information: User's Manual for Setup and Program Run

20.11.3 Selecting a result from the history

To select a result from the history for further calculations:



- ▶ Select **History**
- > The control opens the calculator's history.
- ▶ Select the desired result



- ▶ Select **History**
- > The control closes the calculator's history.

20.11.4 Deleting the history

To delete the calculator's history:



- ▶ Select **History**
- > The control opens the calculator's history.



- ▶ Select **Delete**
- > The control deletes the calculator's history.

20.12 Cutting data calculator

Application

With the cutting data calculator you can calculate spindle speeds and feed rates. You can load the calculated values directly into the NC block.

To avoid, for example, input errors, you can also optionally load values from cutting data tables.

Requirements

- For calculating with cutting data tables
 - Workpiece material defined in table **WMAT.tab**
 - Tool cutting material defined in table **TMAT.tab**
 - Combination of workpiece material and cutting material defined in the cutting data table ***.cut** or in the diameter-dependent cutting data table ***.cutd**
 - The following parameters defined in tool management:
 - **R**: Tool radius
 - **CUT**: Number of cutting edges
 - **TMAT**: Cutting material from **TMAT.tab**
 - **CUTDATA**: Table row from the ***.cut** or ***.cutd** cutting data table

Further information: "Tables for cutting data calculation", Page 612

Further information: User's Manual for Setup and Program Run

Description of function

If you position the cursor on feed rate **F** or spindle speed **S** in an NC block, you can open the cutting data calculator by clicking on the icon in the action bar.

The control displays different content depending on the NC function and the tool parameters defined.

Use the cutting data calculator first with a tool call so that the correct values for the tool are active. You can then calculate further values in, for example, a path function.

Icons and buttons

The **Cutting data calculator** window, depending on the NC block, displays the following icons and buttons:

Icon or button	Meaning
Number, Name, or None	You select the tool information that the control applies to the NC block Only with TOOL CALL
	Tool selection Open selection window for tools
	Select material Open selection window for workpiece material
	Select type of machining Open selection window for machining mode You select the machining mode from the cutting data table that you specify in tool management.
S, VC, or None	You select the value that the control applies for spindle speed in the NC block. Only with a TOOL CALL block
F, FZ, FU, or None	You select the value that the control applies for feed rate in the NC block.
Apply	Transfer calculated values to the NC block

Areas of the cutting data calculator

The screenshot shows the 'Cutting data calculator' window. It is divided into several sections. Section 1 (top) is for tool selection, showing 'Tool' as 'MILL_D32_ROUGH' with a diameter of '32.000 mm' and 'Number of teeth' as '4'. Section 2 (middle) is for workpiece material, showing 'Baustahl_Construction-s' and 'Material class' as '10'. Section 3 (bottom) is for spindle speed, showing 'Spindle speed (S)' as '1989.437 rpm' and 'Cutting speed (VC)' as '200.000 m/min'. Section 4 (bottom) is for feed rate, showing 'Contouring feed rate (F)' as '837.658 mm/min', 'Feed per tooth (FZ)' as '0.105 mm', and 'Revolution feed (FU)' as '0.421 mm'. The window also has an 'Apply' button and a 'Cancel' button.

The **Cutting data calculator** window for a **TOOL CALL**

The **Cutting data calculator** window shows the following areas:

- 1 Tool

The control displays the data of the active tool.

You can change the tool as needed.

If you enable the **Activate cutting data from table** checkbox, the control displays the cutting data tables area.

The control only offers the checkbox if cutting data tables are also stored for the selected tool.
- 2 Cutting data tables

Optional area

You select the workpiece material and machining mode from the cutting data tables. With this information, the control automatically calculates the required values.

Further information: "Calculating cutting data with tables", Page 521
- 3 Spindle speed

If you enter the cutting speed **VC**, the control calculates the spindle speed **S**.

If you load the values from cutting data tables, the control calculates **VC** and **S**.
- 4 Feed rate

If you enter the feed per tooth **FZ**, the control calculates the revolution feed **FU** and vice versa.

If values are defined in the spindle speed area, the control calculates all feed rates.

20.12.1 Calculating the cutting data without tables

To calculate the cutting data without cutting data tables:

- ▶ Edit the desired **TOOL CALL**
- ▶ Select the syntax element for the feed rate or spindle speed
 - 
 - ▶ Select **Cutting data calculator** in the action bar
 - ▶ The control opens the **Cutting data calculator** window.
 - ▶ Enter the cutting speed **VC** from the tool catalog.
 - ▶ The control calculates the spindle speed **S**.
 - ▶ Enter the feed rate from the tool catalog (e.g., **FZ**).
 - ▶ The control calculates the remaining feed rates (e.g., **F** and **FU**).
 - ▶ Select values that the control loads into the NC block (e.g., **S** and **F**)
- 
 - ▶ Press **Apply**
 - ▶ The control applies the calculated values in the NC block.
 - ▶ Calculate any required values in additional NC blocks (e.g., in path functions)

20.12.2 Calculating cutting data with tables

To calculate the cutting data with tables:

- ▶ Edit the desired **TOOL CALL**
- ▶ Select the syntax element for the feed rate or spindle speed
 - 
 - ▶ Select **Cutting data calculator** in the action bar
 - ▶ The control opens the **Cutting data calculator** window.
 - ▶ Enable the **Activate cutting data from table** checkbox
 - ▶ The control shows an additional area.
 - 
 - ▶ Select the workpiece material using the selection window
 - ▶ The control loads the workpiece material and material class.
 - 
 - ▶ Select the machining mode using the selection window
 - ▶ The control calculates the cutting data.
 - ▶ Select values that the control loads into the NC block (e.g., **S** and **F**)
- 
 - ▶ Select **Apply**
 - ▶ The control applies the calculated values in the NC block.
 - ▶ Calculate any required values in additional NC blocks (e.g., in path functions)

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control by default opens the cutting data calculator with the values from the NC block.

If you have not yet defined values or if values cannot be defined in the NC block, the control displays the last values used.

20.13 Issuing notifications with FUNCTION REPORT

Application

With the **FUNCTION REPORT** NC function, the controls issues notifications under program control. You can define the notification text yourself. If the machine manufacturer or another provider has saved notifications as a PO file, you can also output these notifications.

Related topics

- Issuing error messages with **FN 14: ERROR**
Further information: "Output error messages with FN 14: ERROR", Page 412
- Outputting texts with **FUNCTION PRINT**
Further information: "Issuing text with FUNCTION PRINT", Page 524
- Notifications on the information bar
Further information: User's Manual for Setup and Program Run

Description of function

When the NC function **FUNCTION REPORT** is executed, the control displays a message and triggers a reaction.

With the NC function you can program the following reactions:

Syntax element	Message type	Reaction
WARNING	Warning	NC program continues to run
STOP	Error	Program run or simulation stops
ERROR	Error	Program run or simulation is aborted

Input

11 FUNCTION REPORT WARNING TEXT: "Check tool length"	; Issue self-defined notification
11 FUNCTION REPORT WARNING DOMAIN: "SOURCE" ID: "EXAMPLE" ARGS: "Check tool length"	; Issue notification from a PO file with self-defined additional text

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Functions ► FUNCTION REPORT

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION REPORT	Syntax initiator for issuing a message
ERROR, STOP or WARNING	Reaction
TEXT: or DOMAIN:	Enter message text yourself or select it from a PO file
Name or Parameter	Text or PO file When selecting TEXT: Text or string parameter When selecting DOMAIN: , text entry as well as selecting from a selection window are possible
ID:	ID of the text from the PO file Text or string parameter Only if DOMAIN: has been selected
ARGS:	Input additional text to be shown in the details of the notification Text or string parameter Only if DOMAIN: has been selected Optional syntax element

Note

Refer to your machine manual.

Only existing PO files can be output which have been stored by the machine manufacturer or a third-party provider. In order to output the PO files, the machine manufacturer must provide the IDs of the texts.

The PO file must contain the following information:

Meaning	Input
Message text	msgid "EXAMPLE" msgstr "Message text"
Cause	msgid "EXAMPLE_ UR " msgstr "Cause"
Corrective action	msgid "EXAMPLE_ BE " msgstr "Corrective action"

20.14 Issuing text with FUNCTION PRINT

Application

With the NC function **FUNCTION PRINT**, you can issue texts under program control. For example, you can issue instructions on the screen during program run by defining texts directly in the NC function.

Related topics

- Issue notifications with **FUNCTION REPORT**
Further information: "Issuing notifications with FUNCTION REPORT", Page 522
- Issue texts or values of numeric parameters formatted with **FN 16**
Further information: "Outputting text formatted with FN 16: F-PRINT", Page 413
- Define format strings with **FMT**
Further information: "Format strings", Page 439

Description of function

This function allows you to issue text on the screen or write it to a file. You can enter the texts to be issued directly in the NC function or in string parameters.

Further information: "Variables: Q, QL, QR, QS parameters and named parameters", Page 393

When the control runs the NC block it does not interrupt program run. The control displays the issued texts until you clear the output.

The control also issues the texts in the simulation.

Output options

Screen output

If you select **SCREEN**, the control displays the text in the **FN 16 output** window and on the **FN 16** tab of the **Status** workspace.

When you clear the screen output, the control clears and closes the **FN 16 output** window.

Saving the output file

You can save the texts in a separate output file, for example in a TXT file.

When the control runs the NC block, it writes the text to the defined file. If the file does not exist, the control automatically creates it during program run.

If you clear the output, the control deletes the entire content of the file.

Input

11 FUNCTION PRINT FILE:
 "TNC:\nc_prog\nc_doc
 \measured_values.txt" TEXT: QS21

; Write text from **QS21** to the
 "measured_values.txt" file

To navigate to this function:

Insert NC function ► All functions ► Special functions ► Functions ► FUNCTION PRINT

The NC function includes the following syntax elements:

Syntax element	Meaning
FUNCTION PRINT	Syntax initiator for issuing a text
SCREEN or FILE:	Issue text on the screen or write it to a file
File or Parameter	Path under which the control saves the output file Fixed or variable path Selection by means of a selection window The extension of the output file determines the file type of the output (e.g., TXT, A, H). Only if FILE has been selected
TEXT: or CLEAR	Enter output text or clear output
Name or Parameter	Text to be output Text or string parameter Only if TEXT: has been selected

Notes

- The following characters are allowed when entering text:
 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 _ - # ! # \$ % & ' () * + , . / : ; < = > ? @ [\]
- When you write text to a file, the control always appends the text to the end of the file, regardless of the file type. This may prevent the file from being opened.
 HEIDENHAIN recommends using a text file as the output file.
- If you only specify a file name instead of a path in the **FILE:** selection, the control saves the output file in the folder of the NC program.
- The maximum size of the output file is 10 MB.
- The NC functions **FN 16: F-PRINT** and **FUNCTION PRINT** use the same screen output. This means the outputs in the **FN 16 output** window and on the **FN 16** tab of the **Status** workspace may overlap.
- In order to prevent unintentional overlaps with previous outputs, clear the **FN 16 output** window or the output file before a new output.

21

**The Simulation
workspace**

21.1 Fundamentals

Application

In the **Editor** operating mode, you can use the **Simulation** workspace to graphically test whether NC programs are programmed correctly and run without collisions.

In the **Manual** and **Program Run** operating modes, the control shows the current traverse motions of the machine in the **Simulation** workspace.

Requirements

- Tool definitions according to the tool data from the machine
- Workpiece blank definition that is valid for a test run

Further information: "Defining a workpiece blank with BLK FORM", Page 152

Description of function

In the **Editor** operating mode, the **Simulation** workspace can be open for only one NC program at a time. With this NC program the control displays the **Control-in-operation** icon next to the program name. If you want to open the workspace on a different tab, the control prompts you for confirmation. The query depends on the simulation settings and the status of the active simulation.

Further information: "The Simulation settings window", Page 534

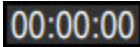
If the simulation of another NC program is currently running, the control shows a window with the name of this NC program above the function bar. If you double-tap or click this window, the control switches from the active tab to the NC program currently being simulated.

The functions available in the simulation depend on the following settings:

- Selected model type, for example **2.5D**
- Selected model quality, for example **Medium**
- Selected mode, for example **Machine**

Icons in the Simulation workspace

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

Icon	Meaning
	Open or close the Visualization options column Further information: "The Visualization options column", Page 530
	Open or close the Workpiece options column Further information: "The Workpiece options column", Page 532
	Open or close the Pre-defined views selection menu Further information: "Pre-defined views", Page 538
	Save as Export simulated workpiece as STL file Further information: "Exporting a simulated workpiece as STL file", Page 539
	Open or close the Simulation settings window Further information: "The Simulation settings window", Page 534
	Advanced checks active Further information: "The Visualization options column", Page 530
	Model quality Further information: "The Simulation settings window", Page 534
	Number or name of the current tool  The display depends on the workspace size.
	Current run time of the program in hh:mm:ss format

The Visualization options column

In the **Visualization options** column, you can define the following display modes and functions:

Icon or toggle switch	Meaning	Requirements
	Select the Machine or Workpiece mode In the Workpiece mode, the control displays the workpiece, the tool, and the tool carrier. Depending on the selected mode, different functions are available, such as a display of the setup situation. If you select the Machine mode, the control additionally displays the setup situation and the machine.	
Workpiece position	Use this function to define the position of the workpiece preset for the simulation. You can use a button to apply a workpiece preset from the preset table. Further information: User's Manual for Setup and Program Run	■ The Editor operating mode
	You can select between the following display modes for the machine: ■ Original: Shaded, opaque representation ■ Semitransparent: Transparent representation ■ Wire-frame model: Representation of the machine contours	
	You can select between the following display modes for the tool: ■ Original: Shaded, opaque representation ■ Semitransparent: Transparent representation ■ Invisible: The object is hidden	
	You can select between the following display modes for the workpiece: ■ Original: Shaded, opaque representation ■ Semitransparent: Transparent representation ■ Invisible: The object is hidden	
	You can show the tool paths during the simulation. The control displays the center-line path of the tools. You can choose between the following display modes for the tool paths: ■ None: Do not show tool paths ■ Feed: Show tool paths with programmed feed rate ■ Feedrate + FMAX: Show tool paths at programmed feed rate and at programmed rapid traverse	■ The Workpiece mode ■ The Editor operating mode
Clamping situation	Use this toggle switch to show the worktable and fixture, if required.	■ The Workpiece mode

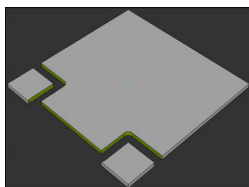
Icon or toggle switch	Meaning	Requirements
Advanced checks	If you activate the Advanced checks toggle switch, the control provides the following checks: ■ Rapid-traverse cut ■ Collision between workpiece and tool ■ Collision between workpiece and machine ■ Fixture collision Further information: "Advanced checks in the simulation", Page 541	■ The Editor operating mode
Program run options	If you activate this toggle switch, the control opens the Program run options window with the following selection options: ■ Skip block If an NC block is preceded by a / character, then the NC block is hidden. If you activate the Skip block toggle switch, the control skips all hidden NC blocks in the simulation. Further information: "Hiding NC blocks", Page 499 If the toggle switch is active, then the control dims the NC blocks to be skipped. Further information: "Appearance of the NC program", Page 120 ■ Pause at M1 If you activate this toggle switch, the control pauses the simulation at each M1 M function in the NC program. Further information: "Overview of miscellaneous functions", Page 383 If the toggle switch is inactive, then the control dims the M1 syntax element. Further information: "Appearance of the NC program", Page 120	■ The Editor operating mode

The Workpiece options column

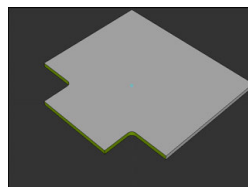
In the **Workpiece options** column, you can define the following simulation functions for the workpiece:

Toggle switch or button	Meaning	Requirements
Measuring	Use this function to measure any points on the simulated workpiece. The control measures the distance between the measured surface and the finished part, using only the 3D model type. Further information: "Measuring function", Page 542	■ The Workpiece mode ■ The 2.5D or 3D model type
Cutout view	Use this function to cut through the simulated workpiece along a plane. Further information: "Cutout view in the simulation", Page 544	■ The Workpiece mode ■ The Editor operating mode ■ The 2.5D model type
Rewind	This function allows you to forward and rewind part of the simulation. Further information: "The Rewind function", Page 545	■ The Editor operating mode ■ NC program or program section simulated and not reset
Highlight workpiece edges	Use this function to highlight the edges of the simulated workpiece.	■ The Workpiece mode ■ The 2.5D model type
Workpiece blank frame	Use this function to show the outside lines of the workpiece blank.	■ The Workpiece mode ■ The Editor operating mode ■ The 2.5D model type
Finished part	Use this function to display a finished part that was defined by means of the BLK FORM FILE NC function. Further information: "Cutout view in the simulation", Page 544	
Software limit switches	Use this function to activate the software limit switches of the machine for the active traverse range in the simulation. By simulating the limit switches you can check whether the working space of the machine is sufficient for the simulated workpiece. Further information: "The Simulation settings window", Page 534	■ The Editor operating mode

Toggle switch or button	Meaning	Requirements
Workpiece coloring	■ Grayscale The control displays the workpiece in various shades of gray. ■ Tool based The control displays the workpiece in color. Each cutting tool is assigned a separate color. ■ Model comparison The control displays a comparison between the workpiece blank and the finished part. Further information: "Model comparison", Page 547 The control displays a heat map on the workpiece:	■ The 2.5D model type ■ Model comparison function in the Workpiece mode only
Reset the workpiece	Use this function to reset the workpiece back to the workpiece blank	■ The 2.5D model type
Reset the tool paths	Use this function to reset the simulated tool paths.	■ The Workpiece mode ■ The Editor operating mode
Remove the chips	Use this function to remove from the simulation those parts of the workpiece that were cut off during machining.	■ The 3D model type



Workpiece before clean-up



Workpiece after clean-up

The Simulation settings window

Editor operating mode

The **Simulation settings** window contains the following areas in the **Editor** operating mode:

Area	Setting
General	■ Model type ■ 2.5D: quick solid-model view ■ None: very fast simulation without solid-model view ■ 3D: exact solid-model view ■ Quality ■ Low: low-quality model, low memory use ■ Medium: normal-quality model, average memory use ■ High: high-quality model, uses much memory ■ Highest: best-quality model, uses very much memory ■ Mode ■ Milling ■ Optimized saving of STL (#152 / #1-04-1) If you activate the toggle switch, the control exports a simplified STL file. During this process, the control removes unnecessary triangles and simplifies the 3D model to max. 20 000 triangles. ■ Prompt when opening a new simulation The toggle switch is active by default. If you open the Simulation workspace in a new tab, the control shows the Close current simulation window. You can exit the active simulation or cancel the process. If the toggle switch is inactive, the control will not show the window. i If you open the Simulation workspace in a new tab while a simulation is running, the control will always show the Close opened simulation window. ■ Active kinemat. Select the kinematics model for the simulation from a selection menu. The machine manufacturer enables the kinematics models. ■ Generate tool-usage file ■ Never Do not generate a tool-usage file ■ Once Generate a tool-usage file for the next simulated NC program ■ Always Generate a tool-usage file for every simulated NC program Further information: User's Manual for Setup and Program Run

Area	Setting
	■ Simulation report If you activate the toggle switch, the control shows errors and warnings for the simulation in the Tests column of the Program workspace. Further information: "The Simulation report area", Page 130
Traverse ranges	■ Traverse ranges In this selection menu you can choose one of the traverse ranges defined by the machine manufacturer, such as Limit1. In each traverse range the machine manufacturer defines different software limit switches for each axis of the machine. For example, the machine manufacturer defines traverse ranges for large machines with two separate working spaces. Further information: "The Workpiece options column", Page 532 ■ Active traverse ranges This function shows the active traverse range and the values defined for within that range.
Tables	You can select your own tables for the simulation in the Editor operating mode. The selected tables are independent of any tables that are active in other operating modes. The following tables are available: ■ Tool table ■ Datum table ■ Preset table Further information: User's Manual for Setup and Program Run With the Reset button, the control selects the same tables for the simulation that are in effect during program run. During programming, you can select table values via a selection window (e.g., a preset). By default, the control shows the program run tables in the selection window. If the Simulation workspace is open, the control shows the tables of the simulation in the selection window.

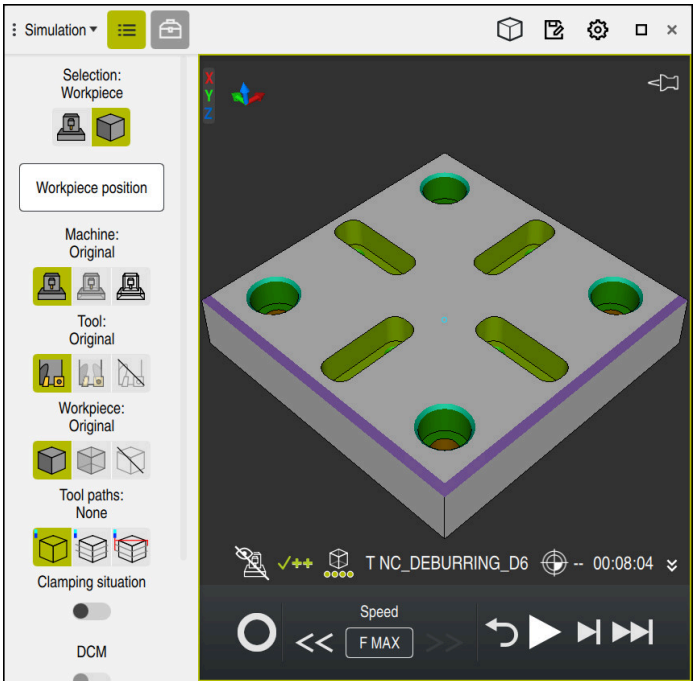
Program Run and Manual operating modes

The **Simulation settings** window contains the following setting in the **Program Run** and **Manual** operating modes:

Area	Setting
General	Model type ■ 2.5D: quick solid-model view ■ None: very fast simulation without solid-model view

You can open the **Simulation settings** window only if there is no active traversing command. In this case, the **Control-in-operation** icon is shown in white.

Action bar



The **Simulation** workspace in the **Editor** operating mode

In the **Editor** operating mode you can test NC programs by simulating them. The simulation helps to detect programming errors or collisions and to check the machining result visually.

The control shows the active tool and the machining time above the action bar.

Further information: User's Manual for Setup and Program Run

The action bar contains the following icons:

Icon	Function
	Control-in-operation: The control uses the Control-in-operation icon to show the current simulation status in the action bar and on the tab of the NC program: ■ White: no movement command ■ Green: active machining, axes are moving ■ Orange: NC program interrupted ■ Red: NC program stopped
	Simulation speed Further information: "Simulation speed", Page 550
	Reset Return to the beginning of the program, reset transformations and the machining time
	Start
	Start in Single Block mode
	Run the simulation up to a certain NC block Further information: "Simulating an NC program up to a certain NC block", Page 551

Simulation of tools

The control visualizes the following entries of the tool table in the simulation:

- L
- LCUTS
- LU
- RN
- T-ANGLE
- R
- R2
- KINEMATIC
- TSHAPE
- R_TIP

- Delta values from the tool table

Delta values from the tool table increase or decrease the size of the simulated tool. Delta values from the NC program shift the tool in the simulation.

Further information: "Tool compensation for tool length and tool radius",
Page 340

Further information: User's Manual for Setup and Program Run

The control displays the tool in the following colors:

- Turquoise: tool length
- Red: length of cutting edge and tool is engaged
- Blue: length of cutting edge and tool is retracted

The tool representation depends on the quality selected in the **Simulation settings** window. The control visualizes round tools as polygons. The lower the quality the fewer corners the polygon has, regardless of the tool diameter. Large tools also produce a more significant chord error, which means that the representation deviates more strongly from a circle.

21.2 Pre-defined views

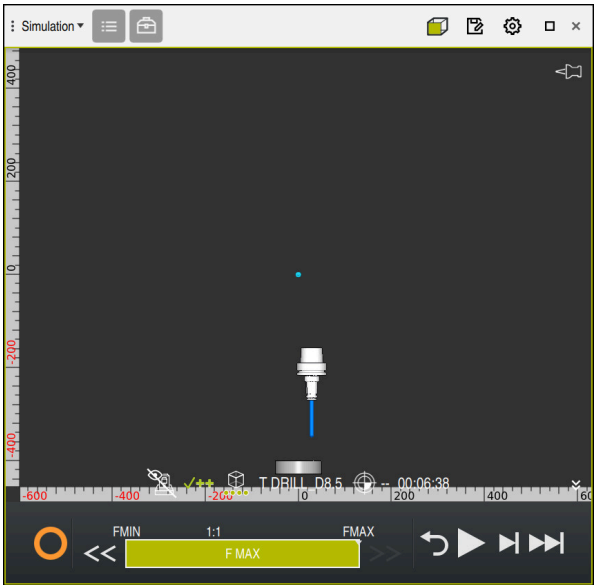
Application

In the **Simulation** workspace, you can choose between various pre-defined views in order to align the workpiece. This allows you to position the workpiece more quickly for the simulation.

Description of function

The control provides the following pre-defined views:

Symbol	Function
	Plan view
	Bottom view
	Front view
	Back view
	Side view (left side)
	Side view (right side)
	Isometric view



Front view of the simulated workpiece in the **Machine** mode

21.3 Exporting a simulated workpiece as STL file

Application

In the simulation you can use the **Save** function to save the current status of the simulated workpiece as a 3D model in STL format.

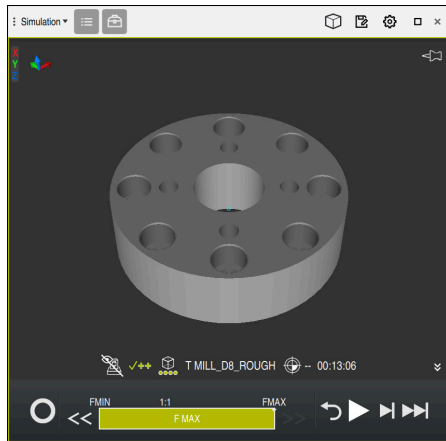
The file size of the 3D model depends on the complexity of the geometry and the selected model quality.

Related topics

- Customizing STL files in **CAD Viewer** (#152 / #1-04-1)

Further information: User's Manual for Setup and Program Run

Description of function



Simulated workpiece

This function can be used only in the **Editor** operating mode.

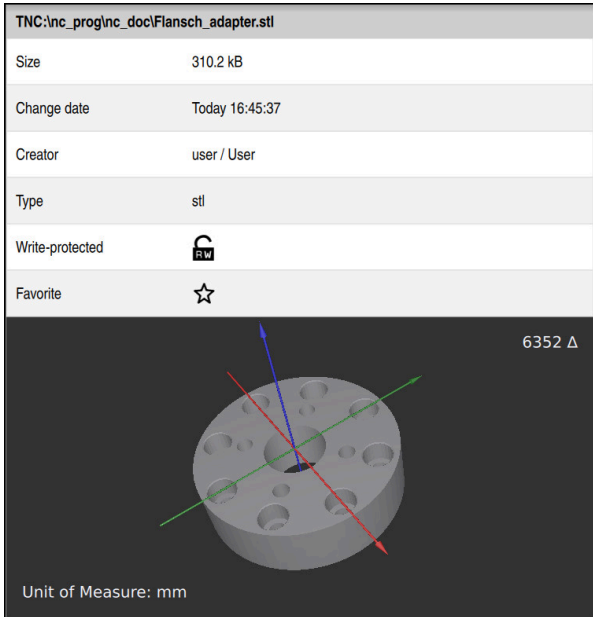
The control can only display STL files with up to 20,000 triangles. If the exported 3D model has too many triangles, due to the excessively high model quality, then you cannot use the exported 3D model on the control.

In this case, reduce the model quality in the simulation.

Further information: "The Simulation settings window", Page 534

You can also use the **3D mesh** function to reduce the number of triangles (#152 / #1-04-1).

Further information: User's Manual for Setup and Program Run



Simulated workpiece as saved STL file

21.3.1 Saving a simulated workpiece as STL file

To save a simulated workpiece as an STL file:



- ▶ Simulate workpiece



- ▶ Select the settings as needed
- ▶ Activate **Optimized saving of STL**, if appropriate (#152 / #1-04-1)



- The control simplifies the STL file when saving it.
- ▶ Select **Save**
- The control opens the **Save as** window.
- ▶ Enter the desired file name
- ▶ Select **Create**
- The control saves the created STL file.

Further information: "The Simulation settings window", Page 534

21.4 Advanced checks in the simulation

Application

The **Advanced checks** function allows you to check in the **Simulation** workspace whether collisions will occur (e.g., between the workpiece and the tool). This avoids unplanned downtimes due to a collision.

Description of function

The **Advanced checks** function can be used only in the **Editor** operating mode.

If you activate the **Advanced checks** toggle switch in the **Visualization options** column, the control opens the **Advanced checks** window.

Further information: "The Visualization options column", Page 530

The **Advanced checks** window allows activating the following tests:

- **Rapid-traverse cut**
The control displays a warning in case material is removed at rapid traverse.
- **Collision between workpiece and tool**
The control displays a warning in case of collisions between the tool carrier or tool shank and the workpiece.
- **Collision between workpiece and machine**
The control displays a warning in case of collisions between the workpiece and the machine (e.g., spindle).
The control does not consider the tool and the workpiece fixture.
- **Fixture collision**
The control displays a warning in case of collisions between the workpiece fixture and the tool including the tool carrier.

The control displays the material removal at rapid traverse and collision objects in red.

The control also considers inactive steps of a stepped tool.

You can activate several tests at the same time.

Notes

- The **Advanced checks** function helps reduce the danger of collision. However, the control cannot consider all possible constellations during operation.
- The **Advanced checks** function in the simulation uses the information from the workpiece blank definition for workpiece monitoring. Even if several workpieces are clamped in the machine, the control can monitor only the active workpiece blank!

Further information: "Defining a workpiece blank with BLK FORM", Page 152

21.5 Measuring function

Application

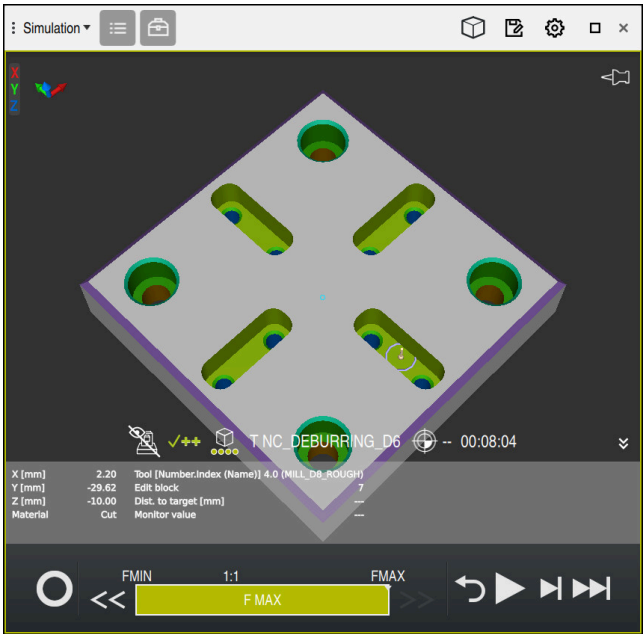
Use the measuring function to measure any points on the simulated workpiece. The control shows various pieces of information about the measured surface.

Requirement

- The **Workpiece** mode

Description of function

If you measure a point on the simulated workpiece, the cursor always locks onto the currently selected surface.



Measured point on simulated workpiece

The control shows the following information about the measured surface:

- Measured positions in the **X**, **Y** and **Z** axes, relative to the workpiece coordinate system **W-CS**
Further information: "Workpiece coordinate system W-CS", Page 312
- Status of the machined surface
 - **Material Cut** = Surface that has been machined
 - **Material NoCut** = Surface that has not been machined
- Cutting tool
- NC block currently running in the NC program
- Distance between the measured surface and the finished part

21.5.1 Measuring the difference between the workpiece blank and the finished part

To measure the difference between the workpiece blank and the finished part:

- ▶ Select an operating mode (e.g., **Editor**)
- ▶ Open an NC program with a workpiece blank and finished part defined in **BLK FORM FILE**
- ▶ Open the **Simulation** workspace



- ▶ Select the **Tool options** column

- ▶ Activate the **Measuring** toggle switch
- ▶ Select the **Workpiece coloring** selection menu
- ▶ Select **Model comparison**



- ▶ The control displays the workpiece blank and finished part defined in the **BLK FORM FILE** function.



- ▶ Start the simulation
- ▶ The control simulates the workpiece.
- ▶ Select the desired point on the simulated workpiece
- ▶ The control displays the difference in the dimension between the simulated workpiece and the finished part.



The control uses the **Model comparison** function to identify dimensional differences between the simulated workpiece and the finished part first in color, starting with differences greater than 0.2 mm.

Notes

- Depending on the selected quality in the **Simulation settings** window, the measurement result possibly does not match the real workpiece. The lower the quality the more strongly the simulated tool deviates from a circle, for example. During the simulation, this deviation (i.e., the chord error), is transferred to the workpiece. Set the highest quality for simulation.

Further information: "Simulation of tools", Page 537

- If you need to compensate for tools, you can use the measuring function to determine the tool to be compensated for.
- If you notice an error in the simulated workpiece, you can use the measuring function to determine the NC block that causes the error.

21.6 Cutout view in the simulation

Application

In the Cutout view you can cut through the simulated workpiece along any axis. This enables you to check holes and undercuts in the simulation, for example.

Requirement

- The **Workpiece** mode

Description of function

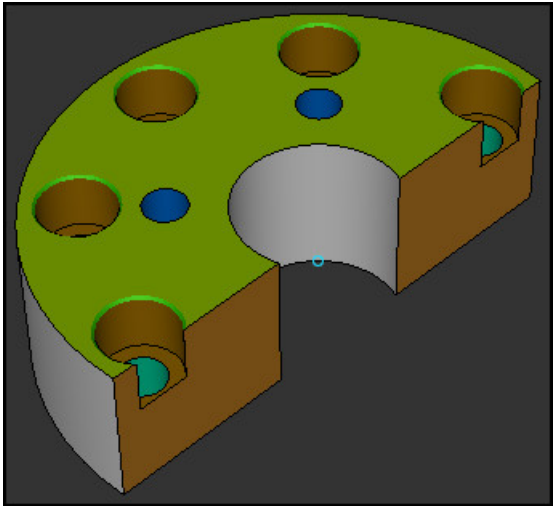
The Cutout view can be used in the **Editor** mode only.

The position of the sectional plane is shown as a percent value when it is shifted in the simulation. The sectional plane is retained until the control is restarted.

21.6.1 Shifting the sectional plane

To shift the sectional plane:

- 
 - ▶ Select the **Editor** operating mode
- 
 - ▶ Open the **Simulation** workspace
- 
 - ▶ Select the **Visualization options** column
- 
 - ▶ Select the **Workpiece** mode
- 
 - ▶ The control shows the workpiece view.
 - ▶ Select the **Workpiece options** column
 - ▶ Activate the **Cutout view** toggle switch
 - ▶ The control activates the **Cutout view**.
 - ▶ The selection menu allows you to select the area which the control shows after the cut such as **+X**
 - ▶ Use the slider to specify the desired percent value
 - ▶ The control simulates the workpiece with the selected sectional settings.



Simulated workpiece in the **Cutout view**

21.7 The Rewind function

Application

With the **Rewind** function, you can wind simulated machining steps forward and backward in order to check the traverses in an NC program and then optimize them, if required. This way, it is easier to estimate how prone to collisions some traverses near the workpiece or fixture are.

Requirements

- The **Editor** operating mode
- Desired program section simulated

Description of function

To activate this function, use the **Rewind** toggle switch in the **Workpiece options** column. While the function is active, the control shows sliders and buttons for winding forward and rewinding in Simulation mode.

You can only view the machining operation between two **TOOL CALL** blocks.

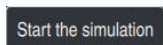
21.7.1 Analyzing a simulation warning

To analyze a warning of the simulation (e.g., a fixture violation for a jaw chuck):

- ▶ In the **Program** workspace, select the warning in the **Simulation report** area of the **Tests** column
- > The control selects the associated NC block in the NC program.



- ▶ In the **Simulation** workspace, select **Run simulation up to block number**
- > The control opens the **Run simulation up to block number** window. The control automatically enters the block number.



- ▶ Select **Start the simulation**
- > The control simulates up to the defined NC block.



- ▶ Select **Start a single block**
- > The control simulates a further NC block.



- ▶ If applicable, open the **Workpiece options** column
- ▶ Activate the **Rewind** toggle switch as needed
- > The control shows a slider and icons for forwarding and rewinding.



You can only rewind the simulation until the last **TOOL CALL** block.



- ▶ Use the slider to rewind approximately to a clamping jaw
- ▶ Press **Slow forward** if needed
- > The control displays the machining at reduced speed.
- ▶ When forwarding, pay attention to fixture violations



- ▶ If necessary, press **Fast forward** to fast-forward to a further clamping jaw
- ▶ Repeat the steps as necessary until the fixture violation is found
- ▶ For example, correct the working depth
- > The warning is analyzed and the NC program is corrected.

Notes

- When you open a new simulation or reset the simulation, the toggle switch appears dimmed.
- If the simulation aborts with an error message, the control does not show the slider.

21.8 Model comparison

Application

With the **Model comparison** function you can compare the workpiece blank and the finished part in STL or M3D format.

Related topics

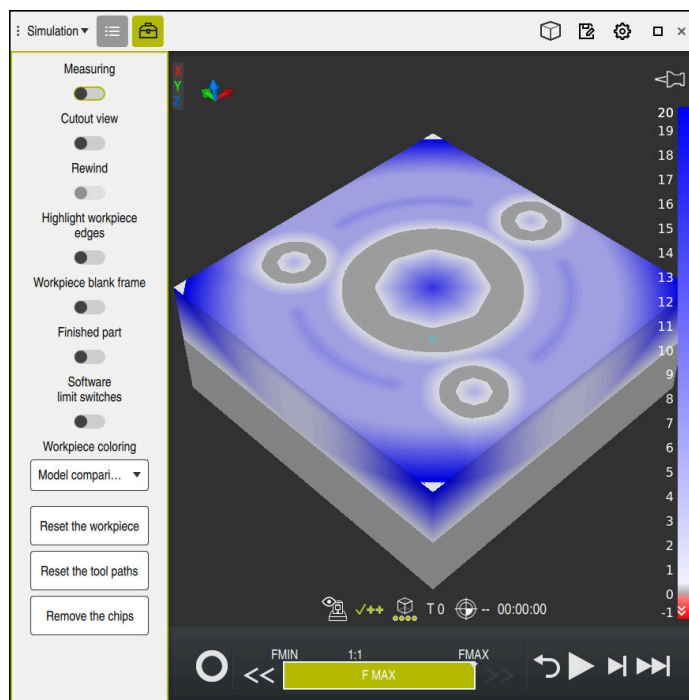
- Programming the blank and finished part with STL files

Further information: "STL file as workpiece blank with BLK FORM FILE",
Page 155

Requirements

- STL file or M3D file of workpiece blank and finished part
- The **Workpiece** mode
- Workpiece blank definition with **BLK FORM FILE**

Description of function



The control uses the **Model comparison** function to show the difference in material between the models being compared. The control uses a color transition from white to blue to show the difference in material. The more material there is covering the finished part model, the deeper the blue is. When material is removed from the finished part model, the control displays this removal in red.

Notes

- The control uses the **Model comparison** function to identify dimensional differences between the simulated workpiece and the finished part, starting with differences greater than 0.2 mm.
- Use the measuring function to measure the exact dimensional difference between the workpiece blank and the finished part.

Further information: "Measuring the difference between the workpiece blank and the finished part", Page 543

- The result depends on the quality of the STL files and the quality selected in the **Simulation settings** window. The lower the quality the more strongly the simulation deviates from the real result. Set the highest quality for simulation.

21.9 Center of rotation in the simulation

Application

By default, the center of rotation in the simulation is at the center of the model. When you zoom in, the center of rotation is always shifted to the center of the model. If you want to rotate the simulation around a specific point, then you can define the center of rotation manually.

Description of function

Use the **Center of rotation** function to manually set the center of rotation for the simulation.

The control shows the **Center of rotation** symbol as follows, depending on the status:

Symbol	Function
	The center of rotation is at the center of the model.
	The symbol blinks. The center of rotation can be shifted.
	The center of rotation was set manually.

21.9.1 Setting the center of rotation to a corner of the simulated workpiece

To set the center of rotation to a corner of the workpiece:

- ▶ Select an operating mode (e.g., **Editor**)
- ▶ Open the **Simulation** workspace
- > The center of rotation is at the center of the model.
 - ▶ Select **Center of rotation**
 - > The control switches the **Center of rotation** symbol. The symbol blinks.
 - ▶ Select a corner of the simulated workpiece
 - > The center of rotation is defined. The control switches the **Center of rotation** symbol to "set".

21.10 Simulation speed

Application

You can use a slider to select any speed for the simulation.



Description of function

This function can be used only in the **Editor** operating mode.

The standard speed for the simulation is set to **FMAX**. If you change the simulation speed, then this change is retained until the control is restarted.

You can change simulation speed before as well as during the simulation.

The control provides the following options:

Button	Meaning
	Activate minimum feed rate (0.01*T)
	Reduce the feed rate
	Feed-rate at 1:1 (real-time)
	Increase the feed rate
	Activate maximum feed rate (FMAX)

21.11 Simulating an NC program up to a certain NC block

Application

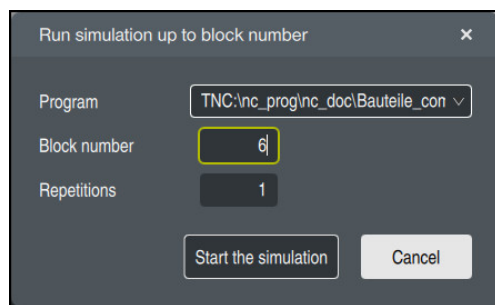
If you want to check a critical point in the NC program then you can simulate the NC program up to a specific NC block that you specify. Once the NC block is reached in the simulation, the control stops the simulation automatically. Starting from this NC block you can then continue the simulation, for example in **Single Block** mode or at a lower simulation speed.

Related topics

- Possibilities in the action bar
Further information: "Action bar", Page 536
- Simulation speed
Further information: "Simulation speed", Page 550

Description of function

This function can be used only in the **Editor** operating mode.



The **Run simulation up to block number** window with a defined NC block

The following settings options are offered in the **Run simulation up to block number** window:

- **Program**
This field offers a selection menu in which you can choose to simulate up to a specific NC block in the active main program or in a called program.
- **Block number**
In the **Block number** field, you enter the number of the NC block up to which the simulation should run. The number of the NC block refers to the NC program selected in the **Program** field.
- **Repetitions**
Use this field if the desired NC block is located within a program-section repeat. Enter in this field up to which iteration of the program-section repeat the simulation should run.
If you enter **1** or **0** in the **Repetitions** field, the control simulates up to the first iteration of the program section (repetition "0").
Further information: "Program-section repeats", Page 181

21.11.1 Simulating an NC program up to a certain NC block

To simulate up to a specific NC block:

- ▶ Open the **Simulation** workspace



- ▶ Select **Run simulation up to block number**
- The control opens the **Run simulation up to block number** window.
- ▶ Use the selection menu in the **Program** field to specify the main program or called program
- ▶ Enter the number of the desired NC block in the **Block number** field
- ▶ If the block involves a program-section repeat, enter the number of the iteration of the program-section repeat in the **Repetitions** field
- ▶ Select **Start the simulation**
- The control simulates the workpiece up to the selected NC block.

Start the simulation

Note

Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.

22

Touch probes

22.1 Calibrating a tool touch probe (#17 / #1-05-1)

22.1.1 Overview

Cycle	Call	Further information
480 CALIBRATE TT (#17 / #1-05-1) ■ Calibrating the tool touch probe	DEF-active	Page 555
484 CALIBRATE IR TT (#17 / #1-05-1) ■ Calibrating the tool touch probe (e.g., infrared tool touch probe)	DEF-active	Page 557

22.1.2 Fundamentals

Application

The following cycles can be used to calibrate the tool touch probe or the infrared tool touch probe.

Touch probe

For the touch probe you use a spherical probe contact

Calibration tool

The calibration tool must be a precisely cylindrical part, for example a calibration pin or cylindrical pin. The control saves all determined calibration values and automatically considers them during all subsequent tool measurements.

The touch probe tool type is not allowed for the calibration.



As the calibration tool, HEIDENHAIN recommends using the calibration pin (CAL_PIN) tool type.

22.1.3 Cycle 480 CALIBRATE TT (#17 / #1-05-1)

ISO programming

G480

Application



Refer to your machine manual!

You calibrate the TT with touch probe cycle **480**. The calibration process runs automatically. The control also measures the center offset of the calibration tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

You calibrate the TT with touch probe cycle **480**.

Cycle sequence

- 1 Clamp the calibration tool. The calibration tool must be a precisely cylindrical part, for example a cylindrical pin
- 2 Manually position the calibration tool in the working plane over the center of the TT
- 3 Position the calibration tool in the tool axis at approximately 15 mm plus set-up clearance over the TT
- 4 The first movement of the tool is along the tool axis. The tool is first moved to clearance height, i.e. set-up clearance + 15 mm.
- 5 The calibration process along the tool axis starts
- 6 This is followed by calibration in the working plane
- 7 The control positions the calibration tool in the working plane at a position of TT radius + set-up clearance + 11 mm
- 8 Then the control moves the tool downwards along the tool axis and the calibration process starts
- 9 During probing, the control moves in a square pattern
- 10 The control saves the calibration values and considers them during subsequent tool measurement
- 11 The control then retracts the stylus along the tool axis to set-up clearance and moves it to the center of the TT

Notes

- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.

Notes about machine parameters

- Use the machine parameter **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300) to define the functionality of the calibration cycle. Refer to your machine manual.
 - Use the machine parameter **centerPos** to define the position of the TT within the machine's working space.
- The TT needs to be recalibrated if you change the position of the TT on the table and/or a **centerPos** machine parameter.
- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height value that the tool tip would lie below the top of the probe contact, the control automatically positions the calibration tool above the top of the probe contact (safety zone from safetyDistToolAx (no. 114203)). Input: -99999.9999...+99999.9999

Example

11 TOOL CALL 12 Z
12 TCH PROBE 480 CALIBRATE TT ~
Q260=+100 ;CLEARANCE HEIGHT

22.1.4 Cycle 484 CALIBRATE IR TT (#17 / #1-05-1)

ISO programming

G484

Application

Cycle **484** allows you to calibrate your tool touch probe (e.g., the wireless infrared TT 460 tool touch probe). You can perform the calibration process with or without manual intervention.

- **With manual intervention:** If you define **Q536** = 0, then the control will stop before the calibration process. You then need to position the calibration tool manually above the center of the tool touch probe.
- **Without manual intervention:** If you define **Q536** = 1, then the control will automatically execute the cycle. You may have to program a prepositioning movement before. This depends on the value of the parameter **Q523 POSITION TT**.

Cycle sequence



Refer to your machine manual.

The machine manufacturer defines the functionality of the cycle.

To calibrate the tool touch probe, program the touch probe cycle **484**. In input parameter **Q536**, you can specify whether you want to run the cycle with or without manual intervention.

Q536 = 0: With manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibration tool
- ▶ Start the calibration cycle
- > The control interrupts the calibration cycle and displays a dialog.
- ▶ Manually position the calibration tool above the center of the tool touch probe.



Ensure that the calibration tool is located above the measuring surface of the probe contact.

- ▶ Press **NC Start** to resume cycle run
- > If you have programmed **Q523 = 2**, the control writes the calibrated position to the machine parameter **centerPos** (no. 114200)

Q536 = 1: Without manual intervention before calibration

Proceed as follows:

- ▶ Insert the calibrating tool
- ▶ Position the calibration tool above the center of the tool touch probe before the start of the cycle.



- Ensure that the calibration tool is located above the measuring surface of the probe contact.
- For a calibration process without manual intervention, you do not need to position the calibration tool above the center of the tool touch probe. The cycle adopts the position from the machine parameters and automatically moves the tool to this position.

- ▶ Start the calibration cycle
- > The calibration cycle is executed without stopping.
- > If you have programmed **Q523 = 2**, then the control writes the calibrated position to the machine parameter **centerPos** (no. 114200).

Notes**NOTICE****Danger of collision!**

If you program **Q536=1**, the tool must be pre-positioned before calling the cycle. The control also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle. There is a danger of collision!

- ▶ Specify whether to stop before cycle start or run the cycle automatically without stopping.

- The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. If you use a cylinder pin of these dimensions, the resulting deformation will only be 0.1 µm per 1 N of probing force. Major inaccuracies may occur if you use a calibration tool whose diameter is too small and/or that protrudes too far from the chuck.
- Before calibrating the touch probe, you must enter the exact length and radius of the calibration tool into the TOOL.T tool table.
- The TT needs to be recalibrated if you change its position on the table.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.

Cycle parameters

Help graphic	Parameter
	Q536 Stop before running (0=Stop)? Define whether the control will stop before the calibration process or whether the cycle will automatically be executed without a stop: 0: Stop before the calibration process. The control prompts you to position the calibration tool manually above the tool touch probe. After moving the tool to the approximate position above the tool touch probe, press NC Start to continue the calibration process or press the CANCEL button to cancel the calibration process. 1: Without stopping before the calibration process. The control starts the calibration process depending on Q523. Before running Cycle 484, you may have to position the tool above the tool touch probe. Input: 0, 1
	Q523 Position of tool probe (0-2)? Position of the tool touch probe: 0: Current position of the calibration tool. The tool touch probe is below the current position of the calibration tool. If Q536 = 0, position the calibration tool manually above the center of the tool touch probe during the cycle. If Q536 = 1, you need to position the calibration tool above the center of the tool touch probe before the start of the cycle. 1: Configured position of the tool touch probe. The control adopts the position from the machine parameter centerPos (no. 114201). You do not need to pre-position the tool. The calibration tool approaches the position automatically. 2: Current position of the calibration tool. See Q523 = 0. 0: The control additionally writes the determined position (where applicable) to the machine parameter centerPos (no. 114201) after calibration. Input: 0, 1, 2

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 484 CALIBRATE IR TT ~	
Q536=+0	;STOP BEFORE EXECUTION ~
Q523=+0	;AT CURRENT POSITION

23

**Touch-probe
cycles for tools
(#17 / #1-05-1)**

23.1 Overview

Measurement of milling cutters

Cycle		Call	Further information
481	CAL. TOOL LENGTH (#17 / #1-05-1) ■ Measuring the tool length	DEF-active	Page 568
482	CAL. TOOL RADIUS (#17 / #1-05-1) ■ Measuring the tool radius	DEF-active	Page 571
483	MEASURE TOOL (#17 / #1-05-1) ■ Measuring the tool length and radius	DEF-active	Page 575

23.2 Fundamentals

23.2.1 Application

In conjunction with the control's tool measurement cycles, the tool touch probe enables you to measure tools automatically: the compensation values for tool length and radius are stored in the tool table and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Measurement of a stationary tool
- Measurement of a rotating tool
- Measurement of individual teeth

Related topics

- Calibrate the tool touch probe

Further information: "Calibrating a tool touch probe (#17 / #1-05-1)", Page 554

23.2.2 Measuring a tool of length 0



Refer to your machine manual!

The optional machine parameter **maxToolLengthTT** (no. 122607) enables the machine manufacturer to define a maximum tool length for the tool measurement cycles.



HEIDENHAIN recommends that you always define tools with their actual tool length if possible.

The tool measuring cycles measure tools automatically. You can also measure tools defined with a length **L** of 0 in the tool table. To do this, the machine manufacturer must define a maximum tool length value in the optional machine parameter **maxToolLengthTT** (no. 122607). The control starts a search in which the actual tool length is roughly determined in the first step. This is followed by a fine measurement.

Cycle sequence

- 1 The tool travels to a clearance height centered above the touch probe.
The clearance height equals the value of the optional machine parameter **maxToolLengthTT** (no. 122607).
- 2 The control performs a rough measurement with the spindle standing still.
When measuring a stationary tool, the control will use the feed rate for probing defined in the machine parameter **probingFeed** (no. 122709).
- 3 The control saves the roughly measured length.
- 4 The control performs a fine measurement with the values from the tool measuring cycle.

Notes

NOTICE

Risk of collision!

If the machine manufacturer fails to define the optional machine parameter **maxToolLengthTT** (no. 122607), there will be no tool search. The control pre-positions the tool with a length of 0. Risk of collision!

- ▶ Observe the machine parameter value in the machine manual.
- ▶ Define tools with the actual tool length **L**

NOTICE

Risk of collision!

Risk of collision if the tool is longer than the value of the optional machine parameter **maxToolLengthTT** (no. 122607)!

- ▶ Observe the machine parameter value in the machine manual

23.2.3 Setting machine parameters



- The touch probe cycles , **481, 482, 483**, can be hidden with the optional machine parameter **hideMeasureTT** (no. 128901).



Programming and operating notes:

- Before you start working with the touch probe cycles, check all machine parameters defined in **ProbeSettings > CfgTT** (no. 122700) and **CfgTTRoundStylus** (no. 114200) or **CfgTTRectStylus** (no. 114300).
- When measuring a stationary tool, the control will use the feed rate for probing defined in the **probingFeed** machine parameter (no. 122709).

Spindle speed setting

When measuring a rotating tool, the control automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

$$n = \text{maxPeriphSpeedMeas} / (r \cdot 0.0063) \text{ where}$$

Abbreviation	Definition
n	Rotational speed [rpm]
maxPeriphSpeedMeas (No. 122712)	Maximum permissible cutting speed in m/min
r	Active tool radius [mm]

Setting of the feed rate

The probing feed rate is calculated as follows:

$$v = \text{measuring tolerance} \cdot n$$

Abbreviation	Definition
v	Probing feed rate [mm/min]
Measuring tolerance	Measuring tolerance [mm], depending on maxPeriphSpeedMeas (no. 122712)
n	Shaft speed [rpm]

probingFeedCalc (no. 122710) determines the calculation of the probing feed rate. The control provides the following options:

- **ConstantTolerance**
- **VariableTolerance**
- **ConstantFeed**

ConstantTolerance:

The measuring tolerance remains constant—regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The lower you set the maximum permissible rotational speed (**maxPeriphSpeedMeas** (no. 122712)) and the permissible tolerance (**measureTolerance1** (no. 122715)), the sooner you will encounter this effect.

VariableTolerance:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The control adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	measureTolerance1
30 to 60 mm	$2 \cdot \text{measureTolerance1}$
60 to 90 mm	$3 \cdot \text{measureTolerance1}$
90 to 120 mm	$4 \cdot \text{measureTolerance1}$

ConstantFeed:

The measuring feed rate remains constant; the measuring error, however, rises linearly with the increase in tool radius:

$$\text{Measuring tolerance} = (r \cdot \text{measureTolerance1}) / 5 \text{ mm) where}$$

Abbreviation	Definition
r	Active tool radius [mm]
measureTolerance1	Maximum permissible error of measurement

Setting for consideration of parallel axes and changes in the kinematics



Refer to your machine manual.

Using the optional machine parameter **calPosType** (no. 122606), the machine manufacturer defines whether the position of parallel axes and changes in the kinematics should be considered for calibration and measuring. A change in kinematics might for example be a head change.

Auxiliary or parallel axes cannot be probed, regardless of the setting of the optional machine parameter **calPosType** (no. 122606).

If the machine manufacturer changes the setting of the optional machine parameter, you need to recalibrate the tool touch probe.

23.2.4 Entries in the tool table for milling tools

Abbr.	Inputs	Dialog
CUT	The number of teeth of the tool for automatic tool measurement or cutting data calculation (maximum of 20 teeth)	Number of teeth?
LTOL	Permitted tool length deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...5.0000	Wear tolerance: length?
RTOL	Permitted tool radius deviation in wear detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...5.0000	Wear tolerance: radius?
DIRECT.	Cutting direction of the tool for automatic tool measurement with a rotating tool. Input: -, +	Cutting direction (M3 = -)?
R-OFFS	Position of tool upon length measurement, offset between the probe contact center and the tool center for automatic tool measurement. Default setting: No value entered (offset = tool radius) Input: -99999.9999...+99999.9999	Tool offset: radius?
L-OFFS	Position of tool upon radius measurement, distance between the probe contact top edge and the tool tip for automatic tool measurement. Is added to the offsetToolAxis machine parameter (no. 122707). Input: -99999.9999...+99999.9999	Tool offset: length?
LBREAK	Permitted tool length deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...9.0000	Breakage tolerance: length?

Abbr.	Inputs	Dialog
RBREAK	Permitted tool radius deviation in breakage detection for automatic tool measurement. If the entered value is exceeded, the control locks the tool in the column TL (status L). Input: 0.0000...9.0000	Breakage tolerance: radius?

Input examples for common tool types

Tool type	CUT	R-OFFS	L-OFFS
Drill	No function	0: No offset required because tool tip is to be measured	
End mill	4: four cutting edges	R: Offset required because the tool diameter is greater than the contact plate diameter of the TT	0: No additional offset required during radius measurement. Offset from offsetToolAxis (no. 122707) used.
Spherical cutter with a diameter of 10 mm	4: four cutting edges	0: No offset required because the south pole of the ball is to be measured.	5: At a diameter of 10 mm, the tool radius will be defined as offset. If this is not the case, the diameter of the spherical cutter will be measured too far down. So the tool diameter will not be correct.

23.3 Measurement of milling cutters (#17 / #1-05-1)

23.3.1 Cycle 481 CAL. TOOL LENGTH (#17 / #1-05-1)

ISO programming

G481

Application



Refer to your machine manual!

For measuring the tool length, program touch probe cycle **481**. Input parameters allow you to select which of the three following methods will be used to measure the tool length:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is stationary.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is stationary.

Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**R-OFFS**).

Cycle for measuring a stationary tool (e.g., for drills)

The control positions the tool to be measured above the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until contact is made. For this measurement, enter 0 in the tool table under Tool offset: radius (**R-OFFS**).

Cycle for measuring individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). You can enter an additional offset in Tool offset: Length (**L-OFFS**) in the tool table. The control probes the tool radially while it is rotating to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation.

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
 - ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded
- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
 - You can run an individual tooth measurement for tools with **up to 20 teeth**.
 - You can use the **CAL_PIN** (calibration pin) tool type only for comparing measured values. To do so, program **Q340** with the value **2**.

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. 0: The measured tool length is written to column L of tool table TOOL.T, and the tool compensation is set to DL = 0. If there is already a value in TOOL.T, it will be overwritten. 1: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation is also available in the Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). 2: The measured tool length is compared to the tool length L from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q115. Nothing is entered under L or DL in the tool table. Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: 0, 1

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 481 CAL. TOOL LENGTH ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;MEASURE INDIVIDUAL TEETH

23.3.2 Cycle 482 CAL. TOOL RADIUS (#17 / #1-05-1)

ISO programming

G482

Application



Refer to your machine manual!

If you want to measure the tool radius, program the touch probe cycle **482**. Select via input parameters by which of two methods the tool radius is to be measured:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

The control pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the face of the milling tool to the upper edge of the touch probe head is defined in **offsetToolAxis** (no. 122707). The control probes the tool radially while it is rotating.

If you have programmed a subsequent measurement of individual teeth, the control will measure the radius of each tooth with the aid of oriented spindle stops.

Further information: "Notes for individual tooth measurement Q341=1", Page 572

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- You can use the **CAL_PIN** (calibration pin) tool type only for comparing measured values. To do so, program **Q340** with the value **2**.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement Q341=1

NOTICE

Caution: Danger to the tool and workpiece!

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
- ▶ Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

$$\epsilon = 90 - \operatorname{atan}\left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}}\right)$$

Abbreviation	Definition
ϵ	Maximum angle of twist
$h[tt]$	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool



On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist. Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

NOTICE

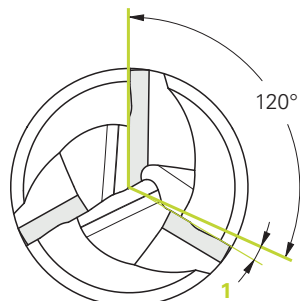
Caution: Possible material damage!

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing wear. The higher the angle deviation and the larger the tool radius, the more probably this behavior can occur. If the control compensates for the tool incorrectly after individual tooth measurement, the workpiece may have to be rejected.

- ▶ Check the workpiece dimensions during subsequent machining operations

Individual tooth measurement of tools with an uneven distribution of teeth may cause the control to identify non-existing breakage and lock the tool.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.



1 Angle deviation

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. 0: The measured tool radius is written to column R of the TOOL.T tool table, and the tool compensation is set to DR = 0. If there is already a value in TOOL.T, it will be overwritten. 1: The measured tool radius is compared to the tool radius R from TOOL.T. The control calculates the deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation is also available in the Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). 2: The measured tool radius is compared to the tool radius from TOOL.T. The control calculates the deviation from the stored value and writes it to Q parameter Q116. Nothing is entered under R or DR in the tool table. Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: 0, 1

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 482 CAL. TOOL RADIUS ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;MEASURE INDIVIDUAL TEETH

23.3.3 Cycle 483 MEASURE TOOL (#17 / #1-05-1)

ISO programming

G483

Application



Refer to your machine manual!

To measure both the length and radius of a tool, program the touch probe cycle **483**. This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Input parameters allow you to select which of the two following methods will be used to measure the tool:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth

Measuring the tool while it is rotating:

The control measures the tool in a fixed programmed sequence. First, if possible, it measures the tool length, and then the tool radius.

Measuring the individual teeth:

The control measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for touch probe cycles **481** and **482**.

Further information: "Notes for individual tooth measurement of radius Q341=1", Page 576

Notes

NOTICE

Danger of collision!

If you set **stopOnCheck** (no. 122717) to **FALSE**, the control does not evaluate the result parameter **Q199** and the NC program is not stopped if the breakage tolerance is exceeded. There is a danger of collision!

- ▶ Set **stopOnCheck** (no. 122717) to **TRUE**
- ▶ You must then take steps to ensure that the NC program stops if the breakage tolerance is exceeded

- Before measuring a tool for the first time, enter the following data on the tool into the TOOL.T tool table: the approximate radius, the approximate length, the number of teeth, and the cutting direction.
- You can use the **CAL_PIN** (calibration pin) tool type only for comparing measured values. To do so, program **Q340** with the value **2**.

Note regarding machine parameters

- In the machine parameter **probingCapability** (no. 122723), the machine manufacturer defines the functionality of the cycle. This parameter allows you to permit tool length measurement with a stationary spindle and at the same time to inhibit tool radius and individual tooth measurements.
- Cylindrical tools with diamond surfaces can be measured while the spindle is stationary. To do so, in the tool table define the number of teeth **CUT** as 0 and adjust the machine parameter **CfgTT**. Refer to your machine manual.

Notes for individual tooth measurement of radius Q341=1

NOTICE

Caution: Danger to the tool and workpiece!

Individual tooth measurement of tools with a large angle of twist may result in a failure of the control to identify tool wear or a broken tool. In this case, tool and workpiece damage may result during subsequent machining operations.

- ▶ Check the workpiece dimensions (for example, by using a workpiece touch probe)
- ▶ Check the workpiece optically in order to exclude broken tools

If the maximum angle of twist is exceeded, you should not carry out individual tooth measurement.

On tools with an even distribution of teeth, a maximum angle of twist can be defined as follows:

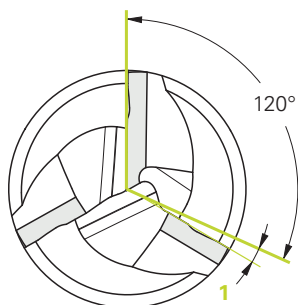
$$\epsilon = 90 - \operatorname{atan}\left(\frac{h[tt]}{\frac{R \times 2 \times \pi}{x}}\right)$$

Abbreviation	Definition
ϵ	Maximum angle of twist
h[tt]	Height of tool touch probe contact
R	Tool radius
x	Number of teeth of tool



On tools with an uneven distribution of teeth, there is no calculation formula for the maximum angle of twist. Check these tools optically in order to exclude breaks. You can measure wear indirectly by measuring the workpiece.

The higher the angle deviation **1** and the larger the tool radius, the more probably this behavior can occur.



1 Angle deviation

Cycle parameters

Help graphic	Parameter
	Q340 Tool measurement mode (0-2)? Define whether and how the measured data will be entered in the tool table. 0: The measured tool length and the measured tool radius are written to columns L and R of the TOOL.T tool table, and the tool compensation is set to DL = 0 and DR = 0. If there is already a value in TOOL.T, it will be overwritten. 1: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored value and enters them into TOOL.T as the delta values DL and DR. The deviation is also available in the Q parameters Q115 and Q116 . If the delta value is greater than the permissible tool length or tool radius tolerance for wear or break detection, the control will lock the tool (status L in TOOL.T). 2: The measured tool length and the measured tool radius are compared to the tool length L and tool radius R in TOOL.T. The control calculates the deviation from the stored values and writes it to the Q parameter Q115 or Q116 . Nothing is entered under L, R, or DL, DR in the tool table. Input: 0, 1, 2
	Q260 Clearance height? Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece preset. If you enter such a small clearance height that the tool tip would lie below the top of the probe contact, the control automatically positions the tool above the top of the probe contact (safety zone from safetyDistStylus). Input: -99999.9999...+99999.9999
	Q341 Probe the teeth? 0=no/1=yes Define whether the control will measure the individual teeth (maximum of 20 teeth) Input: 0, 1

Example

11 TOOL CALL 12 Z	
12 TCH PROBE 483 MEASURE TOOL ~	
Q340=+1	;ENTER DELTA VALUES ~
Q260=+100	;CLEARANCE HEIGHT ~
Q341=+1	;MEASURE INDIVIDUAL TEETH

24

Tables

24.1 The Tables operating mode

Application

In the **Tables** operating mode you can open various tables and edit them as necessary.

Description of function

If you select **Add**, the control displays the **Quick selection new table** and **Open File** workspaces.

In the **Quick selection new table** workspace you can create a new table and open some tables directly.

Further information: "The Quick selection workspaces", Page 360

In the **Open File** workspace, you can open an existing table or create a new table.

Further information: "The Open File workspace", Page 359

Multiple tables can be open at the same time. The control displays each table in a separate workspace.

If a table is selected for program run or simulation, the control shows the status **M** or **S** on the tab of the application. The status of the active application is highlighted in color and for the remaining applications in gray.

You can open the **Table, Form and Document** workspaces in every application.

Further information: "The Table workspace", Page 586

Further information: "The Form workspace for tables", Page 594

You can select various functions by using the context menu (e.g., **Copy**).

Further information: "Context menu", Page 511



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

Buttons

In the **Tables** operating mode, the function bar contains the following buttons that can be used for any table:

Button	Meaning
Undo	The control undoes the last change.
Redo	The control restores the change that was undone.
GOTO record	The control opens the GOTO jump instruction window. The control selects the line number that you defined.
Edit	If the toggle switch is active, you can edit the table.
Mark row	The control marks the currently selected row.

Depending on the selected table, the control provides the following additional buttons in the function bar:

Button	Meaning
Insert rows	The control opens the Insert rows window where you can insert one or more new rows. If you enable the Append checkbox, the control will insert the rows after the last table row.
Reset row	The control resets all data contained in the row.
Delete rows	The control deletes the currently selected row.
Insert tool	The control opens the Insert tool window where you can define the following: ■ Type: ■ Line number (Tool number?) ■ Number of rows ■ Index ■ Append Append rows at the end of the table Further information: User's Manual for Setup and Program Run
Delete tool	The control deletes the tool selected in the tool management. You cannot delete any tools that have been entered into the pocket table. The button is dimmed. Further information: User's Manual for Setup and Program Run
Import	The control imports table contents from a CSV file (e.g., tool data). Further information: "Importing and exporting of table contents", Page 597
Inspect	The control inspects a tool.
Unload	The control unloads a tool.
Load	The controls loads a tool.
Activate the preset	The control activates the currently selected row of the preset table as preset. Further information: User's Manual for Setup and Program Run
Lock record	The control locks the currently selected row of the preset table and thus protects the contents from changes.



Refer to your machine manual.

If necessary, the machine manufacturer adapts the buttons.

If at least one table row is marked, the control contains the following buttons in the action bar:

- **Cut**
- **Copy**
- **Reset**
Only for tables that also provide the **Reset row** button
- **Delete**
- **Select all**
- **Cancel marking**
- **Export**
Export data of the selected lines as a CSV file
Further information: "Importing and exporting of table contents", Page 597
- **Cancel**

24.1.1 Editing the contents of tables

To edit the contents of a table:

- Select the desired table cell



- Activate **Edit**
- The control enables the values for editing.

i To edit a table content, you can also double-tap or double-click the table cell. The control displays the **Editing disabled. Enable?** window. You can enable the values for editing or cancel the process.

i If the **Edit** toggle switch is enabled, you can edit the contents both in the **Table** workspace and in the **Form** workspace.

Notes

- Use the **+**, **-**, *****, **/**, **(**, and **)** keys for calculations in the numerical input fields.
- The control enables you to transfer tables from previous controls to the TNC7 go and to adapt them automatically, if needed.
- When you open a table where columns are missing, for example in case of a tool table from a previous control, the control will display the **Incomplete table layout** window.

When you create a new table in the file manager, the table does not contain information on the required columns yet. When you open the table for the first time, the **Incomplete table layout** window will open in the **Tables** operating mode.

In the **Incomplete table layout** window, a selection menu allows you to select a table template. The control shows which table columns are added or removed, if applicable.

- If you have, for example, processed tables in a text editor, the control provides the **Update TAB / PGM** function. Use this function to complete an incorrect table format.

Further information: "Adapting files", Page 363

i Edit tables only by using the table editor in the **Tables** operating mode to avoid errors (e.g., format errors).

Notes about machine parameters

Refer to your machine manual.

- Using the optional machine parameter **CfgTableCellCheck** (no. 141300), the machine manufacturer can define rules for table columns. The machine parameter allows you to define columns as mandatory fields or reset them automatically to a default value. If this rule is not fulfilled, the control will display an information symbol.
- The machine manufacturer uses the machine parameter **CfgTableCellLock** (no. 135600) to define in which cases individual table cells will be blocked or write-protected. On some machines, you cannot change the tool type once a tool has been inserted into the machine.

24.2 The Create new table window

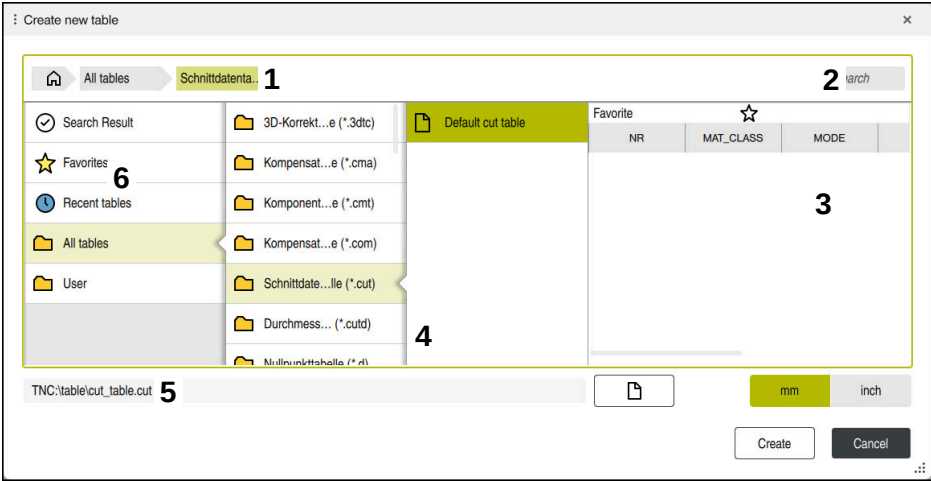
Application

You can create tables using the **Create new table** window in the **Quick selection new table** workspace.

Related topics

- The **Quick selection new table** workspace
Further information: "The Quick selection workspaces", Page 360
- Available file types for tables
Further information: "File types", Page 355

Description of function



The **Create new table** window

The **Create new table** window shows the following areas:

- 1 Navigation path
In the navigation path the control shows the position of the current folder in the folder structure. Use the individual elements of the navigation path to move to a higher folder level. You can change the path or use the history to open a previous path.
- 2 Content columns
The control shows a folder and the available prototypes for each table type.
- 3 Searching
You can search for any strings. The control displays the results under **Search Result**.
- 4 The control shows the following information and functions:
 - Add or remove a favorite
 - Preview
- 5 Unit of measure (mm or inches)
- 6 Path of the table to be created

7 Navigation column

The navigation column offers the following possibilities for navigation:

- **Search Result**

- **Favorites**

The control displays all folders and prototypes that you have marked as favorites.

- **Last functions**

The control shows the eleven most recently used prototypes.

- **All functions**

The control shows all available table types in the folder structure.

Notes

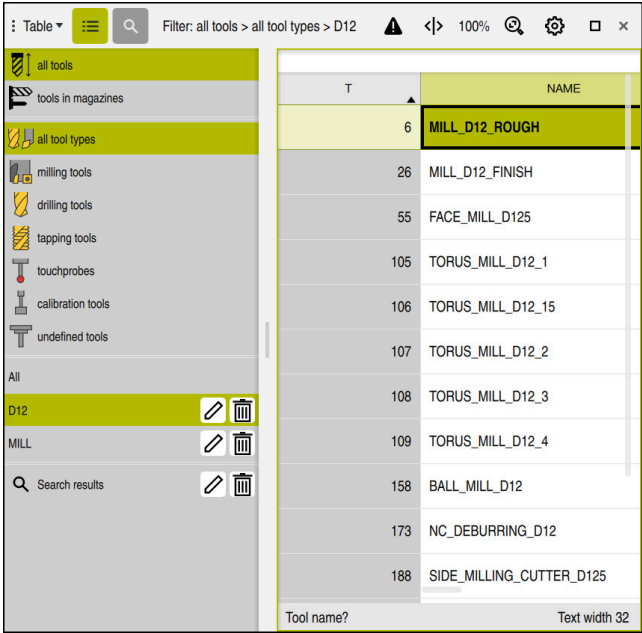
- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., +). Due to SQL commands, these characters can cause problems when data are input or read.
- With the optional machine parameter **CfgTableCreate** (no. 140900), the machine manufacturer can provide additional areas in the navigation column (e.g., tables for the user).
- With the optional machine parameter **dialogText** (no. 105506), the machine manufacturer can define other names for the table types (e.g., tool table instead of **t**).

24.3 The Table workspace

Application

In the **Table** workspace, the control shows the contents of a table. You can search in all tables and filter the table content.

Description of function



The **Table** workspace

In the **Tables** operating mode, the **Table** workspace is open in every application by default.

The control displays the name and path of the file above the header of the table.

When you select the title of a column, the control will sort the table contents by this column in ascending or descending order.

If the table allows it, you can also edit the table contents in this workspace.

If you select a cell, the control displays the following information on the selected parameter in the dialog bar:

- Name
- Text width or input range
- Unit of measure as needed

Refer to your machine manual.
If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

Icons and shortcuts

The **Table** workspace contains the following icons or shortcuts:

Icon or shortcut	Meaning
	Open or close the Filter column Further information: "The Filter column in the Table workspace", Page 588
 [CTRL] + [F]	Open or close the Search column Further information: "The Search column in the Table workspace", Page 590
	■ Activate or deactivate "Rules not met" filter The control displays only the rows that do not meet the rules defined by the machine manufacturer in CfgTable-CellCheck (no. 141300). ■ Open the Data record consistency violations window In these rows, the control shows the icon at the start of the row, even if the filter is inactive. The control displays a window showing how the row violates the rules.
	Edit table characteristics Further information: "Changing table properties of freely definable tables", Page 608
100%	Current size of the content Open or close the Scale selection menu
	Reset scaling Set the font size of the table to 100%
	Open or close settings in the Tables window Further information: "Settings in the Table workspace", Page 591
	Open the Search column and edit the selected filter Only in the Filter column Further information: "The Search column in the Table workspace", Page 590
	Delete the selected filter Only in the Filter column Further information: "The Filter column in the Table workspace", Page 588
[CTRL] + [A]	Mark all rows
[CTRL] + [SPACE]	Mark the active row or end the marking function
[SHIFT] + [UP]	Additionally mark the row above
[SHIFT] + [DOWN]	Additionally mark the row below

The Filter column in the Table workspace

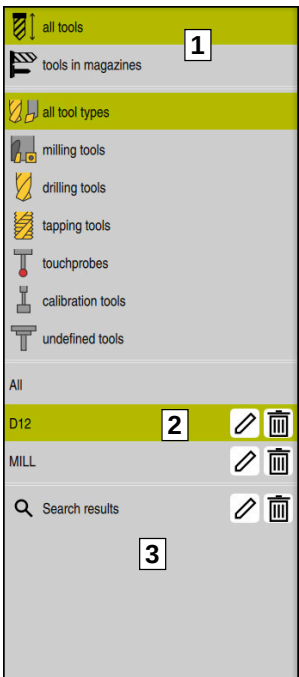
The control provides default filters for the following tables:

- Tool management
- Pocket table
- Presets
- Tool table

For all other tables, you can create user-defined filters.

Further information: "User-defined filters", Page 590

The **Filter** column in the **Table** workspace is divided into several filter groups. The control separates the filter groups with a white double line.



The control provides the following filter groups:

- 1 Default filters
Filters that are available by default for the respective table
In the **Tool management** application, the default filters comprise two filter groups
- 2 User-defined filters
Search processes saved as filters
Further information: "User-defined filters", Page 590
- 3 **Search results**
Results of the **Search** column
Further information: "The Search column in the Table workspace", Page 590

When you tap or click a filter once, the control activates only the selected filter in the respective group.

When you double-tap or click a filter, the control activates the selected filter in addition to the active filters.

Further information: "Connecting conditions and filters", Page 590

The control highlights active filters green.



Refer to your machine manual.

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

Filters in the Tool management

The control provides the following default filters in the **Tool management**:

- All tools
- Magazine tools
- All types
- Milling cutters
- Drills
- Taps
- Thread cutters
- Touch probes (#17 / #1-05-1)
- Calibration tools
- Undefined tools

Filters in the Pocket table

The control provides the following default filters in the **Pocket table**:

- All magazines
- spindle
- main magazine
- all pockets
- empty pockets
- occupied pockets
- Locked pockets

Filters in the Presets table

The control provides the following default filters in the **Presets** table:

- Show all
- Offsets

User-defined filters

You can additionally create user-defined filters by saving a search.

Further information: "The Search column in the Table workspace", Page 590

The control only shows this filter group when you create a user-defined filter. In addition to the user-defined filters, the control displays the **All** filter option.

Further information: "The Filter column in the Table workspace", Page 588

Connecting conditions and filters

The control connects the filters as follows:

- AND operation for several requirements within one filter
You create, for example, a user-defined filter that contains the requirements **R = 8** and **L > 150**. The control filters the table rows when you activate this filter. The control displays only the table rows that meet both requirements at the same time.
- OR operation between filters of the same filter groups
When you activate the default filters **Drills** and **Taps**, for example, the control filters the table rows. The control displays only the table rows that meet at least one of the conditions. The table row must contain either a drill or a tap.
- AND operation between filters of different filter groups
You create, for example, a user-defined filter that contains the requirement **R > 8**. When you activate this filter and the default filter **Milling cutters**, the control filters the table rows. The control displays only the table rows that meet both requirements at the same time.

The Search column in the Table workspace

The control offers a search function in all tables.

You can define multiple search conditions in the **Search** function.

Each condition includes the following information:

- Table column, such as **T** or **NAME**
Use the **Search in** selection menu to select the column.
- Operator if applicable (e.g., **Contains** or **Equal to (=)**)
Use the **Operator** selection menu to select the operator.
- Search term in the **Search for** input field

 If you search the columns using predefined selection values, the control offers a selection menu instead of the input field.

The control provides the following buttons:

Button	Meaning
+	Use Add to add several conditions. The conditions will have a combined effect when you perform the search. You can save several conditions in a user-defined filter.
Search	The control searches the table.
Reset	The control resets the entered conditions and removes any additional conditions.
Save	You can save the entered conditions as a user-defined filter. You can assign any name to the filter. Further information: "User-defined filters", Page 590

An unsaved search works like a user-defined filter. If an unsaved search is active, the control highlights the **Search results** filter group green in the **Filter** column.

Further information: "The Filter column in the Table workspace", Page 588



Refer to your machine manual.

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

Settings in the Table workspace

In the **Tables** window, you can influence the contents shown in the **Table** workspace.

The **Tables** window consists of the following areas:

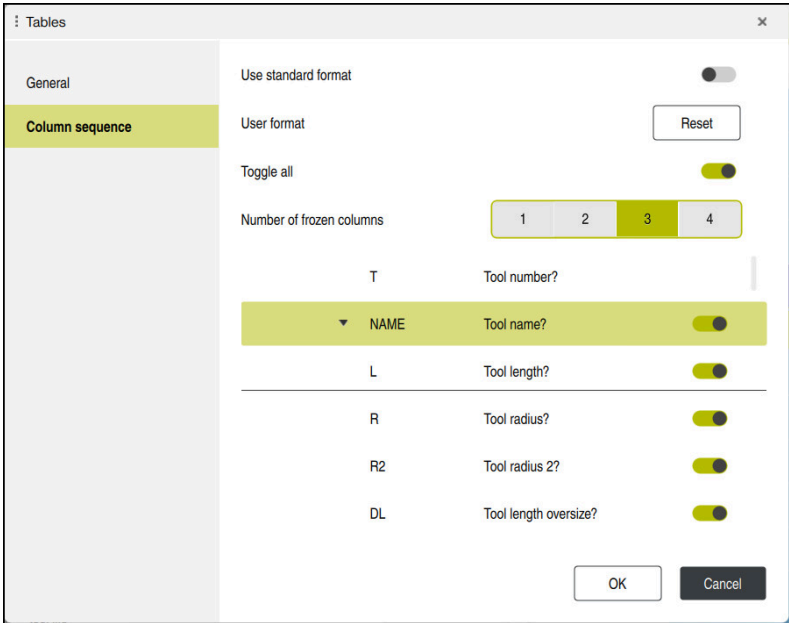
- **General**
- **Column sequence**

The General area

The setting selected in the **General** area is modally effective.

If the **Synchronize table and form** toggle switch is active, the cursor will move synchronously. If, for example, you select a different table column in the **Table** workspace, the control moves the cursor synchronously in the **Form** workspace.

The Column sequence area



The **Tables** window

The **Column sequence** area contains the following settings:

Setting	Meaning
Use standard format	If you activate the toggle switch, the control shows all table columns, indicating them in the standard sequence. If you deactivate the toggle switch, the control restores the previous setting.
User format	If you select the Reset button, the control resets the adaptations to the settings of the standard format.
Toggle all	If you activate the toggle switch, the control shows all table columns. If you deactivate the toggle switch, the control hides all table columns. The first column in each table cannot be hidden.
Number of frozen columns	You define how many table columns the control freezes at the left table edge. You can freeze up to four table columns. These table columns will remain visible even when you navigate further to the right within the table.
Columns of the currently opened table	The control displays all table columns below each other. Use the toggle switches to separately hide or show each table column. The control displays a line below the selected number of frozen columns. When you select a table column, the control displays up and down arrows. Use these arrows to change the sequence of the columns. The respective first column in the table cannot be shifted.

The settings in the **Column sequence** area only apply to the currently opened table.

Virtual columns

In the **Table** workspace, the control can display virtual columns. Virtual columns are not included in the table files but contain calculated values from other data.

The control contains the following virtual columns:

Column	Meaning	Application
MAGAZIN	The control shows whether the tool is currently in the magazine or in the spindle.	■ Tool management ■ Pocket table
TOOL_LIFE	Information on the tool life: ■ +: The current tool life CUR_TIME is at least 5 min below the maximum tool life TIME2. ■ -: The current tool life CUR_TIME is less than 5 min below the maximum tool life TIME2. ■ X: The tool has reached the maximum tool life TIME2. ■ ?: No value defined for the maximum tool life TIME2	■ Tool management ■ Pocket table



Refer to your machine manual.

The machine manufacturer can modify virtual columns and define further columns.

Notes

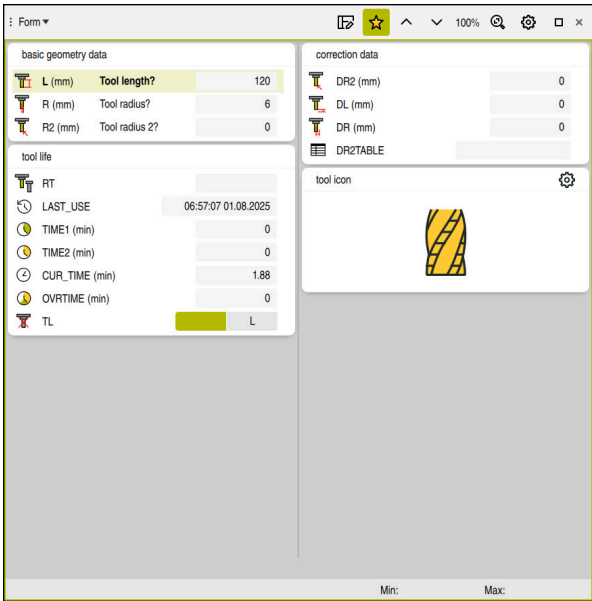
- If you insert the contents of the clipboard while editing is inactive, the control displays the **Editing disabled. Enable?** prompt. If you select **Yes**, the control inserts the copied contents.
- Refer to your machine manual.
Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.
- Since the values in virtual columns are not included in the table file, you cannot read these values with functions such as:
 - NC functions (e.g., **TABDATA READ**)
 - **HEIDENHAIN DNC** (#18 / #3-03-1)
 - **OPC UA NC Server** (#56-61 / #3-02-1*)

24.4 The Form workspace for tables

Application

In the **Form** workspace, the control shows all contents of a selected table row. Depending on the table, you can edit the values in the form.

Description of function



The **Form** workspace in the **Favorites** view

The control displays the following information for each parameter:

- An icon, if applicable
- Name
- Text width or input range
Only in the dialog bar
- Unit of measure as needed
Only in the dialog bar
- Description
- Current value

The control displays the contents of specific tables in groups within the **Form** workspace.



Refer to your machine manual.
If necessary, the machine manufacturer adapts the contents displayed (e.g., the titles of table columns).

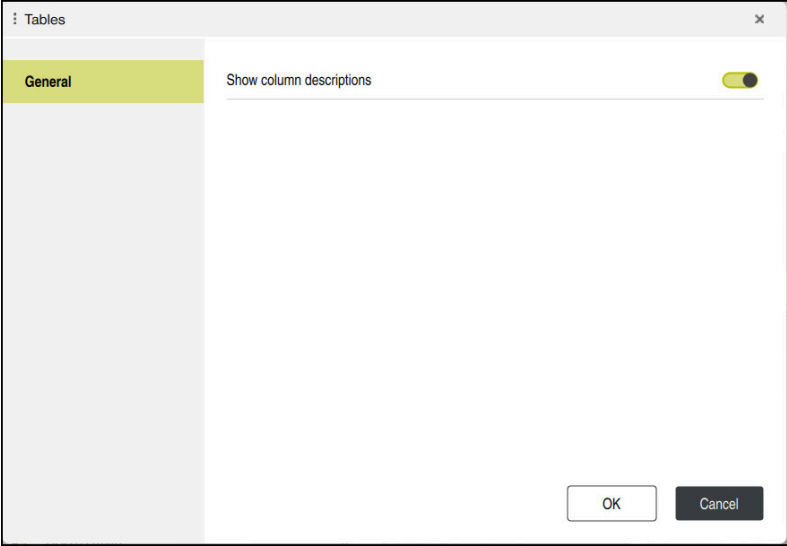
Icons

The **Form** workspace contains the following icons or shortcuts:

Icon or shortcut	Meaning
	Configure the layout You can make the following layout adaptations: ■ Add or remove areas to the Favorites view ■ Rearrange areas using the gripper ■ Add or remove columns
	Favorites In this view, the control shows the areas that are marked as favorites. You can create a user-defined view using the favorites. If the icon is not active, the control shows all areas.
^ [SHIFT] + [UP] ▼ [SHIFT] + [DOWN]	Navigate Navigate between table rows
100%	Current size of the content Open or close the Scale selection menu
	Reset scaling Set font size of the content to 100%
	Settings ■ Open the settings in the Tables window Further information: "Settings in the Form workspace", Page 596 ■ Change the size of the graphic in the Tool Icon area
	Add The control only shows this icon when you are adapting the layout. With this icon you can add the following elements: ■ Column You can divide the workspace into several columns. Further information: "Adding a column in the workspace", Page 596 ■ Area In the Favorites view you can add another area.
	Remove The control only shows this icon when you are adapting the layout. You can delete an empty column with this icon.

Settings in the Form workspace

In the **Tables** window, you can select whether the control will show the parameter descriptions. The selected setting is modally effective.



24.4.1 Adding a column in the workspace

To add a column:

- 
 - ▶ Select **Configure the layout**
 - > The control enables all functions for adapting the layout of the workspace.
- 
 - ▶ In the workspace, swipe to the left
 - ▶ Select **Add**
 - > The control adds a new column.
- 
 - ▶ Move the areas if required
- 
 - ▶ Select **Configure the layout**
 - > The control saves your changes.

Notes

- In the tool manager, the control displays the relevant parameters for the current tool type only and hides all other parameters. If you select the **Configure the layout** icon, the control will also display the hidden form areas.
- The control displays an icon of the selected tool type in the **Tool Icon** area.
Further information: User's Manual for Setup and Program Run
- Refer to your machine manual.
Using the optional machine parameter **freeInputEnabled** (no. 105706), the machine manufacturer specifies whether you can enter custom parameters in selection menus.

24.5 Importing and exporting of table contents

Application

You can import the contents of specific tables into the control and export them from the control. This avoids manual editing efforts and possible typing errors.

Importing tool data is particularly useful in connection with a tool presetter. Via exported contents you can, for example, transfer the tools and presets of the control to the CAM system.

The control uses CSV files for exporting and importing of table contents.

Related topics

- Overview of file types
Further information: "File types", Page 355
- Fundamentals of the **Tool management** application
Further information: User's Manual for Setup and Program Run

Description of function

Importing of data is possible with all tables that contain the **Import** button in the function bar.

To export contents, you have to mark all desired table rows. If a table does not permit exporting, the **Export** button is grayed out in the action bar.

The CSV file has the following structure:

- The first line contains the column names of the table being exported.
- The other lines contain the exported contents. The contents are arranged in the same sequence as the first line. Decimal numbers are separated by a point.

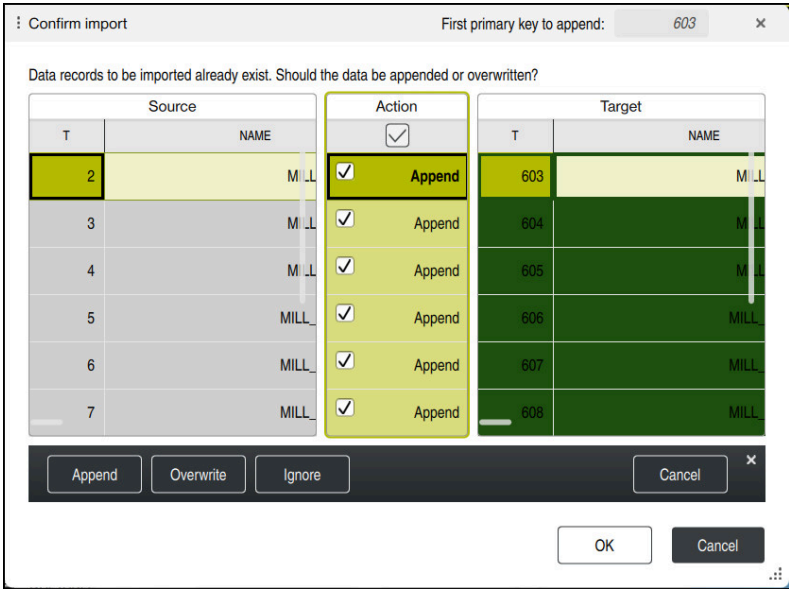
For compatibility with earlier controls, the control provides the following formatting options for exporting:

- **TNC7 (semicolon-separated)** writes the values between double quotation marks and separates the values by semicolons.
Most table calculation programs use this formatting.
- **iTNC 530 / TNC 640 (comma-separated)** writes the values partly between brackets and separates the values by commas

The control is able to import and export data in both formats.

The Confirm import window

When you select a CSV file for importing, the control will display the **Confirm import** window.



The **Confirm import** window with table rows to be appended

The window contains the following areas:

Area	Meaning															
Source	The control shows the values of the CSV file that are to be imported.															
Action	If you double-tap or double-click a row, the following actions for this row can be selected: ■ Append Append row at end of table ■ Overwrite Import row with line number of the CSV file ■ Ignore Do not import row of the CSV file Clicking the checkbox in the header selects all table rows. You can select the desired action for all rows that have been marked in the action bar.															
Target	The control shows the table values that will result from the currently selected action after importing. Depending on the selected action, the control shows the cells in the following colors: <table><tr><th>Color</th><th>Meaning</th><th>Action</th></tr><tr><td>Yellow</td><td>The content of the source differs from the content of the target table.</td><td>Ignore</td></tr><tr><td>Dark gray</td><td>The table will not contain a row with this number after importing.</td><td>Ignore</td></tr><tr><td>Dark green</td><td>The rows will be added in the target file.</td><td>Append</td></tr><tr><td>Red</td><td>The current table content will be overwritten after importing.</td><td>Overwrite</td></tr></table>	Color	Meaning	Action	Yellow	The content of the source differs from the content of the target table.	Ignore	Dark gray	The table will not contain a row with this number after importing.	Ignore	Dark green	The rows will be added in the target file.	Append	Red	The current table content will be overwritten after importing.	Overwrite
Color	Meaning	Action														
Yellow	The content of the source differs from the content of the target table.	Ignore														
Dark gray	The table will not contain a row with this number after importing.	Ignore														
Dark green	The rows will be added in the target file.	Append														
Red	The current table content will be overwritten after importing.	Overwrite														

The control displays an input field in the title bar of the window. If **Append** is selected, you can optionally enter a row number that will be the start number of the imported table rows. This allows you to define new tool numbers, for example.

24.5.1 Exporting table contents

To export table contents:



- ▶ Select the **Tables** operating mode

- ▶ Select the table (e.g., **Tool management**)

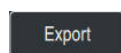
- ▶ Select the row to be exported



- ▶ Select **Mark row**

- > The control marks the selected line and opens the action bar.

- ▶ Select further rows, if required



- ▶ Select **Export**

- > The control opens the **Save as** window.

- ▶ Select a path

- ▶ Enter the file name

- ▶ Select the formatting



- ▶ Select **Save**

- > The control saves the CSV file using the defined path.

24.5.2 Importing table contents

To import table contents:



- ▶ Select the **Tables** operating mode



- ▶ Select the table (e.g., **Tool management**)
- ▶ Activate **Edit**
 - > The control enables the table for editing.



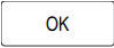
- ▶ Select **Import**
 - > The control opens a selection window.
- ▶ Select the desired CSV file



- ▶ Select **Import**
 - > The control opens the **Confirm import** window.



- ▶ Click the checkbox in the header of the **Action** area
 - > The control marks all rows to be imported and opens the action bar.
- ▶ Select an action:



- **Append**: the control adds the contents as new rows at the end of the table.
 - **Overwrite**: the control overwrites the initial contents with the contents from the CSV file.
 - **Ignore**: the control does not import the contents.
- ▶ Select **OK**
 - > The control imports the contents with the selected action.



If you double-tap or double-click each cell in the **Action** area, the following actions can be selected separately for each row.

NOTICE

Caution: Data may be lost!

When overwriting existing table values with the **Overwrite** function, the control will permanently delete the initial values!

- ▶ Use this function only with table files that are no longer needed

Notes

NOTICE

Caution: Possible material damage!

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

- ▶ Check whether the column names are correct
 - ▶ After importing, check the tool data and adapt them if required.
- The CSV file contains information on the table that the export comes from. Only tables of the same file type allow importing.
 - By default, the control saves the export under **TNC:\system\tooltab**.
 - To be able to import the CSV file, the column with the line numbers must be available.
 - In the **Presets** application, the **Confirm import** window does not include an input field in the title bar. Within the preset table, you can append rows only using the next line number.
 - If rows cannot be overwritten when importing CSV files, the control will display an information symbol. If you select the information symbol, the control displays a pop-up window showing the reason for the problem. The control provides a button with the information symbol. It allows you to filter the contents to find all rows with this problem.

24.6 Accessing table values

24.6.1 Fundamentals

The **TABDATA** functions allow you to access table values. These functions enable automated editing of compensation values from within the NC program, for example. You can access the following tables:

- Tool table ***.t** (read-only access)
- Compensation table ***.tco** (read and write access)
- Compensation table ***.wco** (read and write access)
- Preset table ***.pr** (read and write access)

In each case, the active table is accessed. Read-only access is always possible, whereas write access is possible only during program run. Write access during simulation or during a block scan has no effect.

The control provides the following functions for accessing the table values:

Syntax	Function	Further information
TABDATA READ	Read the value from a table cell	Page 603
TABDATA WRITE	Write a value to a table cell	Page 603
TABDATA ADD	Add a value to a table value	Page 605

If the unit of measure used in the NC program differs from that used in the table, the control converts the values from **millimeters** to **inches**, and vice versa.

Related topics

- Reading values from any desired tables with variable functions
Further information: "Reading table content with FN 28: TABREAD", Page 426
- Writing values to any desired tables with variable functions
Further information: "Writing to a table with FN 27: TABWRITE", Page 423
- Fundamentals regarding variables
Further information: "Basics", Page 393
- Tool table
Further information: User's Manual for Setup and Program Run

24.6.2 Reading table values with TABDATA READ

Application

The function **TABDATA READ** allows you to read a value from a table and save it to a variable.

For example, the **TABDATA READ** function enables you to pre-check the data of the tool to be used to prevent error messages from occurring during program run.

Description of function

Depending on the type of column you want to transfer, you can use numerical or string parameters to save the value. The control automatically converts the table values to the unit of measure used in the NC program.

Input

**11 TABDATA READ Q1 = CORR-TCS
COLUMN "DR" KEY "5"**

; Save the value in row 5, column **DR**, from the compensation table to **Q1**

To navigate to this function:

Insert NC function ► All functions ► FN ► Special functions ► Functions ► TABDATATable access TABDATA ► TABDATA READ

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
READ	Read a table value
Parameter	Variable in which the control stores the value
TOOL, CORR-TCS, CORR-WPL or PRESET	Read the value from the tool table or a compensation table *.tco or *.wco or from the preset table
COLUMN	Column name Text or string parameter
KEY	Row number Number, text, or variable

24.6.3 Writing table values with TABDATA WRITE

Application

Use the function **TABDATA WRITE** to write a value into a table.

You can use the **TABDATA WRITE** function after a touch probe cycle to enter a necessary tool compensation into the compensation table, for example.

Description of function

Depending on the type of column you want to write to, you can use **Q, QL, QR, QS** or named parameters as transfer parameters. Alternatively, you can define the value directly in the NC function **TABDATA WRITE**.

Input

11 TABDATA WRITE CORR-TCS COLUMN "DR" KEY "3" = Q1	; Write the value from Q1 to row 3, column DR , of the compensation table

To navigate to this function:

Insert NC function ▶ All functions ▶ FN ▶ Special functions ▶ Functions ▶ Table access TABDATA ▶ TABDATA WRITE

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
WRITE	Write a table value
CORR-TCS, CORR-WPL or PRESET	Write a value to a compensation table *.tco or *.wco or to the preset table
COLUMN	Column name Text or string parameter
KEY	Row number Number, text, or variable
= or SET UNDEFINED	Write the table value or assign the status undefined
Number, Name or Parameter	Table value Number, text, or variable Only if = has been selected

Notes

NOTICE

Caution: Significant property damage!

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

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

NOTICE

Caution: Data may be lost!

If you use the **TABDATA WRITE** or **FN 27** functions to write to a table cell that contains existing values, the control overwrites these values without confirmation prompting.

- ▶ Write only to empty or no longer needed table cells
- ▶ Create a backup copy of the table

24.6.4 Adding table values with TABDATA ADD

Application

Use the **TABDATA ADD** function to add a value to an existing table value.

You can use the **TABDATA ADD** function to update a tool compensation value after a measurement has been repeated, for example.

Description of function

You can define the value directly within the **TABDATA ADD** NC function or use numerical parameters as transfer parameters.

In order to write into a compensation table, you need to activate the table.

Input

**11 TABDATA ADD CORR-TCS COLUMN
"DR" KEY "3" = Q1**

; Add the value from **Q1** to row 3, column **DR**, of the compensation table

To navigate to this function:

Insert NC function ► **All functions** ► **FN** ► **Special functions** ► **Functions** ► **Table access** ► **TABDATA** ► **TABDATA ADD**

The NC function includes the following syntax elements:

Syntax element	Meaning
TABDATA	Syntax initiator for accessing table values
ADD	Add a value to a table value
CORR-TCS, CORR-WPL or PRESET	Write a value to a compensation table *.tco or *.wco or to the preset table
COLUMN	Column name Text or string parameter
KEY	Row number Number, text, or variable
Number	Value to be added Number or numerical parameter

24.7 Freely definable tables *.tab

Application

In freely definable tables, you can save any information (e.g., measurement results). You can choose from various templates and adapt the table characteristics for your application.

The NC functions **FN 26** to **FN 28** allow you to write information to the table or read information from the table from within the NC program.

Related topics

- Variable functions **FN 26** to **FN 28**
Further information: "NC functions for accessing tables with variables", Page 423

Description of function

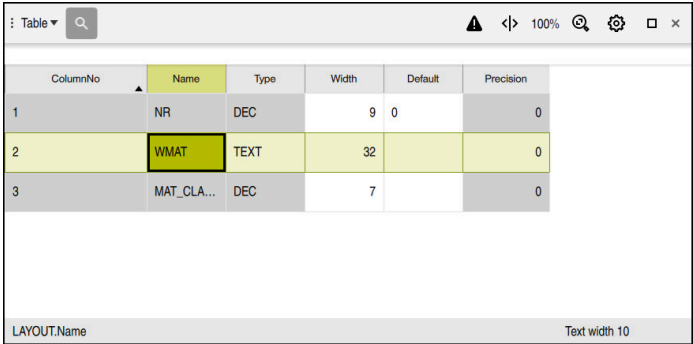
When you create a freely definable table, the control will provide various table templates for selection.

The machine manufacturers can create their own table templates and store them in the control.

After you have created a freely definable table, you can modify its properties. you modify the table properties in the **LAYOUT** application.

Further information: "Changing table properties of freely definable tables", Page 608

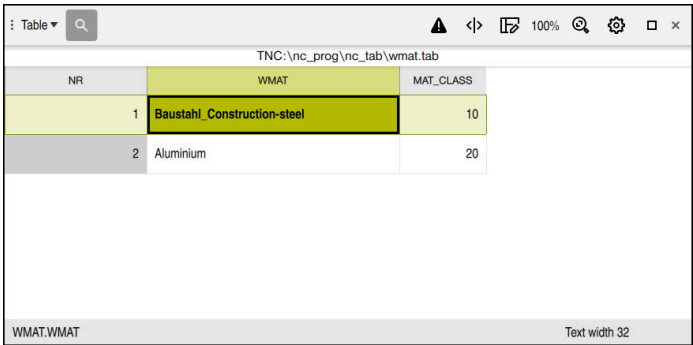
In the **LAYOUT** application, the control shows the columns of the table row by row.



ColumnNo	Name	Type	Width	Default	Precision
1	NR	DEC	9	0	0
2	WMAT	TEXT	32		0
3	MAT_CLA...	DEC	7		0

LAYOUT.Name Text width 10

Freely definable table in the **LAYOUT** application



NR	WMAT	MAT_CLASS
1	Baustahl_Construction-steel	10
2	Aluminium	20

WMAT.WMAT Text width 32

Freely definable table in the **Table** workspace

Properties of a table column

When you change any table properties, each column has the following properties:

Column	Meaning
Name	Name of the column
Width	Maximum number of characters in the column
Default	Default value of each new row Optional input
Type	The control offers the following possible selections in the Type column: ■ TEXT: Text entry ■ SIGN: Algebraic sign + or – ■ BIN: Binary number ■ DEC: Positive integer ■ HEX: Hexadecimal number ■ INT: Integer ■ LENGTH: Floating-point number (mm or inch) i If you write values from an inch program to a freely definable table, the control converts the values. i If the unit of measure is inches, then the column has one more decimal place than you define. ■ FEED: Feed rate (mm/min or 0.1 ipm) ■ IFEED: Feed rate (mm/min or ipm) i If the unit of measure is inches, then the column has one more decimal place than you define. ■ FLOAT: Floating-point number ■ BOOL: Logical value ■ INDEX: Index ■ TSTAMP: Time and date with the format HH:MM:SS DD.MM.YYYY ■ UPTXT: Text entry in capital letters ■ PATHNAME: Path name i In the columns with the data types BIN, DEC or HEX you can enter the values as binary numbers, positive integers or hexadecimal numbers. The control converts the entered values into the column's respective data type.
Precision	Maximum number of decimal places

24.7.1 Changing table properties of freely definable tables

To insert a new column:

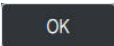
- ▶ Open an empty freely definable table



- ▶ Select **Edit table characteristics**
 - > The control opens the **LAYOUT** application.
- ▶ Activate **Edit**
- ▶ Select **Insert rows**
 - > The control opens the **Insert rows** window.
- ▶ Enter **Column name**
- ▶ Select **Column type**
 - > The control displays a selection menu.



You cannot change the column name and type afterwards.



- ▶ Select the desired column type
 - Further information:** "Properties of a table column", Page 607
- ▶ Select **OK**
 - > The control inserts a new table row at the end of the table.
- ▶ In the **Width** column, define the maximum number of characters of the table column (e.g., **12**).
- ▶ In the **Default** column, define a value if required.
- ▶ In the **Precision** column, define the number of decimal places (e.g., **3**).
- ▶ Select **Save changes**
 - > The control opens the **Save layout changes** window.
- ▶ Select **OK**
 - > The control closes the **LAYOUT** application.

Notes

- The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**). Due to SQL commands, these characters can cause problems when data are input or read.
 - Further information:** "Table access with SQL statements", Page 451
- Column names are not case-sensitive. If the only difference in column names is their case, the control will not open the table, but display an error message.
- The sequence of columns in the **Table** workspace is independent of the sequence of rows in the **LAYOUT** application. You can edit the sequence of columns in the **Table** workspace.
 - Further information:** "Settings in the Table workspace", Page 591

24.8 Point table *.pnt

Application

In a point table, you save randomly distributed points on a workpiece. The control calls a cycle at each point. You can hide individual points and define a clearance height.

Related topics

- Calling point tables, effect with different cycles

Further information: "Point tables", Page 206

Description of function

Parameters in point tables

The point table provides the following parameters:

Parameter	Meaning
NR	Row number in the point table
X	X coordinate of a point
Y	Y coordinate of a point
Z	Z coordinate of a point
FADE	Hide? (yes=ENT/no=NO ENT) Y=Yes: The point is hidden during machining. Points that have been hidden will remain hidden until they are manually shown again. N=No: The point is shown for machining. All points of a point table are shown for machining by default.
CLEARANCE	Clearance height? Safe position in the tool axis to which the control retracts the tool after machining a point. If you do not define a value in the CLEARANCE column, the control will use the value of the cycle parameter Q204 2ND SET-UP CLEARANCE . If you have defined values in both the CLEARANCE column and the Q204 parameter, the control will use the higher of the two values.

24.8.1 Hiding individual points during machining

In the **FADE** column of the point table, you can specify if the defined point will be hidden during the machining process.

To hide points:

- ▶ Select the desired point in the table
- ▶ Select the **FADE** column



- ▶ Activate **Edit**

- ▶ Enter **Y**

- ▶ The control hides the point at the cycle call.

If you enter **Y** in the **FADE** column, you can use the **Skip block** toggle switch to skip this point in **Program Run** operating mode.

Further information: User's Manual for Setup and Program Run

24.9 Datum table *.d

Application

A datum table saves positions on the workpiece. To use a datum table, you must activate it. The datums can be called from within an NC program, for example in order to execute machining processes on several workpieces at the same position. The active row of the datum table serves as the workpiece datum in the NC program.

Related topics

- Contents and creation of a datum table
Further information: "Datum table *.d", Page 610
- Editing a datum table during a program run
Further information: User's Manual for Setup and Program Run
- Preset table
Further information: User's Manual for Setup and Program Run

Description of function

The values of columns **X**, **Y** and **Z** have the effect of a shift in the workpiece coordinate system **W-CS**.

Parameters in datum tables

A datum table provides the following parameters:

Parameter	Meaning
D	Row number in the datum table
X	X coordinate of the datum Transformation relating to the workpiece coordinate system W-CS Further information: "Workpiece coordinate system W-CS", Page 312
Y	Y coordinate of the datum Transformation relating to the workpiece coordinate system W-CS Further information: "Workpiece coordinate system W-CS", Page 312
Z	Z coordinate of the datum Transformation relating to the workpiece coordinate system W-CS Further information: "Workpiece coordinate system W-CS", Page 312
DOC	Comment on shift?

24.9.1 Editing a datum table

You can edit the active datum table during program run.

Further information: User's Manual for Setup and Program Run

To edit a datum table:



- ▶ Activate **Edit**
- ▶ Select the value
- ▶ Edit the value
- ▶ Save the edited value, for example by selecting a different row

NOTICE

Danger of collision!

The control does not consider the changes made to a datum table or compensation table until the values have been saved. You need to activate the datum or compensation value in the NC program again; otherwise, the control will continue using the previous values.

- ▶ Make sure to confirm any changes made to the table immediately (e.g., by pressing the **ENT** key)
- ▶ Activate the datum or compensation value in the NC program again
- ▶ Carefully prove-out the NC program after changing the table values

24.10 Tables for cutting data calculation

Application

The following tables allow you to calculate the cutting data of a tool in the cutting data calculator:

- Table for workpiece materials **WMAT.tab**
Further information: "Table for workpiece materials WMAT.tab", Page 612
- Table for tool materials **TMAT.tab**
Further information: "Table for tool materials TMAT.tab", Page 612
- Cutting data table ***.cut**
Further information: "Cutting data table *.cut", Page 613
- Diameter-dependent cutting data table ***.cutd**
Further information: "Diameter-dependent cutting data table *.cutd", Page 614

Related topics

- Cutting data calculator
Further information: "Cutting data calculator", Page 518
- Tool management
Further information: User's Manual for Setup and Program Run

Description of function

Table for workpiece materials WMAT.tab

In the table for workpiece materials **WMAT.tab**, you define the workpiece material. You must save this table in the **TNC:\table** folder.

The table for workpiece materials **WMAT.tab** provides the following parameters:

Parameter	Meaning
NR	Row number in the table for workpiece material
WMAT	Workpiece material (e.g., aluminum)
MAT_CLASS	Material class Categorize the materials into material classes with the same cutting conditions (e.g., in accordance with DIN EN 10027-2).

Table for tool materials TMAT.tab

In the table for tool materials **TMAT.tab**, you define the tool material. You must save this table in the **TNC:\table** folder.

The table for tool materials **TMAT.tab** provides the following parameters:

Parameter	Meaning
NR	Row number in the table for tool cutting materials
TMAT	Tool material (e.g., solid carbide)
ALIAS1	Additional designation
ALIAS2	Additional designation

Cutting data table *.cut

In the cutting data table ***.cut**, you assign the matching cutting data to the workpiece materials and the tool materials. You must save the table in the **TNC: \system\Cutting-Data** folder.

The cutting data table ***.cut** provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows
MAT_CLASS	Workpiece material from the WMAT.tab table Further information: "Table for workpiece materials WMAT.tab", Page 612 Selection by means of a selection window
MODE	Machining mode (e.g., roughing or finishing)
TMAT	Tool material from the table TMAT.tab Further information: "Table for tool materials TMAT.tab", Page 612 Selection by means of a selection window
VC	Cutting speed in m/min Further information: "Cutting data", Page 166
FTYPE	Type of feed: ■ FU : Feed per revolution FU in mm/rev ■ FZ : Feed per tooth FZ in mm/tooth Further information: "Feed rate F", Page 167 Selection using a selection menu
F	Feed rate value

Diameter-dependent cutting data table *.cutd

In the diameter-dependent cutting data table ***.cutd**, you assign the matching cutting data to the workpiece materials and the tool materials. You must save the table in the **TNC:\system\Cutting-Data** folder.

The diameter-dependent cutting data table ***.cutd** provides the following parameters:

Parameter	Meaning
NR	Sequential number of the table rows
MAT_CLASS	Workpiece material from the WMAT.tab table Further information: "Table for workpiece materials WMAT.tab", Page 612 Selection by means of a selection window
MODE	Machining mode (e.g., roughing or finishing)
TMAT	Tool material from the table TMAT.tab Further information: "Table for tool materials TMAT.tab", Page 612 Selection by means of a selection window
VC	Cutting speed in m/min Further information: "Cutting data", Page 166
FTYPE	Type of feed: ■ FU: Feed per revolution FU in mm/rev ■ FZ: Feed per tooth FZ in mm/tooth Further information: "Feed rate F", Page 1671634175559097 Selection using a selection menu
F_D_0...F_D_9999	Feed rate value for the respective diameter You don't need to define all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate.

Note

In the corresponding folders, the control provides sample tables for automatic cutting data calculation. You can customize these tables and specify your own data, i.e. materials and tools to be used.

25

Overviews

25.1 Special functions defining the machine behavior

With code number 555343, you can enable NC functions that are intended for HEIDENHAIN, the machine manufacturer, and third-party providers only.

The following NC functions influence the machine behavior:

- Kinematics functions:
 - **WRITE KINEMATICS**
 - **READ KINEMATICS**
- PLC functions:
 - **FUNCTION SCOPE**
 - **START**
 - **STORE**
 - **STOP**
 - **READ FROM PLC**
 - **WRITE TO PLC**
 - **WRITE CFG**
 - **PREPARE**
 - **COMMIT TO DISK**
 - **COMMIT TO MEMORY**
 - **DISCARD PREPARATION**
- Variable programming:
 - **FN 19: PLC**
 - **FN 20: WAIT FOR**
 - **FN 29: PLC**
 - **FN 37: EXPORT**
- **CYCL QUERY**

NOTICE

Caution: Significant property damage!

The use of special functions for machine behavior might result in undesired behavior and severe errors (e.g., the control might not be operable any longer). With these NC functions, HEIDENHAIN, the machine manufacturer, and third-party providers have the possibility of modifying the machine behavior under program control. It is not recommended that machine operators or NC programmers use this function. There is a danger of collision during the execution of these NC functions and during the subsequent machining operations!

- ▶ Only use special functions for machine behavior after checking with HEIDENHAIN, the machine manufacturer, or the third-party provider
- ▶ Comply with the documentation from HEIDENHAIN, the machine manufacturer, and third-party providers

Index

A

About the product.....	43
About the User's Manual.....	31
Adding table values.....	605
Additional documentation.....	33
Advanced checks.....	541
Angle encoder.....	107
Application	
Help.....	38
Start/Login.....	59
Auto-completion.....	136
Axis designation.....	106

B

Basic coordinate system.....	311
B-CS.....	311
Blank form.....	152
Block.....	114
Hiding.....	499
Skipping.....	499

C

Calculator.....	516
Calibration	
Tool touch probe.....	554
Calibration of tool touch probe	
Calibration of IR TT.....	557
Calibration of TT.....	555
Cartesian coordinates.....	172
Cartesian coordinate system.....	309
Centering.....	266
CFGREAD.....	438
Circle calculation.....	408
Column Search.....	504
Comment, adding.....	498
Comparison.....	509
Contact.....	42
Context menu.....	511
Context-sensitive help.....	40
Control's user interface.....	57
Control structures	
Case analysis.....	185
ELSE.....	185
ELSE IF.....	185
FOR loop.....	188
IF.....	185
Loops.....	188
Loops, aborting with BREAK..	192
Loops, interrupting.....	192
Loops, skipping with CONTINUE..	193
Program branching.....	185
WHILE loop.....	190
Coordinate definition	
Absolute.....	173
Cartesian.....	172

Incremental.....	174
Coordinate system.....	308
Basics.....	309
Coordinate origin.....	309
Coordinate transformation.....	331
Cycle Axis-specific scaling	
factor.....	326
Datum shift.....	332
Mirroring.....	334
Mirroring cycle.....	323
Reset.....	337
Scaling.....	335
Scaling cycle.....	324
Counter.....	443
Countersinking	
Back boring.....	262
CSV file.....	597
Cutting data.....	166
Cutting data calculator.....	518
Cutting data tables.....	521
Table.....	612
Cutting data table.....	613
Applying.....	521

D

Datum shift.....	332
Datum table.....	320, 610
Columns.....	610
Selecting.....	321
Delta length.....	341
Delta radius.....	342
Delta value.....	340
Diameter-dependent cutting data	
table.....	614
Display unit.....	53
Drilling	
Drilling.....	232
Reaming.....	236, 238
Single-lip deep hole drilling....	254
Universal deep-hole drilling....	247
Universal drilling.....	242
Drilling, centering and thread cycles	
Countersinking and centering....	262
Drilling.....	232
Tapping.....	269
Dwell time.....	379
Cyclic.....	377
Once.....	376

E

Encoder.....	107
Error message	
Entering individually.....	522
Output.....	412
Export version.....	49

F

Feed rate.....	167
File.....	347
Adapting iTNC 530 files.....	363
Characters.....	353
Edit.....	366
Importing iTNC 530 files.....	363
Managing with FUNCTION	
FILE.....	371
Opening with OPEN FILE.....	370
File extension.....	355
File format.....	355
File function.....	351
In NC program.....	370
File management.....	348
Finding.....	351
User-defined filter.....	357
File name.....	353
File path.....	354
Absolute.....	354
Relative.....	354
File type.....	355
Filter	
User-defined.....	357
First steps.....	79
Programming.....	83
FN 16.....	413
Contents and formatting.....	413
Output format.....	413
FN 18.....	420
FN 26.....	423
FN 27.....	423
FN 28.....	426
FN 38.....	422
Form.....	132
For tables.....	594
Freely definable table.....	606
Reading.....	426
Freely definable tables	
Opening.....	423
Writing to.....	423
FUNCTION PRINT.....	524
Function STOP.....	382
Fundamentals	
Programming.....	114

G

Gestures.....	65
GLOBAL DEF.....	445
GOTO.....	496
Graphics.....	527
Grinding mode.....	150
GUI.....	57

H

Hardware.....	53
Help graphic.....	120
Hiding NC blocks.....	499

I		
Icons, miscellaneous.....	73	
I-CS.....	313	
If-Then decision.....	410	
Incremental entries.....	174	
Input		
Absolute.....	173	
Input coordinate system.....	313	
Inserting NC function window....	133	
Integrated product aid		
TNCguide.....	37	
ISO.....	469, 469	
Keys.....	472	
Issuing text		
FUNCTION PRINT.....	524	
iTNC 530		
Adapting files.....	363	
Importing tool table.....	363	
J		
Jumping with GOTO.....	496	
K		
Keyboard.....	53	
Formula.....	495	
NC functions.....	494	
Text.....	495	
Virtual.....	493	
Keys.....	65	
ISO.....	472	
Klartext programming.....	114	
L		
Label.....	178	
Calling.....	179	
Defining.....	178	
Length compensation.....	341	
Licensing terms.....	52	
Linear encoder.....	107	
Loops for program section		
repeats.....	188	
M		
M92 datum M92-ZP.....	108	
Machine coordinate system.....	310	
Machine datum.....	108	
Machine parameter		
Reading out.....	438	
Machining feed rate.....	167	
Main menu.....	76	
Marking.....	511	
M-CS.....	310	
Measuring in the simulation.....	542	
M function.....	381	
For coordinate entries.....	385	
For path behavior.....	388	
Overview.....	383	
Milling cycles		
Milling planes.....	297	
Milling pockets.....	279	
Milling studs.....	289	
Milling mode.....	150	
Milling planes		
Extended face milling.....	297	
Milling pockets		
Rectangular pocket.....	279	
Milling slots		
Slot milling.....	284	
Milling studs		
Rectangular stud.....	289	
Mirroring		
NC function.....	334	
Miscellaneous function.....	381	
For coordinate entries.....	385	
For path behavior.....	388	
Overview.....	383	
Miscellaneous functions		
Fundamentals.....	382	
Model comparison.....	547	
N		
NC block.....	114	
Hiding.....	499	
Skipping.....	499	
NC function		
Changing.....	109	
Editing.....	111	
Inserting.....	109, 133	
NC fundamentals.....	106	
NC program.....	114	
Appearance.....	120	
Call.....	194	
Editing.....	109	
Form.....	132	
Help graphic.....	120	
Search.....	504	
Selecting.....	196	
Settings.....	120	
Structure.....	501	
Structure, creating.....	500	
Using.....	124	
NC sequence.....	201	
NC syntax.....	114	
Nesting.....	182	
New table, creating.....	584	
Notes, types of.....	34	
Notifications		
Issuing.....	522	
O		
Open file.....	359	
Operating elements.....	65	
Operating mode.....	150	
Editor.....	116	
Files.....	348	
Machine.....	59	
Manual.....	59	
Overview.....	59	
Start.....	59	
Tables.....	580	
Operational cycle parameter.....	138	
Orthogonal coordinates.....	172	
P		
Part family.....	405	
Path.....	354	
Absolute.....	354	
Relative.....	354	
Pattern cycles		
Circle.....	220	
Lines.....	223	
PATTERN DEF		
Call.....	209	
Programming.....	209	
Pattern definition		
Cycles.....	219	
PATTERN DEF.....	208	
Point table.....	206	
Pattern definition with PATTERN		
DEF		
frames.....	213	
full circle.....	215	
patterns.....	212	
pitch circle.....	216	
Point.....	210	
Place of operation.....	46	
Point table		
Columns.....	609	
Cycle call.....	207	
Hiding a point.....	609	
Selecting.....	207	
Position encoder.....	107	
Preset		
Activating in NC program.....	315	
Copying in NC program.....	317	
Correcting in NC program.....	318	
Program.....	114	
Appearance.....	120	
Editing.....	109	
Editor.....	118	
Form.....	132	
Help graphic.....	120	
Q parameters.....	393	
Search.....	504	
Settings.....	120	
Structure.....	501	
Structure, creating.....	500	
Text mode.....	135	
Using.....	124	
Program call.....	194	
Cycle PGM CALL.....	199	
Program comparison.....	509	
Program editor		

Text mode.....	135	Search and replace.....	506	System datum, reading.....	420
Program examples		Syntax.....	127	T	
Pattern cycles.....	226	Selected program, calling.....	196	TABDATA.....	602
PATTERN DEF.....	217	Select function		Table	
Programmed dwell time.....	376	Datum table.....	321	Access from within the NC	
Programming example		Selection function.....	194	program.....	602
Milling a pocket and a stud....	295	File.....	370	Access with variables.....	423
Programming examples		NC program.....	196	Adapting.....	363
Coordinate transformation....	329	NC program as cycle.....	144	Creating.....	584
Programming fundamentals.....	114	NC program call.....	194	Cutting data calculation.....	612
Programming possibilities.....	109	Overview.....	194	Datum table.....	610
Programming technique.....	177	SEL PATTERN.....	207	Exporting and importing.....	597
Programming with variables.....	391	Sequence.....	201	Filter.....	588
Program section repeat.....	181	Setting a preset.....	327	Point table.....	609
Program template.....	201	Show file.....	361	SQL access.....	451
Proper and intended operation....	44	Simulation.....	527	Workspace.....	586
Q		Center of rotation.....	549	Table contents, exporting.....	597
Q Info.....	399	Collision test.....	541	Table contents, importing.....	597
Q parameter		Cutout view.....	544	Table values, writing.....	603
String formula.....	432	Measuring.....	542	Tapping	
Q parameter list.....	399	Model comparison.....	547	With floating tap holder.....	269
Q parameters.....	393	Rewind.....	545	Without floating tap holder....	272
Basic calculation method.....	404	Settings.....	528	Target group.....	32
Basics.....	393	Speed.....	550	TCP.....	160
Circle calculation.....	408	STL file, creating.....	539	T-CS.....	314
Formula.....	427	Tool representation.....	537	Template.....	201
Jump.....	410	Skipping NC blocks.....	499	Tests column.....	128
Overview.....	392	Software number.....	49	Simulation report.....	130
Preassigned.....	402	Software option.....	50	Tool check.....	131
System datum, reading.....	420	Speed.....	166	Tool usage.....	130
Text output.....	413	Speed of the simulation.....	550	Text editor.....	366
Trigonometric function.....	407	Spindle orientation.....	380	Representation.....	368
Quick selection.....	360	Spindle speed.....	166	Search.....	504
Editor.....	360	SQL.....	451	Settings.....	368
Tables.....	360	BIND.....	454	Text mode.....	135
R		COMMIT.....	464	Auto-completion.....	136
Radius compensation.....	342	EXECUTE.....	457	Text output	
Reading table values.....	603	FETCH.....	461	FN 16.....	413
Recurring dwell time.....	377	INSERT.....	467	TIP.....	160
Reference point.....	108	Overview.....	452	TLP.....	161
Reference system.....	308	ROLLBACK.....	462	TMAT.....	612
Basic coordinate system.....	311	SELECT.....	455	TNCguide.....	38
Input coordinate system.....	313	UPDATE.....	465	Tool.....	157
Machine coordinate system..	310	STL file as workpiece blank.....	155	Delta value.....	340
Tool coordinate system.....	314	STOP.....	382	Exporting and importing.....	597
Workpiece coordinate system....	312	String formula.....	432	Length compensation.....	341
Right-click.....	511	String parameter.....	432	Overview.....	158
RL/RR/RO.....	344	Structure.....	501	Preset.....	159
S		Creating.....	500	Radius compensation....	342, 344
Safety precaution.....	47	Structure item.....	500	Tool call.....	162
Content.....	34	Subprogram.....	180	Tool change.....	162
Scaling.....	335	Swipe menu.....	351	Tool carrier reference point.....	159
Search.....	504	Syntax.....	114	Tool center point TCP.....	160
Regular expressions.....	507	Syntax element.....	114	Tool change position.....	108
		Syntax emphasis		Tool compensation.....	340
		Text editor.....	368	Tool coordinate system.....	314
		Syntax highlighting.....	120	TOOL DEF.....	169
		Syntax search.....	127		

Tool location point TLP.....	161	Virtual keyboard.....	493
Tool material.....	612		
Tool measurement		W	
Complete measurement.....	575	W-CS.....	312
Fundamentals.....	563	WMAT.....	612
Machine parameters.....	564	Working plane.....	106
Tool length.....	568	Workpiece blank.....	152
Tool radius.....	571	Cuboid.....	153
Tool table.....	566	Cylinder.....	154
Tool pre-selection.....	169	Pipe.....	154
Tool radius compensation.....	344	STL file.....	155
Tool table		Workpiece blank definition.....	152
iTNC 530.....	363	Workpiece coordinate system...	312
Tool tip TIP.....	160	Workpiece counter.....	443
Touch gestures.....	65	Workpiece datum.....	108
Touch probe cycles for tool		Workpiece material.....	612
Measurement of milling		Workpiece preset.....	108
cutter.....	568	Activating in NC program.....	315
Touchscreen.....	53	Copying in NC program.....	317
Transformation.....	331	Correcting in NC program.....	318
Datum shift.....	332	Managing.....	315
Mirroring.....	334	Workspace	
Reset.....	337	Document.....	361
Scaling.....	335	Form for tables.....	594
Traverse movement		Help.....	490
Fundamentals.....	175	Keyboard.....	493
Traverse range, switching.....	150	Main menu.....	76
Trigonometry.....	407	Open file.....	359
Turning mode.....	150	Overview.....	62
		Program.....	118
U		Quick selection.....	360
USB device.....	364	Quick selection in the Editor	
Removing.....	365	operating mode.....	360
User Aids.....	489	Quick selection in the Tables	
User-defined filter.....	357	operating mode.....	360
User interface of the control.....	57	Simulation.....	527
User's Manuals.....	33	Table in the Tables operating	
		mode.....	586
V		Text editor.....	366
Variable.....	391		
Basic calculation method.....	404		
Circle calculation.....	408		
Counter.....	443		
Format strings.....	439		
Formula.....	427		
Information, sending.....	422		
Overview.....	392		
Preassigned.....	402		
SQL statement.....	451		
String formula.....	432		
String parameter QS.....	432		
System datum, reading.....	420		
Text output.....	413		
Trigonometric function.....	407		
Variables			
Basics.....	393		
Checking.....	399		
Jump.....	410		

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany

☎ +49 8669 31-0

FAX +49 8669 32-5061

info@heidenhain.de

Technical support FAX +49 8669 32-1000

Measuring systems ☎ +49 8669 31-3104
service.ms-support@heidenhain.de

NC support ☎ +49 8669 31-3101
service.nc-support@heidenhain.de

NC programming ☎ +49 8669 31-3103
service.nc-pgm@heidenhain.de

PLC programming ☎ +49 8669 31-3102
service.plc@heidenhain.de

APP programming ☎ +49 8669 31-3106
service.app@heidenhain.de

www.heidenhain.com

www.klartext-portal.com

The Information Site for
HEIDENHAIN Controls

Klartext App

Klartext on your
mobile device

Google
Play Store

Apple
App Store



Touch probes and vision systems

HEIDENHAIN provides universal, high-precision touch probe systems for machine tools, for example for the exact determination of workpiece edge positions and for tool measurement. Proven technology, such as a wear-free optical sensor, collision protection, or integrated blower/flusher jets for cleaning the measuring point ensure the reliability and safety of the touch probes when measuring workpieces and tools. For even higher process reliability, the tools can be monitored conveniently with the vision systems and tool-breakage sensor from HEIDENHAIN.



For more details on touch probes and vision systems:

www.heidenhain.com/products/touch-probes-and-vision-systems

