HEIDENHAIN

TNC 320

User's Manual
ISO Programming

NC Software
77185x-18

English (en)
10/2023

Controls and displays

Keys

Keys on the screen

Key	Function
	Select screen layout
	Toggle the display between machine operating mode, programming mode, and a third desktop
	Soft keys for selecting functions on screen
	Switch the soft-key rows

Machine operating modes

Key	Function
	Manual operation
	Electronic handwheel
	Positioning with Manual Data Input
	Program Run, Single Block
	Program Run, Full Sequence

Programming modes

Key	Function
	Programming
	Test Run

Entering and editing coordinate axes and numbers

Key	Function
X ... V	Select the coordinate axes or enter them in the NC program
0 ... 9	Numbers
. +/-	Decimal separator / Reverse algebraic sign
P I	Polar coordinate entry / Incremental values
Q	Q parameter programming / Q parameter status
	Capture actual position
	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END	Conclude the NC block, end your input
CE	Clear entries or error message
DEL	Abort dialog, delete program section

Tool functions

Key	Function
TOOL DEF	Define tool data in the NC program
TOOL CALL	Call tool data

Managing NC programs and files, control functions

Key	Function
PGM MGT	Select or delete NC programs or files, external data transfer
PGM CALL	Define program call, select datum and point tables
MOD	Select MOD functions
HELP	Display help text for NC error messages, call TNCguide
ERR	Display all current error messages
CALC	Show calculator
SPEC FCT	Show special functions
	Currently not assigned

Navigation keys

Key	Function
	Position the cursor
GOTO	Go directly to NC blocks, cycles, and parameter functions
HOME	Navigate to the beginning of a program or table
END	Navigate to the end of the program or table row
PG UP	Navigate up one page
PG DN	Navigate down one page
	Select the next tab in forms
	Up/down one dialog box or button

Cycles, subprograms, and program-section repeats

Programming path contours

Key	Function
APPR DEP	Contour approach and departure
FK	FK free contour programming
L	Straight line
CC	Circle center/pole for polar coordinates
C	Circular arc with center
CR	Circular arc with radius
CT	Circular arc with tangential transition
CHF RND	Chamfer/rounding arc

Potentiometer for feed rate and spindle speed

Feed rate	Spindle speed

Table of contents

1 Basic Information 25
2 First Steps 61
3 Fundamentals 79
4 Tools 133
5 Programming Contours 149
6 Programming Aids 201
7 Miscellaneous Functions 235
8 Subprograms and Program Section Repeats 255
9 Programming Q Parameters 277
10 Special Functions 351
11 Multiple-axis Machining 403
12 Data Transfer from CAD Files 445
13 Tables and Overviews 471

1 Basic Information 25

1.1 About this manual 26

1.2 Control model, software and features 28

Software options 30
New and modified functions with 77185x-18 31
New cycle functions of software 77185x-18 52
Modified cycle functions with 77185x-18 56

2 First Steps 61

2.1 Overview 62

2.2 Switching on the machine 63

Acknowledging the power interruption 63

2.3 Programming the first part 64

Selecting the operating mode 64
Important controls and displays 64
Creating a new NC program / file management 65
Defining a workpiece blank 66
Program layout 67
Programming a simple contour 68
Creating a cycle program 74

3 Fundamentals ... 79

3.1 The TNC 320 ... 80

HEIDENHAIN Klartext and ISO ... 80
Compatibility ... 80

3.2 Visual display unit and operating panel ... 81

Display screen ... 81
Setting the screen layout ... 82
Operating panel ... 82

3.3 Modes of operation ... 84

Manual Operation and El. Handwheel ... 84
Positioning with Manual Data Input ... 84
Programming ... 85
Test Run ... 85
Program Run, Full Sequence and Program Run, Single Block ... 86

3.4 NC fundamentals ... 87

Position encoders and reference marks ... 87
Programmable axes ... 87
Reference systems ... 88
Designation of the axes on milling machines ... 96
Polar coordinates ... 97
Absolute and incremental workpiece positions ... 98
Selecting the preset ... 99

3.5 Creating and entering NC programs ... 100

Structure of an NC program in ISO format ... 100
Defining the workpiece blank: G30/G31 ... 101
Creating a new NC program ... 106
Programming tool movements in ISO ... 107
Actual position capture ... 109
Editing an NC program ... 110
The control's search function ... 114

3.6 File management ... 116

Files ... 116
Displaying externally generated files on the control ... 118
Directories ... 118
Paths ... 118
Overview: Functions of the file manager ... 119
Calling the File Manager ... 120
Selecting drives, directories and files ... 121
Creating a new directory ... 123
Creating new file ... 123
Copying a single file ... 123

Copying files into another directory ... 124
Copying a table ... 125
Copying a directory ... 126
Choosing one of the last files selected ... 126
Deleting a file ... 127
Deleting a directory ... 127
Tagging files ... 128
Renaming a file ... 129
Sorting files ... 129
Additional functions ... 130

4 Tools 133

4.1 Entering tool-related data 134

Feed rate F 134
Spindle speed S 135

4.2 Tool data 136

Requirements for tool compensation 136
Tool number, tool name 136
Tool length L 137
Tool radius R 139
Delta values for lengths and radii 139
Entering tool data into the NC program 139
Calling the tool data 140
Tool change 142

4.3 Tool compensation 145

Introduction 145
Tool length compensation 145
Tool radius compensation 146

5 Programming Contours 149

5.1 Tool movements 150

Path functions 150
FK free contour programming 150
Miscellaneous functions M 150
Subprograms and program section repeats 151
Programming with Q parameters 151

5.2 Fundamentals of path functions 152

Programming tool movements for machining 152

5.3 Approaching and departing a contour 155

Starting point and end point 155
Tangential approach and departure 157
Overview: Types of paths for contour approach and departure 158
Important positions for approach and departure 159
Approaching on a straight line with tangential connection: APPR LT 161
Approaching on a straight line perpendicular to the first contour point: APPR LN 161
Approaching on a circular path with tangential connection: APPR CT 162
Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT 163
Departing in a straight line with tangential connection: DEP LT 164
Departing in a straight line perpendicular to the last contour point: DEP LN 164
Departing on a circular path with tangential connection: DEP CT 165
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT 165

5.4 Path contours — Cartesian coordinates 166

Overview of path functions 166
Programming path functions 166
Straight line in rapid traverse G00 or straight line with feed rate F G01 167
Inserting a chamfer between two straight lines 168
Rounded corners G25 169
Circle center I, J 170
Circular arc around circle center 171
Circular arc G02/G03/G05 with fixed radius 173
Circular arc G06 with tangential transition 175
Superimposing a linear motion on a circular contour 176
Example: Linear movements and chamfers with Cartesian coordinates 177
Example: Circular movements with Cartesian coordinates 178
Example: Full circle with Cartesian coordinates 179

5.5 Path contours – Polar coordinates 180

Overview 180
Datum for polar coordinates: pole I, J 181
Straight line in rapid traverse G10 or straight line with feed rate F G11 181
Circular path G12/G13/G15 around pole I, J 182

Circle G16 with tangential connection ... 182
Helix ... 183
Example: Linear movement with polar coordinates ... 185
Example: Helix ... 186

5.6 Path contours – FK free contour programming ... 187

Fundamentals ... 187
Defining the working plane ... 188
FK programming graphics ... 189
Initiating the FK dialog ... 190
Pole for FK programming ... 190
Free straight line programming ... 191
Free circular path programming ... 192
Input possibilities ... 193
Auxiliary points ... 196
Relative data ... 197
Example: FK programming 1 ... 199

6 Programming Aids 201

6.1 GOTO function 202
Using the GOTO key 202

6.2 Screen keypad 203
Entering text with the screen keypad 203

6.3 Display of NC programs 204
Syntax highlighting 204
Scrollbar 204

6.4 Adding comments 205
Application 205
Entering comments during programming 205
Inserting comments after program entry 205
Entering a comment in a separate NC block 205
Commenting out an existing NC block 206
Functions for editing a comment 206

6.5 Freely editing an NC program 207

6.6 Skipping NC blocks 208
Insert a slash (/) 208
Delete the slash (/) 208

6.7 Structuring NC programs 209
Definition and applications 209
Displaying the program structure window / Changing the active window 209
Inserting a structure block in the program window 210
Selecting blocks in the program structure window 210

6.8 Calculator 211
Operation 211

6.9 Cutting data calculator 214
Application 214
Working with cutting data tables 215

6.10 Programming graphics 218
Activating and deactivating programming graphics 218
Generating a graphic for an existing NC program 219
Block number display ON/OFF 219
Erasing the graphic 219
Showing grid lines 219
Magnification or reduction of details 220

6.11 Error messages 221
Display of errors 221
Opening the error window 221
Detailed error messages 222
INTERNAL INFO soft key 222
GROUPING soft key 223
ACTIVATE AUTOMATIC SAVING soft key 223
Deleting errors 224
Error log 225
Keystroke log 226
Informational texts 227
Saving service files 227
Closing the error window 227

6.12 TNCguide: context-sensitive help 228
Application 228
Using TNCguide 229
Downloading current help files 232

7 Miscellaneous Functions 235

7.1 Entering miscellaneous functions M and STOP 236

Fundamentals 236

7.2 Miscellaneous functions for program run inspection, spindle and coolant 237

Overview 237

7.3 Miscellaneous functions for coordinate entries 238

Programming machine-referenced coordinates: M91/M92 238
Moving to positions in a non-tilted input coordinate system with a tilted working plane: M130 240

7.4 Miscellaneous functions for path behavior 241

Machining small contour steps: M97 241
Machining open contour corners: M98 242
Feed rate factor for plunging movements: M103 243
Feed rate in millimeters per spindle revolution: M136 244
Feed rate for circular arcs: M109/M110/M111 245
Pre-calculating radius-compensated contours (LOOK AHEAD): M120 246
Superimposing handwheel positioning during program run: M118 248
Retraction from the contour in the tool-axis direction: M140 249
Suppressing touch probe monitoring: M141 251
Deleting basic rotation: M143 251
Lifting off the tool automatically from the contour at NC stop: M148 252
Rounding corners: M197 253

8 Subprograms and Program Section Repeats 255

8.1 Labeling subprograms and program section repeats 256

Label 256

8.2 Subprograms 257

Operating sequence 257
Programming notes 257
Programming the subprogram 257
Calling a subprogram 257

8.3 Program-section repeats 258

Label G98 258
Operating sequence 258
Programming notes 258
Programming a program section repeat 259
Calling a program section repeat 259

8.4 Calling an external NC program 260

Overview of the soft keys 260
Operating sequence 261
Programming notes 261
Calling an external NC program 263

8.5 Point tables 265

Creating a point table 265
Hiding single points for the machining process 266
Selecting a point table in the NC program 267
Using point tables 268
Definition 268

8.6 Nesting 269

Types of nesting 269
Nesting depth 269
Subprogram within a subprogram 270
Repeating program section repeats 271
Repeating a subprogram 272

8.7 Programming examples 273

Example: Milling a contour in several infeeds 273
Example: Groups of holes 274
Example: Group of holes with multiple tools 275

9 Programming Q Parameters 277

9.1 Principle and overview of functions 278
Q parameter types 279
Programming notes 281
Calling Q parameter functions 282

9.2 Part families—Q parameters in place of numerical values 283
Application 283

9.3 Describing contours with mathematical functions 284
Application 284
Overview 285
Programming fundamental operations 286

9.4 Trigonometric functions 288
Definitions 288
Programming trigonometric functions 288

9.5 Calculation of circles 290
Application 290

9.6 If-then decisions with Q parameters 291
Application 291
Jump conditions 291
Programming if-then decisions 293

9.7 Entering formulas directly 294
Entering formulas 294
Calculation rules 294
Overview 296
Example: Trigonometric function 298

9.8 Checking and changing Q parameters 299
Procedure 299

9.9 Additional functions 301
Overview 301
D14 Output of error messages 302
D16 – Formatted output of text and Q parameter values 308
D18 – Reading system data 318
D19 Transferring values to PLC 318
D20 NC and PLC synchronization 319
D29 Transferring values to the PLC 320
D37 – EXPORT 320
D38 – Sending information from the NC program 321

9.10 String parameters 323
String processing functions 323
Assigning string parameters 324
Chain-linking string parameters 325
Converting a numerical value to a string parameter 326
Copying a substring from a string parameter 327
Reading system data 328
Converting a string parameter to a numerical value 329
Testing a string parameter 330
Determining the length of a string parameter 331
Comparing the lexical order of two alphanumerical strings 332
Reading out machine parameters 333

9.11 Preassigned Q parameters 335
Values from the PLC: Q100 to Q107 335
Active tool radius: Q108 335
Tool axis: Q109 336
Spindle status: Q110 336
Coolant on/off: Q111 336
Overlap factor: Q112 336
Unit of measure in the NC program Q113 337
Tool length: Q114 337
Measurement result from programmable touch-probe cycles: Q115 to Q119 337
Q parameters Q115 and Q116 for automatic tool measurement 338
Calculated coordinates of the rotary axes: Q120 to Q122 338
Measurement results from touch-probe cycles 339

9.12 Programming examples 343
Example: Rounding a value 343
Example: Ellipse 344
Example: Concave cylinder machined with ball-nose cutter 346
Example: Convex sphere machined with end mill 348

10 Special Functions 351

10.1 Overview of special functions 352

Main menu for SPEC FCT special functions 352
Program defaults menu 353
Functions for contour and point machining menu 353
Menu for defining different ISO functions 354

10.2 Function mode 355

Program function mode 355
Function Mode Set 355

10.3 Machining with polar kinematics 356

Overview 356
Activating FUNCTION POLARKIN 357
Deactivating FUNCTION POLARKIN 360
Example: SL cycles in the polar kinematics 361

10.4 Defining ISO functions 362

Overview 362

10.5 Modifying presets 363

Activating a preset 363
Copying a preset 365
Correcting a preset 366

10.6 Datum table 367

Application 367
Description 367
Creating a datum table 368
Opening and editing a datum table 369
Activating the datum table in your NC program 371
Activating the datum table manually 371

10.7 Compensation table 372

Application 372
Types of compensation tables 372
Creating a compensation table 373
Activate the compensation table 374
Editing a compensation table during program run 375

10.8 Accessing table values 376

Application 376
Reading a table value 376
Writing a table value 377
Adding a table value 378

10.9 Defining a counter 380
Application 380
Defining FUNCTION COUNT 381

10.10 Creating text files 382
Application 382
Opening and exiting a text file 382
Editing texts 383
Deleting and re-inserting characters, words and lines 383
Editing text blocks 384
Finding text sections 385

10.11 Freely definable tables 386
Fundamentals 386
Creating a freely definable table 386
Editing the table format 387
Switching between table and form view 389
D26 Opening a freely definable table 389
D27 writing to a freely definable table 390
D28 reading a freely definable table 392
Adapting the table format 393

10.12 Pulsing spindle speed FUNCTION S-PULSE 394
Program pulsing spindle speed 394
Resetting the pulsing spindle speed 396

10.13 Dwell time FUNCTION FEED DWELL 397
Programming a dwell time 397
Resetting the dwell time 398

10.14 Dwell time FUNCTION DWELL 399
Programming a dwell time 399

10.15 Lift-off tool at NC stop: FUNCTION LIFTOFF 400
Programming liftoff with FUNCTION LIFTOFF 400
Resetting the lift-off function 402

11 Multiple-axis Machining 403

11.1 Functions for multi-axis machining 404

11.2 The PLANE function: Tilting the working plane (option 8) 405

Introduction 405
Overview 407
Defining the PLANE function 408
Position display 408
Resetting PLANE function 409
Defining the working plane with spatial angles: PLANE SPATIAL 410
Defining the working plane with projection angles: PLANE PROJECTED 413
Defining the working plane with Euler angles: PLANE EULER 415
Defining the working plane with two vectors: PLANE VECTOR 417
Defining the working plane via three points: PLANE POINTS 419
Defining the working plane via a single incremental spatial angle: PLANE RELATIV 421
Tilting the working plane through axis angles: PLANE AXIAL 422
Defining the positioning behavior of the PLANE function 424
Automatic tilting into position MOVE/TURN/STAY 425
Selection of tilting possibilities SYM (SEQ) +/– 428
Selection of the transformation type 431
Tilting the working plane without rotary axes 433

11.3 Miscellaneous functions for rotary axes 434

Feed rate in mm/min on rotary axes A, B, C: M116 (option 8) 434
Shorter-path traverse of rotary axes: M126 435
Reducing display of a rotary axis to a value less than 360°: M94 436
Selecting tilting axes: M138 437

11.4 Running CAM programs 438

From 3-D model to NC program 438
Considerations required for post processor configuration 439
Please note the following for CAM programming 441
Possibilities for intervention on the control 443
ADP motion control 443

12 Data Transfer from CAD Files 445

12.1 Screen layout of CAD Viewer 446

CAD Viewer fundamentals 446

12.2 CAD Import (option 42) 447

Application 447
Using CAD Viewer 448
Opening a CAD file 448
Basic settings 449
Setting layers 451
Setting a preset 453
Setting the datum 455
Selecting and saving a contour 459
Selecting and saving machining positions 463

12.3 Generating STL files with 3D mesh (option 152) 467

Positioning the 3D model for rear-face machining 469

13 Tables and Overviews 471

13.1 System data 472

List of D18 functions 472
Comparison: D18 functions 509

13.2 Overview tables 513

Miscellaneous functions 513
User functions 515

13.3 ISO function overview TNC 320 519

1

Basic Information

1.1 About this manual

Safety precautions

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

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

⚠ DANGER

Danger indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **will result in death or severe injury.**

⚠ WARNING

Warning indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in death or serious injury**.

⚠ CAUTION

Caution indicates hazards for persons. If you do not follow the avoidance instructions, the hazard **could result in minor or moderate injury.**

NOTICE

Notice indicates danger to material or data. If you do not follow the avoidance instructions, the hazard **could result in property damage**.

Sequence of information in precautionary statements

All precautionary statements 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.

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

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

tnc-userdoc@heidenhain.de

1.2 Control model, software and features

This manual describes programming functions provided by controls as of the following NC software numbers.

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.

Control model	NC software number
TNC 320	771851-18
TNC 320 Programming Station	771855-18

The machine manufacturer adapts the usable features of the control to his machine by setting appropriate machine parameters. Some of the functions described in this manual may therefore not be among the features provided by the control on your machine tool.

Control functions that may not be available on your machine include:

- Tool measurement with the TT

In order to find out about the actual features of your machine, please contact the machine manufacturer.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the HEIDENHAIN controls. Participation in one of these courses is recommended to familiarize yourself thoroughly with the control's functions.

User's Manual for Programming of Machining Cycles:

All functions provided by the machining cycles are described in the User's Manual for **Programming of Machining Cycles**. Please contact HEIDENHAIN if you need this User's Manual.
ID: 1303429-xx

User's Manual for Programming of Measuring Cycles for Workpieces and Tools:

All functions provided by the touch-probe cycles are described in the User's Manual for **Programming of Measuring Cycles for Workpieces and Tools**. Please contact HEIDENHAIN if you need this User's Manual.
ID: 1303435-xx

User's Manual for Setup, Testing and Running NC Programs:

All information for setting up the machine as well as for testing and running your NC programs is provided in the User's Manual for **Setup, Testing and Running NC Programs**. Please contact HEIDENHAIN if you need this User's Manual.
ID: 1263173-xx

Software options

The TNC 320 features various software options, each of which can be enabled separately by your machine manufacturer. The respective options provide the functions listed below:

Additional Axis (option 0 and option 1)	
Additional axis	Additional control loops 1 and 2
Advanced Function Set 1 (option 8)	
Advanced functions (set 1)	**Machining with rotary tables** ■ Cylindrical contours as if in two axes ■ Feed rate in distance per minute **Coordinate conversions:** Tilting the working plane **Interpolation:** Circular in 3 axes with tilted working plane
HEIDENHAIN DNC (option 18)	
	Communication with external PC applications over COM component
CAD Import (option 42)	
CAD import	■ Support for DXF, STL, STEP and IGES ■ Adoption of contours and point patterns ■ Simple and convenient specification of presets ■ Selecting graphical features of contour sections from conversational programs
Extended Tool Management (option 93)	
Extended tool management	Python-based expansion of tool management ■ Program-specific usage sequence of all tools ■ Program-specific tooling list of all tools
Remote Desktop Manager (option 133)	
Remote operation of external computer units	■ Windows on a separate computer unit ■ Incorporated in the control's interface
CAD Model Optimizer (option 152)	
Optimization of CAD models	Conversion and optimization of CAD models ■ Fixtures ■ Workpiece blank ■ Finished part
Opt. contour milling (option 167)	
Optimized contour cycles	Cycles for machining any pockets and islands using trochoidal milling

Further options available

HEIDENHAIN offers further hardware enhancements and software options that can be configured and implemented only by your machine manufacturer.

For more information, please refer to your machine manufacturer's documentation or the HEIDENHAIN brochure titled **Options and Accessories**.

ID: 827222-xx

VTC User's Manual

All functions of the software for the VT 121 vision system are described in the **VTC User's Manual**. Please contact HEIDENHAIN if you require a copy of this User's Manual.

ID: 1322445-xx

Intended place of operation

The control complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Legal information

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

To call further information on the control:

- Press the **MOD** key
- Select the **General Information** group in the MOD menu
- Select the **License information** MOD function

When using the OPC UA NC server or DNC server, you can influence the behavior of the control. Therefore, before using these interfaces for productive purposes, verify that the control can still be operated without malfunctions or drops in performance. The manufacturer of the software that uses these communication interfaces is responsible for performing system tests.

New and modified functions with 77185x-18

Overview of new and modified software functions

Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.

ID: 1322093-xx

Software options

Further information: User's Manual for **Programming of Machining Cycles**

- CAD Model Optimizer (software option 152) was added for the **CAD Viewer**. With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective fixture and tool holder files, for example, or to position STL files generated from the simulation for another machining operation.

  Further information: "Generating STL files with 3D mesh (option 152)", Page 467
- Software option 167 (Optimized Contour Milling cycles) has been added. The OCM cycles enable the trochoidal milling of closed or open contour geometries that can be combined from several subcontours. High removal rates can be achieved because the control adheres to the programmed path overlap and thus keeps a constant tool contact angle during roughing.
- The **Display Step** software option (option 23) is available in the standard control version. The display step of the axes is no longer limited to four decimal places.

  The machine parameter **displayPace** (no. 101000) allows you to define the display step for the individual axes. The minimum display step of the axes is 0.1 µm or 0.0001°.
- **State Reporting Interface** (software option 137) is no longer available.

New functions

- The **BLK FORM FILE** function allows you to use STL files to define the workpiece blank, and optionally the finished part, by specifying the path of the files. This enables you to use, e.g., 3-D models from the CAD system in your NC program.

  Further information: "Defining the workpiece blank: G30/G31", Page 101
- When programming circular paths with **C**, **CR** and **CT**, the **LIN_** syntax element is now available in order to superimpose a linear motion over the circular motion of an axis. This allows you to program a helix in a simple way.

  In ISO programming, you can define a third axis using the free syntax input in conjunction with the **G02**, **G03**, and **G05** functions.

  Further information: "Path contours — Cartesian coordinates", Page 166
- **FUNCTION CORRDATA** allows you to activate a line of the compensation table. The compensation remains effective until the next tool change or the end of the program.

  Further information: "Activate the compensation table", Page 374
- **FUNCTION MODE SET** allows you to activate settings defined by the machine manufacturer (e.g., changes to the range of traverse) from within the NC program

  Further information: "Function Mode Set", Page 355
- The function **PRESET SELECT** allows you to activate a preset from the preset table. You can choose to retain active transformations and select the preset to which the function should apply.

  Further information: "Activating a preset", Page 363
- The function **PRESET COPY** allows you to copy a preset defined in the preset table to another line. You can optionally activate the copied preset and retain the active transformations.

  Further information: "Copying a preset", Page 365
- The function **PRESET CORR** corrects the active preset.

  Further information: "Correcting a preset", Page 366

- The function **POLARKIN** activates a polar kinematic model. In a polar kinematic model, the control performs movements using a rotary axis and two linear axes. You define the positioning behavior of the rotary axis and you define whether machining is allowed at the center of rotation of the rotary axis.
- The function **TABDATA** allows you to access the tool table and the compensation tables (*.tco and *.wco) during program run. In order to access the compensation tables, you need to activate them.
  - The function **TABDATA READ** allows you to read a value from a table and save it to a Q, QL, QR, or QS parameter.
  - The function **TABDATA WRITE** allows you to write a value from a Q, QL, QR, or QS parameter into a table.
  - The function **TABDATA ADD** allows you to add a value from a Q, QL, or QR parameter to a value contained in a table.

  Further information: "Accessing table values ", Page 376
- The **SELECT FILE** soft key has been added to the selection window provided by the **APPLY FILE NAME** soft key. If the called file is located in the same directory as the file you are calling it from, pressing this soft key loads the name of the file without its path.
- During retraction with **M140 MB MAX**, the control takes into account the safety clearances that can be defined by the machine manufacturer for software limit switches and collision objects. The control reduces the retraction movements by the clearances and stops before the software limit switches.

  Further information: "Retraction from the contour in the tool-axis direction: M140", Page 249

- In the mask file of the **FN 16: F-PRINT** (ISO: **D16**) function, you can define whether the control shows or hides blank lines for undefined QS parameters.

  Further information: "D16 – Formatted output of text and Q parameter values", Page 308
- The function **SYSSTR(ID10321 NR20)** determines the number of the current week in accordance with ISO 8601.

  Further information: "Reading system data", Page 328
- Using the **SYNTAX** soft key, you can enclose path information in quotation marks in order to use any special characters as part of the path (e.g., **/**). The control provides the **SYNTAX** soft key with the following NC functions:
  - **FN 16: F-PRINT** (ISO: **D16**)
  - **FN 26: TABOPEN** (ISO: **D26**)
  - Cycle **12 PGM CALL** (ISO: **G39**)
  - **CALL PGM** (ISO: **%**)
- The **FN 18: SYSREAD** (ISO: **D18**) functions have been extended:
  - **FN 18: SYSREAD (D18) ID10**: Read program information
    - **NR10**: Counts the number of executions of the current program section
  - **FN 18: SYSREAD (D18) ID15**
    - **NR10**: contents of a Q parameter
    - **NR11**: contents of a QL parameter
    - **NR12**: contents of a QR parameter
  - **FN 18: SYSREAD (D18) ID35 NR2**: active radius compensation
  - **FN 18: SYSREAD (D18) ID50**: values in the tool table
    - **NR45**: value in the **RCUTS** column
    - **NR46**: value in the **LU** column
  - **FN 18: SYSREAD (D18) ID210**: Read active coordinate transformations
    - **NR10**: Type of active tilting function
    - **NR11**: Coordinate system for manual movements
  - **FN 18: SYSREAD (D18) ID245 NR1**: Current nominal position of an axis (**IDX**) in the REF system
  - **FN 18: SYSREAD (D18) ID295**: Read data of the machine kinematics
    - **NR5**: Type of use of an axis within the kinematics
  - **FN 18: SYSREAD (D18) ID310**: Read geometrical behavior
    - **NR126**: State of the miscellaneous function **M126**
  - **FN 18: SYSREAD (D18) ID370 NR7**: Reaction of the control if a probing point is not reached during a programmable touch-probe cycle **14xx**
  - **FN 18: SYSREAD (D18) ID610**: Values of various machine parameters for **M120**
    - **NR49**: Mode of filter reduction of one axis (**IDX**)
    - **NR53**: Radial jerk at normal feed rate
    - **NR54**: Radial jerk at high feed rate
  - **FN 18: SYSREAD (D18) ID630**: SIK information of the control

    - **NR3**: SIK generation **SIK1** or **SIK2**
    - **NR4**: Specifies whether and how often a software option (**IDX**) has been enabled on controls with **SIK2**
  - **FN 18: SYSREAD (D18) ID950**: tool-table values for the current tool
    - **NR45**: value in the **RCUTS** column
    - **NR46**: value in the **LU** column
    - **NR47**: value in the **RN** column
    - **NR48**: value in the **R_TIP** column
  - **FN 18: SYSREAD (D18) ID990 NR28**: Current tool spindle angle
  - **FN 18: SYSREAD (D18) ID1070 NR1**: active feed-rate limit through the **F MAX** soft key
  - **FN 18: SYSREAD (D18) ID10010 NR1** and **NR2**: information about the current main program or the called NC program as a text variable
    - **IDX1**: directory path
    - **IDX2**: file name
    - **IDX3**: file type
  - **FN 18: SYSREAD (D18) ID10015**
    - **NR20**: contents of a QS parameter
    - **NR30**: contents of a QS parameter (all characters, except the letters and numbers, are replaced with an underscore (_) character)

  Further information: "System data", Page 472
- The **SELECT COMPENS. TABLE** soft key has been added to the soft-key row of the **PGM CALL** functions. This soft key activates the **SEL CORR-TABLE** function that allows you to activate a compensation table for the NC program.

  Further information: "Activate the compensation table", Page 374

- The control includes the sample tables **WMAT.tab**, **TMAT.tab** and **EXAMPLE.cutd** for automatic cutting data calculation.

  Further information: "Cutting data calculator", Page 214
- If, after a hardware change or an update, an error occurs when the control is booting, the control will automatically open the error window and display a question-type error. The control displays soft keys providing different response options.

  Further information: "Display of errors", Page 221
- In the error window, the **ACTIVATE AUTOMATIC SAVING** soft key has been added to **MORE FUNCTIONS**. This soft key allows you to define up to five error numbers. The control will automatically create a service file upon occurrence of these error numbers.

  Further information: "ACTIVATE AUTOMATIC SAVING soft key", Page 223
- The control saves active NC programs only up to a maximum size of 10 MB each to a service file. NC programs larger than that are not saved.

  Further information: "Saving service files", Page 227
- In the optional machine parameter **CfgClearError** (no. 130200), the machine manufacturer defines whether the control automatically clears warning and error messages when an NC program is selected or restarted.
- The CAD Viewer has been enhanced as follows:
  - When you double-click a layer in the **CAD Viewer**, the control marks the first contour element of this layer.
  - You can transfer the data from the CAD Import clipboard to an NC program as well as to other applications (e.g., **Leafpad**).
  - In **CAD Viewer**, you can choose the **YZ** and **ZX** working planes for milling. You can choose the desired working plane from a selection menu.

  Further information: "Data Transfer from CAD Files", Page 445

New setup functions

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- To install or update software version 18, a control with a hard disk size of at least 30 GB is required.
- The **Test Run** operating mode has been enhanced as follows:
  - In the **Test Run** operating mode, the control uses the active preset.
  - The **RESET THE PRESET** soft key has been added to the **BLANK IN WORK SPACE** menu. For the simulation, you can use this soft key to set the values of the active preset in the principal axis to 0.
  - If you use **TARGET** in the **BLK FORM FILE** function to define a finished part, you can show or hide it by soft key in the **Test Run** operating mode.
  - In **Test Run** operating mode, you can use the **EXPORT WORKPIECE** soft key to export the current status of the motion simulation as a 3D model in STL format.
  - In **Test Run** operating mode, the control provides an extended collision monitoring function that checks for collisions between the workpiece and the tool or tool holder. You can activate extended collision monitoring by soft key.
- The **OPEN COMPENS. TABLES** soft key has been added to the **Program run, single block** and **Program run, full sequence** operating modes. This soft key allows you to open and edit the active datum table and the active compensation tables.
- In the **Program run, single block** and **Program run, full sequence** operating modes, the **ACTUAL POSITON CAPTURE** key allows you to load the current position values of an axis into the datum table.
- The control can execute NC programs with the **SECTION MONITORING** NC function. This NC function may be included in NC programs of the TNC7, but has no function on the TNC 320.
- The control supports USB data media with the NTFS file system.
- The control provides the additional tool **Parole** that allows you to open video files.
- Within the file management, the control hides system files, as well as files and folders whose name begins with a period. If necessary, you can display the files with the **SHOW HIDDEN FILES** soft key.

- The general status display has been enhanced as follows:
  - The control displays a corresponding icon in the general status display when tool radius compensation is active.
  - If a feed-rate limit has been activated with the **F MAX** soft key, the control displays an exclamation mark after the feed-rate value in the general status display.
  - When the **PARAXCOMP DISPLAY** function is active, the control displays a corresponding icon in the general status display.
  - When the **PARAXCOMP MOVE** function is active, the control displays a corresponding icon in the general status display.
  - When the functions **PARAXMODE** oder **POLARKIN** are active, the control displays a corresponding icon in the general status display.
- The additional status display has been enhanced as follows:
  - The control shows the **POS HR** tab of the additional status display without the option 44 being enabled. The control shows the active maximum values of handwheel superimpositioning from **M118**.
  - The **TRANS** tab of the additional status display indicates the active shift in the working plane coordinate system (**WPL-CS**). If the shift comes from a compensation table (***.wco**), the control shows the path to the compensation table as well as the number and, if applicable, the comment of the active row.
- The HR 520 and HR 550 FS handwheels display the offset of handwheel superimpositioning from **M118**. The handwheels also show the active setting of the 3-D ROT menu (option 8).
- The **3-D ROT: AXIS ANGLE SPA. ANGLE** soft key has been added to the 3-D ROT menu. This soft key allows you to select whether the control displays the defined axis values or the current spatial angles.
- The preset table contains default values only in the columns **SPA**, **SPB**, and **SPC**. The machine manufacturer can define default values for the other columns.
- The TS 760 input option has been added to the **TYPE** column of the touch probe table.
- You define the shape of the stylus in the **STYLUS** column of the touch probe table. You define an L-shaped stylus with the **L-TYPE** selection.

- The following tool types have been added:
  - **Face milling cutter,MILL_FACE**
  - **Chamfer mill, MILL_CHAMFER**
  - **Side milling cutter, MILL_SIDE**
- The tool table has been enhanced as follows:
  - In the **RCUTS** column of the tool table, you define the front-face cutting width of a tool (e.g., for indexable inserts).
  - You define the usable length of a tool in the **LU** column of the tool table. The usable length limits the plunging depth of the tool in cycles.
  - You define the neck radius of the tool in the **RN** column of the tool table. This allows the control to display the tool correctly in the simulation (e.g., neck of end mills or side milling cutters).
  - You define a radius at the tip of the tool in the **R_TIP** column of the tool table.
  - You define a database ID for the tool in the **DB_ID** column of the tool table. In a tool database for all machines, you can identify tools with unique database IDs (e.g., within a workshop). This allows you to coordinate the tools of multiple machines more easily.
- You can use M3D and STL files (e.g., from the CAD system) as tool carrier files.
- The **ACTUAL POSITION CAPTURE** soft key allows you to load the actual position of the tool axis into the form view of the tool management as the tool length.
- The **POS. DISP.** soft key enables you to switch the tool table view. The control displays the tool table in combination with the position display or in full screen mode.
- Compensation tables allow you to compensate for tools during program run without having to edit the NC program or the tool tables. The compensation table *.tco is effective in the tool coordinate system as an alternative to the compensation in the tool call.

- The control supports the TS 760 workpiece touch probe.
- A link to the **Firewall settings** HEROS function has been added to the **External access** MOD function.
- A link to the **Certificate and keys** HEROS function has been added to the **External access** MOD function. This function can be used to define settings for secure connections via SSH.
- If the machine manufacturer has defined the parameter **CfgOemInfo** (no. 131700), then the control displays the **Info about machine manufacturer** area in the **General Information** MOD group.
- **Remote Desktop Manager** (option 133) allows you to create private connections if user administration is active. Private connections are visible to and usable by the creator only.
- The HEROS menu has been enhanced as follows:
  - In the HEROS settings, you can adjust the screen brightness of the control.
  - In the **Screenshot settings** window you can define under which path and file name the control saves screenshots. The file name can contain a placeholder (e.g., %N for sequential numbering).

- The user administration has been enhanced as follows:
  - When user administration is active, the file manager displays the **public** directory that can be accessed by every user.

    When you place the cursor on the **public** directory, the control shows the **ADVANCED ACCESS RIGHTS** soft key. This soft key allows the owner of a file to define the access rights of the following users:
    - Owner
    - Group
    - Other users
  - The **useradmin**, **oem**, and **sys** users can deactivate the user administration.
  - When user administration is active, you can set up only secure network connections via SSH. The control automatically disables the LSV2 connections via the serial interfaces (COM1 and COM2) and the network connections without user authentication. If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.
  - When user administration is active, you can create user-specific private network drive connections. **Single Sign On** allows you to connect to an encrypted network drive while logging on to the control.
  - When configuring the user administration, you can use the **Autologin** function to define a user who will automatically be logged on by the control during booting.
- The optional machine parameter **applyCfgLanguage** (no. 101305) allows you to define whether the HEROS operating system adopts the conversational language defined in machine parameter **ncLanguage** (no. 101301) during booting. If you activate this function, you can change the conversational language only in the machine parameters.
- The optional machine parameter **extendedDiagnosis** (no. 124204) allows you to define whether the control saves graphics journal data after a restart. This data is used to diagnose graphics problems.
- The machine parameter **CfgTTRectStylus** (no. 114300) has been added. This parameter allows you to define settings for a tool touch probe with a cuboid probe contact.

Modified functions

- You can use the **RND** element (ISO: **G24**) to connect circles that are perpendicular to the machining plane instead of lying in the machining plane.

  Further information: "Rounded corners G25", Page 169
- 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.

  Further information: "Defining the workpiece blank: G30/G31", Page 101
- The pop-up window for tool selection always shows the content of the **NAME** column, even if you are calling the tool with the tool number.

  Further information: "Calling the tool data", Page 140
- Within **FUNCTION S-PULSE**, you can define a lower and an upper speed limit for the pulsing speed by means of the **FROM-SPEED** and **TO-SPEED** syntax elements.

  Further information: "Pulsing spindle speed FUNCTION S-PULSE", Page 394
- In the **TABDATA WRITE**, **TABDATA ADD** and **FN 27: TABWRITE** (ISO: **D27**) NC functions, you can enter values directly.

  Further information: "Accessing table values ", Page 376

  Further information: "D27 writing to a freely definable table", Page 390
- If you program a precision stop of rotary axes using **M134** or **M135**, the control no longer displays an error message. The control ignores these miscellaneous functions.
- The number range for the machine manufacturer's miscellaneous functions has been increased from 1999 to 9999.
- The **M109** function allows the control to keep the feed rate constant at the cutting edge, including during approach and departure.

  Further information: "Feed rate for circular arcs: M109/M110/M111", Page 245

- The function **M120** for look-ahead calculation of a radius-compensated contour is no longer reset by milling cycles.

  Further information: "Pre-calculating radius-compensated contours (LOOK AHEAD): M120 ", Page 246
- The **FN 10** function also allows you to check QS parameters and texts for inequalities.

  Further information: "Programming if-then decisions", Page 293
- You can use UTF-8 character encoding in the mask file of **FN 16: F-PRINT** (ISO: **D16**).

  Further information: "D16 – Formatted output of text and Q parameter values", Page 308
- The priority of arithmetic operations has been changed in the Q parameter formula.

  Further information: "Calculation rules", Page 294

- While program run is interrupted or has been canceled, you can edit Q and QS parameters with numbers 0 to 99, 200 to 1199 and 1400 to 1999 in the **Q parameter list** window.
- Scrolling in the structure window works in the same way as scrolling in the NC program. You can define the position of the active structure block by soft key.

  Further information: "Structuring NC programs", Page 209
- The control uses the active unit of measure (mm or inches) for calculations in the cutting data calculator.
- The result fields and the diameter field of the cutting data calculator are freely editable.

  Further information: "Cutting data calculator", Page 214
- The CAD Viewer has been enhanced as follows:
  - **CAD Viewer** performs all internal calculations in mm. If you select the inch unit of measure, the **CAD Viewer** converts all values to inches.
  - The **Show sidebar** icon enlarges the Sidebar window to half the size of the screen.
  - The control always shows the **X**, **Y** and **Z** coordinates in the Element Information window. If the 2D mode is active, the control shows the Z coordinate dimmed.
  - **CAD Viewer** also recognizes circles as machining positions that consist of two semicircles.
  - You can save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import (option 42) is not available.
  - Path finding between individual drilling positions has been optimized in the **CAD Viewer**.
  - CAD Import (option 42) subdivides contours that are not located in the working plane, into individual sections. **CAD Viewer** creates straight lines **L** and circular arcs that are as long as possible.

    The resulting NC programs are often much shorter and clearer than NC programs generated by CAM. Thus, the contours are better suited for cycles, such as the OCM cycles (option 167).
  - CAD Import outputs the radii of the circular arcs as comments. At the end of the generated NC blocks, CAD Import displays the smallest radius to help you select the most suitable tool.
  - In the **Find circle centers by diameter range** window, you can filter the data by position depth values.
  - When you select contours and positions in **CAD Viewer**, you can rotate the workpiece using touch gestures. While you are using touch gestures, the control will not display any element information.

  Further information: "Data Transfer from CAD Files", Page 445
- In the compensation tables (*.tco and *.wco), the input range for all columns containing numerical values has been changed from +/– 999.999 to +/– 999.9999.

  Further information: "Compensation table", Page 372

- The **FILTER** soft key in the error window was renamed to **GROUPING**. With this soft key, the control groups warnings and error messages.
  Further information: "GROUPING soft key", Page 223

Modified setup functions

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- If you set the **MEASURING** soft key to **ON**, then the control displays the following additional information:
  - Surface orientation of the current position
  - Workpiece number
  - Workpiece name
  - Note during machining at rapid traverse, thread cutting cycle, or blank form update
- The **BLANK IN WORK SPACE** menu provides a soft key that allows you to load the current machine status. The control loads the following information in addition to the active preset:
  - Active kinematics
  - Active traverse ranges
  - Active machining mode
  - Active traverse limits
- The control uses hatch marking to depict threads in the simulation.
- The simulation takes into account the following columns of the tool table:
  - **R_TIP**
  - **LU**
  - **RN**
- The control takes into account the following NC functions in the **Test Run** operating mode.
  - **FN 27: TABWRITE** (ISO: **D27**)
  - **FUNCTION FILE**
  - **FUNCTION FEED DWELL**
- Any display filter you have set in the file manager will be retained even after a control restart.
- If you create a table, and there is at least one prototype available for this file type, the control displays the window **Select table format**. The control also shows whether the prototype is defined in mm or inches. If the control shows both units of measure, you can select a unit of measure.

  The machine manufacturer defines the prototypes. If the prototype contains values, the control transfers these values to the newly created table.

- If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed. If you press the **END** key again, the control opens the original NC program again with the cursor on the last selected line. With large files this behavior can cause a delay.
- The machine manufacturer defines the sequence for traversing the axes when returning to the contour.
- The control takes into account manual axes when returning to the contour.
- In **Program run, single block** operating mode, the control interprets the workpiece blank definition as a single NC block only.
- The control displays the tool index in the block scan pop-up window if needed.
- The control considers the functions **FN 27: TABWRITE** (ISO: D27) and **FUNCTION FILE** only in the operating modes **Program run, single block** and **Program run, full sequence**.
- The additional status display has been enhanced as follows:
  - The control also displays the number of repetitions on the **Overview** and **LBL** tabs of the additional status display after an internal stop.
  - When the **PARAXCOMP DISPLAY** function or the **PARAXCOMP MOVE** function is active, the control displays **(D)** or **(M)** after the respective axis designations on the **Übersicht** tab and the **POS** tab of the additional status display.
  - On the **TT** tab of the additional status display, the control displays the tilt angle of the tool touch probe as well as information about the cuboid probe contacts.
  - In **Test Run** operating mode, the control displays the **M** tab of the additional status display when the screen layout **PROGRAM STATUS** is active.
- The handwheel functions have been enhanced as follows:
  - The smallest definable speed level on handwheels with display has been changed from 0.1% to 0.01% of the maximum handwheel speed.
  - If a handwheel is active, the control shows the contouring feed rate in the display during program run. If only the currently selected axis is moving, the control shows the axis feed rate.
  - When you activate a handwheel with display, the control automatically activates the override potentiometer of the handwheel.
  - In the operating modes **Manual Operation** and **Positioning w/ Manual Data Input**, you can activate a handwheel with display while a macro or a manual tool change is being executed.
- You can activate or deactivate the **F MAX** soft key for reducing the feed rate. The defined value is retained.

- The manual touch probe functions have been enhanced as follows:
  - By default, the control calculates the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the control calculates the basic rotation in the workpiece coordinate system (W-CS).
  - If you use a manual touch-probe function to automatically probe a stud or hole with an opening angle of 360°, the control returns the workpiece touch probe to the starting position at the end of the probing process.
  - With the **PROBING PL** function, the control shows a help graphic prior to aligning the 3D basic rotation in order to point out the danger of collision during tilting.
- The input range of the columns **SPA**, **SPB**, **SPC**, **A_OFFS**, **B_OFFS**, and **C_OFFS** in the preset table has been changed from +/–999.9999999° to +/–99 999.99999°.
- The minimum input value of the **FMAX** column in the touch probe table has been changed from –9999 to +10.
- The form view of the tool management shows only those input fields that are needed for the selected tool type.
- The maximum input range of the **LTOL** and **RTOL** columns of the tool table has been increased. It was from 0 mm to 0.9999 mm, and is now from 0.0000 mm to 5.0000 mm.
- The maximum input range of the **LBREAK** and **RBREAK** columns of the tool table has been increased. It was from 0 mm to 0.9999 mm, and is now from 0.0000 mm to 9.0000 mm.
- The control no longer supports the ITC 750 additional operating station.
- If the control is accessed from external, it displays a corresponding icon in the header.

  The control uses an icon to indicate whether a connection configuration is secure or non-secure.
- Limits that have been defined in the **Traverse limits** MOD function are also effective for modulo axes.
- **Program run** in the **Machine times** MOD area shows only the times at which at least one axis was moving during program run.
- From within the **Diagnostic functions** MOD group, you can access **TNCdiag** and **Hardware configuration** without a code number.
- The name of a connection in **Remote Desktop Manager** (option 133) must not contain characters other than letters, numbers, and underscores.
- The user interface of the **Network settings** window has been changed. For network configuration, use the **Network Connections** window.

- In the **Certificate and keys** window, you can select a file with additional public SSH keys in the **Externally administered SSH key file** area. This allows you to use SSH keys without having to transfer them to the control.
- You can export and import existing network configurations in the **Network settings** window.
- If you enter a password or code number with Caps Lock active, then the control issues a message.
- The machine manufacturer can define a path for saving the values of the QR parameters. If the values are on the **TNC** drive, you can use the **NC/PLC Backup** HEROS function to back up the QR parameters.
- **PKI Admin** now includes the **Advanced settings** tab.

  You can define whether the server certificate should contain static IP addresses and allow connections without an associated CRL file.
- The user administration has been enhanced as follows:
  - When user administration is active, the **Liberating motion** mode requires the NC.OPModeManual permission (i.e., at least the role of **NC.Programmer**).
  - If you use the **Connection to Windows domain** function when configuring the user administration, you can set up a secure connection by activating the **Use LDAPs** check box.
  - If a remote log-in takes place while user administration is inactive, for example via SSH, the control automatically assigns the **HEROS.LegacyUserNoCtrlfct** role.
  - If you deactivate the user administration and select the **Delete existing user databases** check box, the control also deletes the .home folder in the **TNC:** directory.
  - Your IT administrator can set up a function user to facilitate connectivity to the Windows domain.
  - If you have connected the control to the Windows domain, you can export the required configurations for other controls.

- The machine parameter **spindleDisplay** (no. 100807) has been enhanced. The control can also display the spindle position on the **Overview** tab of the additional status display when the spindle is in jog mode.
- The input range of machine parameter **displayPace** (no. 101000) has been extended. The minimum display step of the axes is 0.000001° or mm.
- If user administration is inactive, the control also automatically blocks non-secure LSV2 or RPC connections. In the optional machine parameters **allowUnsecureLsv2** (no. 135401) and **allowUnsecureRpc** (no. 135402), the machine manufacturer can define whether the control will permit non-secure connections. These machine parameters are included in the **CfgDncAllowUnsecur** (no. 135400) data object.

  When the control detects a non-secure connection, it displays an informational notice.
- Machine parameter **CfgStretchFilter** (no. 201100) has been removed.

New cycle functions of software 77185x-18

Overview of new and modified software functions

Further information about the previous software versions is presented in the **Overview of New and Modified Software Functions** documentation. Please contact HEIDENHAIN if you need this documentation.

ID: 1322093-xx

New cycle functions with 77185x-18

- Cycle **224 DATAMATRIX CODE PATTERN** (ISO: **G224**)
  This cycle allows the control to convert a text into a machine-readable Data Matrix code. The Data Matrix code functions as a point pattern for a previously defined machining cycle.
- Cycle **271 OCM CONTOUR DATA** (ISO: **G271**, option 167)
  This cycle allows you to define the machining information for the OCM cycles. You can define the first pocket of the contour description as an open frame. As a result, the downfeed is performed outside of the material during machining.
- Cycle **272 OCM ROUGHING** (ISO: **G272**, option 167)
  This cycle allows the control to machine the defined contour using trochoidal milling. The control precisely adheres to the programmed path overlap, while keeping a constant tool contact angle and depth of cut.
- Cycle **273 OCM FINISHING FLOOR** (ISO: **G273**, option 167)
  This cycle allows the control to finish the contour floor with a constant path overlap. The finishing allowance for the side is left over.
- Cycle **274 OCM FINISHING SIDE** (ISO: **G274**, option 167)
  This cycle enables finishing along the contour. The control approaches the contour tangentially and machines every subcontour separately. The control does not take into account the finishing allowance defined for the floor in Cycle **271** (**G271**), but moves to full depth.
- Cycle **277 OCM CHAMFERING** (ISO: **G277**, option 167)
  This cycle allows the control to deburr contours that were last defined, roughed, or finish-machined with other OCM cycles.
- Cycle **1271 OCM RECTANGLE** (ISO: **G1271**, option 167)
  This cycle allows you to define a rectangle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1272 OCM CIRCLE** (ISO: **G1272**, option 167)
  This cycle allows you to define a circle that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1273 OCM SLOT / RIDGE** (ISO: **G1273**, option 167)
  This cycle allows you to define a slot that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1274 OCM CIRCULAR SLOT** (ISO: **G1274**, option 167)
  This cycle allows you to define a circular slot that is then used as a pocket or boundary for face milling in conjunction with other OCM cycles.

- Cycle **1278 OCM POLYGON** (ISO: **G1278**, option 167)
  This cycle allows you to define a polygon that is then used as a pocket, island, or boundary for face milling in conjunction with other OCM cycles.
- Cycle **1281 OCM RECTANGLE BOUNDARY** (ISO: **G1281**, option 167)
  This cycle allows you to define a rectangular boundary for islands or open pockets that you previously programmed with the standard OCM shapes.
- Cycle **1282 OCM CIRCLE BOUNDARY** (ISO: **G1282**, option 167)
  This cycle allows you to define a rectangular boundary for islands or open pockets that you previously programmed with the standard OCM shapes.
- The control offers an **OCM cutting data calculator** that allows you to determine the optimum cutting data for Cycle **272 OCM ROUGHING**(ISO: **G272**, option 167). Press the **OCM CUTTING DATA** soft key to open the cutting data calculator during cycle definition. You can load the results directly into the cycle parameters.

Further information: User's Manual for **Programming of Machining Cycles**

New measuring cycle functions with 77185x-18

- Cycle **1400 POSITION PROBING** (ISO: **G1400**)
  This cycle allows you to probe a single position. You can transfer the acquired values to the active row of the preset table.
- Cycle **1401 CIRCLE PROBING** (ISO: **G1401**)
  This cycle allows you to determine the center point of a hole or a stud. You can transfer the acquired values to the active row of the preset table.
- Cycle **1402 SPHERE PROBING** (ISO: **G1402**)
  This cycle allows you to determine the center point of a sphere. You can transfer the acquired values to the active row of the preset table.
- Cycle **1404 PROBE SLOT/RIDGE** (ISO: **G1404**)
  This cycle determines the center and the width of a slot or ridge. The control probes two opposing touch points. You can also define a rotation for the slot or the ridge.
- Cycle **1412 INCLINED EDGE PROBING** (ISO: **G1412**)
  This cycle allows you to ascertain a workpiece misalignment by probing two points on an inclined edge.
- Cycle **1416 INTERSECTION PROBING** (ISO: **G1416**)
  This cycle allows you to determine the intersection of two edges. The cycle requires a total of four touch points and two positions per edge. You can use the cycle in the three object planes **XY**, **XZ** and **YZ**.
- Cycle **1430 PROBE POSITION OF UNDERCUT** (ISO: **G1430**)
  This cycle determines a single position with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus.
- Cycle **1434 PROBE SLOT/RIDGE UNDERCUT** (ISO: **G1434**)
  This cycle determines the center and the width of a slot or ridge with an L-shaped stylus. The control can probe undercuts due to the shape of the stylus. The control probes two opposing touch points.
- Cycle **1493 EXTRUSION PROBING** (ISO: **G1493**)
  This cycle allows you to define an extrusion. With an active extrusion, the control repeats the touch points along a direction over a defined length.

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

Modified cycle functions with 77185x-18

- Within the **CONTOUR DEF** function, you can exclude certain areas **V** (void) from machining. These areas may be contours in cast parts or machining operations from previous steps, for example.
- You can also define subcontours as **LBL** subprograms within the complex **SEL CONTOUR** contour formula.
- In Cycle **12 PGM CALL** (ISO: G39) you can use the **SYNTAX** soft key to place paths within quotation marks. To separate folders and files within paths, both the **** and the **/** character are permitted.
- The parameter **Q357 CLEARANCE TO SIDE** has been added to Cycle **202 BORING** (ISO: **G202**). This parameter allows you to define how far the control retracts the tool at the bottom of the hole in the working plane. This parameter is only effective if parameter**Q214 DISENGAGING DIRECTN** has been defined.
- The parameter **Q357 CLEARANCE TO SIDE** has been added to Cycle **202 BORING** (ISO: **G202**). This parameter allows you to define how far the control retracts the tool at the bottom of the hole in the working plane. This parameter is only effective if parameter**Q214 DISENGAGING DIRECTN** has been defined.
- The parameter **Q373 FEED AFTER REMOVAL** has been added to Cycle **205 UNIVERSAL PECKING** (ISO: **G205**). This parameter is used to define the feed rate for returning to the advanced stop distance after chip removal.
- Cycles **205 UNIVERSAL PECKING** (ISO: **G205**) and **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) check the parameter **Q379 STARTING POINT**. If the value of the starting point is equal to or greater than the value of the parameter **Q201 DEPTH**, then the control issues an error message.
- The parameters **Q429 COOLANT ON** and **Q430 COOLANT OFF** in Cycle **241 SINGLE-LIP D.H.DRLNG** (ISO: **G241**) have been enhanced. You can define a path for a user macro.
- The parameter **Q370 TOOL PATH OVERLAP** has been added to Cycle **208 BORE MILLING** (ISO: **G208**). Use this parameter to define the lateral infeed.
- Cycle **240 CENTERING** (ISO: **G240**) has been extended in order to consider pre-drilled diameters.

  The following parameters have been added:
  - **Q342 ROUGHING DIAMETER**
  - **Q253 F PRE-POSITIONING**: If parameter **Q342** is defined, feed rate for approaching the deepened starting point
- The machine manufacturer can hide the cycles **220 POLAR PATTERN** (ISO: **G220**) and **221 CARTESIAN PATTERN** (ISO: **G221**). We recommend using the **PATTERN DEF** function.

- Cycle **225 ENGRAVING** (ISO: **G225**) has been expanded:
  - The parameter **Q202 MAX. PLUNGING DEPTH** allows you to define the maximum plunging depth.
  - The parameter **Q367 TEXT POSITION** now includes the input options **7**, **8** and **9**. With these values, you can set the reference of the engraving text onto the horizontal centerline.
  - The approach behavior was changed. If the tool is below the **2ND SET-UP CLEARANCE**, the control first positions the tool to the 2nd set-up clearance **Q204** and then to the starting position in the working plane.
- The input value **1** has been added to parameter **Q515 FONT** in Cycle **225 ENGRAVING** (ISO: **G225**). Use this input value to select the **LiberationSans-Regular** font.
- In Cycle **225 ENGRAVING** (ISO: **G225**), you can program system variables that contain the following information for the current main program and the NC program called:
  - Entire file path
  - Directory path
  - File name
  - File type
- Cycle **225 ENGRAVING** (ISO: **G225**) allows you to use system variables to engrave the number of the current week.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), you program a limit that is perpendicular to the milling direction **Q350**, then the control adds the tool radius to the length of the surface in the unlimited direction. As a result, the control completely machines the defined surface without leaving behind any residual material, as would be caused by the tool radius. If the parameter **Q220** (corner radius) is defined, then the control adds both the tool radius and this value to the length of the surface.
- If, in Cycle **233 FACE MILLING** (ISO: **G233**), the parameter **Q389** has been defined with the value 2 or 3 and a lateral limit is defined in addition, then the control approaches the contour or departs from it on an arc with **Q207 FEED RATE MILLING**.
- Cycles **208 BORE MILLING** (ISO: **G208**), **253 SLOT MILLING** (ISO: **G208**) and **254 CIRCULAR SLOT** (ISO: **G254**) monitor a cutting width defined in the column **RCUTS** of the tool table. If the center of a tool that is not a center-cut tool would contact the workpiece surface, the control issues an error message.
- Cycles**251 RECTANGULAR POCKET** (ISO: **G251**), **252 CIRCULAR POCKET** (ISO: **G252**) and **272 OCM ROUGHING** (ISO: **G272**, option 167) take into consideration a cutting width defined in the column **RCUTS** when calculating the plunging path.

- If the defined usable length in column **LU** of the tool table is less than the depth, the control displays an error message.

  The following cycles monitor the usable length LU:
  - All cycles for drilling and boring
  - All cycles for tapping
  - All cycles for the machining of pockets and studs
  - Cycle 22 **ROUGH-OUT** (ISO: **G122**)
  - Cycle 23 **FLOOR FINISHING** (ISO: **G123**)
  - Cycle 24 **SIDE FINISHING** (ISO: **G124**)
  - Cycle 233 **FACE MILLING** (ISO: **G233**)
  - Cycle 272 **OCM ROUGHING** (ISO: **G272**, option 167)
  - Cycle 273 **OCM FINISHING FLOOR** (ISO: **G273**, option 167)
  - Cycle 274 **OCM FINISHING SIDE** (ISO: **G274**, option 167)
- Certain cycles permit you to enter tolerances. In the following cycles, you can define dimensions, tolerances according to EN ISO 286-2 or general tolerances according to ISO 2768-1:
  - Cycle **208 BORE MILLING** (ISO: **G208**)
  - **127x** (Option 167) OCM standard figure cycles
- The following cycles consider the miscellaneous functions **M109** and **M110**:
  - Cycle **22 ROUGH-OUT** (ISO: G122)
  - Cycle **23 FLOOR FINISHING** (ISO: G123
  - Cycle **24 SIDE FINISHING** (ISO: G124
  - Cycle **25 CONTOUR TRAIN** (ISO: G125)
  - Cycle **275 TROCHOIDAL SLOT** (ISO: G275
  - Cycle **276 THREE-D CONT. TRAIN** (ISO: G276)
  - Cycle **274 OCM FINISHING SIDE** (ISO: G274, option 167)
  - Cycle **277 OCM CHAMFERING** (ISO: G277, option 167)

Further information: User's Manual for **Programming of Machining Cycles**

- Cycle **460 CALIBRATION OF TS ON A SPHERE** (ISO: **G460**) determines the radius, length, center offset and spindle angle of an L stylus.
- Cycles **14xx** support probing with an L-shaped stylus.
- The unit of measure of the main program can be seen in the header of the log file of the Probing Cycles **14xx** and **42x**
- In cycles with the number **14xx**, you can now pre-position with a handwheel in semi-automatic mode. After probing, you can move to clearance height manually.

- The Cycles **1420 PROBING IN PLANE** (ISO: **G1420**), **1410 PROBING ON EDGE** (ISO: **G1410**), **1411 PROBING TWO CIRCLES** (ISO: **G1411**) have been enhanced:
  - In these cycles, you can define tolerances according to EN ISO 286-2 or general tolerances according to ISO 2768-1.
  - If you defined the value 2 in parameter **Q1125 CLEAR. HEIGHT MODE**, the control positions the touch probe at rapid traverse **FMAX** from the touch probe table to the set-up clearance.
- By default, Cycles **1410 PROBING ON EDGE** (ISO: **G1410**) and **1411 PROBING TWO CIRCLES** (ISO: **G1411**) calculate the basic rotation in the input coordinate system (I-CS). If the axis angles and tilt angles don't coincide, the cycles calculate the basic rotation in the workpiece coordinate system (W-CS).
- Cycle **441 FAST PROBING** (ISO: **G441**) now features the parameter **Q371 TOUCH POINT REACTION**. This parameter defines the reaction of the control in cases where the stylus is not deflected.
- Using the parameter **Q400 INTERRUPTION** in Cycle **441 FAST PROBING** (ISO: **G441**), you can define whether the control will interrupt program run and display a measuring log. The parameter is effective in conjunction with the following cycles:
  - Touch-probe cycles **46x** for calibrating the workpiece touch probe
  - Touch-probe cycles **14xx** for determining a workpiece misalignment and for acquiring the preset
- Cycles **480 CALIBRATE TT** (ISO: **G480**) and **484 CALIBRATE IR TT** (ISO: **G484**) allow you to calibrate a tool touch probe with cuboid probe contacts.
- The parameter **Q523 TT-POSITION** has been added to Cycle **484 CALIBRATE IR TT** (ISO: **G484**). This parameter allows you to define the position of the tool touch probe and, if desired, to transfer the position to the machine parameter **centerPos** after calibration.
- For rotating tools, Cycle **483 MEASURE TOOL** (ISO: **G483**) first measures the tool length and then the tool radius.
- Using the optional machine parameter **maxToolLengthTT** (no. 122607), the machine manufacturer defines a maximum tool length for tool touch probe cycles.
- 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.

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

2

First Steps

2.1 Overview

This chapter is intended to help you quickly learn to handle the most important procedures on the control. For more information on the respective topic, see the section referred to in the text.

The following topics are covered in this chapter:

- Switching on the machine
- Programming the workpiece

The following topics are covered in the User's Manual for Setup, Testing and Running NC Programs:

- Switching on the machine
- Graphically testing the workpiece
- Setting up tools
- Setting up the workpiece
- Machining the workpiece

2.2 Switching on the machine

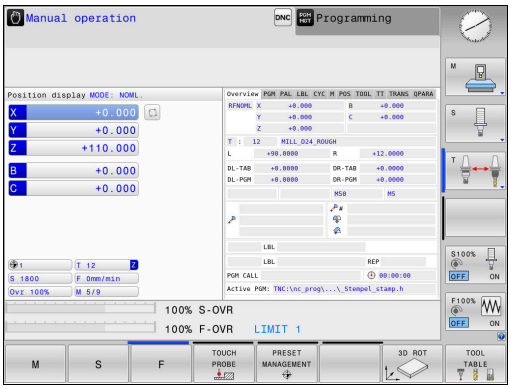
Acknowledging the power interruption

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 on the machine:

- Switch on the power supply for the control and the machine

> The control starts the operating system. This process may take several minutes.

> The control will then display the "Power interrupted" message in the screen header.

CE

- Press the **CE** key

> The control compiles the PLC program.

I

- Switch on the machine control voltage

> The control is in the **Manual operation** mode.

Depending on your machine, you may need to carry out further steps in order to run NC programs.

Further information on this topic

- Switch on the machine
  Further information: User's Manual for **Setup, Testing and Running NC Programs**

2.3 Programming the first part

Selecting the operating mode

You can write NC programs only in the **Programming** operating mode:

- ▶ Press the operating mode key
- > The control switches to the **Programming** operating mode.

Further information on this topic

- Operating modes
  Further information: "Programming", Page 85

Important controls and displays

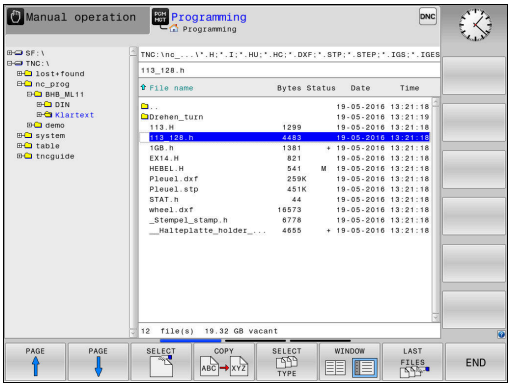
Key	Functions for conversational guidance
ENT	Confirm entry and activate the next dialog prompt
NO ENT	Ignore the dialog question
END	End the dialog immediately
DEL	Abort dialog, discard entries
	Soft keys on the screen with which you select functions appropriate to the active operating state

Further information on this topic

- Writing and editing NC programs
  Further information: "Editing an NC program", Page 110
- Overview of keys
  Further information: "Controls and displays", Page 2

Creating a new NC program / file management

To create a new NC program:



PGM MGT

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

  The file management of the control is arranged much like the file management on a PC with Windows Explorer. The file management enables you to manage data in the control's internal memory.
- Select a folder
- Enter the desired file name with the extension **.I**

ENT

- Press the **ENT** key
- The control asks for the unit of measure of the new NC program.

MM

- Press the soft key of the desired unit of measure: **MM** or **INCH**

The control automatically generates the first and last NC blocks of the NC program. You will not be able to change these NC blocks at a later time.

Further information on this topic

- File management
  Further information: "File management", Page 116
- Creating a new NC program
  Further information: "Creating and entering NC programs", Page 100

Defining a workpiece blank

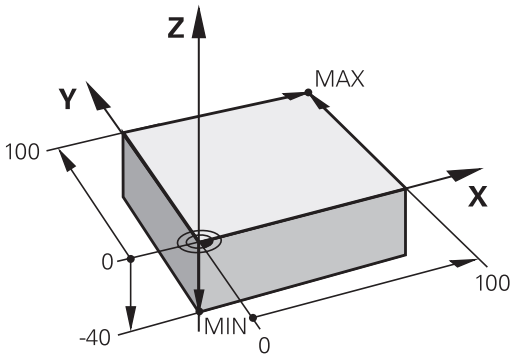
Once you have opened a new NC program, you can define a workpiece blank. You can define a cuboid by entering the MIN and MAX points relative to the selected preset.

After you have selected the desired shape for the blank with the appropriate soft key, the control automatically initiates the workpiece blank definition process and prompts you to enter the required data.

To define a cuboid-shaped blank:

- Press the soft key for the desired shape of the workpiece blank
- **Spindle axis Z, plane XY**: Enter the active spindle axis. G17 is saved as default setting. Accept with the **ENT** key
- **Workpiece blank def.: minimum X**: Enter the smallest X coordinate of the blank relative to the preset (e.g.: 0), and confirm with the **ENT** key
- **Workpiece blank def.: minimum Y**: Enter the smallest Y coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key
- **Workpiece blank def.: minimum Z**: Enter the smallest Z coordinate of the blank relative to the preset (e.g., –40), and confirm with the **ENT** key
- **Workpiece blank def.: maximum X**: Enter the largest X coordinate of the blank relative to the preset (e.g., 100), and confirm with the **ENT** key
- **Workpiece blank def.: maximum Y**: Enter the largest Y coordinate of the blank relative to the preset (e.g., 100), and confirm with the **ENT** key
- **Workpiece blank def.: maximum Z**: Enter the largest Z coordinate of the blank relative to the preset (e.g., 0), and confirm with the **ENT** key

> The control ends the dialog.



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.

Example

```
%NEW G71 *
N10 G30  G17 X+0  Y+0 Z-40*
N20 G31  X+100 Y+100  Z+0*
N99999999 %NEW  G71 *
```

Further information on this topic

- Define workpiece blank
  Further information: "Creating a new NC program", Page 106

Program layout

NC programs should be arranged consistently in a similar manner. This makes it easier to find your place, accelerates programming and reduces errors.

Recommended program layout for simple, conventional contour machining

Example

```
%BSPCONT G71 *
N10 G30  G71 X...  Y... Z...*
N20 G31  X... Y...  Z...*
N30 T5  G17 S5000*
N40 G00  G40 G90  Z+250 M3*
N50 X...  Y...*
N60 G01  Z+10 F3000  M8*
N70 X...  Y... RL  F500*
...
N160 G40  ... X...  Y... F3000  M9*
N170 G00  Z+250 M2*
N99999999 BSPCONT  G71 *
```

1 Call tool, define tool axis
2 Retract the tool; turn on spindle
3 Pre-position the tool in the working plane near the contour starting point
4 Pre-position the tool along the tool axis above the workpiece, or pre-position the tool directly to the cutting depth, and turn on coolant as needed
5 Contour approach
6 Contour machining
7 Contour departure
8 Retract the tool, end the NC program

Further information on this topic

- Contour programming
  Further information: "Programming tool movements for machining", Page 152

Recommended program layout for simple cycle programs

Example

```
%BSBCYC G71 *
N10 G30  G71 X...  Y... Z...*
N20 G31  X... Y...  Z..*
N30 T5  G17 S5000*
N40 G00  G40 G90  Z+250 M3*
N50 G200...*
N60 X...  Y...*
N70 G79  M8*
N80 G00  Z+250 M2*
N99999999 BSBCYC  G71 *
```

1 Call tool, define tool axis
2 Retract the tool; turn on spindle
3 Define the fixed cycle
4 Move to the machining position
5 Call the cycle, and switch on the coolant
6 Retract the tool, end the NC program

Further information on this topic

- Cycle programming
  Further information: User's Manual for **Programming of Machining Cycles**

Programming a simple contour

Suppose you want to mill a single time around the contour shown on the right at a depth of 5 mm. You have already defined the workpiece blank.

After you have opened an NC block with a function key, the control will prompt you to enter all of the data in the header using dialog texts.

To program the contour:

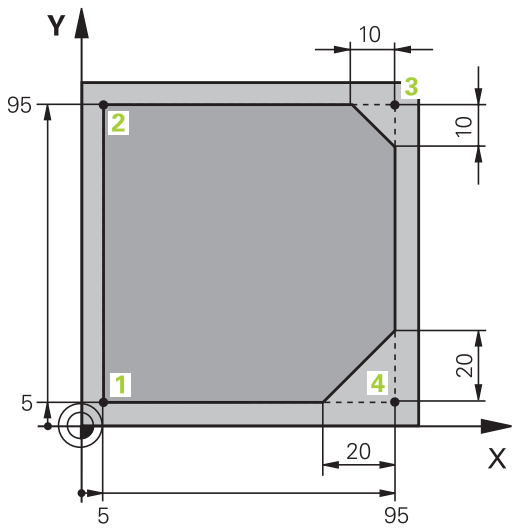
Call the tool

TOOL CALL
- ▶ Press the **TOOL CALL** key
- ▶ Enter the tool data, e.g., tool number 16

ENT
- ▶ Press the **ENT** key

ENT
- ▶ Confirm the tool axis **G17** with the **ENT** key
- ▶ Enter the spindle speed (e.g., 6500)

END
- ▶ Press the **END** key
- > The control completes the NC block.



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.

Retract the tool

L
- Press the **L** key

←
- Press the left arrow key
> The control opens the input range for G functions.

G00
- Press the **G00** soft key
> The control executes the NC block in rapid traverse.

Alternative:

G
- Press the **G** key on the alphanumeric keyboard
- Enter **0**

ENT
- Press the **ENT** key
> The control executes the NC block in rapid traverse.

G90
- Press the **G90** soft key
> The control processes the entered dimensions as absolute dimensions.

Z
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT
- Press the **ENT** key

G40
- Press the **G40** soft key
> The control does not activate radius compensation.
- If needed, enter a miscellaneous function **M**, such as **M3**, turn on spindle

END
- Press the **END** key
> The control saves the positioning block.

Pre-position the tool in the working plane

G
- Press the **G** key on the alphanumeric keyboard
- Enter **0**

ENT
- Press the **ENT** key
> The control executes the NC block in rapid traverse.

X
- Press the **X** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

Y
- Press the **Y** axis key
- Enter the value for the position to be approached (e.g., –20 mm)

ENT
- Press the **ENT** key

G40
- Press the **G40** soft key
> The control does not activate radius compensation.
- If needed, enter a miscellaneous function M

END
- Press the **END** key
> The control saves the positioning block.

Position the tool to the cutting depth

G
- Press the **G** key on the alphanumeric keyboard
- Enter **0**

ENT
- Press the **ENT** key

> The control executes the NC block in rapid traverse.

Z
- Press the **Z** axis key
- Enter the value for the position to be approached (e.g., -5 mm)

ENT
- Press the **ENT** key

G40
- Press the **G40** soft key

> The control does not activate radius compensation.

- Enter a miscellaneous function **M**, such as **M8** to turn coolant on

END
- Press the **END** key

> The control saves the positioning block.

Approach the contour smoothly

L
- Press the **L** key
- Enter the coordinates of the contour starting point **1**

ENT
- Press the **ENT** key

G41
- Press the **G41** soft key

> The control activates radius compensation to the left.

- Enter the value for the machining feed rate (e.g., 700 mm/min)

END
- Press the **END** key

G
- Press the **G** key on the alphanumeric keyboard
- Enter **26**

ENT
- Press the **ENT** key

> The control opens the **G26** command, smooth contour approach.

- Enter the rounding radius of the approach circle (e.g., 8 mm)

END
- Press the **END** key

> The control saves the approach movement.

Machine the contour

- Press the L key
- Enter the changing coordinates of contour point **2** (e.g., **Y 95**)

- Press the **END** key

> The control applies the changed value and retains all of the other information from the previous NC block.

- Press the **L** key
- Approach the changing coordinates of contour point **3** (e.g., **X 95**)

- Press the **END** key

- Press the **CHF** key
- Enter the chamfer width **G24** at contour point **3** (10 mm)

- Press the **END** key

> The control saves the chamfer at the end of the linear block.

- Press the **L** key
- Enter the changing coordinates of contour point **4**

- Press the **END** key

- Press the **CHF** key
- Enter the chamfer width **G24** at contour point **4** (20 mm)

- Press the **END** key

Complete the contour with a smooth departure

- Press the **L** key
- Enter the changing coordinates of contour point **1**

- Press the **END** key

- Press the **G** key on the alphanumeric keyboard
- Enter **27**

- Press the **ENT** key
- The control opens the **G27** command, smooth contour approach.
- Enter the rounding radius of the departure circle (e.g., 8 mm)

- Press the **END** key
- The control saves the departure movement.

- Press the **L** key
- Specify coordinates outside of the workpiece in X and Y (e.g. **, X -20 Y -20**

- Press the **ENT** key

- Press the **G40** soft key
- The control does not activate radius compensation.
- Enter the value for the positioning feed rate (e.g., 3000 mm/min)

- Press the **ENT** key
- If needed, enter a miscellaneous function **M**, such as M9, turn off coolant

- Press the **END** key
- The control saves the entered positioning block.

Retract the tool

G
- Press the **G** key on the alphanumeric keyboard
- Enter **0**

ENT
- Press the **ENT** key

> The control executes the NC block in rapid traverse.

Z
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT
- Press the **ENT** key

G40
- Press the **G40** soft key

> The control does not activate radius compensation.

- Enter a miscellaneous function **M**, such as **M30** for program end

END
- Press the **END** key

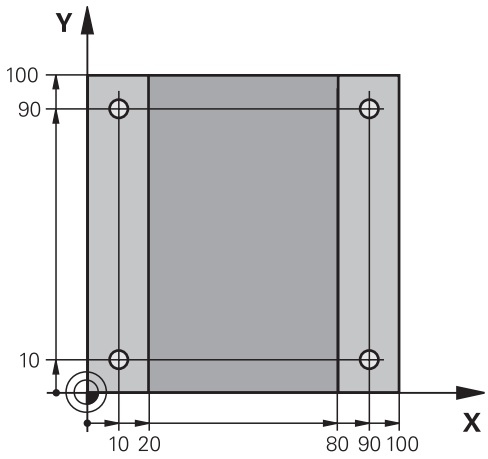
> The control saves the positioning block and ends the NC program.

Further information on this topic

- Complete example with NC blocks
  Further information: "Example: Linear movements and chamfers with Cartesian coordinates", Page 177
- Creating a new NC program
  Further information: "Creating and entering NC programs", Page 100
- Approaching/departing contours
  Further information: "Approaching and departing a contour", Page 155
- Programming contours
  Further information: "Overview of path functions", Page 166
- Tool radius compensation
  Further information: "Tool radius compensation", Page 146
- Miscellaneous functions M
  Further information: "Miscellaneous functions for program run inspection, spindle and coolant ", Page 237

Creating a cycle program

Suppose that you are tasked with drilling the holes shown to the right with a standard drilling cycle (depth: 20 mm). You have already defined the workpiece blank.



Call the tool

TOOL CALL
- Press the **TOOL CALL** key
- Enter the tool data, e.g., tool number 5

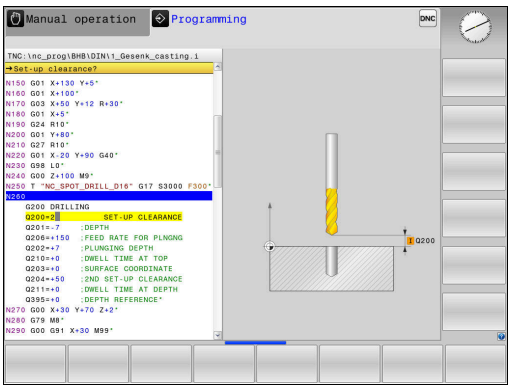
ENT
- Press the **ENT** key

ENT
- Confirm the tool axis **G17** with the **ENT** key
- Enter the spindle speed (e.g., 4500)

END
- Press the **END** key

> The control completes the NC block.

Retract the tool



L
- Press the **L** key

←
- Press the left arrow key

> The control opens the input range for G functions.

G00
- Press the **G00** soft key

> The control executes the NC block in rapid traverse.

Alternative:

G
- Press the **G** key on the alphanumeric keyboard
- Enter **0**

ENT
- Press the **ENT** key

> The control executes the NC block in rapid traverse.

G90
- Press the **G90** soft key

> The control processes the entered dimensions as absolute dimensions.

Z
- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT
- Press the **ENT** key

G40
- Press the **G40** soft key

> The control does not activate radius compensation.

- If needed, enter a miscellaneous function **M**, such as **M3**, turn on spindle

END
- Press the **END** key

> The control saves the positioning block.

Define the cycle

- Press the **CYCL DEF** key

- Press the **DRILLING/ THREAD** soft key

- Press the **200** soft key
- > The control starts the dialog for cycle definition.
- Enter the cycle parameters

- Confirm each entry with the **ENT** key
- > The control displays a graphic illustrating the respective cycle parameter.

Call the cycle at the machining positions

- Press the **G** key on the alphanumeric keyboard
- Enter **0**
- > The control executes the NC block in rapid traverse.

- Press the **ENT** key
- Enter the coordinates of the first position

- Press the **ENT** key

- Press the **G40** soft key
- > The control does not activate radius compensation.
- Enter **M99**, cycle call

- Press the **END** key
- > The control saves the NC block.

- Press the **G** key
- Enter **0**

- Press the **ENT** key
- Enter the coordinates of the second position

- Press the **ENT** key

- Press the **G40** soft key
- > The control does not activate radius compensation.
- Enter **M99**, cycle call

- Press the **END** key
- > The control saves the NC block.
- Program all positions and call them with **M99**

Retract the tool

G

- Press the **G** key on the alphanumeric keyboard
- Enter **0**

ENT

- Press the **ENT** key

> The control executes the NC block in rapid traverse.

Z

- Press the **Z** axis key
- Enter the retraction value (e.g., 250 mm)

ENT

- Press the **ENT** key

G40

- Press the **G40** soft key

> The control does not activate radius compensation.

- Enter a miscellaneous function **M**, such as **M30** for program end

END

- Press the **END** key

> The control saves the positioning block and ends the NC program.

Example

%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	Workpiece blank definition
N20 G31 X+100 Y+100 Z+0*	
N30 T5 G17 S4500*	Tool call
N40 G00 G90 Z+250 G40 M3*	Retract the tool; turn on spindle
N50 G200 DRILLING	Define the cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
N60 G00 X+10 Y+10 G40 M8 M99*	Turn on coolant; call cycle
N70 G00 X+10 Y+90 G40 M99*	Call the cycle
N80 G00 X+90 Y+10 G40 M99*	Call the cycle
N90 G00 X+90 Y+90 G40 M99*	Call the cycle
N100 G00 Z+250 M30*	Retract the tool, end program
N99999999 %C200 G71 *	

Further information on this topic

- Creating a new NC program
  Further information: "Creating and entering NC programs", Page 100
- Cycle programming
  Further information: User's Manual for **Programming of Machining Cycles**

3

Fundamentals

3.1 The TNC 320

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional milling and drilling operations right at the machine in easy-to-use Klartext conversational language. They are designed for milling, drilling, and boring machines, as well as for machining centers, with up to 6 axes. You can also change the angular position of the spindle under program control.

The keyboard and screen layout are clearly arranged in such a way that the functions are fast and easy to use.

HEIDENHAIN Klartext and ISO

HEIDENHAIN Klartext, the dialog-guided programming language for workshops, is an especially easy method of writing programs. Programming graphics illustrate the individual machining steps for programming the contour. If no NC-dimensioned drawing is available, then the FK free contour programming will help. Workpiece machining can be graphically simulated either during a test run or during a program run.

It is also possible to program in ISO format.

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

Compatibility

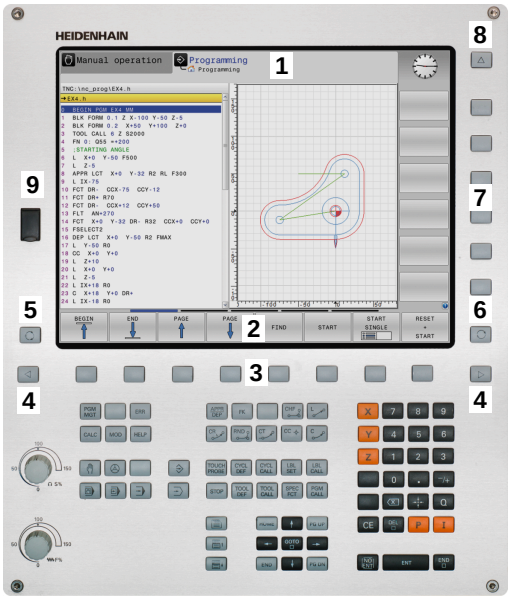
NC programs created on HEIDENHAIN contouring controls (starting from the TNC 150 B) may not always run on the TNC 320. If the NC blocks contain invalid elements, the control will mark these as ERROR blocks or with error messages when the file is opened.

3.2 Visual display unit and operating panel

Display screen

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

The figure at right shows the keys and controls on the VDU:



1 Header

When the control is on, the screen displays the selected operating modes in the header: The machine operating mode at left and the programming mode at right. The currently active mode is displayed in the larger field of the header, where the dialog prompts and messages also appear (exception: if the control only displays graphics).

2 Soft keys

In the footer the control indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The thin bars immediately above the soft-key row indicate the number of soft-key rows that can be called with the keys to the right and left that are used to switch the soft keys. The bar representing the active soft-key row is blue

3 Soft-key selection keys

4 Keys for switching the soft keys

5 Setting the screen layout

6 Key for switchover between machine operating modes, programming modes, and a third desktop

7 Soft-key selection keys for machine manufacturer's

8 Keys for switching the soft keys for machine manufacturer's

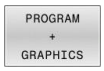
9 USB connection

Setting the screen layout

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

Setting up the screen layout:

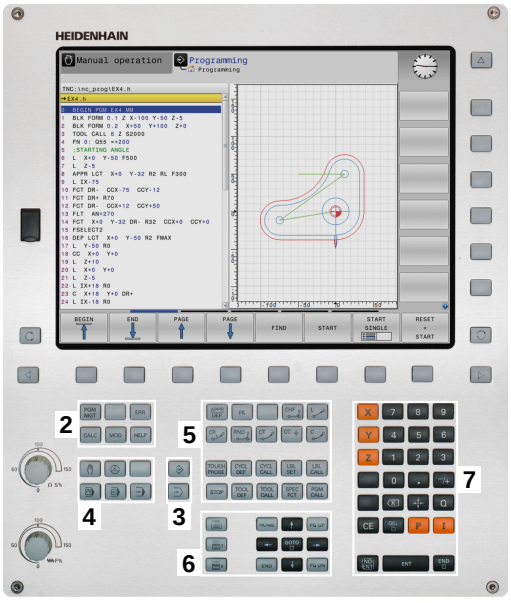
- Press the **screen layout** key: The soft-key row shows the available layout options
  Further information: "Modes of operation", Page 84



- Select the desired screen layout with a soft key

Operating panel

The TNC 320 can be delivered with an integrated operating panel. As an alternative, the TNC 320 is also available with a separate monitor and an external operating panel with alphabetic keyboard.



1 Alphabetic keyboard for entering texts and file names, as well as for ISO programming

2
- File manager
- Calculator
- MOD function
- HELP function
- Show error messages
- Toggle between the operating modes

3 Programming modes

4 Machine operating modes

5 Initiating programming dialogs

6 Navigation keys and **GOTO** jump command

7 Numerical input and axis selection

10 Machine operating panel
More information: Machine manual

The functions of the individual keys are described on the inside front cover.

Refer to your machine manual.

Some machine manufacturers do not use the standard HEIDENHAIN operating panel.

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

Cleaning

Switch the control off before cleaning the keyboard unit.

NOTICE

Caution: danger 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 cleaning agents 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 when it stops functioning properly.

To clean a trackball (if needed):

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

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

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

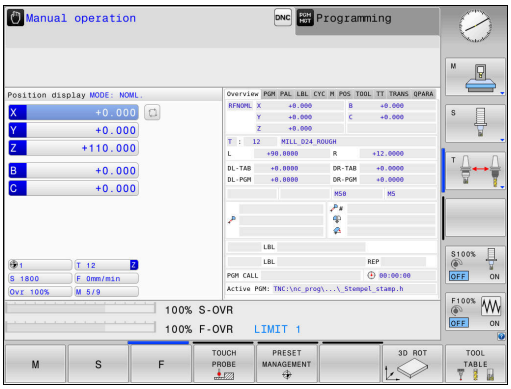
3.3 Modes of operation

Manual Operation and El. Handwheel

In the **Manual operation** mode of operation, you can set up the machine. You can position the machine axes manually or incrementally, and you can set presets.

If option 8 is active, you can tilt the working plane.

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

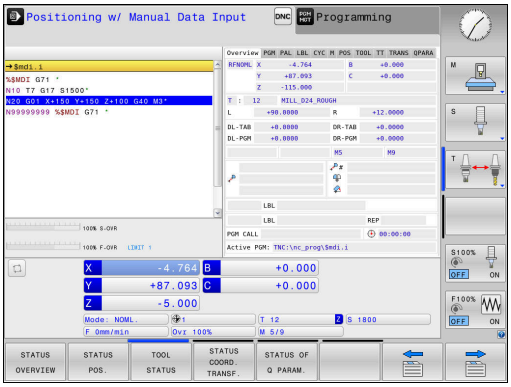


Soft keys for selecting the screen layout

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

Positioning with Manual Data Input

This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

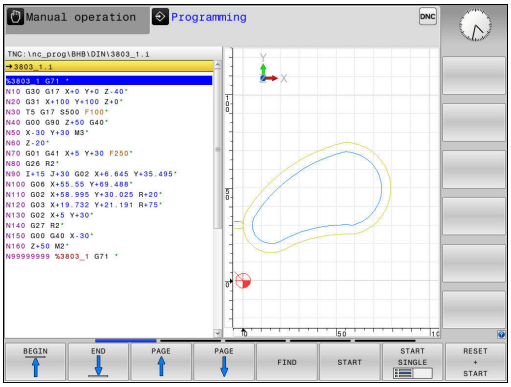


Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece

Programming

In this mode of operation you create NC programs. The FK free programming feature, the various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

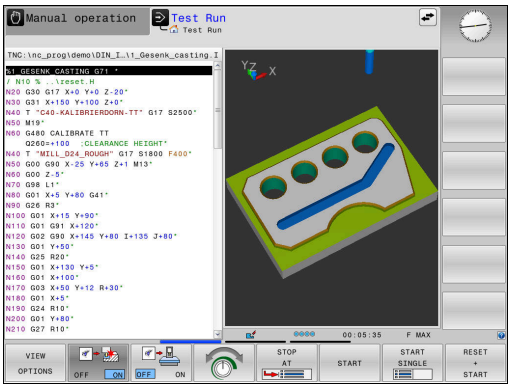


Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: program structure
PROGRAM + GRAPHICS	Left: NC program, right: programming graphics

Test Run

In the **Test Run** operating mode, the control simulates NC programs and program sections in order to check them for errors, such as geometrical incompatibilities, missing or incorrect data within the NC program, or violations of the working space. This simulation is supported graphically in different display modes.



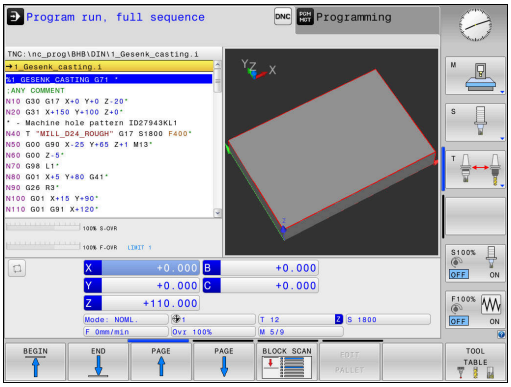
Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece

Program Run, Full Sequence and Program Run, Single Block

In the **Program Run Full Sequence** operating mode, the control runs an NC program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the **Program Run Single Block** operating mode, you execute each NC block separately by pressing the **NC Start** key. With point pattern cycles and **CYCL CALL PAT**, the control stops after each point. The workpiece blank definition will be interpreted as a separate NC block.



Soft keys for selecting the screen layout

Soft key	Window
PGM	NC program
PROGRAM + SECTS	Left: NC program, right: structure
PROGRAM + STATUS	Left: NC program, right: status display
PROGRAM + WORKPIECE	Left: NC program, right: workpiece
WORKPIECE	Workpiece

3.4 NC fundamentals

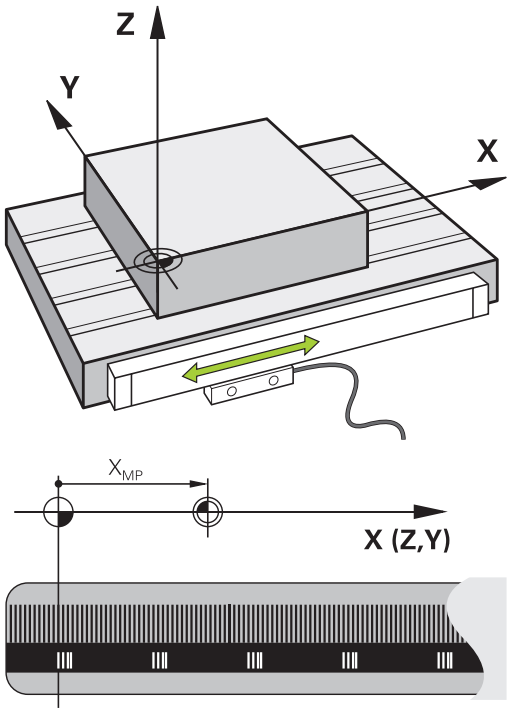
Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.

When a machine axis moves, the corresponding position encoder generates an electrical signal. The control evaluates this signal and calculates the precise actual position of the machine axis.

If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this assignment, incremental position encoders are provided with reference marks. When a reference mark is crossed over, a signal identifying a machine-based reference point is transmitted to the control. This enables the control to re-establish the assignment of the displayed position to the current machine position. For linear encoders with distance-coded reference marks, the machine axes need to move by no more than 20 mm, for angle encoders by no more than 20°.

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.

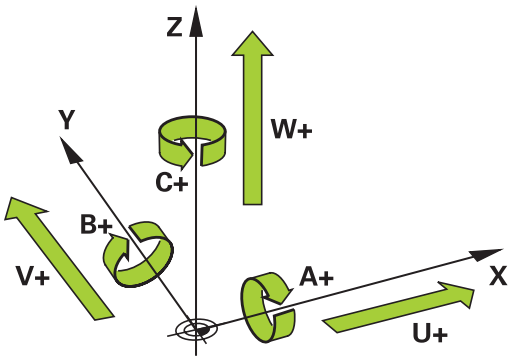


Programmable axes

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

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

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



Refer to your machine manual.

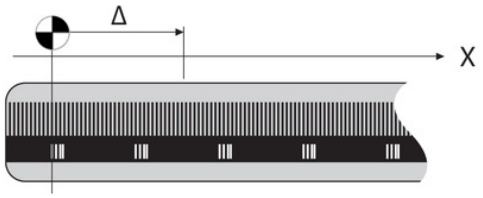
The number, designation and assignment of the programmable axes depend on the machine.

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

Reference systems

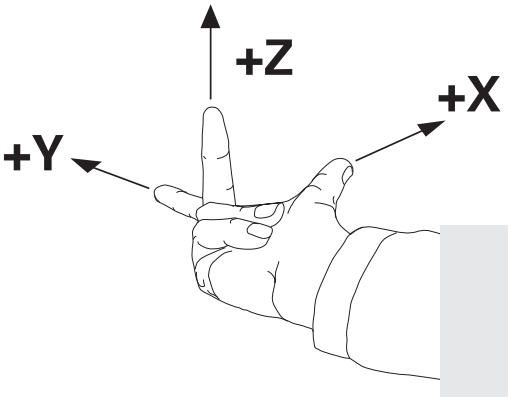
For the control to move an axis in accordance with a defined path, it requires a **reference system**.

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



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

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

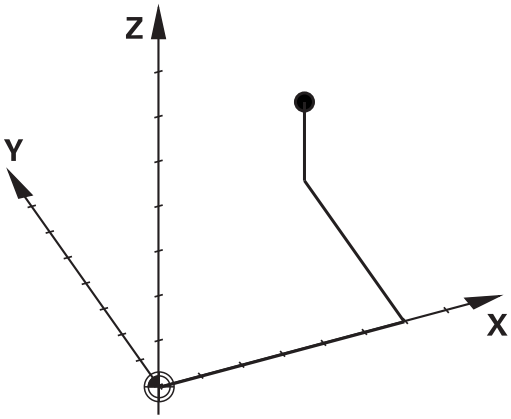


According to the right-hand rule the fingertips point in the positive directions of the three principal axes.

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

In order, for example, for the control to always perform a tool change at the same position, as well as always execute a machining operation referenced to the current workpiece position, the control must be able to differentiate between different reference systems.

The control differentiates between the following reference systems:



- Machine coordinate system M-CS:
  Machine **C**oordinate **S**ystem
- Basic coordinate system B-CS:
  Basic **C**oordinate **S**ystem
- Workpiece coordinate system W-CS:
  Workpiece **C**oordinate **S**ystem
- Working plane coordinate system WPL-CS:
  Working **Pl**ane **C**oordinate **S**ystem
- Input coordinate system I-CS:
  Input **C**oordinate **S**ystem
- Tool coordinate system T-CS:
  Tool **C**oordinate **S**ystem

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

The machine coordinate system is the reference system.

Machine coordinate system M-CS

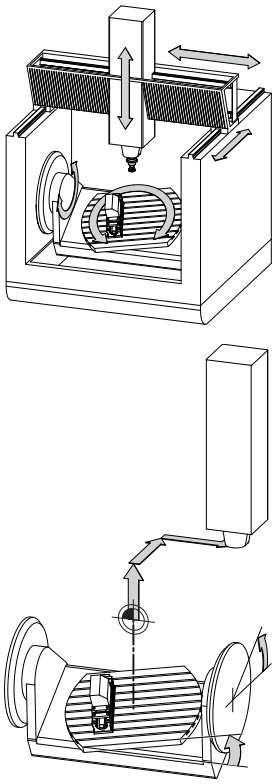
The machine coordinate system corresponds to the description of kinematics and therefore to the actual mechanical design of the machine tool.

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

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

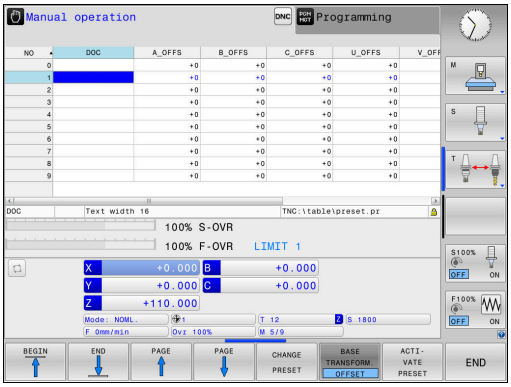
The position of the coordinate origin, the machine datum, is defined by the machine manufacturer during machine configuration. The values in the machine configuration define the zero positions of the encoders and the corresponding machine axes. The machine datum does not necessarily have to be located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

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



Machine datum (MZP)

Soft key	Application
BASE TRANSFORM. OFFSET	The user can define shifts in the machine coordinate system according to the specific axis with use of the **OFFSET** values of the preset table.
DATUM TABLE	The user can define shifts of the rotary and parallel axes according to the specific axis with use of the datum table.
TRANSFOR- MATIONS	The user can define shifts of the rotary and parallel axes according to the specific axis with use of the **TRANS DATUM** function.



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

Further information: User's Manual for **Setup, Testing and Running NC Programs**

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

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

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

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

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

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

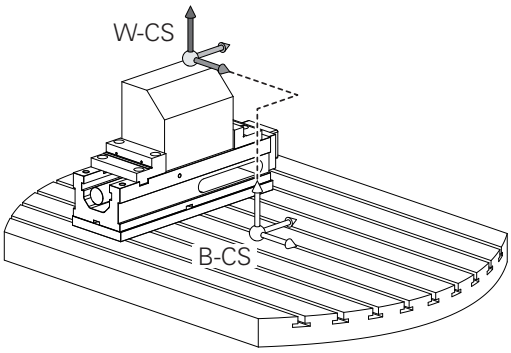
Basic coordinate system B-CS

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

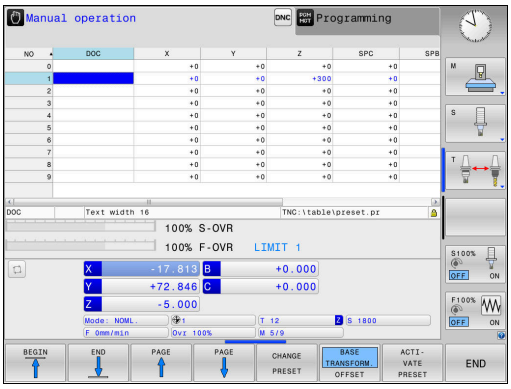
The orientation of the basic coordinate system in most cases corresponds to that of the machine coordinate system. There may be exceptions to this if a machine manufacturer uses additional kinematic transformations.

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

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



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



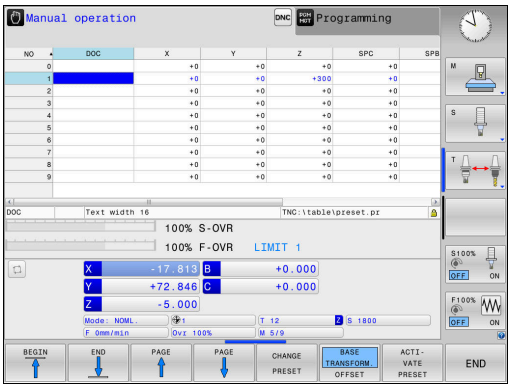
The machine manufacturer configures the **BASE TRANSFORM.** columns of the preset management in accordance with the machine.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Workpiece coordinate system W-CS

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

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



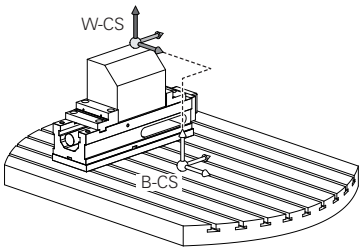
Soft key	Application
BASE TRANSFORM. OFFSET	The user determines the position and orientation of the workpiece coordinate system by using a 3-D touch probe, for example. The control saves the values determined with respect to the basic coordinate system as **BASE TRANSFORM.** values in the preset management.

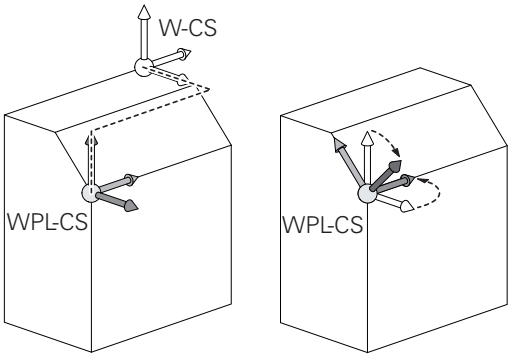
Further information: User's Manual for **Setup, Testing and Running NC Programs**

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

Transformations in the workpiece coordinate system:

- **3D ROT** functions
  - **PLANE** functions
  - Cycle **G80 WORKING PLANE**
- Axes **X**, **Y**, **Z** of Cycle **G53/G54 DATUM SHIFT** or function **TRANS DATUM** (shifting **before** tilting the working plane)
- Columns **X**, **Y**, **Z** of the datum table (shifting **before** tilting the working plane)
- Cycle **G28 MIRRORING** or **TRANS MIRROR** (mirroring **before** tilting the working plane)





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

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

Programming notes:

- Transformations (mirroring and shifting) that are programmed before the **PLANE** functions (except for **PLANE AXIAL**) will change the position of the tilt datum (origin of the working plane coordinate system WPL-CS) and the orientation of the rotary axes
  - If you just program a shift, then only the position of the tilt datum will change
  - If you just program mirroring, then only the orientation of the rotary axes will change
- When used in conjunction with **PLANE AXIAL** and Cycle **G80**, the programmed transformations (mirroring, rotation, and scaling) do not affect the position of the tilt datum or the orientation of the rotary axes

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

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

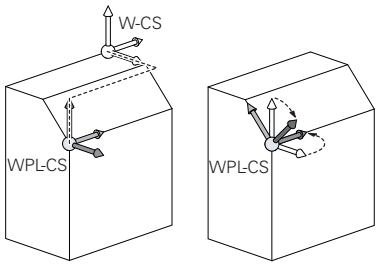
Other transformations are of course possible in the working plane coordinate system

Further information: "Working plane coordinate system WPL-CS", Page 94

Working plane coordinate system WPL-CS

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

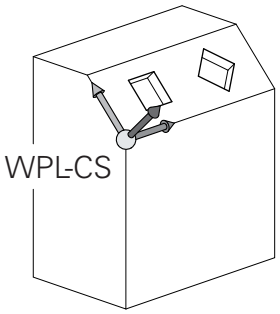
The position and orientation of the working plane coordinate system depend on the active transformations in the workpiece coordinate system.



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

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

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

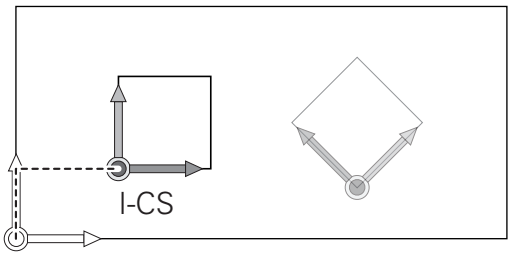


Transformations in the working plane coordinate system:

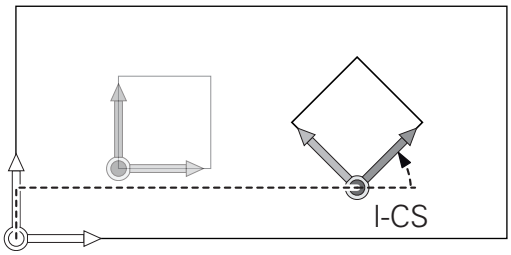
- Axes **X**, **Y**, **Z** of Cycle **G53/G54 DATUM SHIFT** or of the **TRANS DATUM** function
- Cycle **G28 MIRRORING** or **TRANS MIRROR** function
- Cycle **G73 ROTATION** or **TRANS ROTATION** function
- Cycle **G72 SCALING FACTOR** or **TRANS SCALE** function
- **PLANE RELATIVE**

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

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



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



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

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

Input coordinate system I-CS

The input coordinate system is a 3-D Cartesian coordinate system.

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

> Without active transformations in the working plane coordinate system, the position and orientation of the input coordinate system and working plane coordinate system are identical.
>
> In addition, there are no transformations in the workpiece coordinate system on 3-axis machine tools or with pure 3-axis machining. The **BASE TRANSFORM.** values of the active line of the preset table have a direct effect on the input coordinate system with this assumption.

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

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

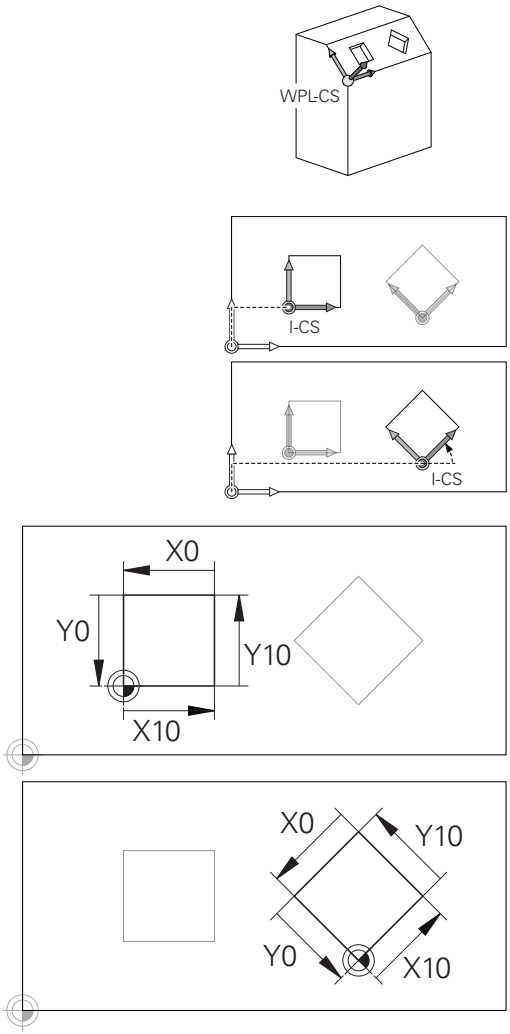
Positioning blocks in input coordinate system:

- Paraxial positioning blocks
- Positioning blocks with Cartesian or polar coordinates
- Cycles

```
N70  X+48*
```

```
N70  G01 X+48 Y+102 Z-1.5 G40*
```

> Orientation of the tool coordinate system can be performed in various reference systems.
>
> **Further information:** "Tool coordinate system T-CS", Page 96



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

Tool coordinate system T-CS

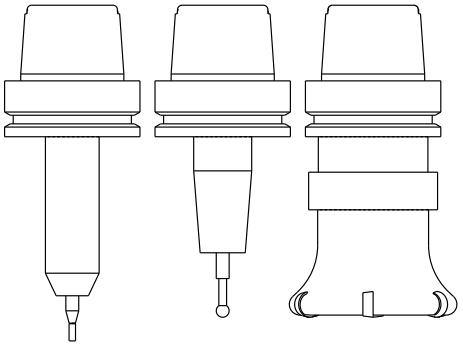
The tool coordinate system is a 3-D Cartesian coordinate system. Its coordinate origin is the tool reference point. The values of the tool table, **L** and **R** with milling tools and **ZL**, **XL** and **YL** with turning tools, reference this point.

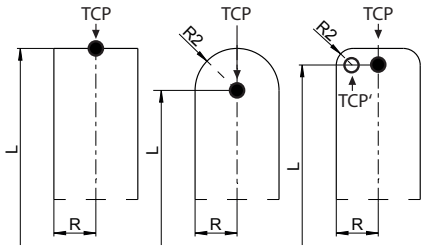
Further information: User's Manual for **Setup, Testing and Running NC Programs**

In accordance with the values from the tool table, the coordinate origin of the tool coordinate system is shifted to the tool center point (TCP).

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

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





Designation of the axes on milling machines

The X, Y and Z axes on your milling machine are also referred to as tool axis, principal axis (1st axis) and secondary axis (2nd axis). The assignment of the tool axis is decisive for the assignment of the principal and secondary axes.

Tool axis	Principal axis	Secondary axis
X	Y	Z
Y	Z	X
Z	X	Y

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

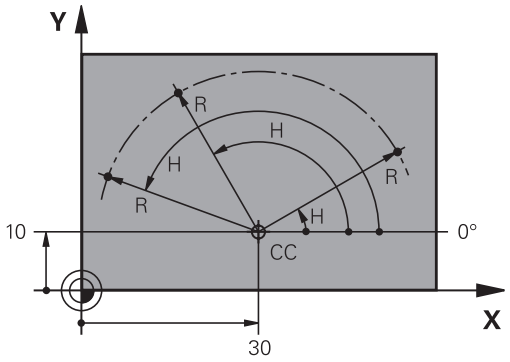
Restricted use of the tool axes **X** and **Y** is possible when prepared and configured by the machine manufacturer.

Polar coordinates

If the production drawing is dimensioned in Cartesian coordinates, you write the NC program using Cartesian coordinates. For parts containing circular arcs or angles, it is often simpler to give the dimensions in polar coordinates.

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

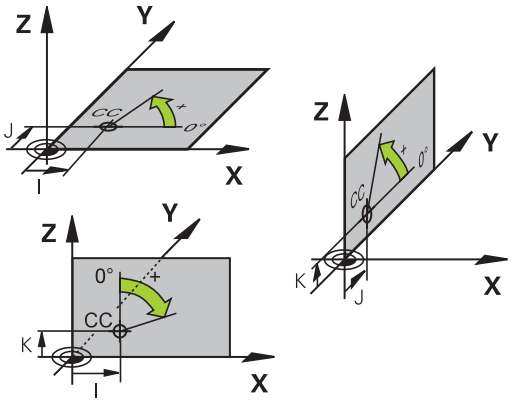
- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the value of the angle between the angle reference axis and the line that connects the circle center CC with the position.



Setting the pole and the angle reference axis

The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle H.

Coordinates of the pole (plane)	Angle reference axis
X/Y	+X
Y/Z	+Y
Z/X	+Z



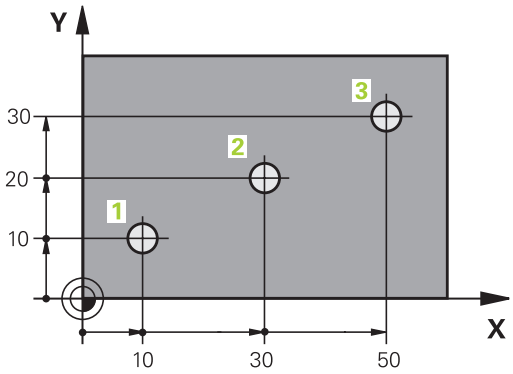
Absolute and incremental workpiece positions

Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is unambiguously defined by its absolute coordinates.

Example 1: Holes dimensioned in absolute coordinates

Hole 1	Hole 2	Hole 3
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm

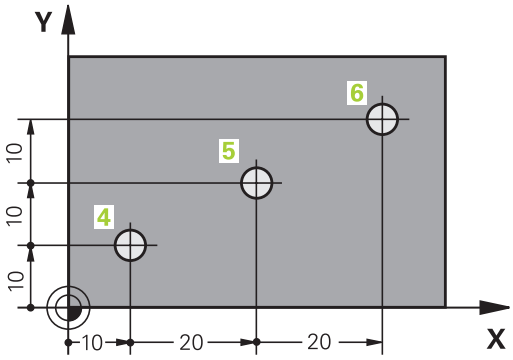


Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When you write an NC program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. This is why they are also referred to as chain dimensions.

To program a position in incremental coordinates, enter the G91 function before the axis.

Example 2: Holes dimensioned in incremental coordinates



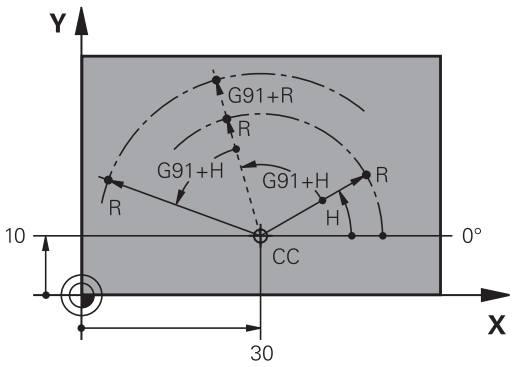
Absolute coordinates of hole 4
X = 10 mm
Y = 10 mm

Hole 5, with respect to 4	Hole 6, with respect to 5
G91 X = 20 mm	G91 X = 20 mm
G91 Y = 10 mm	G91 Y = 10 mm

Absolute and incremental polar coordinates

Absolute coordinates always refer to the pole and the angle reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.



Selecting the preset

A production drawing specifies a certain form element of the workpiece (usually a corner) as the absolute reference point (datum). When setting the preset, first align the workpiece along the machine axes, and move the tool to a known position in each axis relative to the workpiece. For each position, set the display of the control either to zero or to a known position value. You thereby assign the workpiece to the reference system that is applicable for the control's display or your NC program.

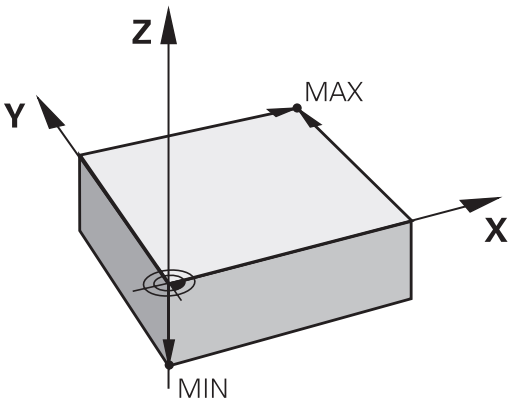
If the production drawing is dimensioned in relative reference points, simply use the coordinate transformation cycles.

Further information: User's Manual for **Programming of Machining Cycles**

If the production drawing is not dimensioned for NC programming, then select a position or corner of the workpiece as a reference point from which the dimensions of the remaining workpiece positions can be determined.

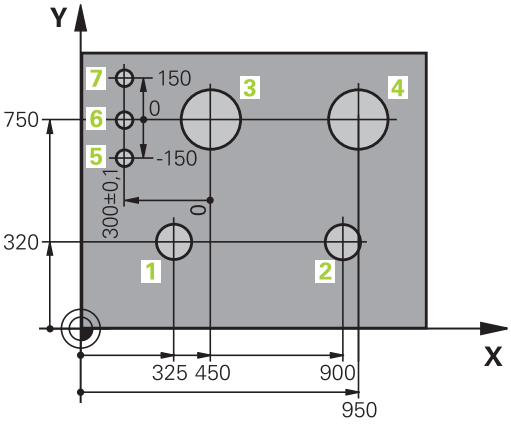
A particularly convenient way of setting the presets is with a 3-D touch probe from HEIDENHAIN.

Further information: User's Manual for **Setup, Testing and Running NC Programs**



Example

The workpiece drawing shows holes (1 to 4), whose dimensions are shown relative to an absolute preset with the coordinates X=0 Y=0. The coordinates of holes 5 to 7 refer to the relative preset with the absolute coordinates X=450 Y=750. A **Datum shift** allows you to temporarily shift the datum to the position X = 450, Y = 750 in order for you to program the holes (5 to 7) without further calculations.



3.5 Creating and entering NC programs

Structure of an NC program in ISO format

An NC program consists of a series of NC blocks. The illustration at right shows the elements of an NC block.

The control numbers the NC blocks of an NC program automatically, depending on the setting in the machine parameter **blockIncrement** (105409). The **blockIncrement** machine parameter (105409) defines the block number increment.

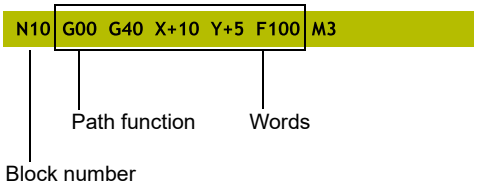
The first NC block of an NC program is identified by **%**, the program name, and the active unit of measure.

The subsequent NC blocks contain information on

- The workpiece blank
- Tool calls
- Approaching a safe position
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

The last block of a program is identified by **N99999999**, the program name, and the active unit of measure.

NC block



NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. There is danger of collision during the approach movement after a tool change!

- If necessary, program an additional safe auxiliary position

Defining the workpiece blank: G30/G31

Immediately after creating a new NC program, you define an unmachined workpiece blank. If you wish to define the blank at a later stage, then press the **SPEC FCT** key, the **PROGRAM DEFAULTS** soft key, and then the **BLK FORM** soft key. The control needs this definition for its graphical simulations.

- You only need to define the workpiece blank if you wish to run a graphic test for the NC program.
- 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 **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: User's Manual for **Setup, Testing and Running NC Programs**

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 can depict various types of blank forms:

Soft key	Function
	Define a rectangular blank
	Define a cylindrical blank
	Define a rotationally symmetric blank of any shape
	Load STL file as workpiece blank Optionally load an additional STL file as finished part

Rectangular blank

The sides of the cuboid lie parallel to the X, Y and Z axes. This blank is defined by two of its corner points:

- MIN point G30: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point G31: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values

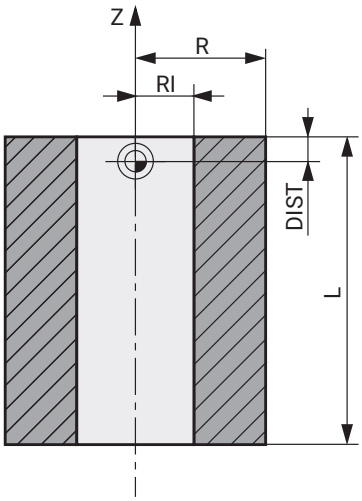
Example

%NEW G71 *	Program begin, name, unit of measure
N10 G30 G17 X+0 Y+0 Z-40*	Spindle axis, MIN point coordinates
N20 G31 X+100 Y+100 Z+0*	MAX point coordinates
N99999999 %NEW G71 *	Program end, name, unit of measure

Cylindrical blank

The cylindrical blank form is defined by the dimensions of the cylinder:

- X, Y or Z: Rotation axis
- D, R: Diameter or radius of the cylinder (with positive algebraic sign)
- L: Length of the cylinder (with positive algebraic sign)
- DIST: Shifting along the rotational axis
- DI, RI: Inside diameter or inside radius for a hollow cylinder



> The parameters **DIST** and **RI** or **DI** are optional and need not be programmed.

Example

%NEW G71 *	Program begin, name, unit of measure
N10 BLK FORM CYLINDER Z R50 L105 DIST+5 RI10*	Spindle axis, radius, length, distance, inside radius
N99999999 %NEW G71 *	Program end, name, unit of measure

Rotationally symmetric blank of any shape

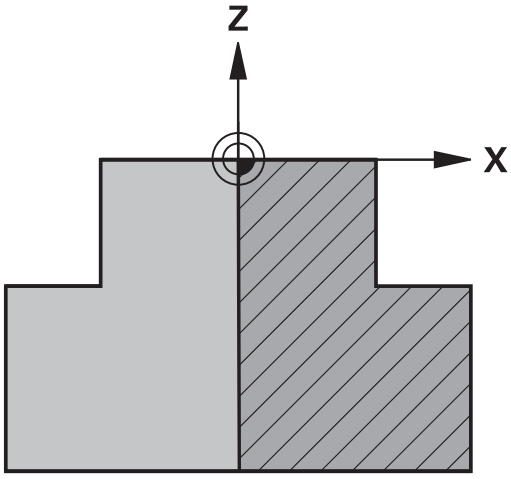
You define the contour of the rotationally symmetric blank in a subprogram. Use X, Y or Z as the rotation axis.

In the workpiece blank definition you refer to the contour description:

- DIM_D, DIM-R: Diameter or radius of the rotationally symmetrical blank form
- LBL: Subprogram with the contour description

The contour description may contain negative values in the rotation axis but only positive values in the reference axis. The contour must be closed, i.e. the contour beginning corresponds to the contour end.

If you define a rotationally symmetric blank with incremental coordinates, the dimensions are then independent of the diameter programming.



> The subprogram can be designated with a number, an alphanumeric name, or a QS parameter.

Example

%NEW G71 *	Program begin, name, unit of measure
N10 BLK FORM ROTATION Z DIM_R LBL1*	Spindle axis, manner of interpretation, subprogram number
N20 M30*	End of main program
N30 G98 L1*	Subprogram start
N40 G01 X+0 Z+1*	Starting point of contour
N50 G01 X+50*	Programming in the positive direction of the principal axis
N60 G01 Z-20*	
N70 G01 X+70*	
N80 G01 Z-100*	
N90 G01 X+0*	
N100 G01 Z+1*	Contour end
N110 G98 L0 *	End of subprogram
N99999999 %NEW G71 *	Program end, name, unit of measure

STL files as workpiece blank and optional finished part

Integrating STL files as workpiece blank and finished part is particularly convenient in combination with CAM programs, where the required 3-D models are available in addition to the NC program.

Missing 3D models, such as semifinished parts with multiple, separate machining steps, can be created directly on the control with the **EXPORT WORKPIECE** soft key in the **Test Run** operating mode.

The file size depends on the complexity of the geometry.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Please note that the STL files are limited in respect of the number of permitted triangles:

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

Binary files are loaded faster by the control.

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 workpiece blank definition you refer to the desired STL files by indicating the path. Use the **SELECT FILE** soft key if you want the control to take over the path information automatically.

If you do not wish to load a finished part, then close the dialog after the workpiece blank definition.

The path of the STL file can also be entered directly as text or with a QS parameter.

Example

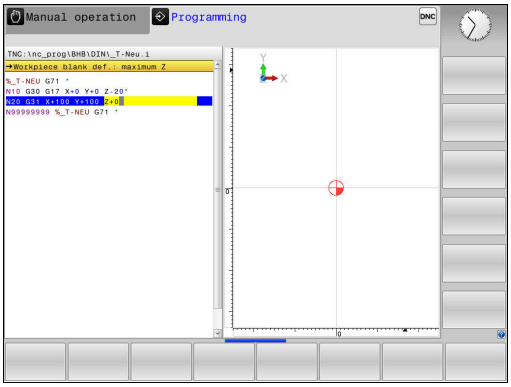
%NEU G71 *	Program beginning, name, unit of measure
N10 BLK FORM FILE "TNC:\...stl" TARGET "TNC:\...stl"*	Indication of path to the workpiece blank, path to the optional finished part
N99999999 %NEU G71 *	Program end, name, unit of measure

If the NC program and the 3-D models are in a folder or in a defined folder structure, relative paths make it easier to move the files later.

Further information: "Programming notes", Page 261

Creating a new NC program

An NC program is always entered in **Programming** mode. Example for creating a program:



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

PGM MGT

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

Select the directory in which you wish to store the new NC program:

FILE NAME = NEW.I

ENT

- Enter the new program name
- Press the **ENT** key

MM

- Select the unit of measure: Press the **MM** or **INCH** soft key
- The control switches the screen layout and initiates the dialog for defining the **BLK FORM** (workpiece blank).

- Select a rectangular workpiece blank: Press the soft key for a rectangular blank form

Working plane in graphic: XY

G17

- Enter the spindle axis, e.g. **G17**

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.

Workpiece blank def.: Minimum

ENT

- Enter in sequence the X, Y and Z coordinates of the MIN point and confirm each of your entries with the **ENT** key

Workpiece blank def.: Maximum

ENT

- Enter in sequence the X, Y and Z coordinates of the MAX point and confirm each of your entries with the **ENT** key

Example

%NEW G71 *	Program begin, name, unit of measure
N10 G30 G17 X+0 Y+0 Z-40*	Spindle axis, MIN point coordinates
N20 G31 X+100 Y+100 Z+0*	MAX point coordinates
N99999999 %NEW G71 *	Program end, name, unit of measure

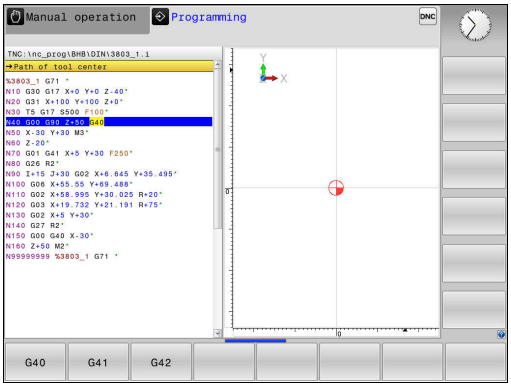
The control automatically generates the first and last NC blocks of the NC program.

If you do not wish to define a workpiece blank, then cancel the dialog at **Working plane in graphic: XY** using the **DEL** key!

Programming tool movements in ISO

To program an NC block, pressing the **SPEC FCT** key. Press the **PROGRAM FUNCTIONS** soft key, and then the **DIN/ISO** soft key. You can also use the gray path function keys to get the corresponding G code.

If you enter ISO functions on a keyboard connected through a USB port, make sure that capitalization is active.



Example of a positioning block

G

ENT

- Press the **G** key
- Enter **1** and press the **ENT** key to open the NC block

COORDINATES ?

X

- **10** (enter the target coordinate for the X axis)

Y

- **20** (enter the target coordinate for the Y axis)

ENT

- Go to the next question with **ENT**.

Path of tool center

G

- Enter **40** and confirm with the **ENT** key to traverse without tool radius compensation

Alternative:

G41

G42

- Move the tool to the left or to the right of the programmed contour: Press the **G41** or **G42** soft key

Feed rate F=?

- **100** (enter a feed rate of 100 mm/min for this path contour)

ENT

- Go to the next question with **ENT**.

MISCELLANEOUS FUNCTION M ?

- **3** (enter the miscellaneous function **M3 Spindle on**)

END

- With the **END** key, the control ends this dialog.

Example

```
N30 G01  G40 X+10  Y+5 F100 M3*
```

Actual position capture

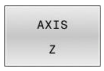
The control enables you to transfer the current tool position into the NC program, for example during

- Positioning-block programming
- Cycle programming

To transfer the correct position values:

- Place the input box at the position in the NC block where you want to insert a position value

- Select the actual-position-capture function
- > In the soft-key row the control displays the axes whose positions can be transferred.



- Select the axis
- > The control writes the current position of the selected axis into the active input box.

In the working plane the control always captures the coordinates of the tool center, even though tool radius compensation is active.

The control takes the active tool length compensation into account and always captures the coordinate of the tool tip in the tool axis.

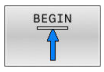
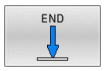
The control keeps the soft-key row for axis selection active until the **actual position capture** key is pressed again. This behavior remains in effect even if you save the current NC block or open a new NC block with a path function key. If you have to choose an input alternative via soft key (e.g. for radius compensation), then the control closes the soft-key row for axis selection.

The **Actual position capture** function is not permitted when the **Tilt working plane** function is active.

Editing an NC program

You cannot edit the active NC program while it is being run.

While you are creating or editing an NC program, you can select any desired line in the NC program or individual words in an NC block with the arrow keys or the soft keys:

Soft key / key	Function
	Go to previous page
	Go to next page
	Go to beginning of program
	Go to end of program
	Change the position of the current NC block on the screen. Press this soft key to display additional NC blocks that are programmed before the current NC block No function if the NC program is fully visible on the screen
	Change the position of the current NC block on the screen. Press this soft key to display additional NC blocks that are programmed after the current NC block No function if the NC program is fully visible on the screen
	Move from one NC block to the next NC block
	Select individual words in an NC block
	Select a specific NC block **Further information:** "Using the GOTO key", Page 202

Soft key / key	Function
CE	■ Set the selected word to zero ■ Erase an incorrect number ■ Delete the (clearable) error message
NO ENT	Delete the selected word
DEL	■ Delete the selected NC block ■ Erase cycles and program sections
INSERT LAST NC BLOCK	Insert the NC block that you last edited or deleted

Inserting an NC block at any desired location

- Select the NC block after which you want to insert a new NC block
- Initiate the dialog

Save changes

The control normally saves changes automatically if you switch the operating mode or if you select the file manager. If you deliberately want to save changes to the NC program:

- Select the soft-key row with the saving functions

STORE

- Press the **STORE** soft key
- The control saves all changes made since the last time you saved the program.

Saving an NC program to a new file

You can save the contents of the currently active NC program under a different program name. Proceed as follows:

- Select the soft-key row with the saving functions

SAVE AS

- Press the **SAVE AS** soft key
- The control opens a window in which you can enter the directory and the new file name.
- Select the target directory if required with the **SWITCH** soft key
- Enter the file name
- Confirm with the **OK** soft key or the **ENT** key, or abort the process by pressing the **CANCEL** soft key

The file saved with **SAVE AS** can also be found in the file manager by pressing the **LAST FILES** soft key.

Undoing changes

You can undo all changes made since the last time you saved the program. Proceed as follows:

- Select the soft-key row with the saving functions

- Press the **CANCEL CHANGE** soft key
- > The control opens a window in which you can confirm or cancel this action.
- Confirm with the **YES** soft key or cancel with the **ENT** key, or press the **NO** soft key to abort

Editing and inserting words

- Select a word in an NC block
- Overwrite it with the new value
- > The dialog is available while the word is highlighted.
- To accept the change, press the **END** key

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different NC blocks

- Select a word in an NC block: Press the arrow key repeatedly until the desired word is highlighted

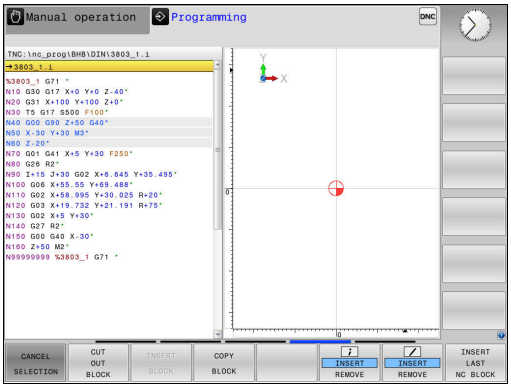
- Select an NC block with the arrow keys
  - Arrow down: search forwards
  - Arrow up: search backwards

The word that is highlighted in the new NC block is the same as the one you selected previously.

If you start a search in a very long NC program, the control shows a progress indicator. You can cancel the search at any time, if necessary.

Marking, copying, cutting and inserting program sections

The control provides the following functions for copying program sections within an NC program or into another NC program:



Soft key	Function
SELECT BLOCK	Switch the marking function on
CANCEL SELECTION	Switch the marking function off
CUT OUT BLOCK	Cut the marked block
INSERT BLOCK	Insert the block that is stored in the buffer memory
COPY BLOCK	Copy the marked block

To copy a program section:

- Select the soft key row containing the marking functions
- Select the first NC block of the section you wish to copy
- Mark the first NC block: Press the **SELECT BLOCK** soft key.
- The control highlights the block in color and displays the **CANCEL SELECTION** soft key.
- Place the cursor on the last NC block of the program section you wish to copy or cut.
- The control shows the marked NC blocks in a different color You can end the marking function at any time by pressing the **CANCEL SELECTION** soft key.
- Copy the selected program section: Press the **COPY BLOCK** soft key. Cut the selected program section: Press the **CUT OUT BLOCK** soft key.
- The control stores the selected block.

> If you want to transfer a program section to another NC program, you now need to select the desired NC program in the file manager.

- Use the arrow keys to select the NC block after which you want to insert the copied/cut section
- Insert the saved program section: Press the **INSERT BLOCK** soft key
- End the marking function: Press the **CANCEL SELECTION** soft key

The control's search function

The search function of the control enables you to search for any text within an NC program and replace it by a new text, if required.

Finding any text

FIND

- ▶ Select the search function
- > The control superimposes the search window and displays the available search functions in the soft-key row.
- ▶ Enter the text to be searched for, e.g.: **TOOL**
- ▶ Select forwards search or backwards search

FIND

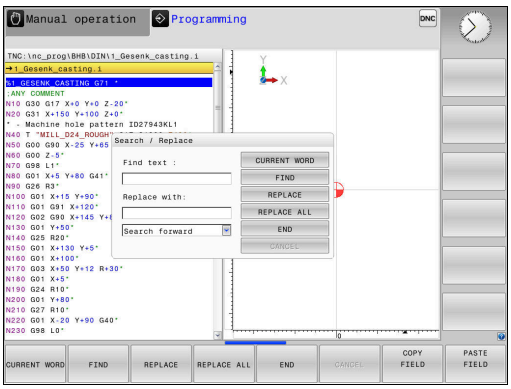
- ▶ Start the search process
- > The control moves to the next NC block containing the text you are searching for

FIND

- ▶ Repeat the search process
- > The control moves to the next NC block containing the text you are searching for

END

- ▶ Terminate the search function: Press the END soft key



Finding/Replacing any text

NOTICE

Caution: Data may be lost!

The **REPLACE** and **REPLACE ALL** functions overwrite all of the found syntax elements without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a result, NC programs may be irreversibly damaged.

- Back up the NC programs, if required, before you start the replacement
- Be appropriately careful when using **REPLACE** and **REPLACE ALL**

The **FIND** and **REPLACE** functions cannot be used in the active NC program while the program is running. These functions are also not available if write protection is active.

- Select the NC block containing the word you wish to find

FIND

- Select the search function
- > The control superimposes the search window and displays the available search functions in the soft-key row.
- Press the **CURRENT WORD** soft key
- > The control loads the first word of the current NC block. If required, press the soft key again to load the desired word.

FIND

- Start the search process
- > The control moves to the next occurrence of the text you are searching for.

REPLACE

- To replace the text and then move to the next occurrence of the text, press the **REPLACE** soft key. Or, to replace all text occurrences, press the **REPLACE ALL** soft key. Or, to skip the text and move to its next occurrence, press the **FIND** soft key

END

- Terminate the search function: Press the END soft key

3.6 File management

Files

Files in the control	Type
NC programs	
in HEIDENHAIN format	.H
in ISO format	.I
Compatible NC programs	
HEIDENHAIN unit programs	.HU
HEIDENHAIN contour programs	.HC
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Dependent data (e.g. structure items)	.DEP
Freely definable tables	.TAB
Texts as	
ASCII files	.A
Text files	.TXT
HTML files, e.g. result logs of touch probe cycles	.HTML
Help files	.CHM
CAD files as	
ASCII files	.DXF .IGES .STEP

When you write an NC program on the control, you must first enter a program name. The control saves the NC program to the internal memory as a file with the same name. The control can also save texts and tables as files.

The control provides a special file management window in which you can easily find and manage your files. Here you can call, copy, rename and erase files.

With the control you can manage and save files up to a total size of **2 GB**.

Depending on the setting, the control generates backup files with the file name extension *.bak after editing and saving NC programs. This reduces the available memory space.

File names

When you store NC programs, tables and texts as files, the control adds an extension to the file name, separated by a point. This extension indicates the file type.

File name	File type
PROG20	.I

File names, drive names and directory names on the control must comply with the following standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (POSIX Standard).

The following characters are permitted:

A B C D E F G H I J K L M N O P Q R S T U V W X Y Z a b c d e f g h i j k l m n o p q r s t u v w x y z 0 1 2 3 4 5 6 7 8 9 _ -

The following characters have special meanings:

Character	Meaning
.	The last period (dot) in a file name is the extension separator
\ and **/**	Directory separators
:	Separates the drive name from the directory

Do not use any other characters. This helps to prevent file transfer problems, etc.

The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**).

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

Further information: "Paths", Page 118

Displaying externally generated files on the control

The control features several software tools which you can use to display the files shown in the table below. Some of the files can also be edited.

File types	Type
PDF files	pdf
Excel tables	xls csv
Internet files	html
Text files	txt ini
Graphic files	bmp gif jpg png

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Directories

To ensure that you can easily find your NC programs and files, we recommend that you organize your internal memory into directories (folders). You can divide a directory into further directories, which are called subdirectories. With the **-/+** key or **ENT** you can show or hide the subdirectories.

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash ****.

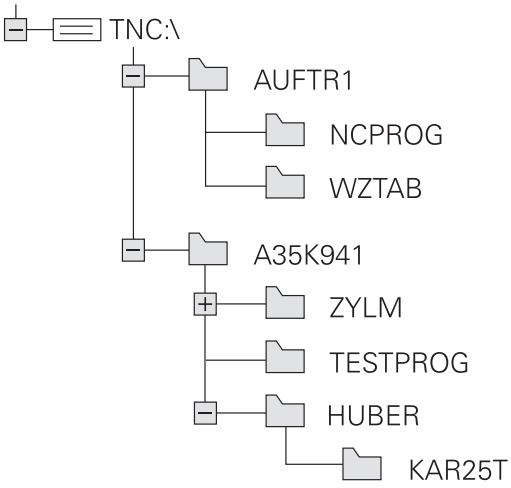
The maximum permitted path length is 255 characters. The path length consists of the drive characters, the directory name and the file name, including the extension.

Example

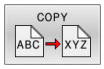
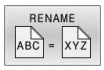
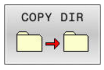
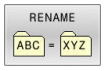
The directory AUFTR1 was created on the **TNC** drive. Then, in the AUFTR1 directory, the directory NCPROG was created and the NC program PROG1.H was copied into it. The NC program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.I

The chart at right illustrates an example of a directory display with different paths.



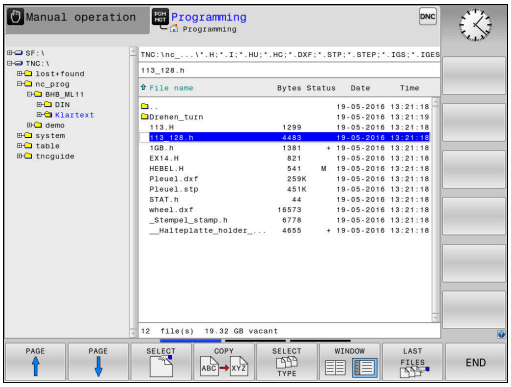
Overview: Functions of the file manager

Soft key	Function	Page
	Copy a single file	123
	Display a specific file type	121
	Create new file	123
	Display the last 10 files that were selected	126
	Delete a file	127
	Tag a file	128
	Rename file	129
	Protect a file against editing and erasure	130
	Cancel file protection	130
	Import file of an iTNC 530	See the User's Manual for Setup, Testing and Running NC Programs
	Customize table view	393
	Manage network drives	See the User's Manual for Setup, Testing and Running NC Programs
	Select the editor	130
	Sort files by properties	129
	Copy a directory	126
	Delete directory with all its subdirectories	
	Refresh directory	
	Rename a directory	
	Create a new directory	

Calling the File Manager

PGM MGT

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



> If you exit an NC program by pressing the **END** key, the control opens the file manager. The cursor is on the NC program that was just closed.
>
> If you press the **END** key again, the control opens the original 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 narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. A drive is the internal memory of the control. Other drives are the interfaces (RS232, Ethernet) to which you can connect a PC for example. A directory is always identified by a folder symbol to the left and the directory name to the right. Subdirectories are shown to the right of and below their parent directories. If there are subdirectories, you can show or hide them using the **-/+** key.

If the directory tree is longer than the screen, navigate using the scroll bar or a connected mouse.

The wide window on the right shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

display	Meaning
File name	File name and file type
Bytes	File size in bytes
Status	File properties:
E	The file has been selected in the **Programming** mode of operation
S	File is selected in the **Test Run** operating mode
M	The file is selected in a Program Run mode of operation
+	File has non-displayed dependent files with the extension DEP, e.g. with use of the tool usage test
	File is protected against erasing and editing
	File is protected against deletion and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited

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

Selecting drives, directories and files

PGM MGT

- Call the file manager by pressing the **PGM MGT** key

Navigate with a connected mouse or use the arrow keys or the soft keys to move the cursor to the desired position on the screen:

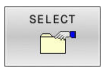
- Moves the cursor from the left to the right window, and vice versa

- Moves the cursor up and down within a window

- Moves the cursor one page up or down within a window

Step 1: Select drive

- Move the highlight to the desired drive in the left window



- Select a drive: Press the **SELECT** soft key, or

- Press the **ENT** key

Step 2: Select a directory

- Move the highlight to the desired directory in the left window

> The right-hand window automatically shows all files stored in the highlighted directory

Step 3: Select a file

- Press the **SELECT TYPE** soft key

- Press the **SHOW ALL** soft key
- Move the highlight to the desired file in the right window

- Press the **SELECT** soft key, or

- Press the **ENT** key

> The control opens the selected file in the operating mode from which you called the file manager.

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

Filtering the display

To filter the displayed files:

- Press the **SELECT TYPE** soft key

- Press the soft key for the desired file type

Alternative:

- Press the **SHOW ALL** soft key

> The control displays all files in this folder.

Alternative:

- Use wildcards, such as **4*.H**

> The control will show all files of file type .h whose name starts with 4.

Alternative:

- Enter the file name extension, e.g. ***.H;*.D**

> The control will show all files of file type .h and .d.

Any display filter you have set will remain effective even after a control restart,

Creating a new directory

- Move the highlight in the left window to the directory in which you want to create a subdirectory

- Press the **NEW DIRECTORY** soft key
- Enter a directory name

- Press the **ENT** key

- Press the **OK** soft key to confirm or

- Press the **CANCEL** soft key to abort

Creating new file

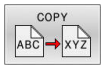
- Select the directory in the left window in which you wish to create the new file
- Position the cursor in the right window

- Press the **NEW FILE** soft key
- Enter the file name with extension

- Press the **ENT** key
- As needed, the control continues the dialog (e.g., select unit of measure).
- Continue the dialog where necessary

Copying a single file

- Move the cursor to the file you wish to copy



- Press the **COPY** soft key to select the copying function
- The control opens a pop-up window.

Copying files into the current directory

- Enter the name of the destination file.

- Press the **ENT** key or the **OK** soft key
- The control copies the file to the active directory. The original file is retained.

Copying files into another directory

- Press the **Target Directory** soft key to select the target directory from a pop-up window

- Press the **ENT** key or the **OK** soft key
- The control copies the file under the same name to the selected directory. The original file is retained.

When you start the copying process with the **ENT** key or the **OK** soft key, the control displays a pop-up window with a progress indicator.

Copying files into another directory

- Select a screen layout with two equally sized windows

In the right window

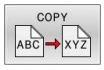
- Press the **SHOW TREE** soft key
- Move the cursor to the directory into which you wish to copy the files, and display the files in this directory with the **ENT** key

In the left window

- Press the **SHOW TREE** soft key
- Select the directory with the files that you want to copy, and display the files with the **SHOW FILES** soft key

- Press the Tag soft key: Call the file tagging functions

- Press the Tag soft key: Position the cursor on the file you wish to copy and tag. You can tag several files in this way, if desired



- Press the Copy soft key: Copy the tagged files into the target directory

Further information: "Tagging files", Page 128

If you have tagged files in both the left and right windows, the control copies from the directory in which the cursor is located.

Overwriting files

If you copy files to a directory in which other files are stored under the same name, the control will ask whether the files in the target directory should be overwritten:

- Overwrite all files (**Existing files** field selected): Press the **OK** soft key, or
- To leave the files as they are, press the **CANCEL** soft key

If you want to overwrite a protected file, select the **Protected files** field, or cancel the process.

Copying a table

Importing lines to a table

If you are copying a table into an existing table, you can overwrite individual lines with the **REPLACE FIELDS** soft key. Prerequisites:

- The target table must exist
- The file to be copied must only contain the lines you want to replace
- Both tables must have the same file extension

NOTICE

Caution: Data may be lost!

The **REPLACE FIELDS** function overwrites all lines of the target file that are contained in the copied table without a confirmation prompt. The original file is not automatically backed up by the control before the replacement process. As a consequence, tables may be irreversibly damaged.

- ▶ Back up the tables, if required, before you start the replacement
- ▶ Be accordingly careful when using **REPLACE FIELDS**

Example

With a tool presetter you have measured the length and radius of ten new tools. The tool presetter then generates the TOOL_Import.T tool table with 10 lines (for the 10 tools).

Proceed as follows:

- ▶ Copy this table from the external data medium to any directory
- ▶ Copy the externally created table to the existing table TOOL.T using the control's file manager.
- > The control asks you whether you want to overwrite the existing TOOL.T tool table.
- ▶ Press the **YES** soft key
- > The control will completely overwrite the current TOOL.T tool table. After this copying process the new TOOL.T table consists of 10 lines.
- ▶ Alternative: Press the **REPLACE FIELDS** soft key
- > The control overwrites the 10 lines in the TOOL.T file. The data of the other lines is not changed.

Extracting lines from a table

You can select one or more lines in a table and save them in a separate table.

Proceed as follows:

- ▶ Open the table from which you want to copy lines
- ▶ Use the arrow keys to select the first line to be copied
- ▶ Press the **MORE FUNCTIONS** soft key
- ▶ Press the **TAG** soft key
- ▶ Select additional lines, if required
- ▶ Press the **SAVE AS** soft key
- ▶ Enter a name for the table in which the selected lines are to be saved

Copying a directory

- Move the highlight in the right window onto the directory you want to copy
- Press the **COPY** soft key
- The control opens the window for selecting the target directory.
- Select the target directory and confirm with the **ENT** key or the **OK** soft key
- The control copies the selected directory and all its subdirectories to the selected target directory.

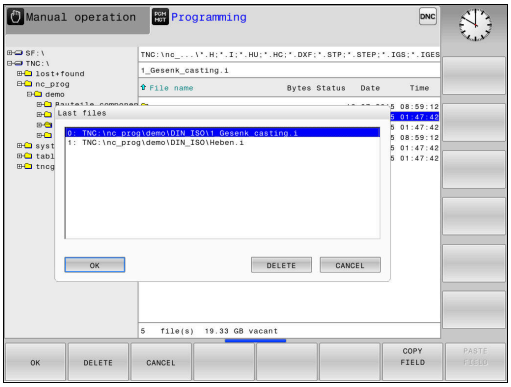
Choosing one of the last files selected

PGM MGT

- Call the file manager: Press the **PGM MGT** key.

LAST FILES

- Display the last ten files selected: Press the **LAST FILES** soft key



Press the arrow keys to move the cursor to the file you wish to select:

- Moves the cursor up and down within a window

OK

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

ENT

- Press the **ENT** key

The **COPY FIELD** soft key allows you to copy the path of a marked file. You can reuse the copied path later (e.g., when calling a program with the **PGM CALL** key).

Deleting a file

NOTICE

Caution: Data may be lost!

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

- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the file you want to delete

- Press the **DELETE** soft key
  - The control asks whether you want to delete the file.
- Press the **OK** soft key
  - The control deletes the file.
- Alternative: Press the **CANCEL** soft key
  - The control aborts the procedure.

Deleting a directory

NOTICE

Caution: Data may be lost!

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

- Regularly back up important data to external drives

Proceed as follows:

- Move the cursor to the directory you want to delete

- Press the **DELETE ALL** soft key
  - The control inquires whether you really intend to delete the directory and all its subdirectories and files.
- Press the **OK** soft key
  - The control deletes the directory.
- Alternative: Press the **CANCEL** soft key
  - The control aborts the procedure.

Tagging files

Soft key	Tagging function
TAG FILE	Tag a single file
TAG ALL FILES	Tag all files in the directory
UNTAG FILE	Untag a single file
UNTAG ALL FILES	Untag all files
COPY ABC XYZ	Copy all tagged files

Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To tag several files:

- Move the cursor to the first file

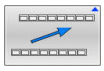
TAG
- To display the tagging functions, press the **TAG** soft key

TAG FILE
- To tag a file, press the **TAG FILE** soft key

- Move the cursor to other files

TAG FILE
- To tag another file, press the **TAG FILE** soft key, etc.

To copy tagged files:


- Leave the active soft-key row

COPY ABC XYZ
- Press the **COPY** soft key

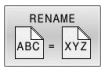
To delete tagged files:

- Leave the active soft-key row

DELETE
- Press the **DELETE** soft key

Renaming a file

- Move the cursor to the file you wish to rename



- Select the function for renaming: Press the **RENAME** soft key
- Enter the new file name; the file type cannot be changed
- To rename: Press the **OK** soft key or the **ENT** key

Sorting files

- Select the folder in which you wish to sort the files

- Press the **SORT** soft key
- Select the soft key with the corresponding display criterion
  - **SORT BY NAME**
  - **SORT BY SIZE**
  - **SORT BY DATE**
  - **SORT BY TYPE**
  - **SORT BY STATUS**
  - **UNSORTED**

Additional functions

Protecting a file and canceling file protection

- Place the cursor on the file you want to protect

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

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

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

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

Selecting the editor

- Place the cursor on the file you want to open

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

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

Connecting and removing USB storage devices

The control automatically detects connected USB devices with a supported file system.

To remove a USB device:

- Move the cursor to the left-hand window
- Press the **MORE FUNCTIONS** soft key

- Remove the USB device

Further information: User's Manual for **Setup, Testing and Running NC Programs**

ADVANCED ACCESS RIGHTS

The **ADVANCED ACCESS RIGHTS** function can only be used in connection with user administration. This function requires the **public** directory.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Upon the first activation of user administration, the **public** directory below the **TNC:** drive will be connected.

> Access rights can only be defined for files located in the **public** directory.
>
> For all files stored on the **TNC:** drive instead of the **public** directory, the **user** function user will automatically be assigned as the owner.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Displaying hidden files

The control hides system files, as well as files and folders whose name begins with a period.

NOTICE

Caution: Possible loss of data!

The control's operating system uses certain hidden folders and files. These folders and files are hidden by default. Any manipulation of the system data within the hidden folders might damage the control's software. If you save your own files to these folders, the system will create invalid paths.

- Always leave hidden folders and files hidden
- Do not use hidden folders and files for saving your own data

If required, you can show the hidden files and folders temporarily, e.g., if a file whose name begins with a period is transferred inadvertently.

To show hidden files and folders:

- Press the **MORE FUNCTIONS** soft key

- Press the **SHOW HIDDEN FILES** soft key

> The control displays the files and folders.

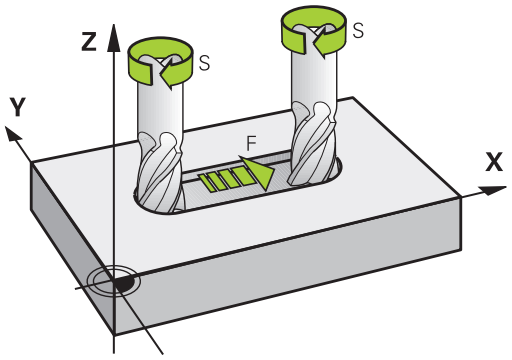
4

Tools

4.1 Entering tool-related data

Feed rate F

The feed rate **F** is the speed at which the tool center point moves. The maximum feed rates can be different for the individual axes and are set in machine parameters.



Input

You can enter the feed rate in the **T** block and in every positioning block.

Further information: "Programming tool movements in ISO", Page 107

You enter the feed rate **F** in mm/min in millimeter programs, and in 1/10 inch/min in inch-programs, for resolution reasons.

Rapid traverse

If you wish to program rapid traverse, enter **G00.**

> Make sure to program rapid traverse movements exclusively with the **G00** 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.

Duration of effect

A feed rate entered as a numerical value remains in effect until an NC block with a different feed rate is reached. **G00** is only effective in the NC block in which it is programmed. After the NC block with **G00** is executed, the feed rate will return to the last feed rate entered as a numerical value.

Changing during program run

You can adjust the feed rate during the program run with the feed rate potentiometer F.

The feed-rate potentiometer only reduces the programmed feed rate, and not the feed rate calculated by the control.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a **T** block (tool call). Instead, you can also define the cutting speed Vc in meters per minute (m/min).

Programmed change

In the NC program, you can change the spindle speed in a **T** block by entering only the new spindle speed.

Proceed as follows:

S

- Press the **S** key on the alphabetic keyboard
- Enter the new spindle speed

In the following cases the control changes only the speed:

- **T** block without tool name, tool number, and tool axis
- **T** block without tool name, tool number, with the same tool axis as in the previous **T** block

In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:

- **T** block with tool number
- **T** block with tool name
- **T** block without tool name or tool number, with a changed tool axis direction

Changing during program run

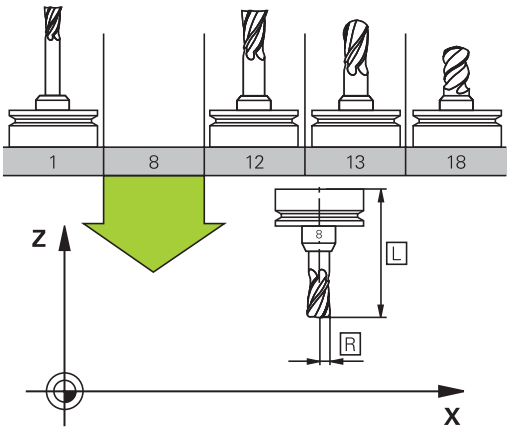
You can adjust the spindle speed during program run with the spindle speed potentiometer S.

4.2 Tool data

Requirements for tool compensation

You usually program the coordinates of path contours as they are dimensioned in the workpiece drawing. To allow the control to calculate the tool center path (i.e. the tool compensation) you must also enter the length and radius of each tool you are using.

You can enter tool data either directly in the NC program with **G99** or separately in a tool tables. In a tool table, you can also enter additional data for the specific tool. The control will consider all the data entered for the tool when executing the NC program.



Tool number, tool name

Each tool is identified by a number between 0 and 32767. If you are working with tool tables, you can also enter a tool name for each tool. Tool names can have up to 32 characters.

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

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

Impermissible characters: <blank space> ! " ‘ ( ) * + : ; < = > ? [/] ^ ` { | } ~

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool T0 should also be defined with L=0 and R=0.

Assign unique tool names!

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

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

  Refer to your machine manual.

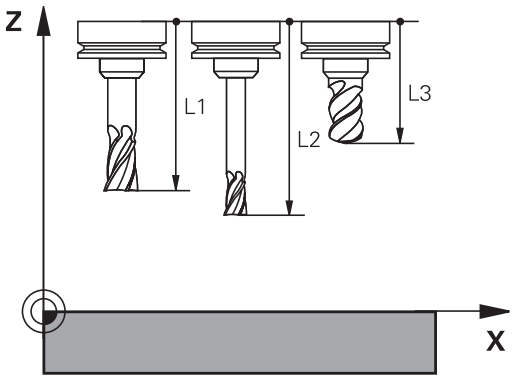
  If there are multiple magazines, the machine manufacturer can specify the search sequence of the tools in the magazines.

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

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

Tool length L

Always enter the tool length **L** as an absolute value based on the tool reference point.



> The absolute length of the touch probe is always referenced to the tool reference point. The machine manufacturer usually defines the spindle nose as the tool reference point.

Measuring the tool length

You can measure your tools in the machine (e.g., with a tool touch probe) or externally with a tool presetter. If such measurements are not possible, you can determine the tool length.

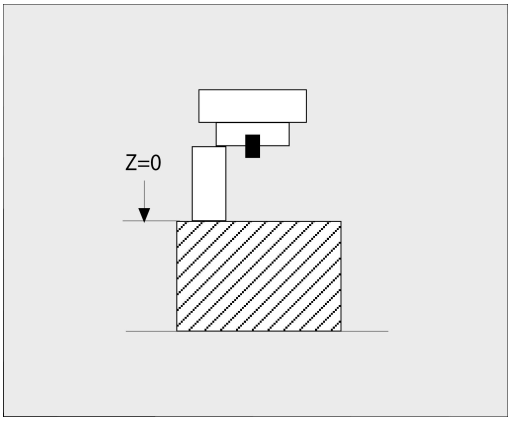
You have the following options for determining the tool length:

- With a gauge block
- With a calibration pin (inspection tool)

> Before you determine tool length, you have to set the preset in the spindle axis.

Determining the tool length with a gauge block

> You can only set the preset with a gauge block if the tool reference point is at the spindle nose.
>
> Place the preset on the surface you want to touch off with the tool. This surface might have to be created first.



To set the datum with a gauge block:

- Place the gauge block on the machine table
- Position the spindle nose next to the gauge block
- Gradually move in **Z+** direction until you can just slide the gauge block under the spindle nose
- Set the preset in **Z**

To determine the tool length:

- Insert the tool
- Touch off the surface

> The control displays the absolute tool length as the actual position in the position display.

Determining the tool length with a calibration pin and a tool setter

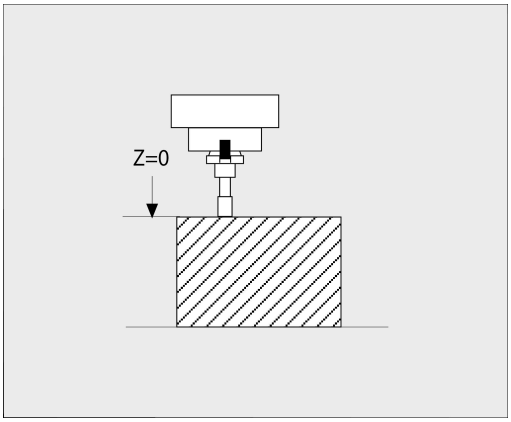
To set the preset with a calibration pin and a tool setter:

- Clamp the tool setter onto the machine table.
- Bring the flexible inner ring of the tool setter to the same height as the fixed outer ring.
- Set the gauge to 0
- Move the calibration pin onto the flexible inner ring.
- Set the preset in **Z**

To determine the tool length:

- Insert the tool
- Move the tool onto the flexible inner ring until the gauge displays 0.

> The control displays the absolute tool length as the actual position in the position display.



Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii

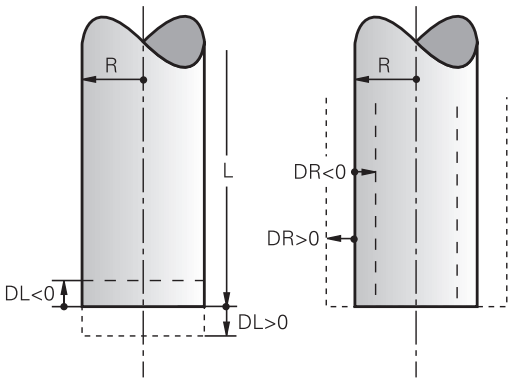
Delta values are offsets in the length and radius of a tool.

A positive delta value represents a tool oversize (**DL**, **DR**>0). For a machining operation with an oversize, enter the value for the oversize in the NC program with **T** or with the help of a compensation table.

A negative delta value describes a tool undersize (**DL**, **DR**<0). An undersize is entered in the tool table for wear.

Delta values are usually entered as numerical values. In a **T** block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to ± 99.999 mm.



Delta values from the tool table influence the graphical representation of the clearing simulation.

Delta values from the NC program do not change the depicted size of the **tool** in the simulation. However, the programmed delta values move the **tool** in the simulation by the amount of the defined value.

Delta values from the **T** block influence the position display depending on the optional machine parameter **progToolCallDL** (no. 124501; branch **CfgPositionDisplay** no. 124500).

Entering tool data into the NC program

Refer to your machine manual.

The machine manufacturer determines the scope of functions of the **G99** function.

The number, length, and radius of a specific tool are defined in the **G99** block of the NC program:

Proceed as follows for the definition:

TOOL DEF

- Press the **TOOL DEF** key.

- **Tool length**: Compensation value for the tool length
- **Tool radius**: Compensation value for the tool radius

Example

```
N40 G99 T5 L+10 R+5*
```

Calling the tool data

Before you can call the tool, you have to define it in a **G99** block or in the tool table.

A **T** in the NC program is programmed with the following data:

TOOL CALL

- Press the **TOOL CALL** key
- **Tool call**: Enter the number or name of the tool. With the **TOOL NAME** soft key, you can enter a name. With the **QS** soft key, you enter a string parameter. The control automatically puts the tool name in quotation marks. You must first assign a tool name to a string parameter. The names refer to an entry in the active tool table TOOL.T.

- Alternative: Press the **SELECT** soft key

> The control opens a window where you can select a tool directly from the TOOL.T tool table.

- To call a tool with other compensation values, enter a decimal point followed by the index you defined in the tool table.
- **Working spindle axis X/Y/Z**: Enter the tool axis
- **Spindle speed S**: Enter the spindle speed S in revolutions per minute (rpm). Alternatively, you can define the cutting speed Vc in meters per minute (m/min). Press the **VC** soft key
- **Feed rate F**: Enter feed rate **F** in millimeters per minute (mm/min). The feed rate is effective until you program a new feed rate in a positioning block or in a **T** block
- **Tool length oversize DL**: Enter the delta value for the tool length
- **Tool radius oversize DR**: Enter the delta value for the tool radius
- **Tool radius oversize DR2**: Enter the delta value for tool radius 2

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.

In the following cases the control changes only the speed:

- **T** block without tool name, tool number, and tool axis
- **T** block without tool name, tool number, with the same tool axis as in the previous **T** block

In the following cases the control runs the tool-change macro and inserts a replacement tool if necessary:

- **T** block with tool number
- **T** block with tool name
- **T** block without tool name or tool number, with a changed tool axis direction

Tool selection in the pop-up window

If you open a pop-up window for tool selection, the control marks all tools available in the tool magazine green.

You can search for a tool in the pop-up window:

GOTO

- Press the **GOTO** key
- Alternative: Press the **FIND** soft key
- Enter the tool name or tool number

ENT

- Press the **ENT** key

> The control goes to the first tool that matches the entered search string.

The following functions can be used with a connected mouse:

- You can sort the data in ascending or descending order by clicking a column of the table head.
- You can arrange the columns in any sequence you want by clicking a column of the table head and then moving it with the mouse key pressed down

The pop-up windows displayed for a tool number search and a tool name search can be configured separately. The sort order and the column widths are retained when the control is switched off.

Tool call

Call tool number 5 in the tool axis Z with a spindle speed of 2500 rpm and a feed rate of 350 mm/min. The tool length and tool radius 2 are to be programmed with an oversize of 0.2 and 0.05 mm, the tool radius with an undersize of 1 mm.

Example

```
N20 T  5.2 G17  S2500 DL+0.2  DR-1*
```

The character **D** preceding **L**, **R** and **R2** designates delta values.

Preselection of tools

Refer to your machine manual.
The preselection of tools with **G51** can vary depending on the individual machine tool.

If you are working with tool tables, use a **G51** block to preselect the next tool. Simply enter the tool number, or a Q parameter, or type the tool name in quotation marks.

Tool change

Automatic tool change

Refer to your machine manual.
The tool change function can vary depending on the individual machine tool.

If your machine tool has automatic tool changing capability, the program run is not interrupted. When the control reaches a tool call with **T**, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

Refer to your machine manual.
The function of **M101** can vary depending on the individual machine tool.

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

If you do not define a replacement tool in the **RT** column and call the tool via its tool name, the control will switch to a tool with the same name once the maximum tool age **TIME2** has been reached.

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

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

NOTICE

Danger of collision!

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

- Use **M101** only for machining operations without undercuts
- Deactivate the tool change with **M102**

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

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

Input parameter BT (block tolerance)

Depending on the NC program, the machining time can increase as a result of the tool life verification and calculation of the automatic tool change. You can influence this with the optional input parameter **BT** (block tolerance).

If you enter the **M101** function, the control continues the dialog by requesting **BT**. Here you define the number of NC blocks (1 to 100) by which the automatic tool change may be delayed. The resulting time period by which the tool change is delayed depends on the content of the NC blocks (e.g. feed rate, path). If you do not define **BT**, the control uses the value 1 or, if applicable, a default value defined by the machine manufacturer.

The higher the value of **BT**, the smaller will be the effect of an extended program duration through **M101**. Please note that this will delay the automatic tool change!

To calculate a suitable initial value for **BT**, use the following formula:

$BT = 10 \div t$

t: average machining time of an NC block in seconds

Round the result up to an integer value. If the calculated result is greater than 100, use the maximum input value of 100.

If you want to reset the current age of a tool (e.g. after changing the indexable inserts), enter the value 0 in the **CUR_TIME** column.

Prerequisites for a tool change with M101

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

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

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

- During execution of machining cycles
- While radius compensation (**G41**/**G42**) is active
- Directly after an approach function **APPR**
- Directly before a departure function **DEP**
- Directly before and after **G24** and **G25**
- During execution of macros
- During execution of a tool change
- Directly after a **T** block or **G99**
- During execution of SL cycles

Overtime for tool life

This function must be enabled and adapted by the machine manufacturer.

The tool condition at the end of planned tool life depends on e.g. the tool type, machining method and workpiece material. In the **OVRTIME** column of the tool table, enter the time in minutes for which the tool is permitted to be used beyond the tool life.

The machine manufacturer specifies whether this column is enabled and how it is used during tool search.

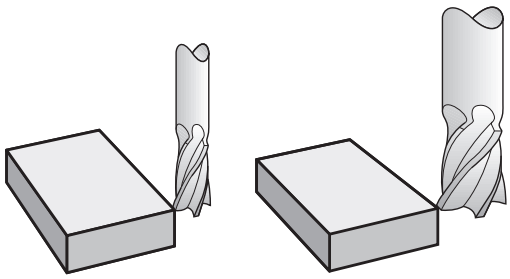
4.3 Tool compensation

Introduction

The control adjusts the tool path by the compensation value for the tool length in the spindle axis. In the machining plane, it compensates the tool radius.

If you are writing the NC program directly on the control, the tool radius compensation is effective only in the working plane.

The control accounts for the compensation value in up to five axes including the rotary axes.



Tool length compensation

Length compensation automatically becomes active as soon as a tool is called. It is canceled as soon as a tool is called with the length L=0 (e.g., **T 0**).

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 **T 0**. There is a risk of collision during subsequent tool positioning movements!

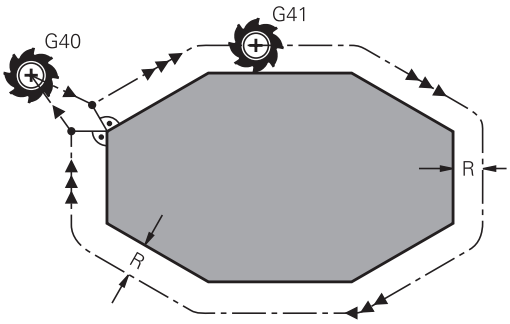
- Always define the actual tool length of a tool (not just the difference)
- Use **T 0** only to empty the spindle

With length compensation, delta values from both the NC program and the tool table are considered.

Compensation value = **L** + $\mathbf{DL}_{TAB}$ + $\mathbf{DL}_{Prog}$ with

L:	Tool length **L** from **G99** block or tool table
$\mathbf{DL}_{TAB}$:	Oversize for length **DL** in the tool table
$\mathbf{DL}_{Prog}$:	Oversize **DL** for length from **T** block or from the compensation table The most recently programmed value becomes active. **Further information:** "Compensation table", Page 372

Tool radius compensation



An NC block can contain the following types of tool radius compensation:

- **G41** or **G42** for radius compensation of any contouring function
- **G40**, if there is no radius compensation

> The control shows an active tool compensation in the general status display.

The radius compensation takes effect as soon as a tool is called and is moved with one of the above-mentioned types of tool radius compensation within a straight-line block or a paraxial movement in the working plane.

> The control automatically cancels radius compensation in the following cases:
>
> - Straight-line block with **G40**
> - **DEP** function for departing from the contour
> - Selection of a new NC program via **PGM MGT**

For radius compensation, the control takes the delta values from both the **T** block and the tool table into account:

Compensation value = **R** + $\mathbf{DR}_{TAB}$ + $\mathbf{DR}_{Prog}$ with

R: Tool radius **R** from **G99** block or tool table

$\mathbf{DR}_{TAB}$: Oversize **DR** for the radius in the tool table

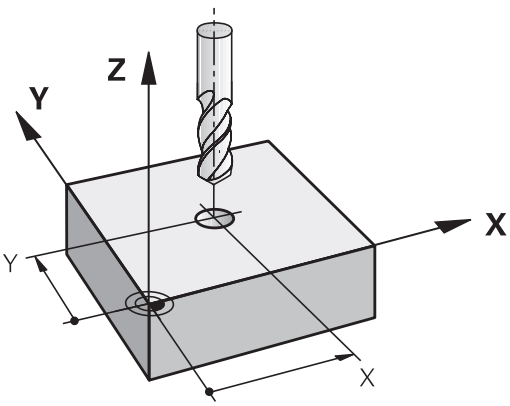
$\mathbf{DR}_{Prog}$: Oversize **DR** for the radius from the **T** block or from the compensation table

Further information: "Compensation table", Page 372

Movements without radius compensation: G40

The tool center moves in the working plane to the programmed coordinate.

Applications: Drilling and boring, pre-positioning

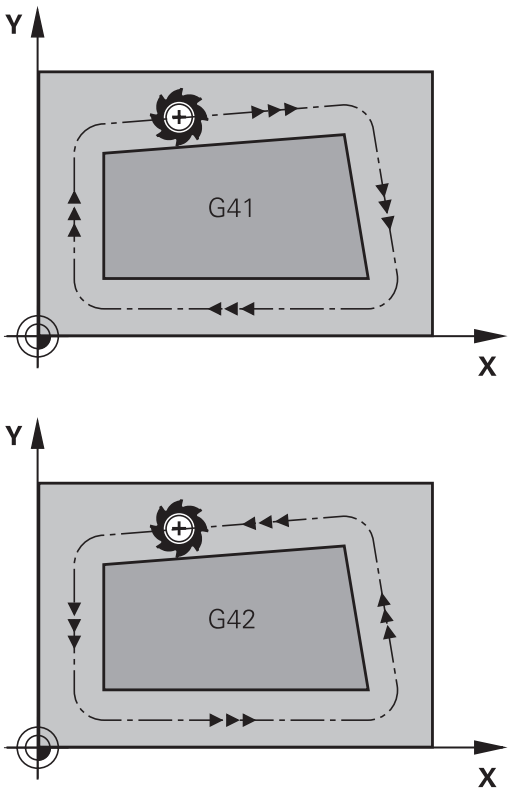


Contouring with radius compensation: G42 and G41

G42: The tool moves to the right of the programmed contour

G41: The tool moves to the left of the programmed contour

The tool center moves along the contour at a distance equal to the radius. **Right** or **left** are to be understood as based on the direction of tool movement along the workpiece contour.



> Between two NC blocks, each with a different tool radius compensation **G42** and **G41**, there must be at least one traversing block in the working plane without tool radius compensation **G40**.
>
> The control does not put radius compensation into effect until the end of the NC block in which it is first programmed.
>
> When radius compensation is activated with **G42/G41**, and in the case of cancellation with **G40**, the control always positions the tool perpendicularly to the programmed start or end point. Position the tool before the first contour point or after the last contour point such that the contour does not incur damage.

Entering radius compensation

Radius compensation is entered in a **G01** block. Enter the coordinates of the target point, and confirm your entry with the **ENT** key.

G41

- Select tool movement to the left of the programmed contour: Press the **G41** soft key, or

G42

- Select tool movement to the right of the contour: Press the **G42** soft key, or

G40

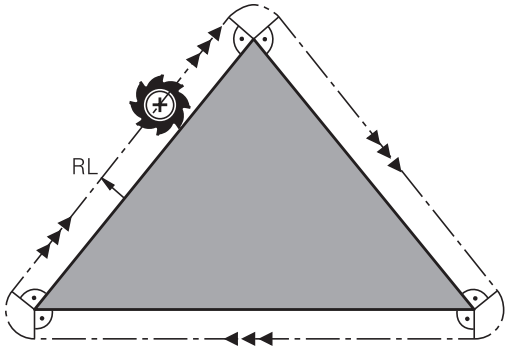
- Select tool movement without radius compensation or cancel radius compensation: Press the **G40** soft key

END

- Terminate the NC block: Press the **END** key

Radius compensation: Machining corners

- Outside corners:
  If you program radius compensation, the control moves the tool around outside corners on a transitional arc. If necessary, the control reduces the feed rate at outside corners during, for example, large changes in direction
- Inside corners:
  The control calculates the intersection of the tool center paths at inside corners under radius compensation. Starting at this point, the tool moves along the next contour element. This prevents damage to the workpiece at the inside corners. As a result, the tool radius for a certain contour cannot be selected to be just any size.

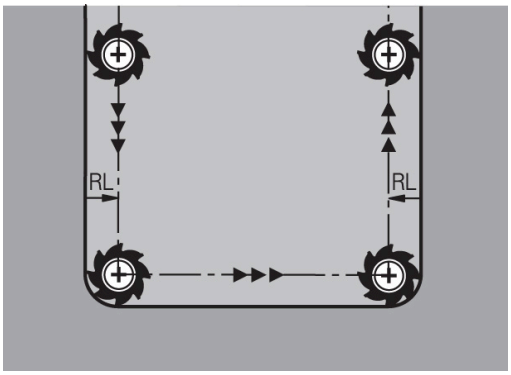


NOTICE

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
- Consider the approach strategy



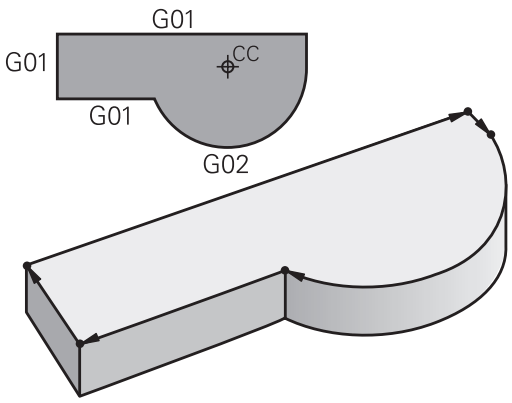
5

Programming Contours

5.1 Tool movements

Path functions

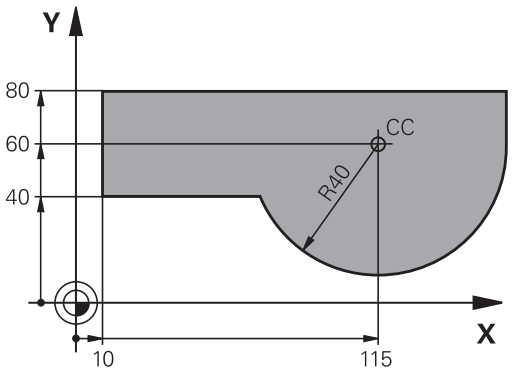
A workpiece contour is usually composed of several contour elements such as straight lines and circular arcs. With the path functions, you can program the tool movements for **straight lines** and **circular arcs**.



FK free contour programming

If a production drawing is not dimensioned for NC and the dimensions given are not sufficient for creating a part program, you can program the workpiece contour with the FK free contour programming. The control calculates the missing data.

With FK programming, you also program tool movements for **straight lines** and **circular arcs**.



Miscellaneous functions M

With the control's miscellaneous functions you can affect

- the program run, e.g., a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- the path behavior of the tool

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program-section repeat. If you wish to execute a specific NC program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have an NC program call a separate NC program for execution.

Further information: "Subprograms and Program Section Repeats", Page 255

Programming with Q parameters

Instead of programming numerical values in an NC program, you enter markers called Q parameters. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

In addition, programming with Q parameters enables you to measure with the 3D touch probe during the program run.

Further information: "Programming Q Parameters", Page 277

5.2 Fundamentals of path functions

Programming tool movements for machining

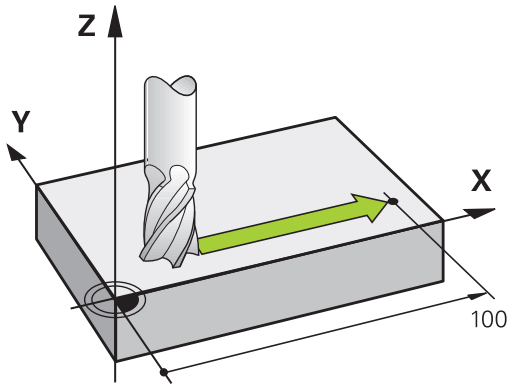
You create an NC program by programming the path functions for the individual contour elements in sequence. You do this by entering the coordinates of the end points of the contour elements given in the production drawing. The control calculates the actual path of the tool from these coordinates, and from the tool data and radius compensation.

The control moves all machine axes programmed in the NC block of a path function simultaneously.

Movement parallel to the machine axes

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

Depending on the individual machine, the machining program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Path contours are programmed as if the tool were moving.



Example

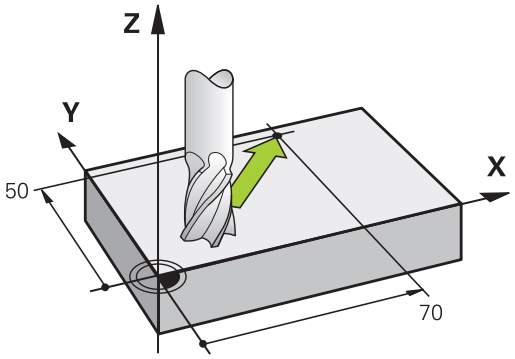
```
N50 G00 X+100*
```

N50	Block number
G00	Path function **straight line at rapid traverse**
X+100	Coordinate of the end point

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

Movement in the main planes

If the NC block contains two coordinates, the control moves the tool in the programmed plane.



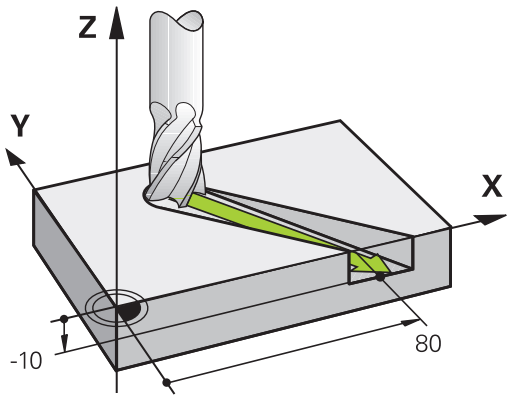
Example

```
N50 G00 X+70  Y+50*
```

The tool retains the Z coordinate and moves on the XY plane to the position X=70, Y=50.

Three-dimensional movement

If the NC block contains three coordinates, the control moves the tool spatially to the programmed position.



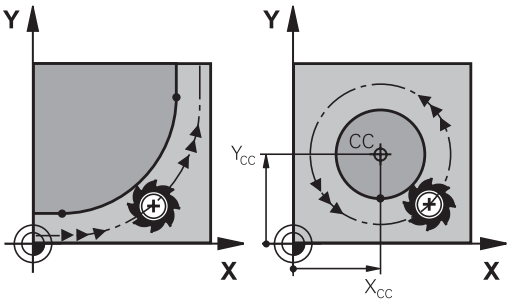
Example

```
N50 G01 X+80  Y+0 Z-10*
```

Circles and circular arcs

The control moves two machine axes simultaneously on a circular path relative to the workpiece. You can define a circular movement by entering the circle center with **I** and **J**.

Use the path functions for circular arcs to program circles in the working plane. You define the main plane based on the spindle axis in the **T**.



Spindle axis	Main plane
(G17)	**XY**, also UV, XV, UY
(G18)	**ZX**, also WU, ZU, WX
(G19)	**YZ**, also VW, YW, VZ

Circular motion in another plane

You can also use the **Tilt the working plane** function or Q parameters to program circular motions that do not lie in the main plane.

Further information: "The PLANE function: Tilting the working plane (option 8)", Page 405

Further information: "Principle and overview of functions", Page 278

Direction of rotation DR for circular movements

When a circular path has no tangential transition to another contour element, enter the direction of rotation as follows:

Clockwise direction of rotation: **G02/G12**

Counterclockwise direction of rotation: **G03/G13**

Radius compensation

The radius compensation must be in the NC block in which you move to the first contour element. You cannot activate radius compensation in an NC block for a circular path. It must be activated beforehand in a straight-line block.

Further information: "Path contours — Cartesian coordinates", Page 166

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

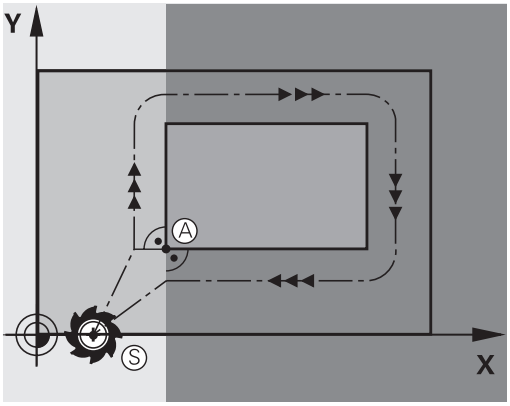
5.3 Approaching and departing a contour

Starting point and end point

The tool approaches the first contour point from the starting point. The starting point must be:

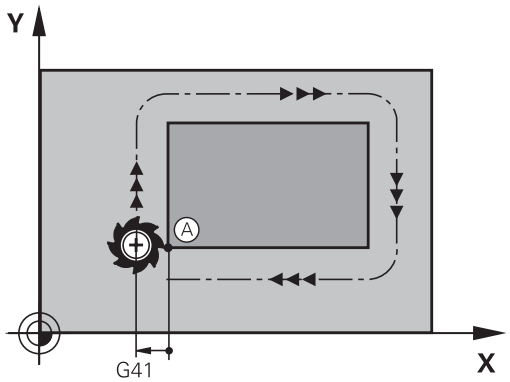
- Programmed without radius compensation
- Approachable without danger of collision
- Close to the first contour point

Example in the figure on the right:
If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.



First contour point

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

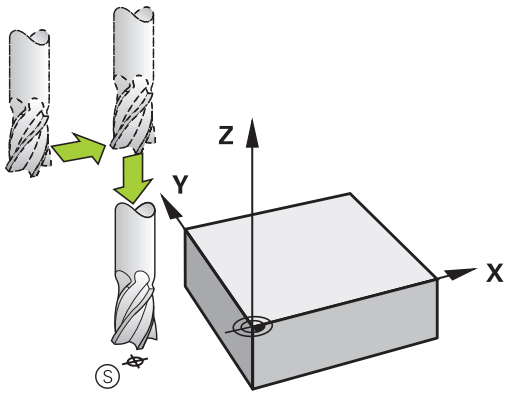


Approaching the starting point in the spindle axis

When the starting point is approached, the tool must be moved to the working depth in the spindle axis. If danger of collision exists, approach the starting point in the spindle axis separately.

Example

```
N40 G00  Z-10*
N30 G01  X+20  Y+30  G41 F350*
```



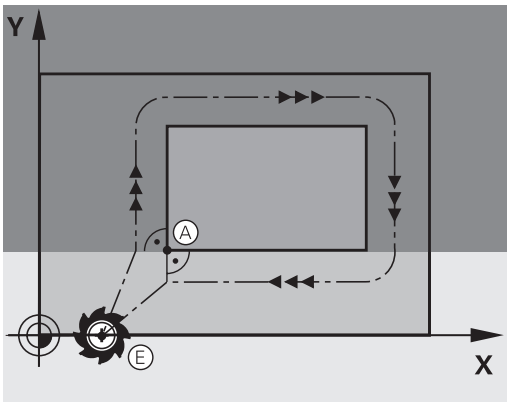
End point

The end point should be selected so that it is:

- Approachable without danger of collision
- Near to the last contour point
- In order to make sure the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element

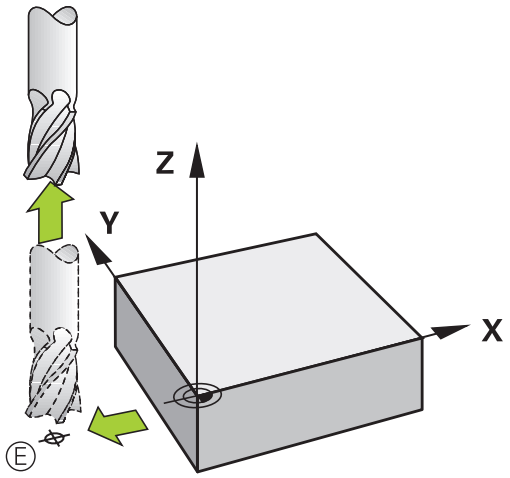
Example in the figure on the right:
If you set the end point in the dark gray area, the contour will be damaged when the end point is approached.

Departing the end point in the spindle axis:
Program the departure from the end point in the spindle axis separately.



Example

```
N50 G01  G40 X+60  Y+70  F700*
N60 G00  Z+250*
```

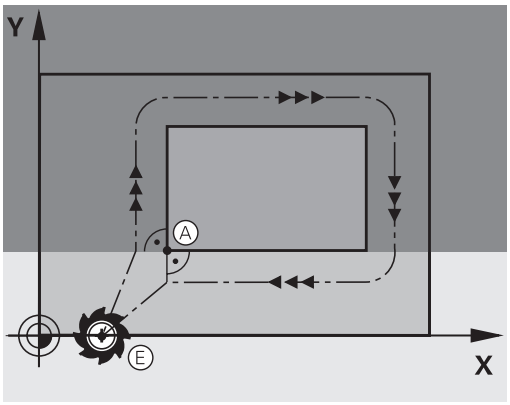


Common starting and end points

Do not program any radius compensation if the starting point and end point are the same.

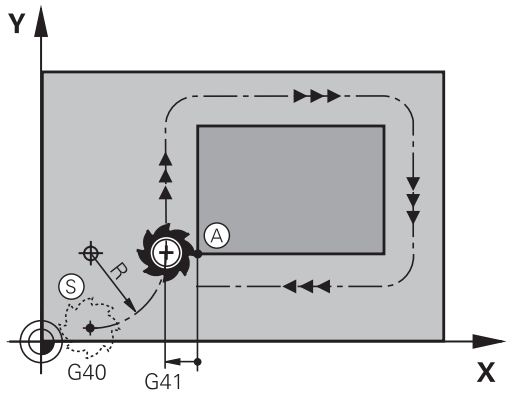
In order to make sure the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

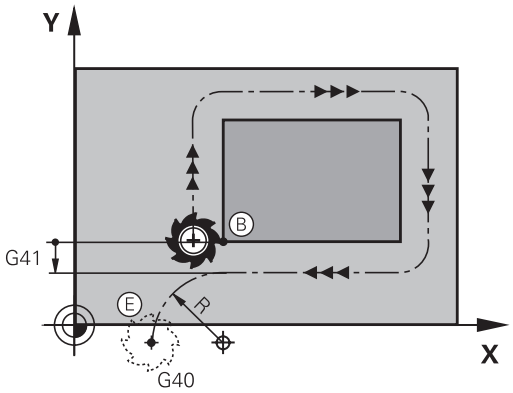
Example in the figure on the right:
If you set the end point in the dark gray area, the contour will be damaged when the contour is approached/departed.



Tangential approach and departure

With **G26** (figure at center right), you can program a tangential approach to the workpiece, and with **G27** (figure at lower right) a tangential departure. In this way you can avoid dwell marks.





Starting point and end point

The starting point and the end point lie outside the workpiece, close to the first and last contour points. They are to be programmed without radius compensation.

Approach

- **G26** is entered after the NC block in which the first contour element is programmed: This will be the first NC block with radius compensation **G41/G42**

Departure

- **G27** after the NC block in which the last contour element is programmed: This will be the last NC block with radius compensation **G41/G42**

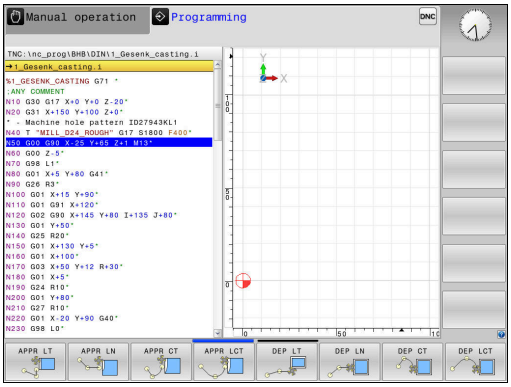
The radius for **G26** and **G27** must be selected so that the control can execute the circular path between the starting point and the first contour point, as well as the last contour point and the end point.

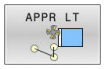
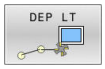
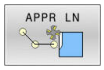
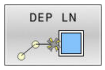
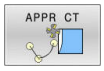
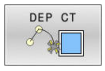
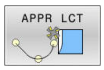
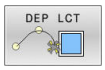
Example

N50 G00 G40 G90 X-30 Y+50*	Starting point
N60 G01 G41 X+0 Y+50 F350*	First contour point
N70 G26 R5*	Tangential approach with radius R = 5 mm
. . .	
Program contour blocks	
. . .	Last contour point
N210 G27 R5*	Tangential departure with radius R = 5 mm
N220 G00 G40 X-30 Y+50*	End point

Overview: Types of paths for contour approach and departure

The functions for contour approach **APPR** and departure **DEP** are activated with the **APPR/DEP** key. You can then select the following path forms with the corresponding soft keys:



Approach	Departure	Function
		Straight line with tangential connection
		Straight line perpendicular to a contour point
		Circular arc with tangential connection
		Circular arc with tangential connection to the contour. Approach and departure to an auxiliary point outside the contour on a tangentially connecting line

Approaching and departing a helix

The tool approaches and departs a helix on its extension by moving in a circular arc that connects tangentially to the contour. You program helical approach and departure with the **APPR CT** and **DEP CT** functions.

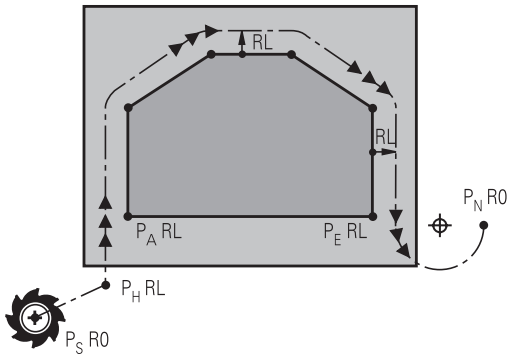
Important positions for approach and departure

NOTICE

Danger of collision!

The control traverses from the current position (starting point P_S) to the auxiliary point P_H at the last feed rate entered. If you programmed **G00** in the last positioning block before the approach function, the control also approaches the auxiliary point P_H at rapid traverse.

- Program a feed rate other than **G00** before the approach function



R0=G40; RL=G41; RR=G42

- Starting point P_S
  You program this position in the block before the APPR block. P_S lies outside the contour and is approached without radius compensation (G40).
- Auxiliary point P_H
  Some of the paths for approach and departure go through an auxiliary point P_H that the control calculates from your input in the APPR or DEP block.
- First contour point P_A and last contour point P_E
  You program the first contour point P_A in the APPR block. The last contour point P_E can be programmed with any path function. If the APPR block also includes the Z coordinate, the control moves the tool simultaneously to the first contour point P_A.
- End point P_N
  The position P_N lies outside of the contour and results from your input in the DEP block. If the DEP block also includes the Z coordinate, the control moves the tool simultaneously to the end point P_N.

Designation	Meaning
APPR	Approach
DEP	Departure
L	Line
C	Circle
T	Tangential (smooth connection)
N	Normal (perpendicular)

NOTICE

Danger of collision!

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

- Program a suitable pre-position
- Check the auxiliary point P_H, the sequence and the contour with the aid of the graphic simulation

With the **APPR LT**, **APPR LN** and **APPR CT** functions, the control moves the tool to the auxiliary point P_H at the last programmed feed rate (which can also be **FMAX**). With the **APPR LCT** function, the control moves to the auxiliary point P_H at the feed rate programmed with the APPR block. If no feed rate is programmed yet before the approach block, the control generates an error message.

Polar coordinates

You can also program the contour points for the following approach/departure functions over polar coordinates:

- APPR LT becomes APPR PLT
- APPR LN becomes APPR PLN
- APPR CT becomes APPR PCT
- APPR LCT becomes APPR PLCT
- DEP LCT becomes DEP PLCT

Select by soft key an approach or departure function, then press the orange **P** key.

Radius compensation

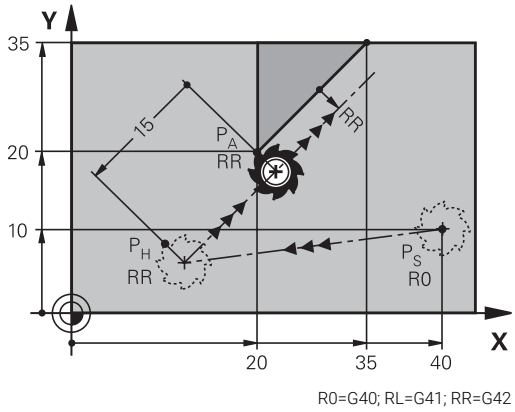
The tool radius compensation is programmed together with the first contour point P_A in the APPR block. The DEP blocks automatically discard the tool radius compensation.

If you program **APPR LN** or **APPR CT** with **G40**, the control stops the machining/simulation with an error message.

This method of function differs from the iTNC 530 control!

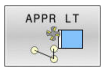
Approaching on a straight line with tangential connection: APPR LT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a straight line that connects tangentially to the contour. The auxiliary point P_H is separated from the first contour point P_A by the distance **LEN**.



R0=G40; RL=G41; RR=G42

- Use any path function to approach the starting point P_S
- Initiate the dialog with the **APPR DEP** key and **APPR LT** soft key



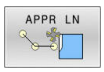
- Coordinates of the first contour point P_A
- **LEN**: Distance from the auxiliary point P_H to the first contour point P_A
- Radius compensation **G41/G42** for machining

Example

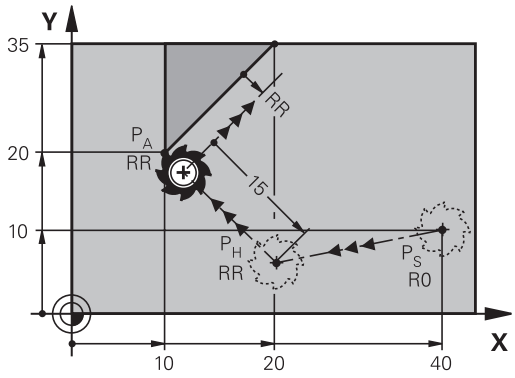
N110 G01 X+40 Y+10 G40 300 M3*	; Approach P_S with **G40**
N120 APPR LT X+20 Y+20 Z-10 LEN15 G42 F100*	; Approach P_A with **G42**; distance P_H to P_A: **LEN15**
N130 G01 X+35 Y+35*	; Complete the first contour element

Approaching on a straight line perpendicular to the first contour point: APPR LN

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the **APPR DEP** key and **APPR LN** soft key:



- Coordinates of the first contour point P_A
- Length: Distance to the auxiliary point P_H. Always enter **LEN** as a positive value
- Radius compensation **G41/G42** for machining



R0=G40; RL=G41; RR=G42

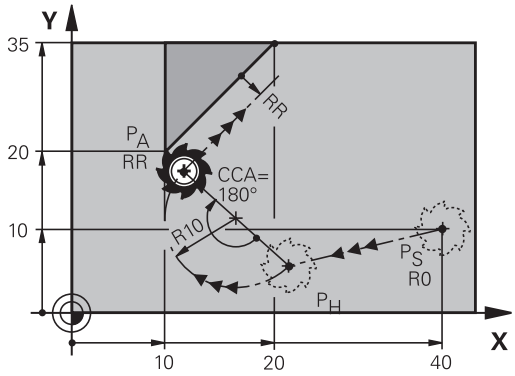
Example

N110 G01 X+40 Y+10 G40 F300 M3*	; Approach P_S with **G40**
N120 APPR LN X+10 Y+20 Z-10 LEN+15 G42 F100*	; Approach P_A with **G42**; P_H to P_A: **LEN+15**
N130 G01 X+20 Y+35*	; Complete the first contour element

Approaching on a circular path with tangential connection: APPR CT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves from PH to the first contour point PA following a circular arc that is tangential to the first contour element.

The arc from P_H to P_A is determined through the radius R and the center angle **CCA**. The direction of rotation of the circular arc is automatically derived from the tool path for the first contour element.



R0=G40; RL=G41; RR=G42

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the **APPR DEP** key and **APPR CT** soft key

- Coordinates of the first contour point P_A
- Radius R of the circular arc
  - If the tool should approach the workpiece in the direction defined by the radius compensation: Enter R as a positive value
  - If the tool should approach the workpiece opposite to the radius compensation: Enter R as a negative value
- Center angle **CCA** of the arc
  - CCA can be entered only as a positive value
  - Maximum input value 360°
- Radius compensation **G41/G42** for machining

Example

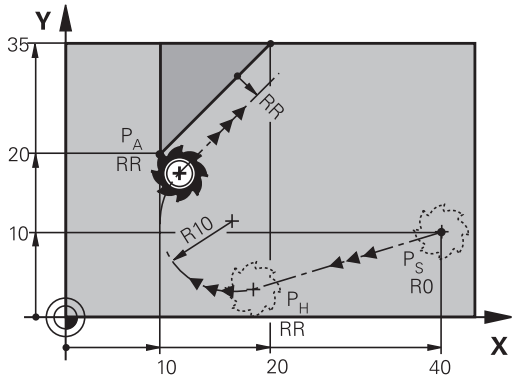
N110 G01 X+40 Y+10 F300 G40 M3*	; Approach P_S with **G40**
N120 APPR CT X+10 Y+20 Z-10 CCA180 R+10 G42 F100*	; Approach P_A with **CCA180** and **G42**; distance P_H to P_A: **R+10**
N130 G01 X+20 Y+35*	; Complete the first contour element

Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a circular arc. The feed rate programmed in the APPR block is effective for the entire path that the control traversed in the approach block (path P_S to P_A).

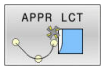
If you have programmed the coordinates of all three principal axes X, Y and Z in the approach block, the control moves the tool from the position defined before the APPR block to the auxiliary point P_H on all three axes simultaneously. Then the control moves the tool from P_H to P_A only in the working plane.

The arc is connected tangentially both to the line P_S - P_H as well as to the first contour element. Once these lines are known, the radius then suffices to completely define the tool path.



R0=G40; RL=G41; RR=G42

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the **APPR DEP** key and **APPR LCT** soft key:



- Coordinates of the first contour point P_A
- Radius R of the circular arc. Enter R as a positive value
- Radius compensation **G41/G42** for machining

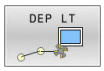
Example

N110 G01 X+40 Y+10 G40 F300 M3*	; Approach P_S with **G40**
N120 APPR LCT X+10 Y+20 Z-10 R10 G42 F100*	; Approach P_A with **G42**; distance P_H to P_A: **R10**
N130 G01 X+20 Y+35*	; Complete the first contour element

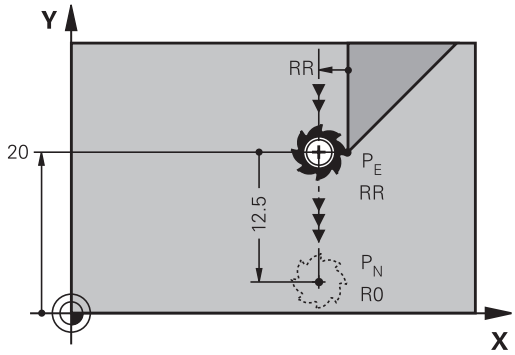
Departing in a straight line with tangential connection: DEP LT

The tool moves on a straight line from the last contour point P_E to the end point P_N. The line lies on the extension of the last contour element. P_N is separated from P_E by the distance **LEN**.

- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR DEP** key and **DEP LT** soft key



- LEN: Enter the distance from the last contour element P_E to the end point P_N.



R0=G40; RL=G41; RR=G42

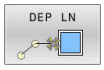
Example

N110 G01 Y+20 G42 100*	; Approach the last contour element P_E with **G42**
N120 DEP LT LEN12.5 100*	; Approach P_N; distance P_E to P_N: **LEN12.5**

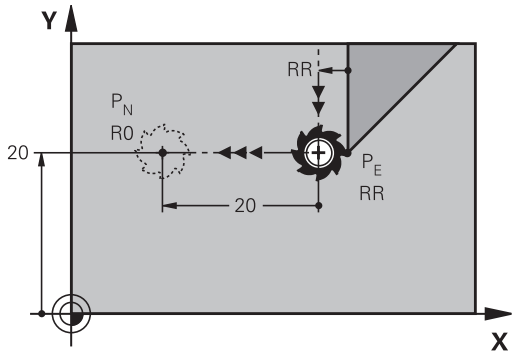
Departing in a straight line perpendicular to the last contour point: DEP LN

The tool moves on a straight line from the last contour point P_E to the end point P_N. The line departs on a perpendicular path from the last contour point P_E. P_N is separated from P_E by the distance **LEN** plus the tool radius.

- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR DEP** key and **DEP LN** soft key



- **LEN**: Enter the distance from the last contour element to P_N. Important: Enter a positive value in **LEN**



R0=G40; RL=G41; RR=G42

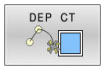
Example

N110 G01 Y+20 G42 F100*	; Approach the last contour element P_E with **G42**
N120 DEP LN LEN+20 F100*	; Approach P_N; distance P_E to P_N: **LEN+20**

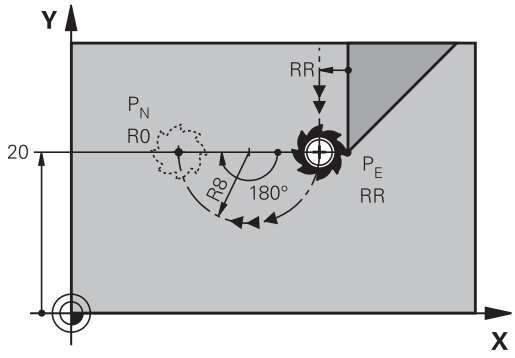
Departing on a circular path with tangential connection: DEP CT

The tool moves on a circular arc from the last contour point P_E to the end point P_N. The circular arc connects tangentially to the last contour element.

- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR DEP** key and **DEP CT** soft key



- Center angle **CCA** of the arc
- Radius R of the circular arc
  - If the tool should depart the workpiece in the direction opposite to the radius compensation: Enter R as a positive value.
  - If the tool should depart the workpiece in the direction **opposite** to the radius compensation: Enter R as a negative value.



R0=G40; RL=G41; RR=G42

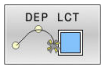
Example

N110 G01 Y+20 G42 100*	; Approach the last contour element P_E with **G42**
N120 DEP CT CCA180 R+8 F100*	; Approach P_N with **CCA180**; distance from P_E to P_N: **R+8**

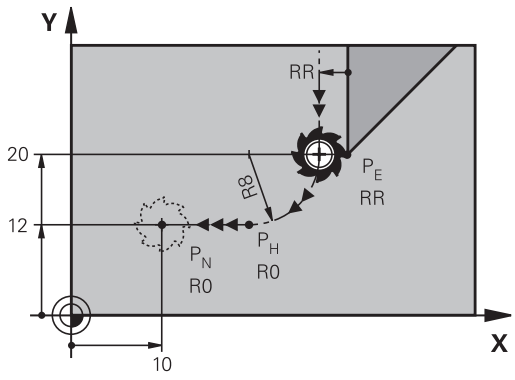
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT

The tool moves on a circular arc from the last contour point P_S to an auxiliary point P_H. It then moves on a straight line to the end point P_N. The arc is tangentially connected both to the last contour element and to the line from P_H to P_N. Once these lines are known, the radius R suffices to unambiguously define the tool path.

- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialog with the **APPR/DEP** key and **DEP LCT** soft key



- Enter the coordinates of the end point P_N
- Radius R of the circular arc. Enter R as a positive value



R0=G40; RL=G41; RR=G42

Example

N110 G01 Y+20 G42 F100*	; Approach the last contour element P_E with **G42**
N120 DEP LCT X+10 Y+12 R8 F100*	; Approach P_N; distance P_E to P_N: **R8**

5.4 Path contours — Cartesian coordinates

Overview of path functions

Key	Function	Tool movement	Required input	Page
L	Straight line **L** **G00** and **G01**	Straight line	Coordinates of the end point	167
CHF	Chamfer **CHF** **G24**	Chamfer between two straight lines	Chamfer side length	168
	Circle center **CC** **I** and **J**	None	Coordinates of the circle center or pole	170
C	Circular arc **C** **G02** and **G03**	Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation	171
CR	Circular arc **CR** **G05**	Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation	173
CT	Circular arc **CT** **G06**	Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point	175
RND	Corner rounding **RND** **G25**	Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding radius R	169
FK	**FK** free contour programming	Straight line or circular path with any connection to the preceding contour element	Input depends on the function	190

Programming path functions

You can program path functions conveniently by using the gray path function keys. In further dialogs, you are prompted by the control to make the required entries.

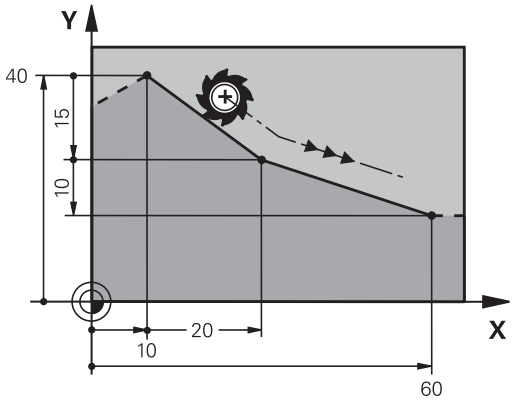
If you enter ISO functions on a keyboard connected through a USB port, make sure that capitalization is active.

At the start of the block the control automatically writes in capitals.

Straight line in rapid traverse G00 or straight line with feed rate F G01

The control moves the tool in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding NC block.

- Press the **L** key to open a program block for a linear movement with feed rate
- **Coordinates** of the end point of the straight line, if necessary
- **Radius compensation G40/G41/G42**
- **Feed rate F**
- **Miscellaneous function M**



Movement at rapid traverse

A straight line block for a rapid traverse motion (**G00** block) can also be initiated with the **L** key:

- Press the **L** key to open a program block for a linear movement
- Press the left arrow key to switch to the input range for G codes
- Press the **G00** soft key if you want to enter a rapid traverse motion

Example

```
N110 G00 G90 G40  Z+100 M3*
N120 G01 G41  X+10 Y+40  F200*
N130 G91  X+20 Y-15*
N140 G90  X+60 G91  Y-10*
```

Actual position capture

You can also generate a straight-line block (**G01** block) by using the **actual position capture** key:

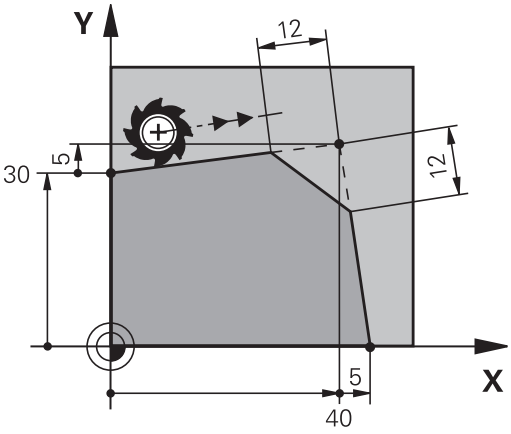
- In **Manual Operation** mode, move the tool to the position you want to capture
- Switch the screen display to programming
- Select the NC block after which you want to insert the straight line block

- Press the **actual position capture** key
- > The control generates a straight-line block with the actual position coordinates.

Inserting a chamfer between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

- The line blocks before and after the **G24** block must be in the same working plane as the chamfer.
- The radius compensation before and after the **G24** block must be the same
- The chamfer must be machinable with the current tool



CHF

- **Chamfer side length**: Length of the chamfer, and if necessary:
- **Feed rate F** (effective only in **G24** block)

```
N70 G01  G41 X+0  Y+30 F300  M3*
N80 X+40  G91 Y+5*
N90 G24  R12 F250*
N100 G91  X+5 G90  Y+0*
```

You cannot start a contour with a **G24** block.

A chamfer is possible only in the working plane.

The corner point is cut off by the chamfer and is not part of the contour.

A feed rate programmed in the **G24** block is effective only in that CHF block. After the **G24** block, the previous feed rate becomes effective again.

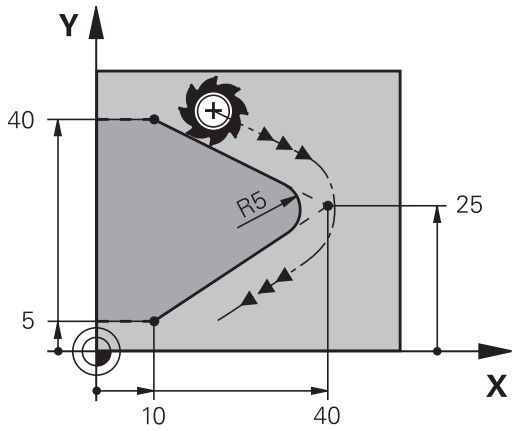
Rounded corners G25

The **G25** function creates rounding arcs at contour corners.

The tool moves on an arc that connects tangentially to both the preceding and subsequent contour elements.

The rounding arc must be machinable with the called tool.

- **Rounding radius**: Enter the radius, and if necessary:
- **Feed F** (effective only in the **G25** block)




```
N50 G01  X+10 Y+40  G41 F300  M3*
N60 G01  X+40 Y+25*
N70 G25  R5 F100*
N80 G01  X+10 Y+5*
```

In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the plane.

The tool will not move to the corner point.

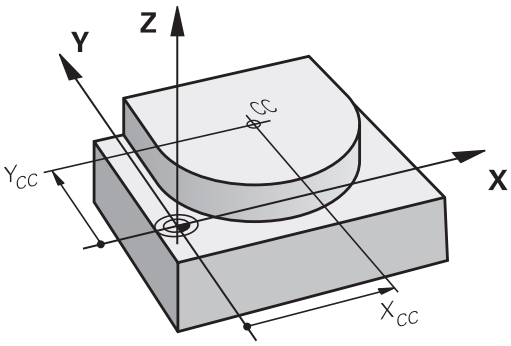
A feed rate programmed in the **G25** block is effective only in that **G25** block. After the **G25** block, the previous feed rate becomes effective again.

You can also use an **G25** block for a tangential contour approach.

Circle center I, J

You can define a circle center for circles that you program with the **G02**, **G03**, or **G05** function. This is done in the following ways:

- Enter the Cartesian coordinates of the circle center in the working plane, or
- Use the position last programmed, or
- Take over the coordinates with the **Actual-position-capture** key



SPEC FCT

- To program the circle center, press the **SPEC FCT** key
- Press the **PROGRAM FUNCTIONS** soft key
- Press the **DIN/ISO** soft key
- Press the **I** or **J** soft key
- Enter coordinates for the circle center or, if you want to use the last programmed position, enter **G29**

```
N50 I+25  J+25*
```

or

```
N10 G00 G40  X+25 Y+25*
N20 G29*
```

The program blocks 10 and 20do not refer to the illustration.

Validity

The circle center definition remains in effect until you program a new circle center.

Entering the circle center incrementally

If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

The only effect of **I** and **J** is to define a position as circle center: The tool does not move to this position.

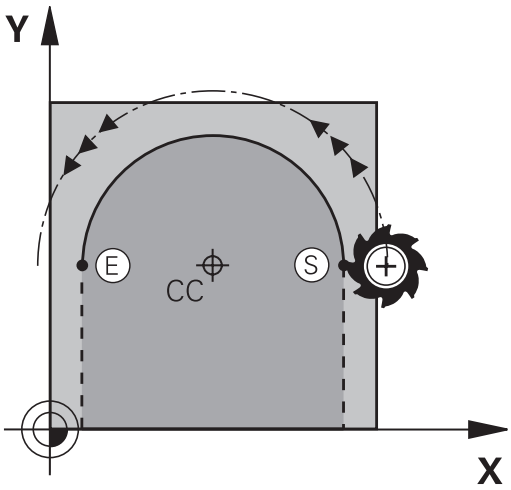
The circle center is also the pole for polar coordinates.

Circular arc around circle center

Before programming a circular arc C, you must first specify the circle center **I, J**. The last programmed tool position will be the starting point of the arc.

Direction of rotation

- Clockwise: **G02**
- Counterclockwise: **G03**
- Without programmed direction: **G05**. The control moves along the circular path with the last programmed direction of rotation.



▶ Move the tool to the starting point of the circle

J

I

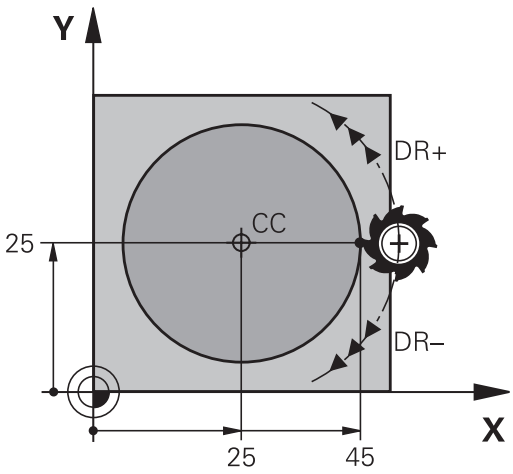
▶ Enter the **coordinates** of the circle center

C

▶ Enter the **coordinates** of the arc end point, if necessary:

▶ **Feed F**

▶ **Miscellaneous function M**




```
N50 I+25  J+25*
N60 G01 G42  X+45 Y+25  F200 M3*
N70 G03  X+45 Y+25*
```

Circular motion in another plane

The control normally makes circular movements in the active working plane. However, you can also program circular arcs that do not lie in the active working plane.

Example

```
N30 T1 G17 S4000*
N50 I+25  K+25*
N60 G01 G42  X+45 Y+25 Z+25 F200 M3*
N70 G03  X+45 Z+25*
```

By simultaneously rotating these circular movements you can create spatial arcs (arcs in three axes).

Full circle

For the end point, program the same coordinates as for the starting point.

The starting and end points of the arc must lie on the circle.

The maximum value for input tolerance is 0.016 mm. Set the input tolerance in the machine parameter **circleDeviation** (no. 200901).

Smallest possible circle that the control can traverse: 0.016 mm.

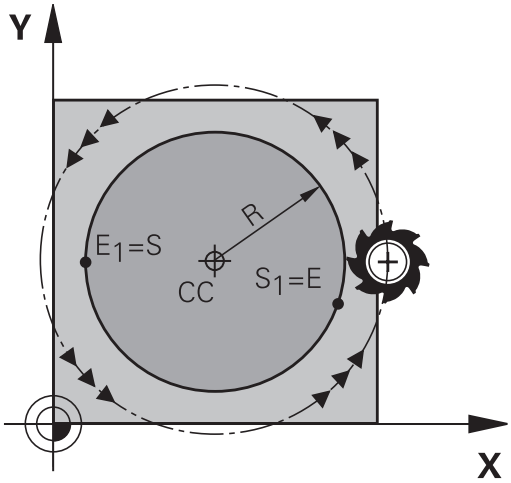
Circular arc G02/G03/G05 with fixed radius

The tool moves on a circular path with the radius R.

Direction of rotation

- Clockwise: **G02**
- Counterclockwise: **G03**
- Without programmed direction: **G05**. The control moves along the circular path with the last programmed direction of rotation.

- **Coordinates** of the arc end point
- Note on **Radius R**: The algebraic sign determines the size of the arc!
- **Miscellaneous function M**
- **Feed F**



Full circle

For a full circle, program two semicircle blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.

Central angle CCA and arc radius R

The starting and end points on the contour can be connected with four arcs of the same radius:

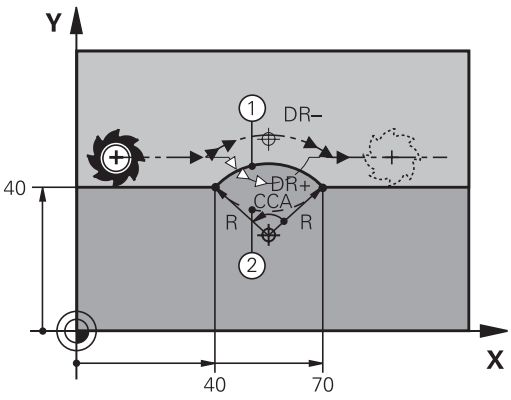
Smaller arc: CCA<180°
Enter the radius with a positive sign, i.e. R>0

Larger arc: CCA>180°
Enter the radius with a negative sign, i.e. R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation **G02** (with radius compensation **G41**)

Concave: Direction of rotation **G03** (with radius compensation **G41**)



The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.

The maximum radius is 99.9999 m.

You can also enter rotary axes A, B and C.

The control normally makes circular movements in the active working plane. However, you can also program circular arcs that do not lie in the active working plane. By simultaneously rotating these circular movements you can create spatial arcs (arcs in three axes).

N100 G01 G41 X+40 Y+40 F200 M3*	
N110 G02 X+70 Y+40 R+20*	; Circular path 1

or

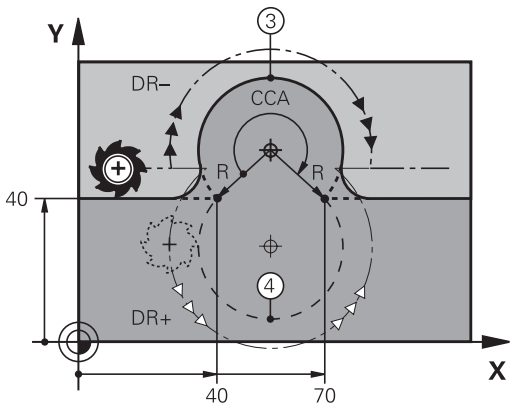
N110 G03 X+70 Y+40 R+20*	; Circular path 2

or

N110 G02 X+70 Y+40 R-20*	; Circular path 3

or

N110 G03 X+70 Y+40 R-20*	; Circular path 4

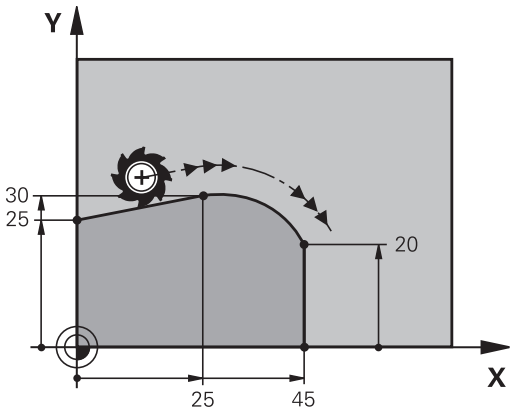


Circular arc G06 with tangential transition

The tool moves on an arc that connects tangentially to the previously programmed contour element.

A connection between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the **G06** block. This requires at least two positioning blocks.



CT

- **Coordinates** of the arc end point, and if necessary:
- **Feed F**
- **Miscellaneous function M**

```
N70 G01 G41  X+0 Y+25  F300 M3*
N80 X+25  Y+30*
N90 G06  X+45 Y+20*
N100 G01 Y+0*
```

A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc!

Superimposing a linear motion on a circular contour

It is possible to superimpose a linear motion on a circular contour defined in Cartesian coordinates, e.g. in order to create a helix.

Superimposed linear motions are possible for the following types of circular contours:

- Circular contour **C**

  Further information: "Circular arc around circle center ", Page 171
- Circular contour **CR**

  Further information: "Circular arc G02/G03/G05 with fixed radius", Page 173
- Circular contour **CT**

  Further information: "Circular arc G06 with tangential transition", Page 175

> The tangential transition is effective only for the axes in the circular plane, and not also for the superimposed linear motion.

As an alternative, you can superimpose a circular contour defined in polar coordinates on a linear motion.

Further information: "Helix", Page 183

Input notes

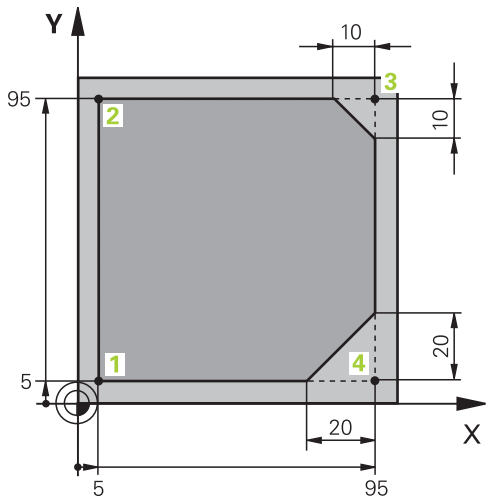
You can define a circular contour **G02**, **G03**, or **G05** by specifying three axes using free syntax input.

Further information: "Freely editing an NC program", Page 207

Example

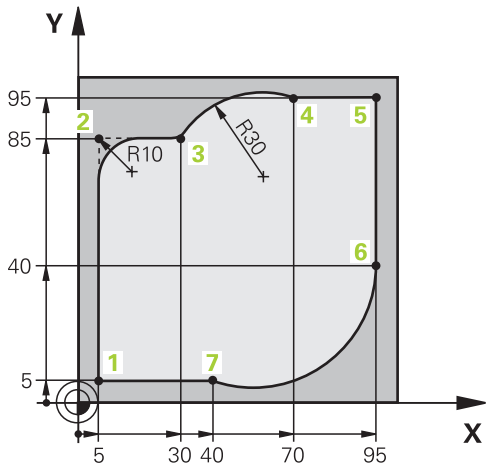
N110 G03 X+50 Y+50 Z-3 R +50*	; Circular contour with linear Z-axis superimposition

Example: Linear movements and chamfers with Cartesian coordinates



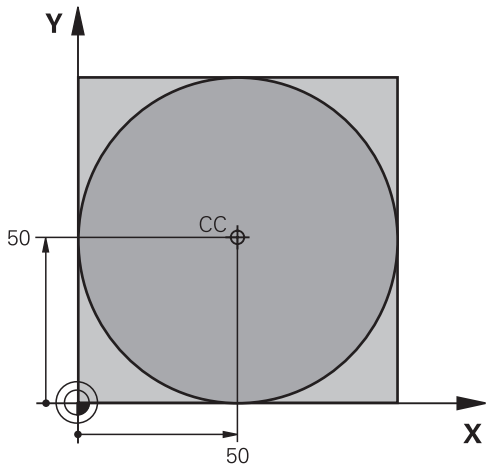
%LINEAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Define the workpiece blank for machining simulation
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S4000*	Call the tool in the spindle axis and with spindle speed
N40 G00 G40 G90 Z+250*	Retract the tool in the spindle axis at rapid traverse
N50 X-10 Y-10*	Pre-position the tool
N60 G01 Z-5 F1000 M3*	Move to working depth at feed rate F = 1000 mm/min
N70 G01 G41 X+5 Y+5 F300*	Approach the contour at point 1, activate radius compensation G41
N80 G26 R5 F150*	Tangential approach
N90 Y+95*	Move to point 2
N100 X+95*	Program the first straight line for corner 3
N110 G24 R10*	Program a chamfer with length 10 mm
N120 Y+5*	Program the second straight line for corner 3 and the first straight line for corner 4
N130 G24 R20*	Program a chamfer with length 20 mm
N140 X+5*	Program the second straight line for corner 4 and approach the last contour point
N150 G27 R5 F500*	Tangential exit
N160 G40 X-20 Y-20 F1000*	Retract the tool in the working plane, cancel radius compensation
N170 G00 Z+250 M2*	Retract the tool, end program
N99999999 %LINEAR G71 *	

Example: Circular movements with Cartesian coordinates



%CIRCULAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Define the workpiece blank for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S4000*	Call the tool in the spindle axis and with spindle speed
N40 G00 G40 G90 Z+250*	Retract the tool in the spindle axis at rapid traverse
N50 X-10 Y-10*	Pre-position the tool
N60 G01 Z-5 F1000 M3*	Move to working depth at feed rate F = 1000 mm/min
N70 G01 G41 X+5 Y+5 F300*	Move to the contour at point 1 with radius compensation G41
N80 G26 R5 F150*	Tangential approach
N90 Y+85*	Program the first straight line for corner 2
N100 G25 R10*	Program a rounding with R = 10 mm, feed rate F = 150 mm/min
N110 X+30*	Move to point 3: starting point of the circular path
N120 G02 X+70 Y+95 R+30*	Move to point 4: end point of the circular path with G02 and radius R = 30 mm
N130 G01 X+95*	Move to point 5
N140 Y+40*	Move to point 6
N150 G06 X+40 Y+5*	Move to point 7: end point of the circular path, arc with tangential connection to point 6; the control calculates the radius automatically
N160 G01 X+5*	Move to last contour point 1
N170 G27 R5 F500*	Depart the contour on a circular path with tangential connection
N180 G40 X-20 Y-20 F1000*	Retract the tool in the working plane, cancel radius compensation
N190 G00 Z+250 M2*	Retract the tool in the tool axis, end of program
N99999999 %CIRCULAR G71 *	

Example: Full circle with Cartesian coordinates



%C-CC G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S3150*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 I+50 J+50*	Define the circle center
N60 X-40 Y+50*	Pre-position the tool
N70 G01 Z-5 F1000 M3*	Move to working depth
N80 G41 X+0 Y+50 F300*	Approach starting point, radius compensation G41
N90 G26 R5 F150*	Tangential approach
N100 G02 X+0*	Move to the circle end point (= circle starting point)
N110 G27 R5 F500*	Tangential exit
N120 G01 G40 X-40 Y-50 F1000*	Retract the tool in the working plane, cancel radius compensation
N130 G00 Z+250 M2*	Retract tool in the tool axis, end of program
N99999999 %C-CC G71 *	

5.5 Path contours – Polar coordinates

Overview

With polar coordinates you can define a position in terms of its angle **H** and its distance **R** relative to a previously defined pole **I, J**.

Polar coordinates are useful with:

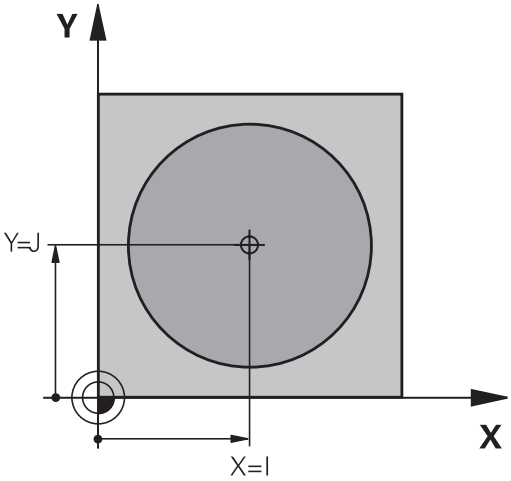
- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

Key	Tool movement	Required input	Page
L + P	Straight line	Polar radius, polar angle of the straight-line end point	181
C + P	Circular path around circle center/pole to arc end point	Polar angle of the arc end point,	182
CR + P	Circular path corresponding to active direction of rotation	Polar angle of the circle end point	182
CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point	182
C + P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis	183

Datum for polar coordinates: pole I, J

You can set the pole (I, J) at any point in the NC program, before indicating positions in polar coordinates. Set the pole in the same way as you would program the circle center.



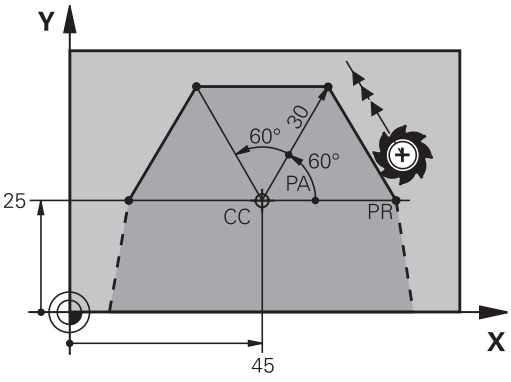
SPEC FCT

- To program a pole, press the **SPEC FCT** key.
- Press the **PROGRAM FUNCTIONS** soft key
- Press the **DIN/ISO** soft key
- Press the **I** or **J** soft key
- **Coordinates**: Enter Cartesian coordinates for the pole or, if you want to use the last programmed position, enter **G29**. Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.

```
N110 I+30  J+10*
```

Straight line in rapid traverse G10 or straight line with feed rate F G11

The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding NC block.



L

- **Polar coordinate radius R**: Enter the distance from the pole CC to the straight-line end point

P

- **Polar-coordinates angle H**: Angular position of the straight-line end point between –360° and +360°

The sign of **H** depends on the angle reference axis:

- If the angle from the angle reference axis to **R** is counterclockwise: **H**>0
- If the angle from the angle reference axis to **R** is clockwise: **H**<0

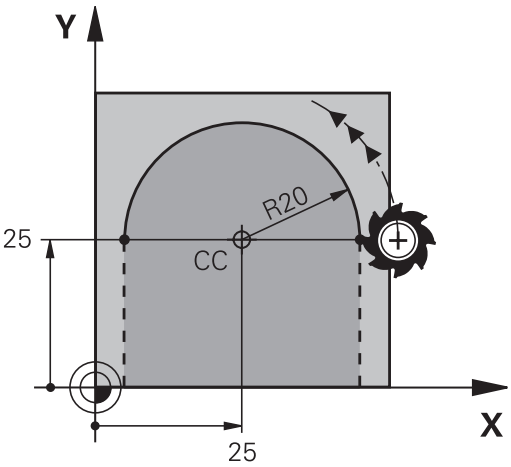
```
N120 I+45  J+45*
N130 G11 G42  R+30 H+0  F300 M3*
N140 H+60*
N150 G91  H+60*
N160 G90  H+180*
```

Circular path G12/G13/G15 around pole I, J

The polar coordinate radius **R** is also the radius of the arc. **R** is defined by the distance from the starting point to the pole **I, J**. The last programmed tool position will be the starting point of the arc.

Direction of rotation

- Clockwise: **G12**
- Counterclockwise: **G13**
- Without programmed direction: **G15**. The control moves along the circular path with the last programmed direction of rotation.



C

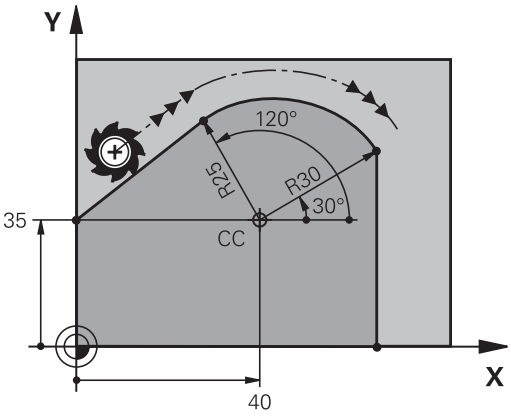
P

- **Polar-coordinates angle H**: Angular position of the arc end point between –99999.9999° and +99999.9999°

```
N180 G11 G42  R+20 H+0  F250 M3*
N190 I+25  J+25*
N200 G13  H+180*
```

Circle G16 with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.



CT

P

- **Polar coordinate radius R**: Distance between the arc end point and the pole **I, J**
- **Polar coordinate angle H**: Angular position of the arc end point.

The pole is **not** the center of the contour circle!

Example

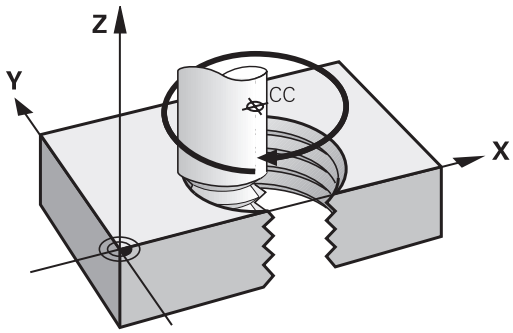
```
N120 G01 G42  X+0 Y+35  F250 M3*
N130 I+40  J+35*
N140 G11  R+25 H+120*
N150 G16  R+30 H+30*
N160 G01  Y+0*
```

Helix

A helix is a combination of a circular motion defined in polar coordinates and a linear motion perpendicular to this plane. Program the circular path in a main plane.

As an alternative, you can superimpose a circular contour defined in Cartesian coordinates on a linear motion.

Further information: "Superimposing a linear motion on a circular contour", Page 176



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

Thread revolutions n:	Thread revolutions + thread overrun at the start and end of the thread
Total height h:	Thread pitch P times thread revolutions n
Incremental total angle **G91 H**:	Thread revolutions x 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z:	Pitch P times (thread revolutions + thread overrun at start of thread)

Shape of the helix

The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

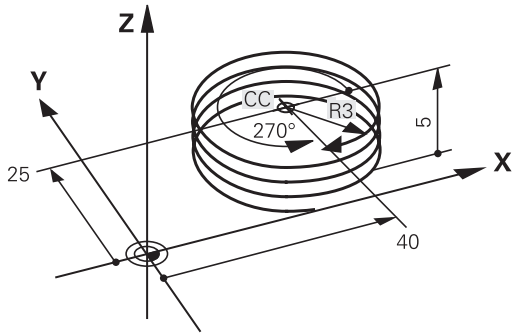
Internal thread	Work direction	Direction of rotation	Radius compensation
Right-hand	Z+	**G13**	**G41**
Left-hand	Z+	**G12**	**G42**
Right-hand	Z–	**G12**	**G42**
Left-hand	Z–	**G13**	**G41**
External thread			
Right-hand	Z+	**G13**	**G42**
Left-hand	Z+	**G12**	**G41**
Right-hand	Z–	**G12**	**G41**
Left-hand	Z–	**G13**	**G42**

Programming a helix

In **G13**, define a positive total incremental angle **G91 H** and in **G14**, a negative total angle. The tool may otherwise move on a wrong path.

For the total angle **G91 h** you can enter a value of -99 999.9999° to +99 999.9999°.

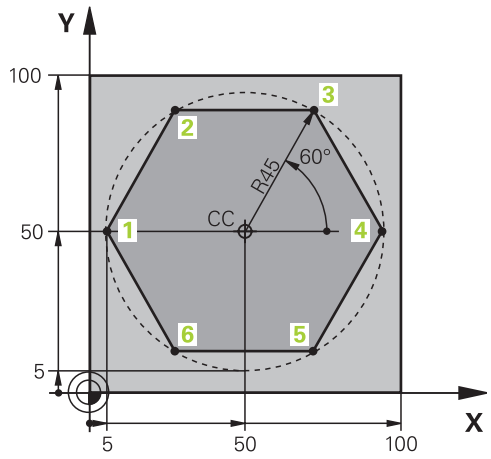
- **Polar coordinates angle:** Enter the total angle of tool traverse along the helix in incremental dimensions.
- **After entering the angle, specify the tool axis with an axis selection key**
- **Coordinate**: Enter the coordinate for the height of the helix in incremental dimensions
- **Enter the radius compensation** according to the table



Example: Thread M6 x 1 mm with 5 revolutions

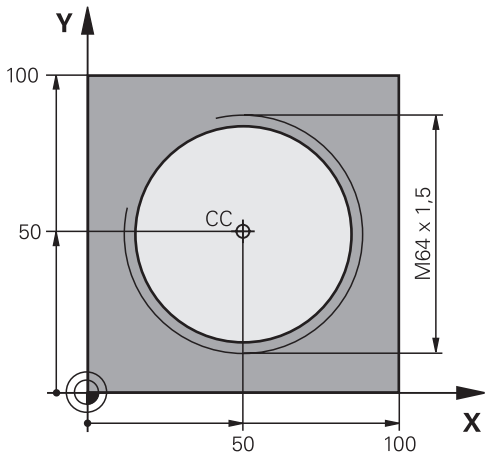
```
N120 G01  Z+0 F100  M3*
N130 I+40  J+25*
N140 G11 G41  R+3 H+270*
N150 G12 G91  H-1800 Z+5*
```

Example: Linear movement with polar coordinates



%LINEARPO G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 G90 X+100 Y+100 z+0*	
N30 T1 G17 S4000*	Tool call
N40 G00 G40 G90 Z+250*	Define the preset for polar coordinates
N50 I+50 J+50*	Retract the tool
N60 G10 R+60 H+180*	Pre-position the tool
N70 G01 Z-5 F1000 M3*	Move to working depth
N80 G11 G41 R+45 H+180 F250*	Approach the contour at point 1
N90 G26 R5*	Approach the contour at point 1
N100 H+120*	Move to point 2
N110 H+60*	Move to point 3
N120 H+0*	Move to point 4
N130 H-60*	Move to point 5
N140 H-120*	Move to point 6
N150 H+180*	Move to point 1
N160 G27 R5 F500*	Tangential exit
N170 G40 R+60 H+180 F1000*	Retract the tool in the working plane, cancel radius compensation
N180 G00 Z+250 M2*	Retract in the spindle axis, end of program
N99999999 %LINEARPO G71 *	

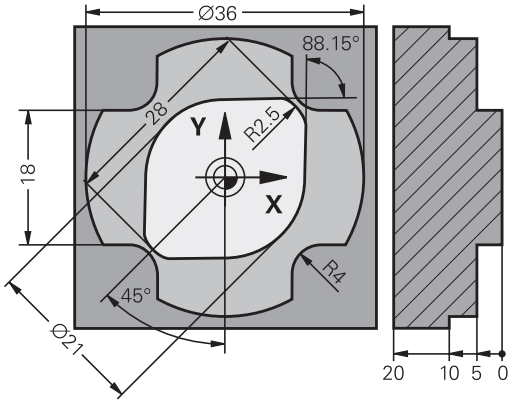
Example: Helix



%HELIX G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S1400*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 X+50 Y+50*	Pre-position the tool
N60 G29*	Transfer the last programmed position as the pole
N70 G01 Z-12,75 F1000 M3*	Move to working depth
N80 G11 G41 R+32 H+180 F250*	Approach first contour point
N90 G26 R2*	Connection
N100 G13 G91 H+3240 Z+13,5 F200*	Helical traverse
N110 G27 R2 F500*	Tangential exit
N120 G01 G40 G90 X+50 Y+50 F1000*	Retract the tool, end of program
N130 G00 Z+250 M2*	
N99999999 %HELIX G71 *	

5.6 Path contours – FK free contour programming

Fundamentals



Workpiece drawings that are not dimensioned for NC often contain unconventional coordinate data that cannot be entered with the gray path function keys.

You can enter such dimensional data directly by using the free contour programming function FK, e.g.

- If there are known coordinates on or in proximity to the contour element
- If coordinate data refers to another contour element
- If directional data and data regarding the course of the contour are known

The control derives the contour from the known coordinate data and supports the programming dialog with the interactive FK programming graphics. The figure at upper right shows a workpiece drawing for which FK programming is the most convenient programming method.

Programming notes

You must enter all available data for every contour element. Even the data that does not change must be entered in every NC block—otherwise it will not be recognized.

Q parameters are permissible in all FK elements, except in elements with relative references (e.g. **RX** or **RAN**), or in elements that are referenced to other NC blocks.

If both FK blocks and conventional blocks are entered in an NC program, the FK contour must be fully defined before you can return to conventional programming.

Program all of the contours before combining them (e.g., with the SL cycles). You thereby ensure that the contours are correctly defined and avoid unnecessary error messages.

The control needs a fixed point that it can use as the basis for all calculations. Use the gray path function keys to program a position that contains both coordinates of the working plane immediately before programming the FK contour. Do not enter any Q parameters in this NC block.

If the first NC block of an FK contour is an **FCT** or **FLT** block, you must program at least two NC blocks with the gray path function keys before it. This fully defines the approach direction.

Do not program an FK contour immediately after a **L** command.

You cannot combine the cycle call **M89** with FK programming.

Defining the working plane

The FK free contour programming feature can only be used for programming contour elements that lie in the working plane.

The control defines the working plane for FK programming according to the following hierarchy:

- Through the tool axis selected in the function **G30/G31**
- Through the plane defined in a **FPOL** block
- Through the defined working plane that is determined in the **TOOL CALLT** block (e.g. **G17** = X/Y plane)

If none of this applies, then the standard **X/Y** plane is active.

Display of the FK soft key depends on the spindle axis specified when defining the workpiece blank. If for example you enter spindle axis **G17** in the workpiece blank definition, the control only shows FK soft keys for the X/Y 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.

Switch the working plane

If you need a different working plane from the currently active plane, then proceed as follows:

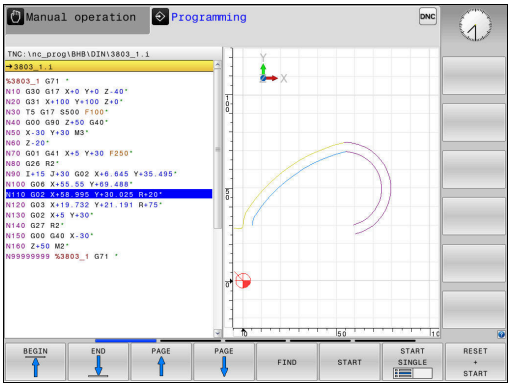
- ▶ Press the **PLANE XY ZX YZ** soft key
- > The control then displays the FK soft keys in the newly selected plane.

FK programming graphics

To use graphical support during FK programming, select the **PROGRAM + GRAPHICS** screen layout.

Further information: "Programming", Page 85

Program all of the contours before combining them (e.g., with the SL cycles). You thereby ensure that the contours are correctly defined and avoid unnecessary error messages.



Incomplete coordinate data often is not sufficient to fully define a workpiece contour. In this case, the control indicates the possible solutions in the FK graphic. You can then select the contour that matches the drawing.

The control uses various colors in the FK graphics:

- **blue:** uniquely specified contour element
  The last FK element is only shown in blue after the departure movement.
- **violet:** not yet uniquely specified contour element
- **ocher:** tool midpoint path
- **red:** rapid traverse
- **green:** more than one solution is possible

If the data permit several possible solutions and the contour element is displayed in green, select the correct contour element as follows:

SHOW SOLUTION

- Press the **SHOW SOLUTION** soft key repeatedly until the correct contour element is displayed. Use the zoom function if you cannot distinguish between possible solutions in the standard view

SELECT SOLUTION

- If the displayed contour element matches the drawing, then select this contour element with the **SELECT SOLUTION** soft key

If you do not yet wish to define a green contour element, then press the **START SINGLE** soft key to continue the FK dialog.

You should define the green contour elements as soon as possible with **SELECT SOLUTION** to limit ambiguity for the subsequent contour elements.

Showing block numbers in the graphic window

To show a block number in the graphic window:

SHOW BLOCK NO. OFF ON

- Set the **SHOW BLOCK NO.** soft key to **ON**

Initiating the FK dialog

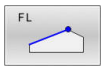
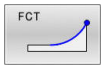
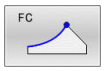
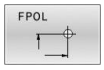
To open the FK dialog:

FK

- Press the **FK** key

> The control then displays the soft-key row with the FK functions.

If you initiate the FK dialog with one of these soft keys, the control shows additional soft-key rows. You can use them to enter known coordinates, directional data, and data regarding the course of the contour.

Soft key	FK element
	Straight line with tangential connection
	Straight line without tangential connection
	Circular arc with tangential connection
	Circular arc without tangential connection
	Pole for FK programming
	Select the working plane

Terminating the FK dialog

To exit the soft-key row for FK programming:

END

- Press the **END** soft key

Alternative:

FK

- Press the **FK** key again

Pole for FK programming

FK

- To display the soft keys for free contour programming, press the **FK** key

FPOL

- To initiate the dialog for defining the pole, press the **FPOL** soft key

> The control displays the axis soft keys of the active working plane.

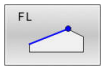
- Enter the pole coordinates using these soft keys

The pole for FK programming remains active until you define a new one using FPOL.

Free straight line programming

Straight line without tangential connection

- To display the soft keys for free contour programming, press the **FK** key



- To initiate the dialog for free programming of straight lines, press the **FL** soft key
- The control displays additional soft keys.
- Enter all known data in the NC block by using these soft keys
- The FK graphic displays the programmed contour element in violet until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green. **Further information:** "FK programming graphics", Page 189

Straight line with tangential connection

If the straight line connects tangentially to another contour element, initiate the dialog with the soft key:

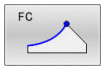
- To display the soft keys for free contour programming, press the **FK** key

- To initiate the dialog, press the **FLT** soft key
- Enter all known data in the NC block by using the soft keys

Free circular path programming

Circular arc without tangential connection

- To display the soft keys for free contour programming, press the **FK** key

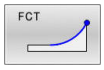


- To initiate the dialog for free programming of circular arcs, press the **FC** soft key
- The control displays soft keys with which you can enter direct data on the circular arc or data on the circle center.
- Enter all known data in the NC block by using these soft keys
- The FK graphic displays the programmed contour element in violet until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green.
  Further information: "FK programming graphics", Page 189

Circular arc with tangential connection

If the circular arc connects tangentially to another contour element, initiate the dialog with the **FCT** soft key:

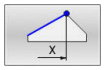
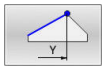
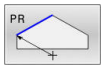
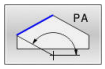
- To display the soft keys for free contour programming, press the **FK** key

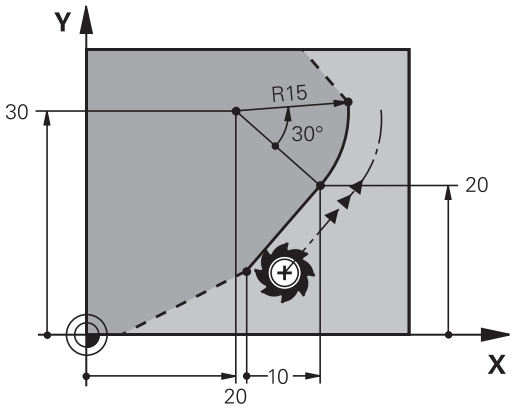


- To initiate the dialog, press the **FCT** soft key
- Enter all known data in the NC block by using the soft keys

Input possibilities

End point coordinates

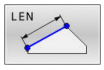
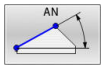
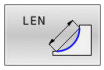
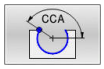
Soft keys		Known data
		Cartesian coordinates X and Y
		Polar coordinates referenced to FPOL

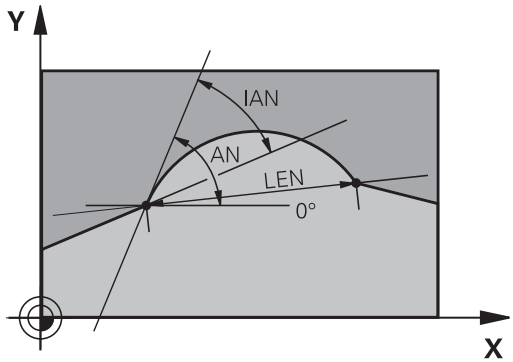


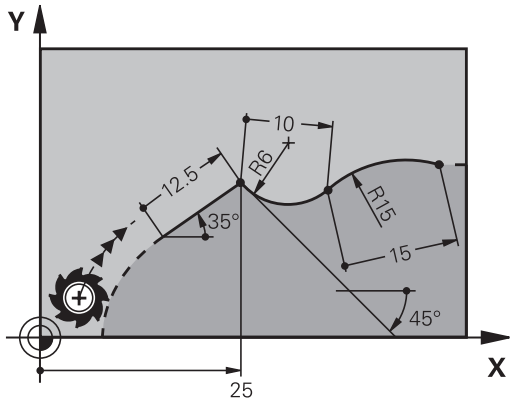
Example

```
N70 FPOL  X+20 Y+30*
N80 FL  IX+10 Y+20  G42 F100*
N90 FCT PR+15 IPA+30  DR+ R15*
```

Direction and length of contour elements

Soft keys	Known data
	Length of a straight line
	Gradient angle of a straight line
	Chord length LEN of an arc
	Gradient angle AN of an entry tangent
	Center angle of an arc





NOTICE

Danger of collision!

Incremental gradient angles **IAN** are referenced by the control to the direction of the previous traversing block. NC programs from previous control models (including iTNC 530) are not compatible. There is danger of collision during the execution of imported NC programs!

- Check the sequence and contour with the aid of the graphic simulation
- Adapt imported NC programs if required

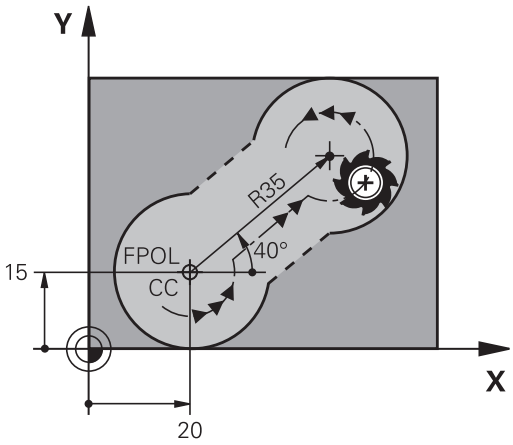
Example

```
N20 FLT  X+25 LEN 12.5  AN+35 G41  F200*
N30 FC  DR+ R6  LEN 10 AN-45*
N40 FCT  DR- R15  LEN 15*
```

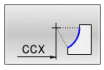
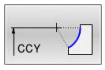
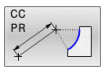
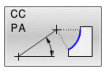
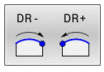
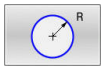
Circle center CC, radius and direction of rotation in the FC/FCT block

The control calculates a circle center for free-programmed arcs from the data you enter. This makes it possible to program full circles in an NC block with FK programming.

If you wish to define the circle center in polar coordinates you must use FPOL, not **CC**, to define the pole. FPOL is entered in Cartesian coordinates and remains in effect until the TNC encounters an NC block in which another **FPOL** is defined.



> A programmed or automatically calculated circle center or pole is effective only in connected conventional or FK sections. If an FK section splits up two conventionally programmed sections, the information about a circle center or pole will be lost. The two conventionally programmed sections must each have their own (if necessary, identical) CC blocks. Conversely, this information will also be lost if there is a conventional section between two FK sections.

Soft keys		Known data
		Circle center in Cartesian coordinates
		Center point in polar coordinates
		Rotational direction of the arc
		Radius of an arc

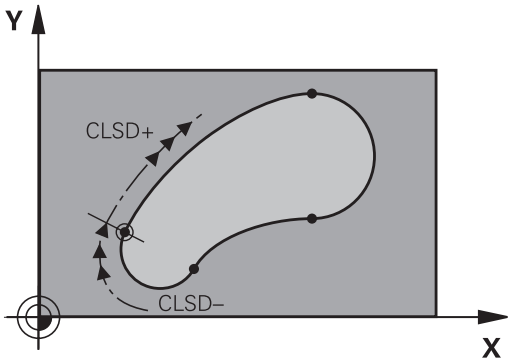
Example

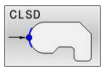
```
N10 FC  CCX+20 CCY+15  DR+ R15*
N20 FPOL  X+20 Y+15*
N30 FL  AN+40*
N40 FC  DR+ R15  CCPR+35 CCPA+40*
```

Closed contours

You can identify the beginning and end of a closed contour with the **CLSD** soft key. This reduces the number of possible solutions for the last contour element.

Enter **CLSD** as an addition to another contour data entry in the first and last NC blocks of an FK section.



Soft key	Known data	
	Beginning of contour:	CLSD+
	End of contour:	CLSD–

Example

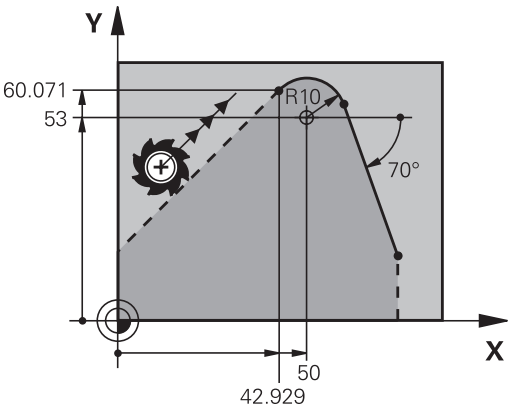
```
N10 G01  X+5 Y+35  G41 F500  M3*
N20 FC  DR- R15  CLSD+ CCX+20  CCY+35*
...
N30 FCT  DR- R+15  CLSD-*
```

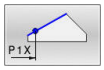
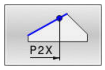
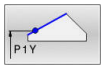
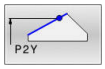
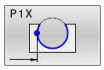
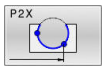
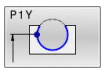
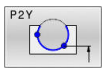
Auxiliary points

For both free-programmed straight lines and free-programmed circular arcs, you can enter the coordinates of auxiliary points that are located on the contour or in its proximity.

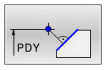
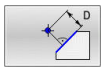
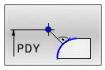
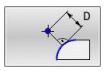
Auxiliary points on a contour

The auxiliary points are located on the straight line, the extension of the straight line, or on the circular arc.



Soft keys		Known data
		X coordinate of an auxiliary point P1 or P2 of a straight line
		Y coordinate of an auxiliary point P1 or P2 of a straight line
		X coordinate of an auxiliary point P1, P2 or P3 of a circular path
		Y coordinate of an auxiliary point P1, P2 or P3 of a circular path

Auxiliary points near a contour

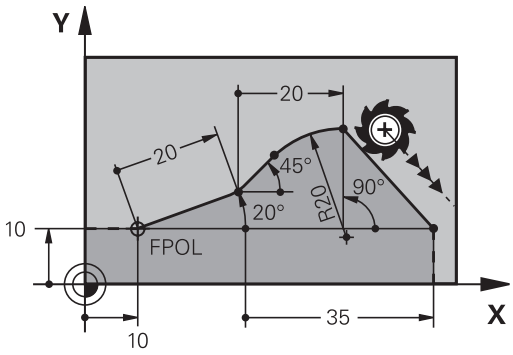
Soft keys		Known data
		X and Y coordinates of the auxiliary point near a straight line
		Distance of auxiliary point to straight line
		X and Y coordinates of an auxiliary point near a circular arc
		Distance of auxiliary point to circular arc

Example

```
N10 FC  DR- R10  P1X+42.929 P1Y+60.071*
N20 FLT  AN-70 PDX+50  PDY+53 D10*
```

Relative data

Relative data are values based on another contour element. The soft keys and program words for relative entries begin with the letter **R**. The figure on the right shows the dimensional data that should be programmed as relative data.



The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the NC block number of the contour element on which the data are based.

The block number of the contour element on which the relative data are based can only be located up to 64 positioning blocks before the NC block in which you program the reference.

If you delete an NC block on which relative data are based, the control will display an error message. Change the NC program first before you delete this NC block.

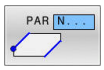
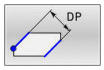
Data relative to NC block N: End point coordinates

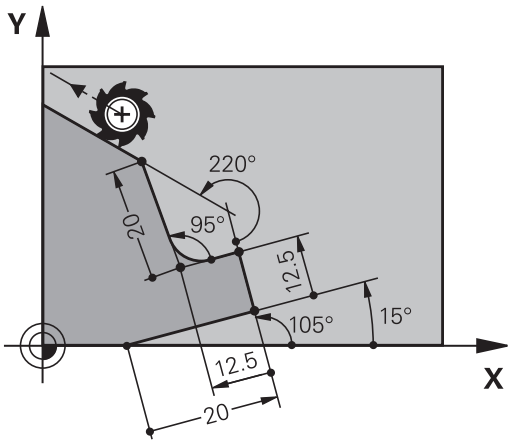
Soft keys	Known data
RX N... RY N...	Cartesian coordinates relative to NC block N
RPR N... RPA N...	Polar coordinates relative to NC block N

Example

```
N10 FPOL  X+10 Y+10*
N20 FL  PR+20 PA+20*
N30 FL  AN+45*
N40 FCT  IX+20 DR-  R20 CCA+90  RX 20*
N50 FL  IPR+35 PA+0  RPR 20*
```

Data relative to NC block N: Direction and distance of the contour element

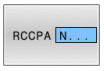
Soft key	Known data
	Angle between a straight line and another element or between the entry tangent of the arc and another element
	Straight line parallel to another contour element
	Distance from a straight line to a parallel contour element

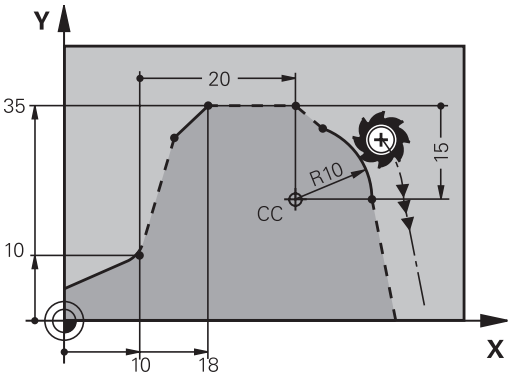


Example

```
N10 FL  LEN 20 AN+15*
N20 FL  AN+105 LEN 12.5*
N30 FL  PAR 10  DP 12.5*
N40 FSELECT 2*
N50 FL  LEN 20 IAN+95*
N60 FL  IAN+220 RAN 20*
```

Data relative to NC block N: Circle center CC

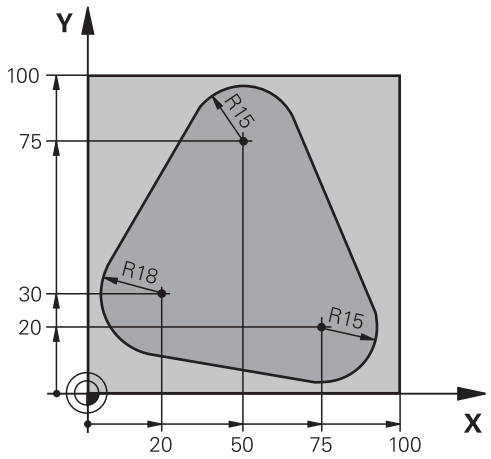
Soft key	Known data
	Cartesian coordinates of the circle center relative to NC block N
	Polar coordinates of the circle center relative to NC block N



Example

```
N10 FL  X+10 Y+10  G41*
N20 FL  ...*
N30 FL  X+18 Y+35*
N40 FL  ...*
N50 FL  ...*
N60 FC  DR- R10  CCA+0 ICCX+20  ICCY-15 RCCX10  RCCY30*
```

Example: FK programming 1



%FK1 G71 *	
N10 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N20 G31 X+100 Y+100 Z+0*	
N30 T 1 G17 S500*	Tool call
N40 G00 G90 Z+250 G40 M3*	Retract the tool
N50 G00 X-20 Y+30 G40*	Pre-positioning the tool
N60 G01 Z-10 G40 F1000*	Move to working depth
N70 APPR CT X+2 Y+30 CCA90 R+5 G41 F250*	Approach the contour on a circular arc with tangential connection
N80 FC DR- R18 CLSD+ CCX+20 CCY+30*	FK contour section:
N90 FLT*	Program all known data for each contour element
N100 FCT DR- R15 CCX+50 CCY+75*	
N110 FLT*	
N120 FCT DR- R15 CCX+75 CCY+20*	
N130 FLT*	
N140 FCT DR- R18 CLSD- CCX+20 CCY+30*	
N150 DEP CT CCA90 R+5 F2000*	Depart the contour on a circular arc with tangential connection
N160 G00 X-30 Y+0*	
N170 G00 Z+250 M2*	Retract the tool, end of program
N99999999 %FK1 G71 *	

6

Programming Aids

6.1 GOTO function

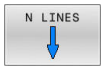
Using the GOTO key

Jumping with the GOTO key

Use the **GOTO** key to jump to a specific location in the NC program, regardless of the active operating mode.

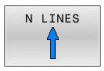
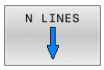
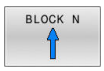
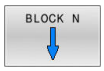
Proceed as follows:

- Press the **GOTO** key
- The control opens a pop-up window.
- Enter a number



- Select the jump statement by soft key, e.g. move down the number of lines entered

The control provides the following options:

Soft key	Function
	Move up the number of lines entered
	Move down the number of lines entered
	Jump to the block number entered
	Jump to the block number entered

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

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Quick selection with the GOTO key

With the **GOTO** key, you can open the Smart Select window that makes it easy for you to select special functions or cycles.

To select special functions:

- Press the **SPEC FCT** key

- Press the **GOTO** key
- The control displays a pop-up window showing a structural view of the special functions
- Select the desired function

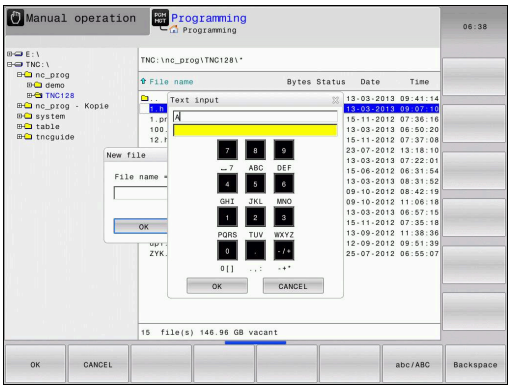
Further information: User's Manual for **Programming of Machining Cycles**

Opening the selection window with the GOTO key

When the control provides a selection menu, you can use the **GOTO** key to open the selection window. This allows you to view the available entries.

6.2 Screen keypad

If you are using the compact version of the control (without alphabetic keyboard), you can enter letters and special characters with the screen keypad or with an alphabetic keyboard connected to the USB port.



Entering text with the screen keypad

To use the screen keypad:

GOTO

- Press the **GOTO** key if you want to enter letters, e.g. a program name or directory name, using the screen keypad.

> The control opens a window in which the numeric keypad of the control is displayed with the corresponding letters assigned.

8

- Press the numerical key until the cursor is on the desired letter
- Wait until the control transfers the selected character before you enter the next character

OK

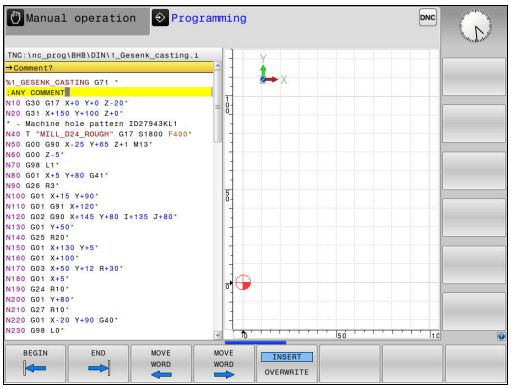
- Use the **OK** soft key to load the text into the open dialog field

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

6.3 Display of NC programs

Syntax highlighting

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



Color highlighting of syntax elements

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

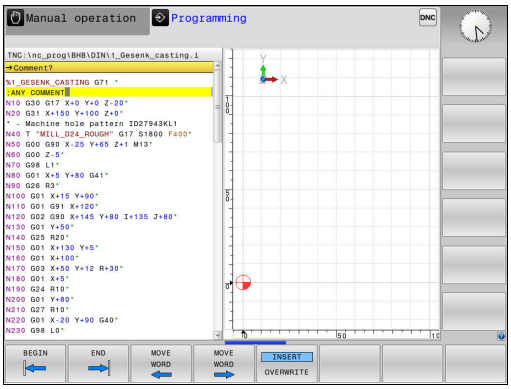
Scrollbar

Screen content can be shifted with the mouse using the scroll bar at the right edge of the program window. In addition, the size and position of the scrollbar indicates program length and cursor position.

6.4 Adding comments

Application

You can add comments to an NC program to explain program steps or make general notes.



> The control shows long comments in different ways, depending on the machine parameter **lineBreak** (no. 105404). It either wraps the comment lines or displays the **>>** symbol to indicate additional content.
>
> The last character in a comment block must not be a tilde sign (**~**).

You can add comments in different ways.

Entering comments during programming

> To use this function you will need a alphabetic keyboard connected via USB.

- Enter the data for an NC block
- Press the semicolon key **;** on the alphabetic keyboard
- The control displays the dialog prompt **Comment?**
- Enter the comment
- Press the **END** key to conclude the NC block

Inserting comments after program entry

> To use this function you will need a alphabetic keyboard connected via USB.

- Select the NC block to which you want to add the comment
- Select the last word in the NC block with the right arrow key:
- Press the semicolon key **;** on the alphabetic keyboard
- The control displays the dialog prompt **Comment?**
- Enter the comment
- Press the **END** key to conclude the NC block

Entering a comment in a separate NC block

> To use this function you will need a alphabetic keyboard connected via USB.

- Select the NC block after which you want to insert the comment
- Initiate the programming dialog with the semicolon key (**;**) on the alphabetic keyboard
- Enter your comment and conclude the NC block by pressing the **END** key

Commenting out an existing NC block

To change an existing NC block into a comment:

- Select the NC block to be commented out

- Press the **INSERT COMMENT** soft key
- The control inserts a semicolon **;** at the beginning of the block.
- Press the **END** key

Changing a comment into an NC block

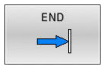
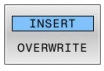
To change a commented-out NC block into an active NC block:

- Select the comment block you want to change

- Press the **REMOVE COMMENT** soft key

  Alternative:
- Press the **>** key on the alphabetic keyboard
- The control removes the semicolon **;** at the beginning of the block.
- Press the **END** key

Functions for editing a comment

Soft key	Function
	Jump to beginning of comment
	Jump to end of comment
	Jump to the beginning of a word. Use a space to separate words
	Jump to the end of a word. Use a space to separate words
	Switch between insert and overwrite mode

6.5 Freely editing an NC program

Certain syntax elements, such as LN blocks, cannot be entered directly in the NC editor by using the available keys and soft keys.

To prevent the use of an external text editor, the control offers the following possibilities:

- Free syntax input using the control's integrated text editor
- Free syntax input using the **?** key in the NC editor

Free syntax input using the control's integrated text editor

To add syntax to an existing NC program:

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

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

- ▶ Press the **SELECT EDITOR** soft key
- > The control opens a selection window.

- ▶ Select the **TEXT EDITOR** option
- ▶ Confirm your selection with **OK**
- ▶ Add the desired syntax

The control does not check the syntax in the text editor. Check your entries in the NC editor when you are finished.

Free syntax input using the ? key in the NC editor

To use this function you will need a alphabetic keyboard connected via USB.

To add syntax to an existing, open NC program:

- ▶ Enter **?**
- > The control opens a new NC block.

- ▶ Add the desired syntax
- ▶ Confirm your entry with **END**

After confirmation, the control checks the syntax. Errors will result in **ERROR** blocks.

6.6 Skipping NC blocks

Insert a slash (/)

You can optionally hide NC blocks.

To hide NC blocks in the **Programming** mode:

- Select the desired NC block

- Press the **INSERT** soft key

> The control inserts a slash (/).

Delete the slash (/)

To show NC blocks again in the **Programming** mode:

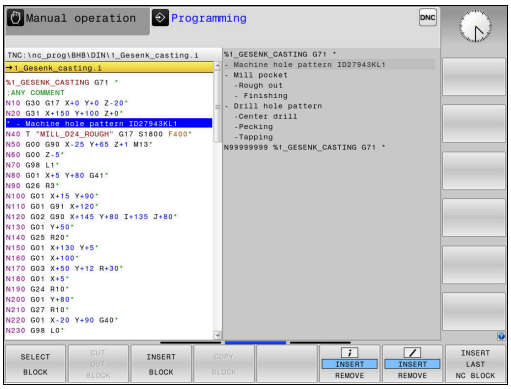
- Select the hidden NC block

- Press the **REMOVE** soft key

> The control removes the slash (/).

6.7 Structuring NC programs

Definition and applications



The control enables you to comment NC programs in structuring blocks. Structuring blocks are texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex NC programs in a clear and comprehensible manner.

This function is particularly convenient if you want to change the NC program later. Structuring blocks can be inserted into the NC program at any point.

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

The control manages the inserted structure items in a separate file (extension: .SEC.DEP). This speeds navigation in the program structure window.

The **PROGRAM + SECTS** screen layout can be selected in the following operating modes:

- **Program run, single block**
- **Program run, full sequence**
- **Programming**

Displaying the program structure window / Changing the active window

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

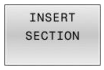
- Change the active window: Press the **CHANGE WINDOW** soft key

Inserting a structure block in the program window

- Select the NC block after which you want to insert the structuring block

- Press the **SPEC FCT** key

- Press the **PROGRAM- MING AIDS** soft key



- Press the **INSERT SECTION** soft key
- Enter the structuring text

- Change the structuring depth (indenting) via soft key

You can indent structure items only during editing.

You can also insert structure blocks with the key combination **Shift + 8**.

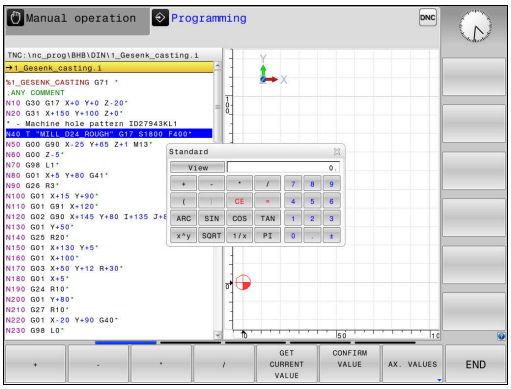
Selecting blocks in the program structure window

If you are scrolling through the program structure window block by block, the control at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

6.8 Calculator

Operation

The control features a calculator with the most important mathematical functions.



- To show the calculator, press the **CALC** key
- Select the arithmetic functions: Select the command via soft key or enter it with an alphanumeric keyboard
- To close the calculator, press the **CALC** key

Calculator function	Command (soft key)
Addition	+
Subtraction	–
Multiplication	*
Division	/
Calculating with parentheses	( )
Arc cosine	ARC
Sine	SIN
Cosine	COS
Tangent	TAN
Exponent	X^Y
Square root	SQRT
Inverted function	1/x
PI (3.14159265359)	PI
Add value to buffer memory	M+
Save value to buffer memory	MS
Retrieve buffer memory contents	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check the algebraic sign	SGN
Calculate the absolute value	ABS

Calculator function	Command (soft key)
Truncate decimal places	INT
Truncate digits before the decimal point	FRAC
Modulo	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Show angular value in radians (default: angular value in degrees)	RAD
Select numerical value notation	DEC (decimal) or HEX (hexadecimal)

Transferring the calculated value into the NC program

- With the arrow keys, select the word into which the calculated value is to be transferred
- Show the calculator by pressing the **CALC** key, and perform the desired calculation
- Press the **CONFIRM VALUE** soft key

> The control transfers the value into the active input field and closes the calculator.

You can also transfer values from an NC program into the calculator. When you press the **GET CURRENT VALUE** soft key or the **GOTO** key, the control transfers the value from the active input field to the calculator.

The calculator remains in effect even after a change in operating modes. Press the **END** soft key to close the calculator.

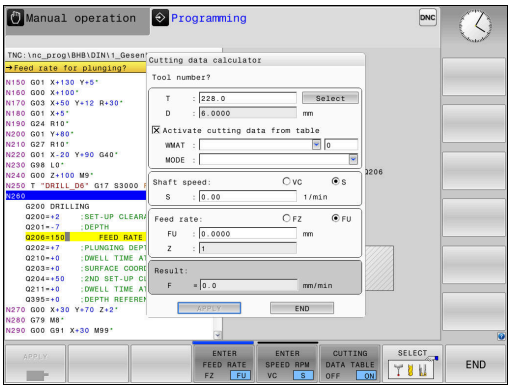
Functions in the pocket calculator

Soft key	Function
AX. VALUES	Transfer the nominal or reference value of the respective axis position into the calculator
GET CURRENT VALUE	Transfer the numerical value from the active input field into the calculator
CONFIRM VALUE	Transfer the numerical value from the calculator into the active input field
COPY FIELD	Copy the numerical value from the calculator
PASTE FIELD	Insert the copied numerical value into the calculator
CUTTING DATA CALCULATOR	Open the cutting data calculator

You can also move the calculator with the arrow keys of your alphabetic keyboard. If you have connected a mouse you can also position the calculator with this.

6.9 Cutting data calculator

Application



With the cutting data calculator you can calculate the spindle speed and the feed rate for a machining process. Then you can load the calculated values into an opened feed rate or spindle speed dialog box in the NC program.

To open the cutting data calculator, press the **CUTTING DATA CALCULATOR** soft key.

The control shows the soft key if you

- Press the **CALC** key
- When defining spindle speeds, press the **CALC** key
- Define feed rates
- Press the **F** soft key in **Manual Operation** mode
- Press the **S** soft key in **Manual Operation** mode

Display modes of the cutting data calculator

The cutting data calculator is displayed with different input fields depending on whether you calculate a spindle speed or a feed rate:

Window for spindle speed calculation:

Abbrev.	Meaning
T:	Tool number
D:	Diameter of the tool
VC:	Cutting speed
S=	Result for spindle speed

If you open the speed calculator in a dialog where the tool is already defined, the speed calculator automatically applies the tool number and diameter. You only need to enter **VC** in the dialog field.

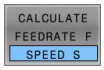
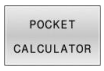
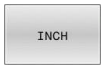
Window for feed rate calculation:

Abbrev.	Meaning
T:	Tool number
D:	Diameter of the tool
VC:	Cutting speed
S:	Spindle speed
Z:	Number of teeth
FZ:	Feed per tooth
FU:	Feed per revolution
F=	Result for feed rate

You can transfer the feed rate from the **T** block into subsequent NC blocks by pressing the **F AUTO** soft key. If you have to change the feed rate later, you only need to adjust the feed rate value in the **T** block.

Functions of the cutting data calculator

You have the following possibilities depending on where you open the cutting data calculator:

Soft key	Function
	Transfer the value from the cutting data calculator into the NC program
	Toggle between feed-rate calculation and spindle-speed calculation
	Toggle between feed per tooth and feed per revolution
	Activate or deactivate working with cutting data tables
	Select a tool from the tool table
	Move the cutting data calculator in the direction of the arrow
	Switch to the calculator
	Use inch values in the cutting data calculator
	Close the cutting data calculator

Working with cutting data tables

Application

If you store tables for materials, cutting materials, and cutting data on the control, then the cutting data calculator can use the values in these tables.

Proceed as follows before working with automatic calculation of the spindle speed and feed rate:

- Enter the type of workpiece material in the table WMAT.tab
- Enter the type of cutting material in the file TMAT.tab
- Enter the combination of workpiece material and cutting material in a cutting data table
- Define the tool with the necessary values in the tool table
  - Tool radius
  - Number of teeth
  - Cutting material
  - Cutting data table

Workpiece material WMAT

Define the workpiece materials in the WMAT.tab table. You must save this table in the directory **TNC:\table**.

This table contains the **WMAT** column for material and a **MAT_CLASS** column where you can categorize the materials by material classes with the same cutting conditions, e.g. according to DIN EN 10027-2.

Enter the workpiece material as follows in the cutting data calculator:

- Select the cutting data calculator
- Select **Activate cutting data from table** in the pop-up window
- Select **WMAT** from the selection menu

TNC:\table\WMAT.TAB

NR	WMAT	MAT_CLASS
1		10
2	1.0038	10
3	1.0044	10
4	1.0114	10
5	1.0177	10
6	1.0143	10
7	St 37-2	10
8	St 37-3 N	10
9	X 14 CrMo S 17	20
10	1.1404	20
11	1.4305	20
12	V2A	21
13	1.4301	21
14	AlCu4PBMg	100
15	Aluminium	100
16	PTFE	200

Cutting material TMAT

Cutting materials are defined in the TMAT.tab table. You must save this table in the directory **TNC:\table**.

You assign the cutting material in the **TMAT** column of the tool table. You can create columns with other names, such as **ALIAS1** and **ALIAS2** in order to enter alternative names for the same cutting material.

Cutting data table

Define the combinations of workpiece material and cutting material with the corresponding cutting data in a table with the file extension .CUT. You must save this table in the directory **TNC:\system\Cutting-Data**.

You assign the appropriate cutting data table in the **CUTDATA** column of the tool table.

TNC:\system\Cutting-Data\CUTDATA.CUT

NR	MAT_CLASS	MODE	TMAT	VC	FTYPE
0	10	Rough	HSS	28	
1	10	Rough	VHM	70	
2	10	Finish	HSS	30	
3	10	Finish	VHM	70	
4	10	Rough	HSS coated	78	
5	10	Finish	HSS coated	82	
6	20	Rough	VHM	90	
7	20	Finish	VHM	82	
8	100	Rough	HSS	150	
9	100	Finish	HSS	145	
10	100	Rough	VHM	450	
11	100	Finish	VHM	440	
12					
13					
14					

> Using the simplified cutting data table, you can determine speeds and feed rates using cutting data that are independent of the tool radius (e.g., **VC** and **FZ**).
>
> If you require specific cutting data depending on the tool radius for your calculations, use the diameter-dependent cutting data table.
>
> **Further information:** "Diameter-dependent cutting data table ", Page 217

The cutting data table contains the following columns:

- **MAT_CLASS**: Material class
- **MODE**: Machining mode, such as finishing
- **TMAT**: Cutting material
- **VC**: Cutting speed
- **FTYPE**: Type of feed rate **FZ** or **FU**
- **F**: Feed rate

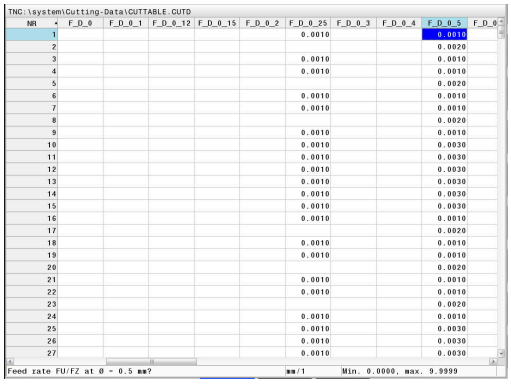
Diameter-dependent cutting data table

In many cases the diameter of the tool determines which cutting data you can use. Use the cutting data table with the file extension .CUTD for this purpose. You must save this table in the directory **TNC:\system\Cutting-Data**.

You assign the appropriate cutting data table in the **CUTDATA** column of the tool table.

The diameter-dependent cutting data table contains the following additional columns:

- **F_D_0**: Feed rate for Ø 0 mm
- **F_D_0_1**: Feed rate for Ø 0.1 mm
- **F_D_0_12**: Feed rate for Ø 0.12 mm
- ...



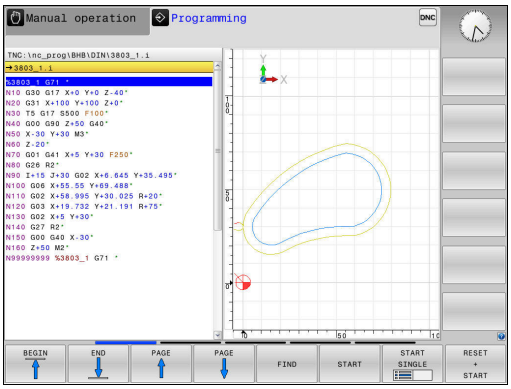
> You don't need to fill in all columns. If a tool diameter is between two defined columns, the control linearly interpolates the feed rate.

Note

In the corresponding folders, the control provides sample tables for automatic cutting data calculation. You can customize theses tables and specify your own data, i.e. materials and tools to be used.

6.10 Programming graphics

Activating and deactivating programming graphics



While you are writing an NC program, you can have the control generate a 2D pencil-trace graphic of the programmed contour.

- Press the **Screen layout** key
- Press the **PROGRAM + GRAPHICS** soft key
- The control shows the NC program to the left and graphics to the right.

- Set the **AUTO DRAW** soft key to **ON**
- While you are entering the program lines, the control generates each programmed movement in the graphics window in the right screen half.

If you do not want the control to generate graphics during programming, then set the **AUTO DRAW** soft key to **OFF**.

If **AUTO DRAW** is set to **ON**, then the control ignores the following program content when creating 2D pencil-trace graphics:

- Program section repetitions
- Jump commands
- M functions, such as M2 or M30
- Cycle calls
- Warnings due to locked tools

Therefore, only use automatic drawing during contour programming.

The control resets the tool data when you reopen an NC program or press the **RESET + START** soft key.

The control uses various colors in the programming graphics:

- **blue:** completely defined contour element
- **violet:** not yet completely defined contour element, can still be modified by e.g. an RND
- **light blue:** holes and threads
- **ocher:** tool midpoint path
- **red:** rapid traverse

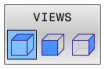
Further information: "FK programming graphics", Page 189

Generating a graphic for an existing NC program

- Use the arrow keys to select the NC block up to which you want the graphic to be generated, or press **GOTO** and enter the desired block number

- Reset previously active tool data, and generate graphic: Press the **RESET + START** soft key

Additional functions:

Soft key	Function
	Reset previously active tool data. Generate programming graphics
	Generate programming graphic blockwise
	Generate a complete programming graphic, or complete it after **RESET + START**
	Stop the programming graphics. This soft key only appears while the control is generating the programming graphics
	Selecting views ■ Plan view ■ Front view ■ Side view
	Display or hide tool paths
	Display or hide tool paths in rapid traverse

Block number display ON/OFF

- Shift the soft-key row

- Show block numbers: Set the **SHOW BLOCK NO.** soft key to **ON**
- Hide block numbers: Set the **SHOW BLOCK NO.** soft key to **OFF**

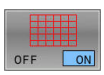
Erasing the graphic

- Shift the soft-key row

- Erase the graphic: Press the **CLEAR GRAPHICS** soft key

Showing grid lines

- Shift the soft-key row

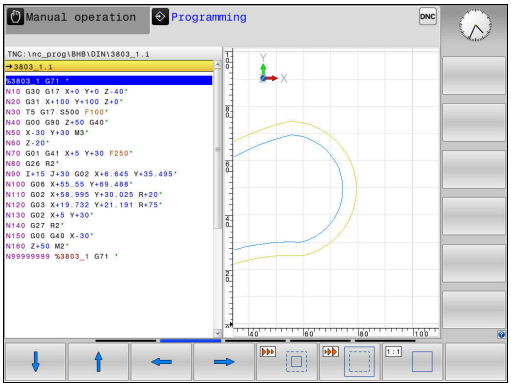


- Show grid lines: Press the **Show grid lines** soft key

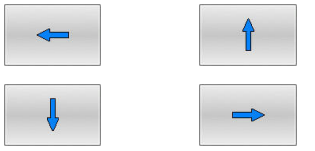
Magnification or reduction of details

You can select the graphics display

- ▶ Shift the soft-key row



The following functions are available:

Soft key	Function
	Shift section
	Reduce section
	Enlarge section
	Reset section

The **RESET BLK FORM** soft key allows you to restore the original section.

You can also use the mouse to change the graphic display. The following functions are available:

- To shift the displayed model, hold down the center mouse button or the mouse wheel, and move the mouse. If you press the shift key at the same time, then you will be able to shift the model only horizontally or vertically.
- To zoom in on a certain area, mark a zoom area by holding the left mouse button down. After you release the left mouse button, the control zooms in on the defined area.
- To rapidly magnify or reduce any area, rotate the mouse wheel backwards or forwards.

6.11 Error messages

Display of errors

The control displays error messages in the following cases, for example:

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

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

The control uses the following icons and text colors for different error classes:

Icon	Text color	Error class	Meaning
	Red	Error Prompt	The control displays a dialog with several options you can select from. **Further information:** "Detailed error messages", Page 222
	Red	Reset error	The control must be restarted. This message cannot be cleared.
	Red	Error	To continue, you must clear this message. An error message can only be cleared after the cause has been eliminated.
	Yellow	Warning	You can continue without clearing the message. Most warnings can be cleared at any time; in some cases, the cause has to be eliminated first.
	Blue	Information	You can continue without clearing the message. You can clear the information at any time.
	Green	Note:	You can continue without clearing the message. The control displays the note until you press the next valid key.

The table rows are ordered by priority. The control displays a message in the header until it is cleared or replaced by a higher-priority message (higher error class).

The control displays long and multi-line error messages in abbreviated form. The complete information on all pending errors is shown in the error window.

An error message that contains an NC block number was caused by an error in the indicated NC block or in the preceding NC block.

Opening the error window

When you open the error window, the complete information on all pending errors will be shown.

ERR

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

Detailed error messages

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

- Open the error window
- Position the cursor on the corresponding error message

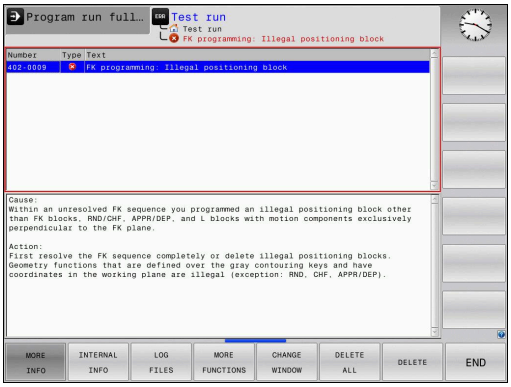
MORE INFO

- Press the **MORE INFO** soft key

> The control opens a window with information on the error cause and corrective action.

MORE INFO

- Exit the info: Press the **MORE INFO** soft key again



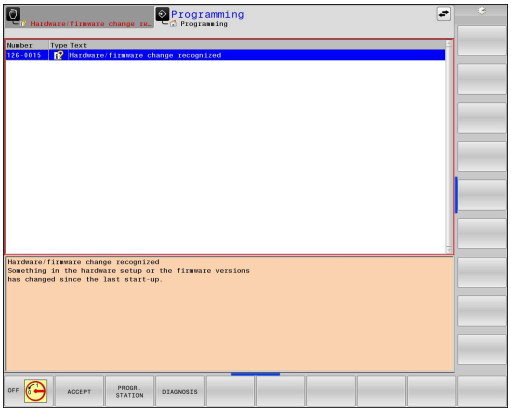
High-priority error messages

When an error message occurs at switch-on of the control due to hardware changes or updates, the control will automatically open the error window. The control displays an error of the question type.

You can correct this error only by pressing the corresponding soft key to acknowledge the question. If necessary, the control continues the dialog until the cause or correction of the error has been clearly determined.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

If a rare **processor check error** should occur, the control will automatically open the error window. You cannot correct such an error.



Proceed as follows:

- Shut down the control
- Restart

INTERNAL INFO soft key

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

- Open the error window
- Position the cursor on the corresponding error message

INTERNAL INFO

- Press the **INTERNAL INFO** soft key

> The control opens a window with internal information about the error.

INTERNAL INFO

- Exit the detailed information: Press the **INTERNAL INFO** soft key again

GROUPING soft key

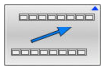
If you activate the **GROUPING** soft key, the control displays all warnings and error messages with the same error number in the same line of the error window. This makes the list of messages shorter and easier to read.

To group the error messages:

- Open the error window

- Press the **MORE FUNCTIONS** soft key

- Press the **GROUPING** soft key
- > The control groups identical warnings and error messages.
- > The number of occurrences of the individual messages is indicated in parentheses in the respective line.



- Press the **GO BACK** soft key

ACTIVATE AUTOMATIC SAVING soft key

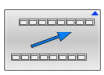
The **ACTIVATE AUTOMATIC SAVING** soft key allows you to specify error numbers that cause the control to save a service file if an error with that number occurs.

- Open the error window

- Press the **MORE FUNCTIONS** soft key

- Press the **ACTIVATE AUTOMATIC SAVING** soft key
- > The control opens the **ACTIVATE AUTOMATIC SAVING** pop-up window.
- Define the entries
  - **Error number**: Enter the desired error number
  - **active:** Enable this option to automatically create the service file
  - **Comment:** Enter a comment on this error number, if required

- Press the **STORE** soft key
- > If an error with the specified error number occurs, a service file will be saved automatically.



- Press the **GO BACK** soft key

Deleting errors

The control can automatically clear pending warning or error messages when an NC program is selected or restarted. The machine manufacturer specifies in the optional machine parameter **CfgClearError** (no. 130200) whether these messages will automatically be cleared.

The factory default setting of the control defines that warning and error messages in the **Test Run** and **Programming** operating modes will be cleared automatically from the error window. Messages issued in the machine operating modes will not be cleared.

Clearing errors outside of the error window

CE

- Press the **CE** key

> The control clears the errors or notes being displayed in the header.

In certain situations you cannot use the **CE** key for clearing the errors because the key is used for other functions.

Clearing errors

- Open the error window
- Position the cursor on the corresponding error message

DELETE

- Press the **DELETE** soft key

DELETE ALL

- As an alternative, clear all errors: Press the **DELETE ALL** soft key

If the cause of the error has not been corrected, the error message cannot be cleared. In this case, the error message remains in the window.

Error log

The control stores errors occurred and important events (e.g., system start) in an error log. The capacity of the error log is limited. When the log is full, the control uses a second file. When this is also full, the first error log is deleted and newly written etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history.

- Open the error window

- Press the **LOG FILES** soft key

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

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

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

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

Keystroke log

The control stores each key pressed and important events (e.g., system start) in a keystroke log. The capacity of the keystroke log is limited. When the keystroke log is full, the control switches to a second keystroke log. When this is also full, the first keystroke log is deleted and newly written, etc. If required, switch from **CURRENT FILE** to **PREVIOUS FILE** to view the history of the inputs.

- Press the **LOG FILES** soft key

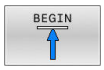
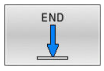
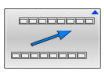
- Open the keystroke log file: Press the **KEYSTROKE LOG** soft key

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

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

The control saves each key pressed during operation in a keystroke log. The oldest entry is at the beginning, and the most recent entry is at the end of the file.

Overview of the keys and soft keys for viewing the log

Soft key/ Keys	Function
	Go to beginning of keystroke log
	Go to end of keystroke log
	Find text
	Current keystroke log
	Previous keystroke log
	Up/down one line
	Return to main menu

Informational texts

If an operating error occurred, e.g. pressing an impermissible key or entering a value outside of a validity range, the control displays an information text in the header to inform you of the operating error. The control deletes this information text with the next valid entry.

Saving service files

If necessary, you can save the current situation of the control and make it available to a service technician for evaluation. A group of service files is saved (error and keystroke logs as well as other files that contain information about the current status of the machine and the machining).

> In order to facilitate sending service files via email, the control will only save active NC programs with a size of up to 10 MB in the service file. If the NC program is larger, it will not be added to the created service file.

When the same name is entered several times in the **SAVE SERVICE FILES** function, the control saves up to five files and deletes the file with the oldest timestamp, as needed. Make a backup of the service files you created (e.g., by moving them to a different folder).

Saving service files

ERR
- ▶ Open the error window

LOG FILES
- ▶ Press the **LOG FILES** soft key

SAVE SERVICE FILES
- ▶ Press the **SAVE SERVICE FILES** soft key
- > The control opens a pop-up window in which you can enter a file name or a complete path for the service file.

OK
- ▶ Press the **OK** soft key
- > The control saves the service file.

Closing the error window

To close the error window:

END
- ▶ Press the **END** soft key

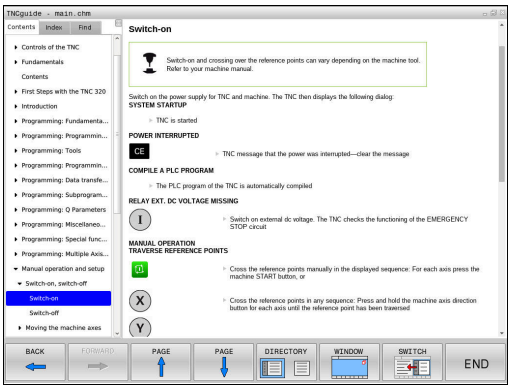
ERR
- ▶ Alternative: Press the **ERR** key
- > The control closes the error window.

6.12 TNCguide: context-sensitive help

Application

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

Further information: "Downloading current help files", Page 232



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

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

The following user documentation is available in **TNCguide**:

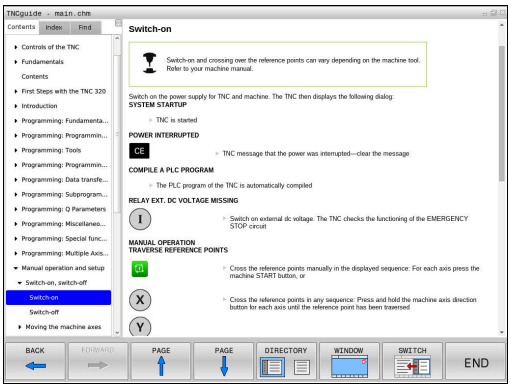
- User's Manual for Klartext Programming (**BHBKlartext.chm**)
- User's Manual for ISO programming (**BHBIso.chm**)
- User's Manual for Setup, Testing and Running NC Programs (**BHBoperate.chm**)
- User's Manual for Programming of Machining Cycles (**BHBcycle.chm**)
- User's Manual for Programming of Measuring Cycles for Workpieces and Tools (**BHBtchprobe.chm**)
- List of All Error Messages **(errors.chm)**

In addition, the **main.chm** "book" file is available, in which all existing .chm files are shown in one place.

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

Using TNCguide

Calling TNCguide



You have several options for starting **TNCguide**:

- Use the **HELP** key
- First click the help symbol in the lower right-hand corner of the screen, then click the appropriate soft key
- Open a help file (.chm file) via the file management. The control can open any .chm file, even if it is not saved in the control's internal memory

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

For many soft keys there is a context-sensitive call through which you can go directly to the description of the soft key's function. This functionality requires using a mouse.

Proceed as follows:

- ▶ Select the soft-key row containing the desired soft key
- ▶ Click with the mouse on the help symbol that the control displays just above the soft-key row
- > The mouse pointer turns into a question mark.
- ▶ Move the question mark to the soft key for which you want an explanation
- > The control opens **TNCguide**. If there is no entry point for the selected soft key, then the control opens the book file **main.chm**. You can search for the desired explanation using full text search or by using the navigation.

Even if you are editing an NC block, context-sensitive help is available:

- ▶ Select any NC block
- ▶ Select the desired word
- ▶ Press the **HELP** key.
- > The control opens the Help system and shows the description of the active function. This does not apply for miscellaneous functions or cycles from your machine manufacturer.

Navigating in TNCguide

It's easiest to use the mouse to navigate in **TNCguide**. A table of contents appears on the left side of the screen. Clicking on the rightward pointing triangle opens subordinate sections, and clicking on the respective entry opens the corresponding page. You can use it in the same way as Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

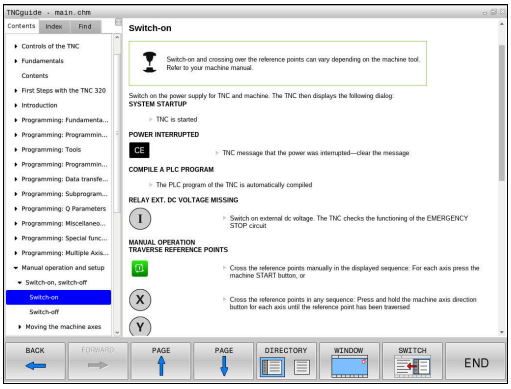
Soft key/ Keys	Function
	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely
	■ If the table of contents at left is active: Expand the table of contents ■ If the text window at right is active: No function
	■ If the table of contents at left is active: Collapse the table of contents ■ If the text window at right is active: No function
ENT	■ If the table of contents at left is active: Use the cursor key to show the selected page ■ If the text window at right is active: If the cursor is on a link, jump to the linked page
	■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the right side of the window ■ If the text window at right is active: Jump back to the left side of the window
	■ If the table of contents at left is active: Select the entry above it or below it ■ If the text window at right is active: Jump to next link
BACK	Select the page last shown
FORWARD	Page forward if you have used the **Select page last shown** function
PAGE	Go back one page
PAGE	Go forward one page

Soft key/ Keys	Function
DIRECTORY	Display or hide table of contents
WINDOW	Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the control window
SWITCH	The focus is returned to the control application so that you can operate the control while **TNCguide** is open. If the full screen is active, the control reduces the window size automatically before the focus changes
END	Exit **TNCguide**

Subject index

The most important subjects in the Manual are listed in the subject index (**Index** tab). You can select them directly by mouse or with the arrow keys.

The left side is active.



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

  Alternative:
- Enter the first few characters
- > The control synchronizes the subject index and creates a list in which you can find the subject more easily.
- Use the **ENT** key to call the information on the selected keyword

Full-text search

On the **Find** tab, you can search all of **TNCguide** for a specific word.

The left side is active.

- Select the **Find** tab
- Activate the **Find:** entry field
- Enter the search word
- Press the **ENT** key
- > The control lists all sources containing the word.
- Use the arrow keys to navigate to the desired source
- Press the **ENT** key to go to the selected source

The full-text search only works for single words.

If you activate the **Search only in titles** function, the control searches only through headings and ignores the body text. To activate the function, use the mouse or select it and then press the space bar to confirm.

Downloading current help files

You'll find the help files for your control software on the HEIDENHAIN website:
http://content.heidenhain.de/doku/tnc_guide/html/en/index.html

Navigate to the suitable help file as follows:

- TNC controls
- Series (e.g., TNC 300)
- Desired NC software number, such as TNC 320 (77185x-18)

> 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.

- Select the desired language version from the **TNCguide online help (CHM files)** table
- Download the ZIP file
- Extract the ZIP file
- Move the extracted CHM files to the **TNC:\tncguide\en** directory or the respective language subdirectory on the control

> When using **TNCremo** to transfer the CHM files to the control, select the binary mode for files with the **.chm** extension.

Language	TNC directory
German	**TNC:\tncguide\de**
English	**TNC:\tncguide\en**
Czech	**TNC:\tncguide\cs**
French	**TNC:\tncguide\fr**
Italian	**TNC:\tncguide\it**
Spanish	**TNC:\tncguide\es**
Portuguese	**TNC:\tncguide\pt**
Swedish	**TNC:\tncguide\sv**
Danish	**TNC:\tncguide\da**
Finnish	**TNC:\tncguide\fi**
Dutch	**TNC:\tncguide\nl**
Polish	**TNC:\tncguide\pl**
Hungarian	**TNC:\tncguide\hu**
Russian	**TNC:\tncguide\ru**
Chinese (simplified)	**TNC:\tncguide\zh**

Language	TNC directory
Chinese (traditional)	**TNC:\tncguide\zh-tw**
Slovenian	**TNC:\tncguide\sl**
Norwegian	**TNC:\tncguide\no**
Slovak	**TNC:\tncguide\sk**
Korean	**TNC:\tncguide\kr**
Turkish	**TNC:\tncguide\tr**
Romanian	**TNC:\tncguide\ro**

7

Miscellaneous Functions

7.1 Entering miscellaneous functions M and STOP

Fundamentals

With the control's miscellaneous functions—also called M functions—you can affect:

- the program run, e.g. a program interruption
- the machine functions, such as switching spindle rotation and coolant supply on and off
- the path behavior of the tool

You can enter up to four M (miscellaneous) functions at the end of a positioning block or in a separate NC block. The control displays the following dialog question: **Miscellaneous function M ?**

You usually enter only the number of the miscellaneous function in the programming dialog. With some miscellaneous functions, the dialog is extended so that you can enter the required parameters for this function.

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

Effectiveness of 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.

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

Some miscellaneous functions are effective block-by-block, i.e. only in the NC block in which the miscellaneous function has been programmed. When a miscellaneous function takes effect modally, you have to cancel this miscellaneous function again in a subsequent NC block (e.g., by using **M9** to switch off coolant that was switched on with **M8**). If miscellaneous functions are still active at the end of the program, the control will rescind the miscellaneous functions.

If multiple M functions were programmed in a single NC block, the execution sequence is as follows:

- M functions taking effect at the start of the block are executed before those taking effect at the end of the block
- If all M functions are effective at the start or end of the block, execution takes place in the sequence as programmed

Entering a miscellaneous function in a STOP block

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

STOP

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

Example

```
N87 G38*
```

7.2 Miscellaneous functions for program run inspection, spindle and coolant

Overview

Refer to your machine manual.

The machine manufacturer can influence the behavior of the miscellaneous functions described below.

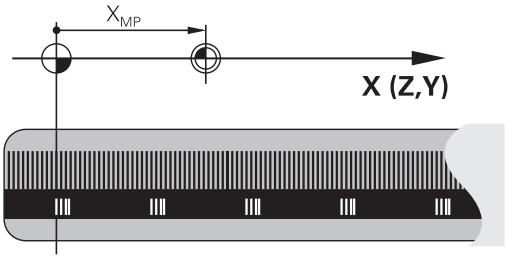
M	Effect	Effective at block	Start	End
M0	Program STOP Spindle STOP			■
M1	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (function defined by the machine manufacturer)			■
M2	STOP program run Spindle STOP Coolant off Return jump to block 0 Clear status display Functional scope depends on machine parameter **resetAt** (no. 100901)			■
M3	Spindle ON clockwise		■	
M4	Spindle ON counterclockwise		■	
M5	Spindle STOP			■
M8	Coolant ON		■	
M9	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M2			■

7.3 Miscellaneous functions for coordinate entries

Programming machine-referenced coordinates: M91/ M92

Scale datum

On the scale, a reference mark indicates the position of the scale datum.



Machine datum

The machine datum is required for the following tasks:

- Define the axis traverse limits (software limit switches)
- Approach machine-referenced positions (such as tool change positions)
- Set a workpiece preset

The distance in each axis from the scale datum to the machine datum is defined by the machine manufacturer in a machine parameter.

Standard behavior

The control references the coordinates to the workpiece datum.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Behavior with M91 – Machine datum

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

> 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**. If the active NC program does not contain a position programmed with **M91**, the coordinates reference the current tool position.

The coordinate values on the control's screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Behavior with M92 – Additional machine reference point

Refer to your machine manual.

In addition to the machine datum, the machine manufacturer can also define an additional machine-based position as a reference point (machine preset).

For each axis, the machine manufacturer defines the distance between the machine preset and the machine datum.

If you want the coordinates in positioning blocks to be based on the machine preset, enter M92 into these NC blocks.

Radius compensation remains the same in blocks that are programmed with **M91** or **M92**. The tool length will **not** be taken into account.

Effect

M91 and M92 are effective only in the blocks in which M91 and M92 have been programmed.

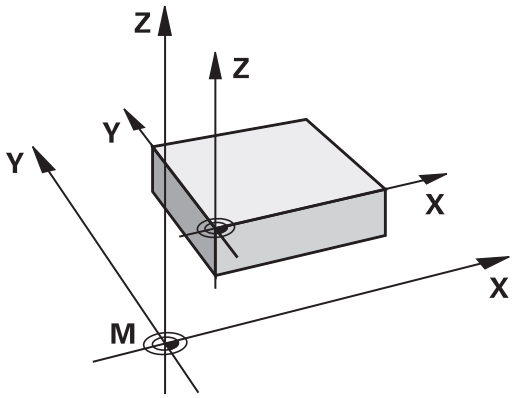
M91 and M92 take effect at the start of block.

Workpiece preset

If you want the coordinates to always be referenced to the machine datum, you can inhibit presetting for one or more axes.

If presetting is inhibited for all axes, the control does not display the **SET PRESET** soft key in the **Manual operation** operating mode.

The figure shows coordinate systems with the machine and workpiece datum.



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the defined preset.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

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

Standard behavior with a tilted working plane

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

Further information: "Working plane coordinate system WPL-CS", Page 94

Behavior with M130

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

M130 ignores only the **Tilt working plane** function, but takes into account active transformations before and after tilting. This means that, when calculating the position, the control considers the axis angles of the rotary axes that are not in their zero position.

Further information: "Input coordinate system I-CS", Page 95

NOTICE

Danger of collision!

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

- Use the simulation to check the sequence and positions

Programming notes

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

Effect

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

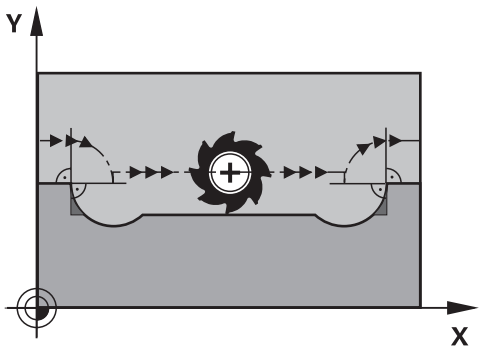
7.4 Miscellaneous functions for path behavior

Machining small contour steps: M97

Standard behavior

The control inserts a transition arc at outside corners. For very small contour steps, the tool would damage the contour.

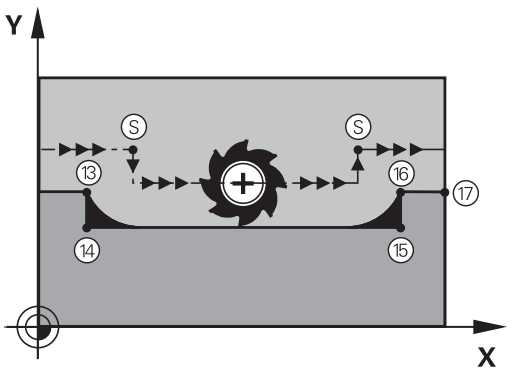
In such cases, the control interrupts the program run and generates the **Tool radius too large** error message.



Behavior with M97

The control determines a path intersection for the contour elements —such as inner corners—and moves the tool above this point.

Program **M97** in the same NC block as the outside corner.



> HEIDENHAIN recommends using the more powerful function **M120** instead of **M97**. **Further information:** "Pre-calculating radius-compensated contours (LOOK AHEAD): M120 ", Page 246

Effect

M97 is effective only in the NC block in which **M97** is programmed.

> The control does not completely finish the corner when it is machined with **M97**. You may need to rework the contour corner with a smaller tool.

Example

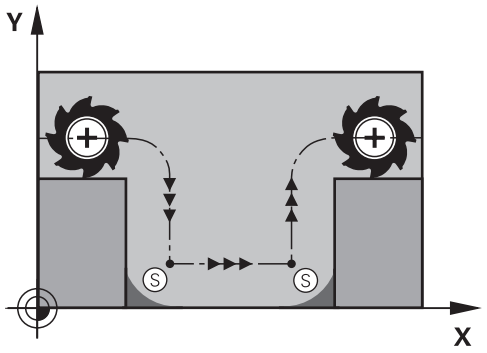
N50 G99 G01 ... R+20*	Large tool radius
...	
N130 X ... Y ... F ... M97*	Move to contour point 13
N140 G91 Y-0.5 ... F ...*	Machine small contour step 13 to 14
N150 X+100 ...*	Move to contour point 15
N160 Y+0.5 ... F ... M97*	Machine small contour step 15 to 16
N170 G90 X ... Y ... *	Move to contour point 17

Machining open contour corners: M98

Standard behavior

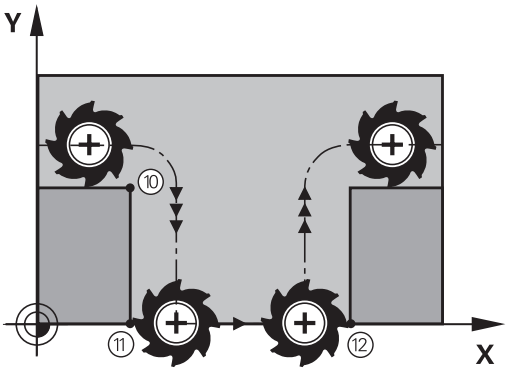
The control calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining.



Behavior with M98

With the **M98** miscellaneous function, the control temporarily suspends radius compensation to ensure that both corners are completely machined:



Effect

M98 is effective only in the NC blocks in which **M98** is programmed.

M98 becomes effective at the end of the block.

Example: Move to the contour points 10, 11 and 12 in succession

```
N100 G01  G41 X ...  Y ... F ...*
N110 X ...  G91 Y ...  M98*
N120 X+ ...*
```

Feed rate factor for plunging movements: M103

Standard behavior

The control moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103

The control reduces the feed rate when the tool moves in the negative direction of the tool axis. The feed rate for plunging FZMAX is calculated from the last programmed feed rate FPROG and a factor F%:

FZMAX = FPROG x F%

Programming M103

If you program **M103** in a positioning block, the control continues the dialog by prompting you for the F factor.

Effect

M103 becomes effective at the start of the block.
Cancel **M103**: Program **M103** once again without a factor.

> **M103** is also in effect with an active tilted working plane coordinate system **WPL-CS**. The feed rate reduction is then active during infeed movements in the virtual tool axis **VT**.

Example

The feed rate for plunging is to be 20% of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
N170 G01 G41 X+20 Y+20 F500 M103 F20*	500
N180 Y+50*	500
N190 G91 Z-2.5*	100
N200 Y+5 Z-5*	141
N210 X+50*	500
N220 G90 Z+5*	500

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The control moves the tool at the feed rate F in mm/min programmed in the NC program

Behavior with M136

In NC programs based on inch units, **M136** is not allowed in combination with **FU** or **FZ**.

The workpiece spindle is not permitted to be controlled when **M136** is active.

It is not possible to combine **M136** with an oriented spindle stop. The control cannot calculate the feed rate because the spindle does not rotate during an oriented spindle stop.

With **M136**, the control does not move the tool in mm/min, but rather at the feed rate F in millimeters per spindle revolution programmed in the NC program. If you change the spindle speed by using the potentiometer, the control changes the feed rate accordingly.

Effect

M136 becomes effective at the start of the block.

You can cancel **M136** by programming **M137**.

Feed rate for circular arcs: M109/M110/M111

Standard behavior

The control applies the programmed feed rate to the path of the tool center.

Behavior for circular arcs with M109

For inside and outside machining of circular arcs, the control keeps the feed rate at the cutting edge constant.

NOTICE

Caution: Danger to the tool and workpiece!

If the **M109** function is active, the control might significantly increase the feed rate when machining very small outside corners (acute angles). There is a risk of tool breakage or workpiece damage during machining.

- Do not use **M109** for machining very small outside corners (acute angles)

Behavior for circular arcs with M110

With circular arcs, the control only keeps the feed rate constant for inside machining operations. The feed rate will not be adjusted for outside machining of circular arcs.

If you program **M109** or **M110** with a number > 200 before calling a machining cycle, the adjusted feed rate will also be effective for circular contours within these machining cycles. The initial state is restored after finishing or canceling a machining cycle.

Effect

M109 and **M110** become effective at the start of the block. **M109** and **M110** can be canceled with **M111**.

Pre-calculating radius-compensated contours (LOOK AHEAD): M120

Standard behavior

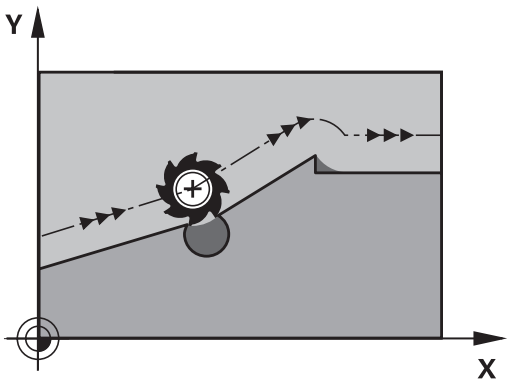
If the tool radius is larger than the contour step that needs to be machined with radius compensation, then the control interrupts program run and issues an error message. **M97** inhibits the error message, but this results in dwell marks and will also move the corner.

Further information: "Machining small contour steps: M97", Page 241

The control might damage the contour in case of undercuts.

Behavior with M120

The control checks radius-compensated contours for undercuts and tool path intersections, and calculates the tool path in advance from the current NC block. Areas of the contour that would be damaged by the tool will not be machined (shown darker in the figure). You can also use **M120** to calculate the tool radius compensation for digitized data or data from an external programming system. This means that you can compensate for deviations from the theoretical tool radius.



The number of NC blocks (99 max.) to be calculated in advance can be defined with **LA** (**L**ook **A**head) following **M120**. Note that the larger the number of NC blocks you choose, the higher the block processing time will be.

Input

If you define **M120** in a positioning block, the control continues the dialog and prompts you for the number of **LA** NC blocks to be calculated in advance.

Effect

Program the function **M120** in an NC block that also contains an **G41** or **G42** radius compensation. This way, you can achieve consistent programming, resulting in clearly structured programs.

M120 can be reset by the following NC functions:

- **M120 LA0**
- **M120** without **LA**
- Radius compensation **G40**
- Departure functions (e.g., **DEP LT**)

M120 becomes effective at the start of the block and remains effective beyond the milling cycles.

Constraints

- After an external or internal stop, you have to use a block scan to be able to re-approach the contour. Before the block scan, you need to cancel **M120**— otherwise the control will issue an error message.
- If you want to approach the contour on a tangential path, you must use the function **APPR LCT**. The NC block with **APPR LCT** must contain only the coordinates of the working plane.
- If you want to depart the contour on a tangential path, you must use the function **DEP LCT**. The NC block with **DEP LCT** must contain only the coordinates of the working plane.
- If radius compensation is active and you execute the following functions, the control aborts program run and displays an error message:
  - **PLANE** functions (option 8)
  - **CALL PGM%**
  - Cycle **12G39 PGM CALL**
  - Cycle **32G62 TOLERANCE**
  - Cycle **19G80 WORKING PLANE**

Superimposing handwheel positioning during program run: M118

Standard behavior

Refer to your machine manual.
Your machine manufacturer must have prepared the control for this function.

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

Behavior with M118

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

Input

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

Effect

To cancel handwheel positioning, program **M118** once again without coordinate input or end the NC program with **M30** / **M2**.

If the program aborts, handwheel positioning will also be canceled.

M118 becomes effective at the start of the block.

Example

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

```
N250 G01  G41 X+0  Y+38.5 F125  M118 X1  Y1 B5*
```

When programmed in an NC program, **M118** is always effective in the machine coordinate system.

The **POS HR** tab of the additional status display shows the **Max. val.** defined in **M118**.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

The function **Handwheel superimposed** is also effective in the **Positioning w/ Manual Data Input** operating mode!

Retraction from the contour in the tool-axis direction: M140

Standard behavior

In the **Program Run Single Block** and **Program Run Full Sequence** operating modes, the control moves the tool as defined in the NC program.

Behavior with M140

With **M140 MB** (move back), you can retract the tool from the contour by a programmable distance in the direction of the tool axis.

Input

If you enter **M140** in a positioning block, the control continues the dialog and prompts you for the path the tool should use for retracting from the contour. Enter the desired path that the tool should follow when retracting from the contour, or press the **MB MAX** soft key to move to the limit of the traverse range.

> In the optional machine parameter **moveBack** (no. 200903), the machine manufacturer defines how far before a limit switch or a collision object a retraction movement **MB MAX** should end.

In addition, you can program the feed rate at which the tool traverses the entered path. If you do not enter a feed rate, the control moves the tool along the entered path at rapid traverse.

Effect

M140 is effective only in the NC block in which it is programmed.

M140 becomes effective at the start of the block.

Example

NC block 250: Retract the tool by 50 mm from the contour

NC block 251: Move the tool to the limit of the traverse range

```
N250 G01  X+0 Y+38.5  F125 M140  MB50*
N251 G01  X+0 Y+38.5  F125 M140  MB MAX*
```

M140 is also in effect with a tilted working plane. For machines with head rotation axes the control moves the tool in the tool coordinate system **T-CS**.

With **M140 MB MAX** the control retracts the tool only in the positive direction in the tool axis.

The control gleans the necessary information about the tool axis for **M140** from the tool call.

NOTICE

Danger of collision!

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

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

Suppressing touch probe monitoring: M141

Standard behavior

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

Behavior with M141

The control moves the machine axes even if the touch probe is deflected. This function is required if you wish to write your own measuring cycle in order to retract the touch probe by means of a positioning block after it has been deflected.

NOTICE

Danger of collision!

The miscellaneous function **M141** suppresses the corresponding error message if the stylus is deflected. The control does not perform an automatic collision check with the stylus. Based on these two types of behavior, you must check whether the touch probe can retract safely. There is a risk of collision if you choose the wrong direction for retraction.

- Carefully test the NC program or program section in the **Program run, single block** operating mode

M141 works only for movements with straight-line blocks.

Effect

M141 is effective only in the NC block in which **M141** is programmed.

M141 becomes effective at the start of the block.

Deleting basic rotation: M143

Standard behavior

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

Behavior with M143

The control deletes a basic rotation from the NC program.

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

Effect

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

M143 becomes effective at the start of the block.

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

Lifting off the tool automatically from the contour at NC stop: M148

Standard behavior

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

Behavior with M148

Refer to your machine manual.

This function must be configured and enabled by your machine manufacturer.

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

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

Further information: User's Manual for **Setup, Testing and Running NC Programs**

LIFTOFF takes effect in the following situations:

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

When lifting the tool off with **M148**, the control will not necessarily lift it off in the tool axis direction.

The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the liftoff direction. If you program **M148**, the control will automatically liftoff the tool in the direction defined by the **FUNCTION LIFTOFF** function.

Effect

M148 remains in effect until deactivated with **M149** or **FUNCTION LIFTOFF RESET**.

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

Rounding corners: M197

Standard behavior

With active radius compensation, the control inserts a transition arc at outside corners. This may lead to rounding of that edge.

Behavior with M197

With the **M197** function, the contour at the corner is tangentially extended and a smaller transition arc is then inserted. When you program the **M197** function and then press the **ENT** key, the control opens the **DL** input field. In **DL**, you define the length the control by which the control extends the contour elements. With **M197**, the corner radius is reduced, the corner is rounded less and the traverse movement is still smooth.

Effect

The **M197** function acts blockwise and is only effective on outside corners.

Example

```
G01 X... Y... RL M197 DL0.876*
```

8

Subprograms and Program Section Repeats

8.1 Labeling 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.

Label

Subprograms and program section repeats start with **G98 L** in the NC program (an abbreviation for LABEL).

A LABEL contains a number between 1 and 65535 or a name to be defined by you. LABEL names can have up to 32 characters.

Permitted characters: # $ % & , - _ . 0 1 2 3 4 5 6 7 8 9 @ a b c d e f g h i j k l m n o p q r s t u v w x y z- 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

Impermissible characters: <blank> ! " ' ( ) * + : ; < = > ? [/] ^ ` { | } ~

You may assign each LABEL number, or each LABEL name, only once in the NC program using the **LABEL SET** key, or by entering **G98**. The quantity of label names that may be entered is limited only by the amount of internal memory.

Do not use a label number or label name more than once!

Label 0 (**G98 L0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

Before creating your NC program, compare the subprogram and program section repeat programming techniques using if-then decisions.

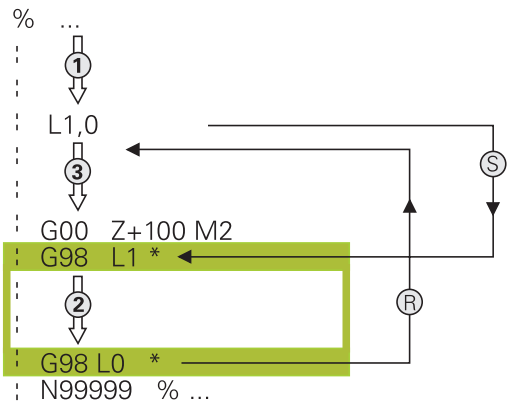
You can thereby avoid possible misunderstandings and programming errors.

Further information: "If-then decisions with Q parameters", Page 291

8.2 Subprograms

Operating sequence

1 The control executes the NC program up to the block in which a subprogram is called with **Ln,0**
2 The subprogram is then executed until the subprogram end **G98 L0**
3 The control then resumes the NC program from the NC block after the subprogram call **Ln,0**



Programming notes

- A main program can contain any number of subprograms
- You can call subprograms in any sequence and as often as desired
- A subprogram cannot call itself
- 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

Programming the subprogram

LBL SET

- To mark the beginning: Press the **LBL SET** key
- Enter the subprogram number. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the text
- Mark the end: Press the **LBL SET** key and enter the label number **0**

Calling a subprogram

LBL CALL

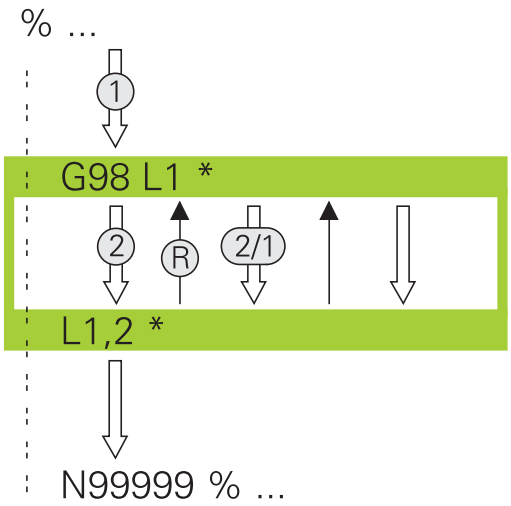
- Call a subprogram: Press the **LBL CALL** key
- Enter the subprogram number of the subprogram you wish to call. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.

L 0 is not permitted (Label 0 is only used to mark the end of a subprogram).

8.3 Program-section repeats

Label G98

The beginning of a program section repeat is marked by the label **G98 L**. The end of a program section repeat is identified by **Ln,m**.



Operating sequence

1 The control executes the NC program up to the end of the program section (**Ln,m**)
2 Then the program section between the called LABEL and the label call **Ln,m** is repeated the number of times entered after **m**
3 The control then resumes the NC program after the last repetition.

Programming notes

- You can repeat a program section up to 65 534 times in succession
- 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.

Programming a program section repeat

LBL SET

- To mark the beginning, press the **LBL SET** key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the **LBL NAME** soft key to switch to text entry.
- Enter the program section

Calling a program section repeat

LBL CALL

- Call a program section: Press the **LBL CALL** key
- Enter the program section number of the program section to be repeated. If you want to use a LABEL name, press the **LBL NAME** soft key to switch to text entry
- Enter the number of repeats **REP** and confirm with the **ENT** key.

8.4 Calling an external NC program

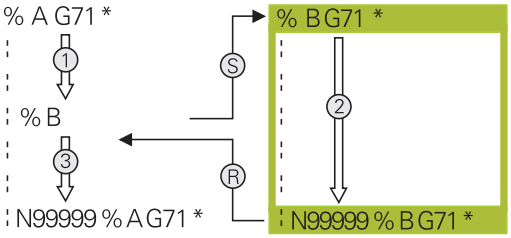
Overview of the soft keys

When you press the **PGM CALL** key, the control displays the following soft keys:

Soft key	Function	Description
CALL PROGRAM	Call an NC program with **%**	Page 263
SELECT DATUM TABLE	Select a datum table with **%:TAB:**	Page 371
SELECT POINT TABLE	Select a point table with **%:PAT:**	Page 267
SELECT CONTOUR	Select a contour program with **%:CNT:**	See the User's Manual for Programming of Machining Cycles
SELECT PROGRAM	Select an NC program with **%:PGM:**	Page 264
CALL SELECTED PROGRAM	Call the last selected file with **%<>%**	Page 264
SELECT CYCLE	Select any NC program with **G: :** as a machining cycle	See the User's Manual for Programming of Machining Cycles

Operating sequence

1 The control executes the NC program up to the block in which another NC program is called with **%**.
2 Then the other NC program is run from beginning to end.
3 The control then resumes the calling NC program with the NC block behind the program call.



Programming notes

- The control does not require any labels to call an NC program.
- The called NC program must not use **%** to call the calling NC program (an endless loop would ensue).
- The called NC program must not contain the miscellaneous function **M2** or **M30**. If you have defined subprograms with labels in the called NC program, then you can replace M2 or M30 with the jump function **D09 P01 +0 P02 +0 P03 99**.
- If you want to call an ISO program, enter the file type .I after the program name.
- You can also call an NC program with Cycle **G39**.
- You can also call any NC program with the **Select the cycle** function (**G: :**).
- As a rule, Q parameters are globally effective when used with a program call, such as **CALL PGM** (**%**). So please note that changes made to Q parameters in the called NC program also influence the calling NC program. If applicable, use QL 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.

Checking the called NC programs

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 control checks the called NC programs:

- If the called NC program contains the miscellaneous functions **M2** or **M30**, then the control displays a warning. The control automatically clears the warning as soon as you select another NC program.
- The control checks the called NC programs for completeness before execution. If the **N99999999** NC block is missing, the control aborts with an error message.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Path information

If the NC program you want to call is located in the same directory as the NC program you are calling it from, then you only need to enter the program name.

If the called NC program is not located in the same directory as the NC program you are calling it from, you must enter the complete path, e.g. **TNC:\ZW35\HERE\PGM1.H**

Alternatively, you can program relative paths:

- Starting from the folder of the calling NC program, one folder level up **..\PGM1.H**
- Starting from the folder of the calling NC program, one folder level down **DOWN\PGM2.H**
- Starting from the folder of the calling NC program, one folder level up and into another folder **..\THERE\PGM3.H**

Use the **SYNTAX** soft key to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.

Further information: "File names", Page 117

If the complete path is enclosed in quotation marks, you can use both **** and **/** to separate the folders and files.

Calling an external NC program

Calling with program calls

Use the NC function **%** to call an external NC program. The control runs the external NC program from the position where it was called in the NC program.

Proceed as follows:

- Press the **PGM CALL** key

- Press the **CALL PROGRAM** soft key
- > The control starts the dialog for defining the NC program to be called.
- Enter the path name with the keyboard

Alternative:

- Press the **SELECT FILE** soft key
- > The control displays a selection window in which you can select the NC program to be called.
- Press the **ENT** key

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. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Call with SELECT PROGRAM and CALL SELECTED PROGRAM

The function **%:PGM:** allows you to select an external NC program that you can separately call at a different position in the NC program. The control runs the external NC program from the position at which you called it in the NC program using **CALL SELECTED PGM%<>%**.

The **%:PGM:** function is also permitted with string parameters, so that you can dynamically control program calls.

To select the NC program:

PGM CALL

- Press the **PGM CALL** key

SELECT PROGRAM

- Press the **SELECT PROGRAM** soft key
- The control starts the dialog for defining the NC program to be called.

SELECT FILE

- Press the **SELECT FILE** soft key
- The control displays a selection window in which you can select the NC program to be called.
- Press the **ENT** key

> 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. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

To call the selected NC program:

PGM CALL

- Press the **PGM CALL** key

CALL SELECTED PROGRAM

- Press the **CALL SELECTED PROGRAM** soft key
- The control uses **%<>%** to call the NC program that was selected last.

> If an NC program that was called using **%<>%** is missing, then the control interrupts the execution or simulation with an error message. In order to avoid undesired interruptions during program run, you can use the function **D18** (**ID10 NO110** and **NO111**) to check all paths at the beginning of the program.
> **Further information:** "D18 – Reading system data", Page 318

8.5 Point tables

Application

With a point table you can execute one or more cycles in sequence on an irregular point pattern.

Creating a point table

To create a point table:

- Select the **PROGRAMMING** operating mode

- Press the **PGM MGT** key
- The control opens the file manager.
- Select the desired folder in your folder structure
- Enter the name and file type (***.pnt**)

- Confirm with the **ENT** key

- Press the **MM** or **INCH** soft key.
- The control opens the table editor and shows an empty point table.

- Press the **INSERT LINE** soft key
- The control inserts a new row in the point table.
- Enter the coordinates of the desired machining position
- Repeat the process until all desired coordinates have been entered

Configuring the point table display

To configure the display of a point table:

- Open the desired point table

  Further information: "Creating a point table", Page 265

SORT/ HIDE COLUMNS

- Press the **SORT/ HIDE COLUMNS** soft key
- > The control opens the **Column sequence** window.
- Configure how the table will be displayed

OK

- Press the **OK** soft key
- > The control will display the table as defined in the selected configuration.

> If you enter the code number 555343, the control will display the **EDIT FORMAT** soft key. With this soft key, you can change the table properties.

Hiding single points for the machining process

In the **FADE** column of the point table, you can specify if the defined point is to be hidden during the machining process.

To hide points:

- Select the desired point from the table
- Select the **FADE** column

ENT

- Activate hiding with the **ENT** key

NO ENT

- Deactivate hiding with the **NO ENT** key

Selecting a point table in the NC program

To select a point table in your NC program:

- In the **Programming** operating mode, select the NC program for which you want to activate the point table.

PGM CALL

- Press the **PGM CALL** key

SELECT POINT TABLE

- Press the **SELECT POINT TABLE** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- Select the point table from the folder structure
- Press the **OK** soft key

If the point table is not stored in the same directory as the NC program, you must enter the complete path.

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. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

```
110 %:PAT: "TNC:\nc_prog\positions.pnt"*
```

Using point tables

To call a cycle at the points defined in the point table, program the cycle call with **G79 PAT**.

With **G79 PAT**, the control will process the point table that you defined last.

To use a point table:

CYCL CALL

- Press the **CYCL CALL** key

CYCLE CALL PAT

- Press the **CYCL CALL PAT** soft key
- Enter the feed rate, e.g. **F MAX**

> The control will use this feed rate to traverse between the points of the point table. If you do not define a feed rate, the control will use the feed rate that was defined last.

- Enter a miscellaneous function if required
- Press the **END** key

Notes

- If you want to move at reduced feed rate when pre-positioning in the tool axis, program the **M103** miscellaneous function.
- With **G79 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 **%**.

Definition

File type	Definition
***.pnt**	Points table

8.6 Nesting

Types of nesting

- Subprogram calls in subprograms
- Program-section repeats within a program-section repeat
- Subprogram calls within program-section repeats
- Program-section repeats within subprograms

Subprograms and program-section repeats can call external NC programs as well.

Nesting depth

The nesting depth defines, among other things, how often program sections or subprograms may contain further subprograms or program section repeats.

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for external NC programs: 19, for which a **G79** has the effect of calling an external program
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example

%UPGMS G71 *	
...	
N17 L "UP1",0*	Subprogram at label G98 L1 is called
...	
N35 G00 G40 Z+100 M2*	Last program block of the main program with M2
N36 G98 L "UP1"	Beginning of subprogram UP1
...	
N39 L2,0*	Subprogram at label G98 L2 is called
...	
N45 G98 L0*	End of subprogram 1
N46 G98 L2*	Beginning of subprogram 2
...	
N62 G98 L0*	End of subprogram 2
N99999999 %UPGMS G71 *	

Program execution

1 Main program UPGMS is executed up to NC block 17
2 Subprogram UP1 is called, and executed up to NC block 39
3 Subprogram 2 is called, and executed up to NC block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
4 Subprogram UP1 is called, and executed from NC block 40 up to NC block 45. End of subprogram 1 and return jump to the main program UPGMS.
5 Main program UPGMS is executed from NC block 18 up to NC block 35. Program end and return jump to NC block 0

Repeating program section repeats

Example

%REPS G71 *	
...	
N15 G98 L1*	Beginning of program section repeat 1
...	
N20 G98 L2*	Beginning of program section repeat 2
...	
N27 L2,2*	Program section call with two repeats
...	
N35 L1,1*	The program section between this NC block and G98 L1
...	(NC block 15) is repeated once
N99999999 %REPS G71 *	

Program execution

1 Main program REPS is executed up to NC block 27
2 The program section between NC block 27 and NC block 20 is repeated twice
3 Main program REPS is executed from NC block 28 up to NC block 35
4 The program section between NC block 35 and NC block 15 is repeated once (including the program section repeat between NC block 20 and NC block 27)
5 Main program REPS is executed from NC block 36 up to NC block 50. Program end and return jump to NC block 0

Repeating a subprogram

Example

%UPGREP G71 *	
...	
N10 G98 L1*	Beginning of program section repeat 1
N11 L2,0*	Subprogram call
N12 L1,2*	Program section call with two repeats
...	
N19 G00 G40 Z+100 M2*	Last NC block of the main program with M2
N20 G98 L2*	Beginning of subprogram
...	
N28 G98 L0*	End of subprogram
N99999999 %UPGREP G71 *	

Program execution

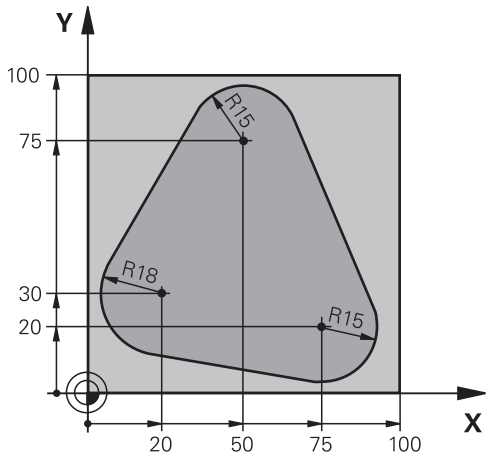
1 Main program UPGREP is executed up to NC block 11
2 Subprogram 2 is called and executed.
3 The program section between NC block 12 and NC block 10 is repeated twice. This means that subprogram 2 is repeated twice
4 Main program UPGREP is executed from NC block 13 up to NC block 19. Program end and return jump to NC block 0

8.7 Programming examples

Example: Milling a contour in several infeeds

Program run:

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Contour milling
- Repeat infeed and contour-milling

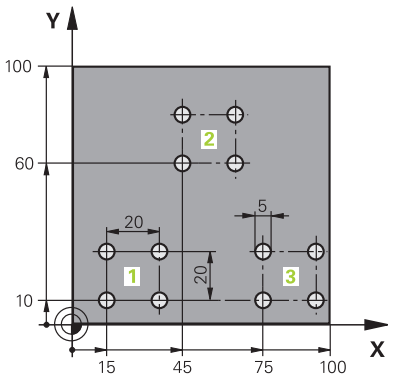


%PGMREP G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S3500*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 I+50 J+50*	Set pole
N60 G10 R+60 H+180*	Pre-position in the working plane
N70 G01 Z+0 F1000 M3*	Pre-position to the workpiece surface
N80 G98 L1*	Set label for program section repeat
N90 G91 Z-4*	Incremental downfeed (in the air)
N100 G11 G41 G90 R+45 H+180 F250*	First contour point
N110 G26 R5*	Contour approach
N120 H+120*	
N130 H+60*	
N140 H+0*	
N150 H-60*	
N160 H-120*	
N170 H+180*	
N180 G27 R5 F500*	Contour departure
N190 G40 R+60 H+180 F1000*	Retract tool
N200 L1,4*	Return jump to label 1; section is repeated a total of 4 times
N200 G00 Z+250 M2*	Retract the tool, end of program
N99999999 %PGMWDH G71 *	

Example: Groups of holes

Program run:

- 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

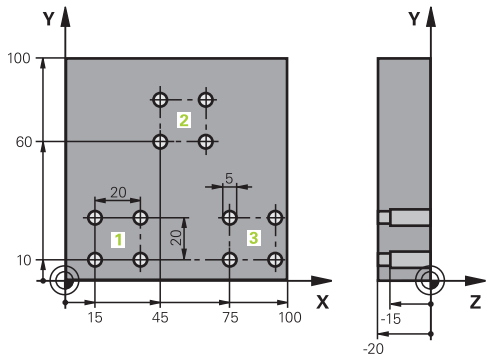


%SP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S3500*	Tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 G200 DRILLING	Cycle definition: Drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=300 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=2 ;2ND SET-UP CLEARANCE	
Q211=0 ;DWELL TIME AT DEPTH	
Q395=0 ;DEPTH REFERENCE	
N60 X+15 Y+10 M3*	Move to starting point for group 1
N70 L1,0*	Call the subprogram for the group
N80 X+45 Y+60*	Move to starting point for group 2
N90 L1,0*	Call the subprogram for the group
N100 X+75 Y+10*	Move to starting point for group 3
N110 L1,0*	Call the subprogram for the group
N120 G00 Z+250 M2*	End of main program
N130 G98 L1*	Beginning of subprogram 1: Group of holes
N140 G79*	Call cycle for 1st hole
N150 G91 X+20 M99*	Move to 2nd hole, call cycle
N160 Y+20 M99*	Move to 3rd hole, call cycle
N170 X-20 G90 M99*	Move to 4th hole, call cycle
N180 G98 L0*	End of subprogram 1
N99999999 %UP1 G71 *	

Example: Group of holes with multiple tools

Program run:

- Program the fixed cycles in the main program
- Call the complete hole pattern (subprogram 1) in the main program
- Approach the groups of holes (subprogram 2) in subprogram 1
- Program the group of holes only once in subprogram 2



%SP2 G71 *	
N10 G30 G17 X+0 Y+0 Z-40*	
N20 G31 G90 X+100 Y+100 Z+0*	
N30 T1 G17 S5000*	Centering drill tool call
N40 G00 G40 G90 Z+250*	Retract the tool
N50 G200 DRILLING	Cycle definition: Centering
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=3 ;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	
N60 L1,0*	Call subprogram 1 for the entire hole pattern
N70 G00 Z+250 M6*	Tool change
N80 T2 G17 S4000*	Drill tool call
N90 D0 Q201 P01 -25*	New depth for drilling
N100 D0 Q202 P01 +5*	New plunging depth for drilling
N110 L1,0*	Call subprogram 1 for the entire hole pattern
N120 G00 Z+250 M6*	Tool change
N130 T3 G17 S500*	Reamer tool call
N140 G201 REAMING	Cycle definition: Reaming
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
N150 L1,0*	Call subprogram 1 for the entire hole pattern

N160 G00 Z+250 M2*	End of main program
N170 G98 L1*	Beginning of subprogram 1: Entire hole pattern
N180 G00 G40 G90 X+15 Y+10 M3*	Move to starting point for group 1
N190 L2,0*	Call subprogram 2 for the group
N200 X+45 Y+60*	Move to starting point for group 2
N210 L2,0*	Call subprogram 2 for the group
N220 X+75 Y+10*	Move to starting point for group 3
N230 L2,0*	Call subprogram 2 for the group
N240 G98 L0*	End of subprogram 1
N250 G98 L2*	Beginning of subprogram 2: Group of holes
N260 G79*	Call cycle for 1st hole
N270 G91 X+20 M99*	Move to 2nd hole, call cycle
N280 Y+20 M99*	Move to 3rd hole, call cycle
N290 X-20 G90 M99*	Move to 4th hole, call cycle
N300 G98 L0*	End of subprogram 2
N310 %UP2 G71 *	

9

Programming Q Parameters

9.1 Principle and overview of functions

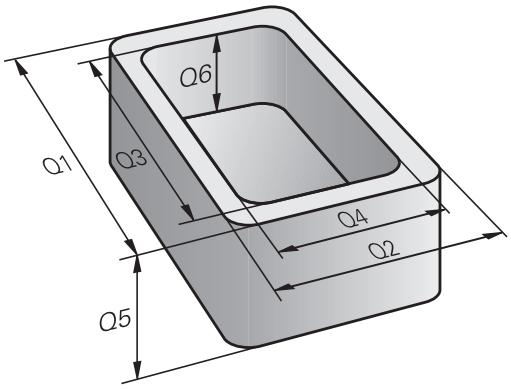
With Q parameters you can program entire families of parts in a single NC program by programming variable Q parameters instead of fixed numerical values.

Q parameters can be used in the following ways:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

The control offers more ways to use Q parameters:

- Program contours that are defined through mathematical functions
- Making the execution of machining steps dependent on logical conditions



Q parameter types

Q parameters for numerical values

Variables always consist of letters and numbers. The letters determine the type of variable and the numbers its range.

For more information, see the table below:

Variable type	Variable range	Meaning
Q parameters:		Q parameters affect all NC programs in the control's memory.
	0 to 99	User-defined Q parameters, if there are no overlaps with the HEIDENHAIN SL cycles Q 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. For this reason, use the Q parameter range 1200 to 1399 for machine manufacturer 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 1399	Q parameters for functions defined by the machine manufacturer (e.g., cycles)
	1400 to 1999	User-defined Q parameters
QL parameters:		QL parameters are active locally within an NC program.
	0 to 499	User-defined QL parameters
QR parameters:		QR parameter affect all NC programs in the control's memory; they are retained even after a restart of the control.
	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)

QR parameters will be included in backups.

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.

Q parameters for texts

Additionally, QS parameters (**S** stands for string) are available and enable you to process texts on the control.

The following characters can be used within QS 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 ; ! # $ % & ' ( ) + , - . / : < = > ? @ [ ] ^ _ ` *

Variable type	Variable range	Meaning
QS parameters:		QS parameters affect all NC programs in the control's memory.
	0 to 99	User-defined QS parameters, if there are no overlaps with the HEIDENHAIN cycles 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. For this reason, use the QS parameter range 1200 to 1399 for machine manufacturer 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

Programming notes

NOTICE

Danger of collision!

HEIDENHAIN cycles, manufacturer cycles and third-party functions use Q parameters. You can also program Q parameters within NC programs. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and suppliers.
- Check the machining sequence using a graphic simulation

You can mix Q parameters and numerical values within an NC program.

Variables can be assigned numerical values between –999 999 999 and +999 999 999. The input range is limited to 16 digits, of which 9 may be before the decimal point. The control can calculate numerical values up to 10^{10}.

You can assign a maximum of 255 characters to **QS** parameters.

The control automatically assigns some Q and QS parameters the same data, e.g., the Q parameter **Q108** is automatically assigned the current tool radius.

Further information: "Preassigned Q parameters", Page 335

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.

Using the **SET UNDEFINED** syntax element, you can assign the **undefined** status to your variables. For example, if you program a position using an undefined Q parameter, the control will ignore this movement. If you use an undefined Q parameter in the calculation steps of your NC program, the control will display an error message and stop the program run.

Calling Q parameter functions

When you are writing an NC program, press the **Q** key (in the numeric keypad for numerical input and axis selection, below the **+/-** key). The control then displays the following soft keys:

Soft key	Function group	Page
BASIC ARITHM.	Basic arithmetic (assign, add, subtract, multiply, divide, square root)	284
TRIGO-NOMETRY	Trigonometric functions	288
JUMP	If/then conditions, jumps	291
DIVERSE FUNCTION	Other functions	301
FORMULA	Entering formulas directly	294
CONTOUR FORMULA	Function for machining complex contours	See the User's Manual for Programming of Machining Cycles

If you define or assign a Q parameter, then the control shows the **Q**, **QL** and **QR** soft keys. You can use these soft keys to select the desired parameter type. Then you define the parameter number.

If you have a alphabetic keyboard connected via the USB port, you can press the **Q** key to open the dialog for entering a formula.

9.2 Part families—Q parameters in place of numerical values

Application

The Q parameter function **D0: ASSIGN** allows you to assign numerical values to Q parameters. You then use a Q parameter in place of the numerical value in the NC program.

Example

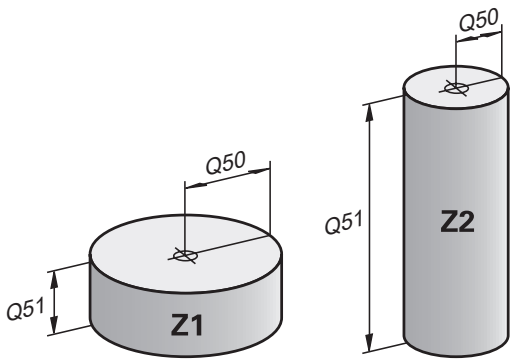
N150 D00 Q10 P01 +25*	Assign
...	Q10 is assigned the value 25
N250 G00 X +Q10*	Corresponds to G00 X +25

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

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

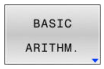


9.3 Describing contours with mathematical functions

Application

The Q parameters listed below enable you to program basic mathematical functions in an NC program:

- Select the Q parameter function: Press the **Q** key in the numeric keypad
- The Q parameter functions are displayed in the soft key row.



- Press the **BASIC ARITHM.** soft key
- The control displays the soft keys for basic mathematical functions

Overview

Soft key	Function
D0 X = Y	**D00**: Assignment Example: **D00 Q5 P01 +60 *** Q5 = 60 Assign a value or the **Undefined** status
D1 X + Y	**D01**: Addition Example: **D01 Q1 P01 -Q2 P02 -5 *** Q1 = −Q2+(−5) Calculate and assign the sum of two values
D2 X - Y	**D02**: Subtraction Example: **D02 Q1 P01 +10 P02 +5 *** Q1 = +10−(+5) Calculate and assign the difference of two values.
D3 X * Y	**D03**: Multiplication Example: **D03 Q2 P01 +3 P02 +3 *** Q2 = 3*3 Calculate and assign the product of two values.
D4 X / Y	**D04**: Division Example: **D04 Q4 P01 +8 P02 +Q2 *** Q4 = 8/Q2 Calculate and assign the quotient of two values Restriction: You cannot divide by 0
D5 SQRT	**D05**: Square root Example: **D05 Q20 P01 4 *** Q20 = √4 Calculate and assign the square root of a number Restriction: You cannot calculate a square root from a negative value

You can enter the following to the right of the **=** sign:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Programming fundamental operations

Example: Assignment

```
N16 D00 Q5 P01 +10*
N17 D03 Q12 P01 +Q5  P02 +7*
```

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

D0 X = Y

- To select the **ASSIGN** Q parameter function: Press the **D0 X=Y** soft key
- > The control asks you for the number of the result parameter.
- Enter **5** (number of Q parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the value or parameter.
- Enter **10** (value)

ENT

- Confirm with the **ENT** key
- > As soon as the control reads the NC block, the value **10** is assigned to the parameter **Q5**.

Example: Multiplication

Q

- Select the Q parameter function: Press the **Q** key

BASIC ARITHM.

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key

D3 X * Y

- To select the **MULTIPLICATION** Q parameter function, press the **D3 X * Y** soft key
- > The control asks you for the number of the result parameter.
- Enter **12** (number of Q parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the first value or parameter.
- Enter **Q5** (parameter)

ENT

- Confirm with the **ENT** key
- > The control asks you for the second value or parameter.
- Enter **7** for the second value

ENT

- Confirm with the **ENT** key

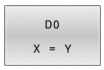
Resetting Q parameters

Example

```
16 D00: Q5 SET UNDEFINED*
17 D00: Q1 = Q5*
```

- Select the Q parameter function: Press the **Q** key

- Select basic mathematical functions by pressing the **BASIC ARITHM.** soft key



- To select the ASSIGN Q parameter function: Press the **D0 X = Y** soft key
- The control asks you for the number of the result parameter.
- Enter **5** (number of Q parameter)

- Confirm with the **ENT** key
- The control asks you for the value or parameter.

- Press **SET UNDEFINED**

The **D00** function also supports transfer of the value **Undefined**. If you try to transfer the undefined Q parameter without **D00**, the control shows the error message **Invalid value**.

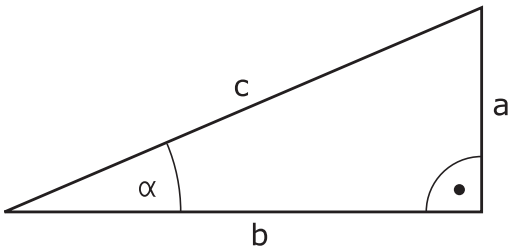
9.4 Trigonometric functions

Definitions

Sine: sin α = opposite side/hypotenuse
sin α = a/c

Cosine: cos α = adjacent side/hypotenuse
cos α = b/c

Tangent: tan α = opposite side/adjacent side
tan α = a/b or tan α = sin α/cos α



where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The control can find the angle from the tangent:
α = arctan(a/b) or α = arctan(sin α/cos α)

Example:
a = 25 mm
b = 50 mm
α = arctan(a/b) = arctan 0.5 = 26.57°
Furthermore:
$a^2+b^2 = c^2$ (where $a^2 = a*a$)
$c = \sqrt{(a^2+b^2)}$

Programming trigonometric functions

You can also calculate trigonometric functions with Q parameters.

Q

- ▶ Select the Q parameter function: Press the **Q** key in the numeric keypad
- > The Q parameter functions are displayed in the soft key row.

TRIGO- NOMETRY

- ▶ Press the **TRIGO- NOMETRY** soft key
- > The control displays the soft keys for trigonometric functions.

Overview

Soft key	Function
D6 SIN(X)	**D06**: Sine Example: **D06 Q20 P01 -Q5 *** Q20 = sin(–Q5) Calculate and assign the sine of an angle in degrees
D7 COS(X)	**D07**: Cosine Example: **D07 Q21 P01 -Q5 *** Q21 = cos(–Q5) Calculate and assign the cosine of an angle in degrees
D8 X LEN Y	**D08**: Root of the sum of squares Example: **D08 Q10 P01 +5 P02 +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).
D13 X ANG Y	**D13**: angle Example: **D13 Q20 P01 +10 P02 -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 < angle < 360°)

9.5 Calculation of circles

Application

The control can use the functions for calculating circles to calculate the circle center and the circle radius from three or four given points on the circle. The calculation is more accurate if four points are used.

Application: These functions can be used, for example, if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Soft key	Function
D23 3 POINTS OF CIRCLE	**D23**: Circle data from three points on the circle Example: **D23 Q20 P01 Q30*** The control saves the determined values in the Q parameters **Q20** to **Q22**.

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**

Soft key	Function
D24 4 POINTS OF CIRCLE	**D24**: Circle data from four points on the circle Example: **D24 Q20 P01 Q30*** The control saves the determined values in the Q parameters **Q20** to **Q22**.

The control checks the values in the Q parameters **Q30** to **Q37** 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**

D23 and **D24** not only assign a value to the results variable to the left of the equal sign, but also to the subsequent variables.

9.6 If-then decisions with Q parameters

Application

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.

> Before creating your NC program, compare the if-then decisions with the subprogram and program section repeat programming techniques.
>
> You can thereby avoid possible misunderstandings and programming errors.
>
> **Further information:** "Labeling subprograms and program section repeats", Page 256

If the condition is not fulfilled, the control continues with the next NC block.

If you want to call an external NC program, then program a program call with **%** after the label.

Jump conditions

Unconditional jump

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

```
D09 P01 +10 P02 +10 P03 1*
```

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.

Conditioning jumps with counters

The jump function allows you to repeat a machining operation any number of times. A Q parameter serves as a counter that increments by 1 at every program section repeat.

The jump function allows you to compare the counter with the number of desired machining operations.

> These jumps differ from the subprogram and program section repeat programming techniques.
>
> On the one hand, for example, jumps require no completed program section ending with L0. On the other hand, jumps do not take these return jump labels into consideration!

Example

%COUNTER G71 *	
;	
N20 Q1 = 0	Loaded value: Initialize counter
N30 Q2 = 3	Loaded value: Number of jumps
;	
N50 G98 L99*	Label
N60 Q1 = Q1 + 1	Initialize counter: New Q1 value = Old Q1 value + 1
N70 D12 P01 +Q1 P02 +Q2 P03 99*	Run program jumps 1 and 2
N80 D09 P01 +Q1 P02 +Q2 P03 99*	Run program jump 3
;	
N99999999 %COUNTER G71 *	

Programming if-then decisions

Possibilities for jump inputs

The following inputs are possible for the condition **IF**:

- Numbers
- Texts
- Q, QL, QR
- **QS** (string parameter)

You have three possibilities for entering the jump address **GOTO**:

- **LBL NAME**
- **LBL NUMBER**
- **QS**

The if-then decisions appear when the **JUMP** soft key is pressed. The control displays the following soft keys:

Soft key	Function
D9 IF X EQ Y GOTO	**D09**: jump if equal Example: **D09 P01 +Q1 P02 +Q3 P03 “UPCAN25“ *** If both values are equal, the control jumps to the defined label.
D9 IF X EQ Y GOTO IS UNDEFINED	**D09**: jump if undefined Example: **D09 P01 +Q1 IS UNDEFINED P03 “UPCAN25“ *** If the variable is undefined, the control jumps to the defined label.
D9 IF X EQ Y GOTO IS DEFINED	**D09**: jump if defined Example: **D09 P01 +Q1 IS DEFINED P03 “UPCAN25“ *** If the variable is defined, the control jumps to the defined label.
D10 IF X NE Y GOTO	**D10**: jump if not equal Example: **D10 P01 +10 P02 -Q5 P03 10 *** If both values are not equal, the control jumps to the defined label.
D11 IF X GT Y GOTO	**D11**: jump if greater than Example: **D11 P01 +Q1 P02 +10 P03 QS5 *** If the first value is greater than the second value, the control jumps to the defined label.
D12 IF X LT Y GOTO	**D12**: jump if less than Example: **D12 P01 +Q5 P02 +0 P03 “ANYNAME“ *** If the first value is less than the second value, the control jumps to the defined label.

9.7 Entering formulas directly

Entering formulas

Using soft keys, you can enter mathematical formulas containing multiple calculation operations directly into the NC program.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Select **Q**, **QL**, or **QR**
- > The control displays the available mathematical operations in the soft-key row.

Calculation rules

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).

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	**+**, **-**

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

Example: Perform multiplication/division before addition/ subtraction

N120 Q1 = 5 * 3 + 2 * 10	= 35

- 1st calculation: 5 * 3 = 15
- 2nd calculation: 2 * 10 = 20
- 3rd calculation: 15 + 20 = 35

Example: Calculate power before addition/subtraction

N130 Q2 = SQ 10 - 3^3	= 73

- 1st calculation: 10 squared = 100
- 2nd calculation: 3 to the power of 3 = 27
- 3rd calculation: 100 – 27 = 73

Example: Calculate function before power

N140 Q4 = SIN 30 ^ 2	= 0.25

- 1st calculation: Calculate sine of 30 = 0.5
- 2nd calculation: 0.5 squared = 0.25

Example: Evaluate expression in parentheses before function

N150 Q5 = SIN (50 - 20)	= 0.5

- 1st calculation: Perform operations in parentheses first: 50 – 20 = 30
- 2nd calculation: Calculate sine of 30 = 0.5

Overview

The control displays the following soft keys:

Soft key	Logical function	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
(	**Open parenthesis** Example: **Q12 = Q1 * (Q2 + Q3)**	Expression in parentheses
)	**Close parenthesis** Example: **Q12 = Q1 * (Q2 + Q3)**	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
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	**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

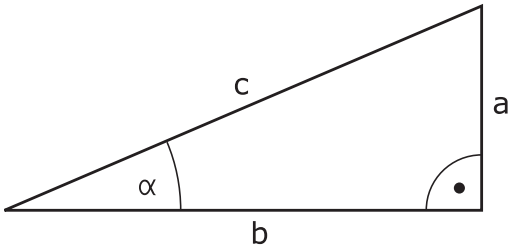
Soft key	Logical function	Operator
PI	**Use the “pi” constant** π = 3.14159 Example: **Q15 = PI**	
LN	**Calculate the natural logarithm (LN)** Base = e = 2.7183 Example: **Q15 = LN Q11**	Function
LOG	**Calculate the logarithm** Base = 10 Example: **Q33 = LOG Q22**	Function
EXP	**Use the exponential function (e ^ n)** Base = e = 2.7183 Example: **Q1 = EXP Q12**	Function
NEG	**Negate** Multiply by −1 Example: **Q2 = NEG Q1**	Function
INT	**Calculate an integer** Truncate decimal places Example: **Q3 = INT Q42** The **INT** function does not round off—it simply truncates the decimal places. **Further information:** "Example: Rounding a value", Page 343	Function
ABS	**Calculate the absolute value** Example: **Q4 = ABS Q22**	Function
FRAC	**Calculate a fraction** Truncate the digits before the decimal point Example: **Q5 = FRAC Q23**	Function
SGN	**Check the algebraic sign** Example: **Q12 = SGN Q50** If **Q50 = 0**, then **SGN Q50 = 0** If **Q50 < 0**, then **SGN Q50 = -1** If **Q50 > 0**, then **SGN Q50 = 1**	Function
%	**Calculate the modulo value (division remainder)** Example: **Q12 = 400 % 360** Result: **Q12 = 40**	Function

Example: Trigonometric function

The lengths of the opposite side a in parameter **Q12** and the adjacent side b in **Q13** are given.

The angle **α** is to be calculated.

Calculate the angle **α** from the opposite side a and the adjacent side b by means of the arc tangent; assign result **Q25**:



Q
- Press the **Q** key

FORMULA
- Press the **FORMULA** soft key
- > The control asks you for the number of the result parameter.
- Enter **25**

ENT
- Press the **ENT** key

▷
- Scroll through the soft-key row

ATAN
- Press the **ATAN** arc tangent function soft key

◁
- Scroll through the soft-key row

(
- Press the **Opening parenthesis** soft key

Q
- Enter **12** (the parameter number)

/
- Select division

Q
- Enter **13** (the parameter number)

)
- Press the **Closing parenthesis** soft key

END
- Press the **END** key to conclude the formula entry

Example

```
N10  Q25 = ATAN (Q12/Q13)
```

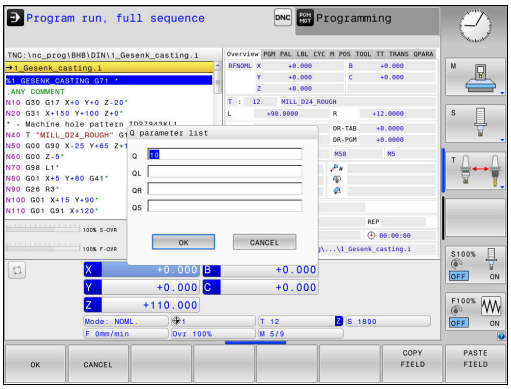
9.8 Checking and changing Q parameters

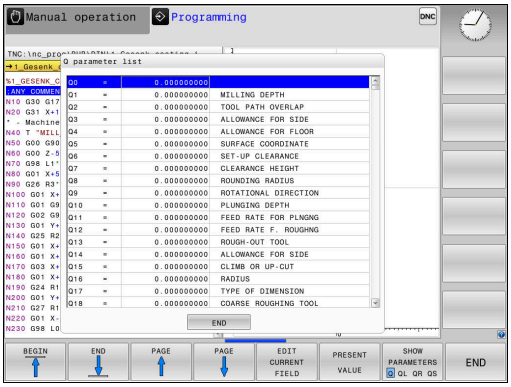
Procedure

You can check Q parameters in all operating modes, and also edit them.

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

- To call the Q parameter functions, press the **Q INFO** soft key or the **Q** key
- > The control lists all of the parameters and their corresponding current values.
- Use the arrow keys or the **GOTO** key to select the desired parameter.
- If you want to change the value, then press the **EDIT CURRENT FIELD** soft key, enter the new value, and confirm with the **ENT** key
- If you want to leave the value unchanged, then press the **PRESENT VALUE** soft key or close the dialog with the **END** key





If you want to check or edit local, global or string parameters, press the **SHOW PARAMETERS Q QL QR QS** soft key. The control then displays the specific parameter type. The functions previously described also apply.

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.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

This status is reached after an NC block has been executed, for example in **Program run, single block**

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.

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

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

- If needed, interrupt the program run (e.g., by pressing the **NC STOP** key and the **INTERNAL STOP** soft key), or stop the test run

- Display the soft key row for screen layout

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

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

- Press the **Q PARAMETER LIST** soft key.
- The control opens a pop-up window.
- For each parameter type (Q, QL, QR, QS), define the parameter numbers you wish to check. Separate single Q parameters with a comma, and connect sequential Q parameters with a hyphen, e.g. 1,3,200-208. The input range per parameter type is 132 characters

The display in the **QPARA** tab always contains eight decimal places. The result of **Q1 = COS 89.999** is shown by the control as 0.00001745, for example. Very large or very small values are displayed by the control in exponential notation. The result of **Q1 = COS 89.999 * 0.001** is shown by the control as +1.74532925e-08, where e-08 corresponds to the factor of 10^{-8}.

9.9 Additional functions

Overview

The additional functions appear when the **DIVERSE FUNCTION** soft key is pressed. The control displays the following soft keys:

Soft key	Function	Page
D14 ERROR=	**D14** Display error messages	302
D16 F-PRINT	**D16** Formatted output of texts or Q parameter values	308
D18 SYS-DATUM READ	**D18** Read system data	318
D19 PLC=	**D19** Transfer values to the PLC	318
D20 WAIT FOR	**D20** NC and PLC synchronization	319
D26 OPEN THE TABLE	**D26** Open a freely definable table	389
D27 WRITE TO TABLE	**D27** Write to a freely definable table	390
D28 READ TABLE	**D28** Read from a freely definable table	392
D29 PLC LIST=	**D29** Transfer up to eight values to the PLC	320
D37 EXPORT	**D37** Export local Q parameters or QS parameters to a calling NC program	320
D38 TRANSMIT	**D38** Send information from the NC program	321

D14 Output of error messages

With the **D14** function, you can output error messages under program control. The messages are pre-defined by the machine manufacturer or by HEIDENHAIN.

If, during program run or during simulation, the control executes the **D14** function, it will interrupt program run and display the defined message. You must then restart the NC program.

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 machine manufacturer assigns and defines the error numbers up to 999 and from 3000 to 9999.

Example

The control is intended to display a message if the spindle is not switched on.

```
N180 D14  P01 1000*
```

The following is a complete list of the **D14** error messages. Please be aware that not all error messages might be available, depending on the model of your control.

Error message predefined by HEIDENHAIN

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRROR IMAGE not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Input value incorrect
1012	Incorrect sign
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory input

Error number	Text
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong rpm
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Q218 must be greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2

Error number	Text
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal to 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter Q351 unequal to 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3D ROT not permitted
1076	Activate 3D ROT
1077	Enter depth as negative
1078	Q303 in meas. cycle undefined!
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory meas. points
1082	Incorrect clearance height
1083	Contradictory plunge type
1084	This fixed cycle not allowed
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not allowed
1090	Enter an infeed not equal to 0
1091	Switchover of Q399 not allowed
1092	Tool not defined
1093	Tool number not permitted
1094	Tool name not permitted

Error number	Text
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed
1100	Kinematic access not possible
1101	Meas. pos. not in traverse range
1102	Preset compensation not possible
1103	Tool radius too large
1104	Plunging type is not possible
1105	Plunge angle incorrectly defined
1106	Angular length is undefined
1107	Slot width is too large
1108	Scaling factors not equal
1109	Tool data inconsistent
1110	MOVE not possible
1111	Presetting not allowed!
1112	Thread angle too small!
1113	3D ROT status is contradictory!
1114	Configuration is incomplete
1115	No turning tool is active
1116	Tool orientation is inconsistent
1117	Angle not possible!
1118	Radius too small!
1119	Thread runout too short!
1120	Contradictory meas. points
1121	Too many limits
1122	Machining strategy with limits not possible
1123	Machining direction not possible
1124	Check the thread pitch!
1125	Angle cannot be calculated
1126	Eccentric turning not possible
1127	No milling tool is active
1128	Insufficient length of cutting edge
1129	Gear definition is inconsistent or incomplete
1130	No finishing allowance provided
1131	Line does not exist in table
1132	Probing process not possible
1133	Coupling function not possible

Error number	Text
1134	Machining cycle is not supported by this NC software
1135	Touch probe cycle is not supported by this NC software
1136	NC program aborted
1137	Touch probe data incomplete
1138	LAC function not possible
1139	Rounding radius or chamfer is too large!
1140	Axis angle not equal to tilt angle
1141	Character height not defined
1142	Excessive character height
1143	Tolerance error: Workpiece rework
1144	Tolerance error: Workpiece scrap
1145	Faulty dimension definition
1146	Illegal entry in compensation table
1147	Transformation not possible
1148	Tool spindle incorrectly configured
1149	Offset of the turning spindle unknown
1150	Global program settings are active
1151	Faulty configuration of OEM macros
1152	The combination of programmed oversizes is not possible
1153	Measured value not captured
1154	Check the monitoring of the tolerance
1155	Hole is smaller than the stylus tip
1156	Preset cannot be set
1157	Alignment of a rotary table is not possible
1158	Alignment of rotary axes is not possible
1159	Infeed limited to length of cutting edge
1160	Machining depth defined as 0
1161	Tool type is unsuitable
1162	Finishing allowance not defined
1163	Machine datum could not be written
1164	Spindle for synchronization could not be ascertained
1165	Function is not possible in the active operating mode
1166	Oversize defined too large
1167	Number of teeth not defined
1168	Machining depth does not increase monotonously

Error number	Text
1169	Infeed does not decrease monotonously
1170	Tool radius not defined correctly
1171	Mode for retraction to clearance height not possible
1172	Gear wheel definition incorrect
1173	Probing object contains different types of dimension definition
1174	Dimension definition contains impermissible characters
1175	Actual value in dimension definition faulty
1176	Starting point of hole too deep
1177	Dimension def.: Nominal value missing for manual pre-positioning
1178	A replacement tool is not available
1179	OEM macro is not defined
1180	Measurement not possible with auxiliary axis
1181	Start position not possible with modulo axis
1182	Function only possible if door is closed
1183	Number of possible records exceeded
1184	Inconsistent machining plane due to axis angle with basic rotation
1185	Transfer parameter contains an impermissible value
1186	Tooth width RCUTS is defined too large
1187	Usable length LU of the tool is too small
1188	The defined chamfer is too large
1189	Chamfer angle cannot be machined with the active tool
1190	The allowances do not define any stock removal
1191	Spindle angle not unique

D16 – Formatted output of text and Q parameter values

Fundamentals

With the function **D16**, you can output formatted fixed and variable numbers and 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

Procedure

In order to output fixed or variable numbers and texts, the following is required:

- Source file

  The source file determines the contents and formatting.
- NC function **D16**

  The control creates the output file using the NC function **D16**.

  The maximum size of the output file is 20 kB.

Creating a text file

In order to output formatted text and the values of the Q parameters, use the control's text editor to create a text file. In this file, you can define the format and Q parameters to be output.

Proceed as follows:

- Press the **PGM MGT** key

- Press the **NEW FILE** soft key
- Create a file with the extension **.A**

Available functions

Use the following formatting functions for creating a text file:

Please note that the input is case-sensitive.

Formatting characters	Meaning
"..."	Identifies 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 Q, QL and QR parameters ■ **F**: Float (32-bit floating-point number) ■ **D**: Double (64-bit floating-point number) ■ **I**: Integer (32-bit integer)

Formatting characters	Meaning
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 QS 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

Example

Input	Meaning
"X1 = %+9.3 F", Q31;	Format for the Q parameter: ■ **X1 =**: Output the text **X1 =** ■ **%**: Specify the format ■ **+**: Number right-aligned ■ **9.3**: Total of 9 characters; 3 of them are decimal places ■ **F**: Floating (decimal number) ■ **Q31**: Output the value from **Q31** ■ **;**: End of block

The following functions allow you to include the following additional information in the protocol log file:

Keyword	Meaning
CALL_PATH	Output the path name of the NC program that contains the **D16** function (e.g., **"TouchProbe: %S",CALL_PATH;**)
M_CLOSE	Close the file written to with **D16**
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_RUSSIAN	Outputs text only for Russian conversational language

Keyword	Meaning
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
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
HOUR	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

Example

Example of a text file to define the output format:

```
“MEASURING LOG OF IMPELLER CENTER OF GRAVITY“;
“DATE: %02d.%02d.%04d“,DAY,MONTH,YEAR4;
“TIME: %02d:%02d:%02d“,HOUR,MIN,SEC;
“NO. OF MEASURED VALUES: = 1“;
"X1 = %9.3F", Q31;
"Y1 = %9.3F", Q32;
"Z1 = %9.3F", Q33;
L_GERMAN;
"Werkzeuglänge beachten";
L_ENGLISH;
"Remember the tool length";
```

Example

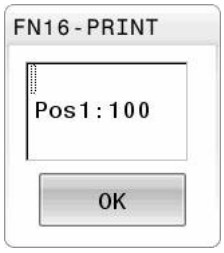
Example of a format file that generates an output file with variable contents:

```
“TOUCHPROBE“;
“%S“,QS1;
M_EMPTY_HIDE;
“%S“,QS2;
“%S“,QS3;
M_EMPTY_SHOW;
“%S“,QS4;
M_CLOSE;
```

Example of an NC program that defines only **QS3**:

N110 Q1 = 100	; Assign the value **100** to **Q1**
N120 QS3 = "Pos 1: " || TOCHAR(DAT+Q1)*	; Convert the numerical value of **Q1** to an alphanumeric value and assign it to the defined string
N130 D16 P01 TNC:\D16.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**:



Activating D16 output in an NC program

Use the function **D16** to define the output file.

The control creates the output file in the following cases:

- End of program **G71**
- Cancellation of program with the **NC STOP** key
- **M_CLOSE** keyword in the format file

Enter the path to the text file and the path to the output file in the D16 function.

Proceed as follows:

- Press the **Q** key.

- Press the **DIVERSE FUNCTION** soft key

- Press the **D16 F-PRINT** soft key

- Press the **SELECT FILE** soft key
- Select the source, i.e. the text file in which the output file is defined

- Confirm with the **ENT** key
- Select the target, i.e. the output path

There are two ways to define the output path:

- Directly in the **D16** function
- In the machine parameters, under **CfgUserPath** (no. 102200)

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. The **APPLY FILE NAME** soft key provided in the selection window of the **SELECT FILE** soft key is available for this.

Specifying the path in the D16 function

If you enter only the file name as the path for the log file, the control saves the log file in the directory in which the NC program with the **D16** function is located.

As an alternative to complete paths, you can program relative paths:

- Starting from the folder of the calling file one folder level down
  D16 P01 MASKE\MASKE1.A/ PROT\PROT1.TXT
- Starting from the folder of the calling file one folder level up and in another folder **D16 P01 ..\MASKE\MASKE1.A/ ..\PROT1.TXT**

Use the **SYNTAX** soft key to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.

Further information: "File names", Page 117

If the complete path is enclosed in quotation marks, you can use both **** and **/** to separate the folders and files.

Operating and programming notes:

- If you define a path both in the machine parameters and in the **D16** function, the path in the **D16** function has priority.
- If you output the same file more than once in the NC program, the control appends the current output to the end of the contents of the previously output file.
- In the **D16** block, program the format file and the log file, each with the extension for the file type.
- The file name extension of the log file determines the file type of the output (e.g., TXT, A, XLS, HTML).
- Use **D18** to retrieve information that is relevant and interesting in log files, such as the number of the touch-probe cycle last used.
  Further information: "D18 – Reading system data", Page 318

Defining the output path in machine parameters

If you wish to save the measurement results to a certain directory, you can define the output path for the log file in the machine parameters.

To change the output path:

MOD
- Press the **MOD** key
- Enter the code number 123

↓
- Select the machine parameter **CfgUserPath** (no. 102200)

↓
- Select the machine parameter **fn16DefaultPath** (no. 102202)
- > The control opens a pop-up window.
- Select the output path for the machine operating modes

↓
- Select the machine parameter **fn16DefaultPathSim** (no. 102203)
- > The control opens a pop-up window.
- Select the output path for the **Programming** and **Test Run** operating modes

Enter the source or the target with parameters

You can enter the paths of the source and the output files as variable values. For this purpose, the desired variables must have been defined in the NC program.

Further information: "Assigning string parameters", Page 324

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

If you want use a QS parameter to output a path to a log file, then use the function **%RS**. This ensures that the control does not interpret the special characters as formatting characters.

Example

```
N90 D16 P01 TNC:\MASKE\MASKE1.A/ TNC:\PROT1.TXT
```

The control creates the PROT1.TXT file:

MEASURING LOG OF IMPELLER CENTER OF GRAVITY
DATE: 15.07.2015
TIME: 08:56:34
NO. OF MEASURED VALUES: = 1
X1 = 149.360
Y1 = 25.509
Z1 = 37.000
Remember the tool length

Displaying messages on the control screen

You can use the **D16** 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.

Example

```
N110 D16 P01 TNC:\MASKE-
     \MASKE1.A / SCREEN:
```
; Display the output file with **FN 16** on the control screen

If the message has more lines than can fit in the pop-up window, you can use the arrow keys to scroll through the window.

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.

If you want to overwrite the previous pop-up window, program the **M_CLOSE** or **M_TRUNCATE** keyword.

Closing the pop-up window

You can close the window in the following ways:

- By pressing the **CE** key
- Defining the **SCLR:** output path (Screen Clear)

Example

```
N90 D16 P01 TNC:\MASKE\MASKE1.A / SCLR:
```

You can also use the **D16** function to close the pop-up window. In this case, no text file is required.

Example

```
N90 D16 P01 / SCLR:
```

Exporting messages

With the **D16** 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 **D16** function.

Example

N110 D16 P01 TNC:\MSK-\MSK1.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 messages

You can use the **D16** function to print output files to a connected printer.

The connected printer must be PostScript-enabled.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

The control will only print the output file if the source 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.

Example

N110 D16 P01 TNC:\MASKE-\MASKE1.A / PRINTER:-\PRINT1	; Print output file with **FN 16**

D18 – Reading system data

With the **D18** function, you can read system data and save them to Q parameters. The selection of the system datum occurs via a group number (ID no.), a system data number, and, if necessary, an index.

The read values of the function **D18** are always output by the control in **metric** units regardless of the NC program's unit of measure.

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: "System data", Page 472

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
N55 D18 Q25 ID210 NR4 IDX3*
```

D19 Transferring values to PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers 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 risk of collision during the execution of the function and during the subsequent machining!

- 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

The **D19** function transfers up to two fixed or variable values to the PLC.

D20 NC and PLC synchronization

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers 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 risk of collision during the execution of the function and during the subsequent machining!

- 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

With the **D20** function, you can synchronize the NC and the PLC during program run. The control stops program run until the condition you specified in the **D20** block has been met.

The **SYNC** function is used whenever you read system data (e.g., with **D18**). The system data need to be synchronized with the current date and time. Use the **D20** to stop the look-ahead calculation. When the control encounters **D20**, it will only calculate the NC block after it has executed the NC block that contains **D20**.

Example: Pause internal look-ahead calculation, read current position in the X axis

N11 D20 SYNC	; Stop internal look-ahead calculation with **FN 20**
N12 D18 Q1 ID270 NR1 IDX1*	; Determine the position of the X axis with **FN 18**

D29 Transferring values to the PLC

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers 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 risk of collision during the execution of the function and during the subsequent machining!

- 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

The **D29** function transfers up to eight fixed or variable values to the PLC.

D37 - EXPORT

NOTICE

Danger of collision!

Changes to the PLC can result in undesired behavior and serious errors (e.g., the control becomes inoperable). For this reason, access to the PLC is password-protected. This function allows HEIDENHAIN, the machine manufacturer, and third-party providers 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 risk of collision during the execution of the function and during the subsequent machining!

- 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

You need the **D37** function if you want to create your own cycles and integrate them in the control.

D38 – Sending information from the NC program

The function **D38** 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).

The syntax consists of two parts:

- **Format of transmitted text**: Output text with optional placeholders for variable values (e.g., **%f**)

  Input may be in the form of QS parameters.

  Both fixed and variable numbers and texts are case-sensitive, so enter them correctly.

- **Datum for placeholder in text**: List of max. seven Q, QL, or QR variables (e.g., **Q1**)

Data transmission is through a standard TCP/IP computer network.

For more detailed information, consult the RemoTools SDK manual.

Example

Document the values from **Q1** and **Q23** in the log.

```
D38* /"Q-Parameter Q1: %f Q23: %f" P02 +Q1 P02 +Q23*
```

Example

Define the output format for the variable values.

```
D38* /"Q-Parameter Q1: %05.1f" P02 +Q1*
```

> The control outputs the variable value as a five-digit number, of which one digit is a decimal place. The output will be padded with leading zeroes as needed.

```
D38* /"Q-Parameter Q1: % 7.3f" P02 +Q1*
```

> The control outputs the variable value as a seven-digit number, of which three digits are decimal places. The output will be padded with blank spaces as needed.

To obtain **%** in the output text, enter **%%** at the desired position.

Example

In this example, you will send information to StateMonitor.

With the function **D38**, you can, for example, enter job data.

The following requirements must be met in order to use this function:

- StateMonitor version 1.2

  Job management with JobTerminal (option 4) is possible with StateMonitor version 1.2 or higher
- The job has been entered in StateMonitor
- Machine tool has been assigned

The following stipulations apply to this example:

- Job number 1234
- Working step 1

D38* /"JOB:1234_STEP:1_CREATE"*	Create job
D38* /"JOB:1234_STEP:1_CREATE_ITEMNAME: HOLDER_ITEMID:123_TARGETQ:20" *	Alternative: Create job with part name, part number, and required quantity
D38* /"JOB:1234_STEP:1_START"*	Start job
D38* /"JOB:1234_STEP:1_PREPARATION"*	Start preparation
D38* /"JOB:1234_STEP:1_PRODUCTION"*	Production
D38* /"JOB:1234_STEP:1_STOP"*	Stop job
D38* /"JOB:1234_STEP:1_ FINISH"*	Finish job

You can also report the quantity of workpieces of the job.

With the **OK**, **S**, and **R** placeholders, you can specify whether the quantity of reported workpieces has been machined correctly or not.

With **A** and **I** you define how StateMonitor interprets the response. If you transfer absolute values, StateMonitor overwrites the previously valid values. If you transfer incremental values, StateMonitor increments the quantity.

D38* /"JOB:1234_STEP:1_OK_A:23"*	Actual quantity (OK) absolute
D38* /"JOB:1234_STEP:1_OK_I:1"*	Actual quantity (OK) incremental
D38* /"JOB:1234_STEP:1_S_A:12"*	Scrap (S) absolute
D38* /"JOB:1234_STEP:1_S_I:1"*	Scrap (S) incremental
D38* /"JOB:1234_STEP:1_R_A:15"*	Rework (R) absolute
D38* /"JOB:1234_STEP:1_R_I:1"*	Rework (R) incremental

9.10 String parameters

String processing functions

You can use the **QS** parameters to create variable character strings. You can output such character strings for example through the **D16** function to create variable logs.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 255 characters to a string parameter. You can also check and process the assigned or imported values using the functions described below. As in Q parameter programming, you can use a total of 2000 QS parameters.

Further information: "Principle and overview of functions", Page 278

The **STRING FORMULA** and **FORMULA** Q parameter functions contain various functions for processing the string parameters.

Soft key	Functions of the STRING FORMULA	Page
DECLARE STRING	Assigning string parameters	324
CFGREAD	Read out the machine parameter values	333
STRING FORMULA	Chain-linking string parameters	325
TOCHAR	Converting a numerical value to a string parameter	326
SUBSTR	Copy a substring from a string parameter	327
SYSSTR	Read system data	328

Soft key	Formula string functions	Page
TONUMB	Converting a string parameter to a numerical value	329
INSTR	Checking a string parameter	330
STRLEN	Finding the length of a string parameter	331
STRCOMP	Compare alphabetic priority	332

If you use the **STRING FORMULA** function, the result is always an alphanumeric value. If you use the **FORMULA** function, the result is always a numeric value.

Assigning string parameters

Before using string variables, you must first assign the variables. Use the **DECLARE STRING** command to do so.

- Press the **SPEC FCT** key

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **STRING FUNCTIONS** soft key

- Press the **DECLARE STRING** soft key

Example

N110 DECLARE STRING QS10 = "workpiece" *	; Assign alphanumeric value to **QS10**

Chain-linking string parameters

With the concatenation operator (string parameter **| |** string parameter) you can make a chain of two or more string parameters.

- Press the **SPEC FCT** key

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **STRING FUNCTIONS** soft key

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the concatenated string. Confirm with the **ENT** key.
- Enter the number of the string parameter in which the **first** substring is saved. Confirm with the **ENT** key
- The control shows the concatenation symbol **| |**
- Press the **ENT** key
- Enter the number of the string parameter in which the **second** substring is saved. Confirm with the **ENT** key
- Repeat the process until you have selected all the required substrings. Conclude with the **END** key

Example: QS10 is to include the complete text of QS12 and QS13

N110 QS10 = QS12 || QS13 *	; Concatenate contents of **QS12** and **QS13** and assign them to the QS parameter **QS10**

Parameter contents:

- **QS12**: **Status:**
- **QS13**: **Scrap**
- **QS10**: **Status: Scrap**

Converting a numerical value to a string parameter

With the **TOCHAR** function, the control converts a numerical value into a string parameter. This enables you to chain numerical values with string variables.

- Show the soft-key row with special functions

- Open the function menu

- Press the String functions soft key

- Press the **STRING FORMULA** soft key

- Select the function for converting a numerical value to a string parameter
- Enter the number or the desired Q parameter to be converted by the control, and confirm with the **ENT** key
- If desired, enter the number of digits after the decimal point that the control should convert, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Convert parameter Q50 to string parameter QS11, use 3 decimal places

N110 QS11 = TOCHAR (DAT +Q50 DECIMALS3)*	; Convert a numerical value from **Q50** to an alphanumeric value and assign it to the QS parameter **QS11**

Copying a substring from a string parameter

The **SUBSTR** function copies a definable range from a string parameter.

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Open the function menu

STRING FUNCTIONS

- Press the String functions soft key

STRING FORMULA

- Press the **STRING FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the character string. Confirm with the **ENT** key.

SUBSTR

- Select the function for copying a substring
- Enter the number of the QS parameter from which the substring is to be copied. Confirm with the **ENT** key
- Enter the number of the place starting from which to copy the substring, and confirm with the **ENT** key
- Enter the number of characters to be copied, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

The first character of a text string starts internally at the 0-position.

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)

N110 QS13 = SUBSTR (SRC_QS10 BEG2 LEN4)*	; Assign substring from **QS10** to the QS parameter **QS13**

Reading system data

With the **SYSSTR** NC function, you can read system data and save the contents in QS parameters. Select the system datum by means of a group number **(ID)** and a number **(NR)**.

Optionally, you can enter **IDX** and **DAT**.

Group name, ID no.	Number	Meaning
Program information, 10010	1	Path of the current main program or pallet program
	2	Path of the currently executed NC program
	3	Path of the NC program selected with Cycle **G39 PGM CALL**
	10	Path of the NC program selected with **%:PGM**
Channel data, 10025	1	Name of the current channel (e.g., **CH_NC**)
Values programmed in the tool call, 10060	1	Current tool name The NC function saves the tool name only if the tool has been called using its tool name.
Current system time, 10321	1 to 16, 20	■ 1: D.MM.YYYY h:mm:ss ■ 2: D.MM.YYYY h:mm ■ 3: D.MM.YY hh:mm ■ 4: YYYY-MM-DD hh:mm:ss ■ 5: YYYY-MM-DD hh:mm ■ 6: YYYY-MM-DD h:mm ■ 7: YY-MM-DD h:mm ■ 8: DD.MM.YYYY ■ 9: D.MM.YYYY ■ 10: D.MM.YY ■ 11: YYYY-MM-DD ■ 12: YY-MM-DD ■ 13: hh:mm:ss ■ 14: h:mm:ss ■ 15: h:mm ■ 16: DD.MM.YYYY hh:mm ■ 20: XX "XX" stands for the two-digit number of the current calendar week that—in accordance with ISO 8601 — is characterized by the following: ■ It comprises seven days ■ It begins with Monday ■ It is numbered sequentially ■ The first calendar week (week 01) is the week with the first Thursday of the Gregorian year.
Touch-probe data, 10350	50	Type of the active TS workpiece touch probe
	70	Type of the active TT tool touch probe
	73	Name of the active TT workpiece touch probe from the **activeTT** machine parameter

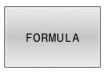
Group name, ID no.	Number	Meaning
	2	Path of the currently selected pallet table
NC software version, 10630	10	Number of the NC software version
Tool data, 10950	1	Current tool name
	2	Content of the **DOC** column of the current tool
	4	Tool-carrier kinematics of the current tool

Converting a string parameter to a numerical value

The **TONUMB** function converts a string parameter to a numerical value. The value to be converted should be only numerical.

> The QS parameter to be converted must contain only one numerical value. Otherwise, the control will output an error message.

- Select Q parameter function



- Press the **FORMULA** soft key
- Enter the number of the string parameter in which the control is to save the numerical value. Confirm with the **ENT** key.

- Shift the soft-key row

- Select the function for converting a string parameter to a numerical value
- Enter the number of the QS parameter to be converted by the control, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

Example: Convert string parameter QS11 to a numerical parameter Q82

N110 Q82 = TONUMB (SRC_QS11)*	; Convert alphanumeric value from **QS11** to a numerical value and assign it to **Q82**

Testing a string parameter

The **INSTR** function checks whether (and where) a string parameter is contained in another string parameter.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter for the result and confirm with the **ENT** key

> The control saves the place at which the text to be searched for begins. It is saved in the parameter.

◁

- Shift the soft-key row

INSTR

- Select the function for checking a string parameter
- Enter the number of the QS parameter in which the text to be searched for is saved. Confirm with the **ENT** key
- Enter the number of the QS parameter to be searched for by the control, and confirm with the **ENT** key
- Enter the number of the place at which the control is to start search the substring, and confirm with the **ENT** key.
- Close the parenthetical expression with the **ENT** key and confirm your entry with the **END** key

The first character of a text string starts internally at the 0-position.

If the control cannot find the required substring, it will save the total length of the string to be searched (counting starts at 1) in the result parameter.

If the substring to be searched for appears multiple times, then the control returns the first place at which it finds the substring.

Example: Search through QS10 for the text saved in parameter QS13. Begin the search at the third place.

N370 Q50 = INSTR (SRC_QS10 SEA_QS13 BEG2)*	; Search **QS10** for substring from **QS13**

Determining the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the determined string length, and confirm with the **ENT** key

◁

- Shift the soft-key row

STRLEN

- Select the function for finding the text length of a string parameter
- Enter the number of the QS parameter whose length is to be determined, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your input with the **END** key

Example: Find the length of QS15

N110 Q52 = STRLEN (SRC_QS15)*	; Determine the number of characters in **QS15** and assign it to **Q52**

If the selected QS parameter has not been defined, the control returns the value **-1**.

Comparing the lexical order of two alphanumerical strings

With the **STRCOMP** NC function, you can compare the lexical order of the content of two QS parameters.

Q

- Select Q parameter function

FORMULA

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the result of comparison, and confirm with the **ENT** key.

◁

- Shift the soft-key row

STRCOMP

- Select the function for comparing string parameters
- Enter the number of the first QS parameter that the control is to compare, and confirm with the **ENT** key
- Enter the number of the second QS parameter that the control is to compare, and confirm with the **ENT** key
- Close the parenthetical expression with the **ENT** key and confirm your input with the **END** key

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).

Example: Compare the lexical order of QS12 and QS14

```
N110 Q52 = STRCOMP
     ( SRC_QS12 SEA_QS14 )*
```

; Compare the lexical order of the values of **QS12** and **QS14**

Reading out machine parameters

With the **CFGREAD** NC function, you can read out machine parameter contents of the control as numerical or alphanumeric values. The read-out numerical values are always given in metric form.

To read a machine parameter, you need to determine the following contents in the configuration editor of the control:

Icon	Type	Meaning	Example
	Key	Group name of the machine parameter The group name can be specified optionally	CH_NC
	Entity	Parameter object The name always begins with **Cfg**	**CfgGeoCycle**
	Attribute	Name of the machine parameter	**displaySpindleErr**
	Index	List index of the machine parameter The list index can be specified optionally	[0]

> You can change the display of the existing parameters in the configuration editor for the machine parameter. By default, the parameters are displayed with short, explanatory texts.
>
> **Further information:** User's Manual for **Setup, Testing and Running NC Programs**

Each time you want to read out a machine parameter with the **CFGREAD** NC function, you must first define a QS parameter with attribute, entity and key.

The control queries the following parameters in the **CFGREAD** NC function:

- **KEY_QS**: Group name (key) of the machine parameter
- **TAG_QS**: Object name (entity) of the machine parameter
- **ATR_QS**: Name (attribute) of the machine parameter
- **IDX**: Index of the machine parameter

Reading a numerical value of a machine parameter

Store the value of a machine parameter as a numerical value in a Q parameter:

- Select Q parameter function

- Press the **FORMULA** soft key
- Enter the number of the Q parameter in which the control is to save the machine parameter
- Press the **ENT** key
- Select the **CFGREAD** function
- Enter the numbers of the string parameters for key, entity, and attribute
- Press the **ENT** key
- Enter the number for the index, or skip the dialog with **NO ENT**, whichever applies
- Close the parenthesized expression with the **ENT** key
- Press the **END** key to conclude entry

Example: Read overlap factor as Q parameter

Parameter settings in the configuration editor

```
ChannelSettings
CH_NC
    CfgGeoCycle
        pocketOverlap
```

Example

N110 QS11 = "CH_NC"	; Assign the key to the QS parameter **QS11**
N120 QS12 = "CfgGeoCycle"	; Assign the entity to the QS parameter **QS12**
N130 QS13 = "pocketOverlap"	; Assign the attribute to the QS parameter **QS13**
N140 Q50 = CFGREAD(KEY_QS11 TAG_QS12 ATR_QS13)	Read out the contents of the machine parameter

9.11 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.

NOTICE

Danger of collision!

HEIDENHAIN cycles, manufacturer cycles and third-party functions use Q parameters. You can also program Q parameters within NC programs. If, when using Q parameters, the recommended Q parameter ranges are not used exclusively, then this can lead to overlapping (reciprocal effects) and thus cause undesired behavior. Danger of collision during machining!

- Only use Q parameter ranges recommended by HEIDENHAIN.
- Comply with the documentation from HEIDENHAIN, the machine manufacturer, and suppliers.
- Check the machining sequence using a graphic simulation

Preassigned variables, such as Q and QS parameters in the range of 100 to 199, must not be used as calculated parameters in NC programs.

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.

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

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

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**.

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 **%**), 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.

Measurement result from programmable touch-probe cycles: Q115 to Q119

The control assigns the measurement result of a programmable touch-probe cycle to the following Q parameters.

For these Q parameters, the control does not take the radius and length of the stylus into account.

The help graphics of the touch-probe cycles show whether the control saves a measurement result in a variable or not.

The control assigns the coordinate axis values after probing to the Q parameters **Q115** to **Q119**:

Q parameters	Axis coordinates
Q115	**TOUCH POINT IN X**
Q116	**TOUCH POINT IN Y**
Q117	**TOUCH POINT IN Z**
Q118	**TOUCH POINT 4TH AXIS** (e.g., A axis) The machine manufacturer defines the 4th axis
Q119	**TOUCH POINT 5TH AXIS** (e.g., B axis) The machine manufacturer defines the 5th axis

Q parameters Q115 and Q116 for automatic tool measurement

The control assigns the deviation of the actual value from the nominal value in automatic tool measurements (e.g., with a TT 160) to the Q parameters **Q115** and **Q116**:

Q parameters	Deviation of actual from nominal value
Q115	Tool length
Q116	Tool radius

After probing, the Q parameters **Q115** and **Q116** might contain other values.

Calculated coordinates of the rotary axes: Q120 to Q122

The control assigns the calculated coordinates of the rotary axes to the Q parameters **Q120** to **Q122**:

Q parameters	Rotary axis coordinates
Q120	**AXIS ANGLE IN THE A AXIS**
Q121	**AXIS ANGLE IN THE B AXIS**
Q122	**AXIS ANGLE IN THE C AXIS**

Measurement results from touch-probe cycles

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

The control assigns the measured actual values to the Q parameters **Q141** to **Q149**:

Q parameters	Measured actual values
Q141	MEASURED ERROR A AXIS
Q142	MEASURED ERROR B AXIS
Q143	MEASURED ERROR C AXIS
Q144	ERROR OF OPTIM. A AXIS
Q145	ERROR OF OPTIM. B AXIS
Q146	ERROR OF OPTIM. C AXIS
Q147	OFFSET IN A AXIS
Q148	OFFSET IN B AXIS
Q149	OFFSET IN C AXIS

The control assigns the measured actual values to the Q parameters **Q150** to **Q160**:

Q parameters	Measured actual values
Q150	MEASURED ANGLE
Q151	ACTL. VALUE, REF AXIS
Q152	ACTL.VALUE, MINOR AXIS
Q153	ACTUAL VALUE, DIAMETER
Q154	ACT.VAL. PCKT REF AX.
Q155	ACT.VAL. PKT MINOR AX.
Q156	ACTUAL VALUE OF LENGTH
Q157	ACTL.VAL., CENTERLINE
Q158	PROJECTD. ANGLE A AXIS
Q159	PROJECTD. ANGLE B AXIS
Q160	COORD., MEASURING AXIS Coordinate of the axis selected in the cycle

The control assigns the calculated deviation values to the Q parameters **Q161** to **Q167**:

Q parameters	Calculated deviation
Q161	**ERROR, CENTR, REF AX.** Deviation of center in main axis
Q162	**ERROR, CENTR, MINOR AX** Deviation of center in the secondary axis
Q163	**ERROR OF DIAMETER**
Q164	**ERROR, PCKT., REF AX.** Deviation of pocket length in the main axis
Q165	**ERROR, CENTR, MINOR AX** Deviation of pocket width in the secondary axis
Q166	**ERROR OF LENGTH** Deviation of the measured length
Q167	**ERROR OF CENTERLINE** Deviation of the centerline position

The control assigns the determined spatial angle values to the Q parameters **Q170** to **Q172**:

Q parameters	Determined spatial angles
Q170	**SPATIAL ANGLE A**
Q171	**SPATIAL ANGLE B**
Q172	**SPATIAL ANGLE C**

The control assigns the determined workpiece status to the Q parameters **Q180** to **Q182**:

Q parameters	Workpiece status
Q180	**WORKPIECE IS GOOD**
Q181	**WORKPIECE NEEDS REWORK**
Q182	**WORKPIECE IS SCRAP**

The control reserves the Q parameters **Q190** to **Q192** for the results of tool measurements with a laser measuring system.

The control reserves the Q parameters **Q195** to **Q198** for internal use:

Q parameters	Reserved for internal use
Q195	**MARKER FOR CYCLES**
Q196	**MARKER FOR CYCLES**
Q197	**MARKER FOR CYCLES** Cycles with position pattern
Q198	**NO., LAST TCH-PRB CYC** Number of the last active touch-probe cycle

The value of the Q parameter **Q199** depends on the status of tool measurement with a tool touch probe:

Q parameters	Status of tool measurement with a tool touch probe
Q199 = 0.0	: Tool is within tolerance.
Q199 = 1.0	Tool is worn (**LTOL**/**RTOL** is exceeded)
Q199 = 2.0	Tool is broken (**LBREAK**/**RBREAK** is exceeded)

Measurement results from 14xx touch-probe cycles

The control assigns the measured actual values resulting from the **14xx** touch-probe cycles to the Q parameters **Q950** to **Q967**:

Q parameters	Measured actual values
Q950	**P1 measured main axis**
Q951	**P1 measured minor axis**
Q952	**P1 measured tool axis**
Q953	**P2 measured main axis**
Q954	**P2 measured minor axis**
Q955	**P2 measured tool axis**
Q956	**P3 measured main axis**
Q957	**P3 measured minor axis**
Q958	**P3 measured tool axis**
Q961	**Measured SPA** Spatial angle **SPA** in the working plane coordinate system **WPL-CS**
Q962	**Measured SPB** Spatial angle **SPB** in the **WPL-CS**
Q963	**Measured SPC** Spatial angle **SPC** in the **WPL-CS**
Q964	**Meas. basic rotation** Rotational angle in the input coordinate system **I-CS**
Q965	**Meas. table rotation**

Q parameters	Measured actual values
Q966	**Measured diameter 1**
Q967	**Measured diameter 2**

The control assigns the deviations calculated in connection with the **14xx** touch-probe cycles to the Q parameters **Q980** to **Q997**:

Q parameters	Measured deviations
Q980	**P1 error main axis**
Q981	**P1 error minor axis**
Q982	**P1 error tool axis**
Q983	**P2 error main axis**
Q984	**P2 error minor axis**
Q985	**P2 error tool axis**
Q986	**P3 error main axis**
Q987	**P3 error minor axis**
Q988	**P3 error tool axis**
Q994	**Error: basic rotation** Angle in the input coordinate system **I-CS**
Q995	**Meas. table rotation**
Q996	**Error: diameter 1**
Q997	**Error: diameter 2**

The value of the Q parameter **Q183** depends on the workpiece status as measured by the 14xx touch-probe cycles:

Q parameters	Workpiece status
Q183 = –1	Not defined
Q183 = 0	Pass
Q183 = 1	Rework
Q183 = 2	Scrap

9.12 Programming examples

Example: Rounding a value

The **INT** function truncates the decimal places.

In order for the control to round correctly, rather than simply truncating the decimal places, add the value 0.5 to a positive number. For a negative number you must subtract 0.5.

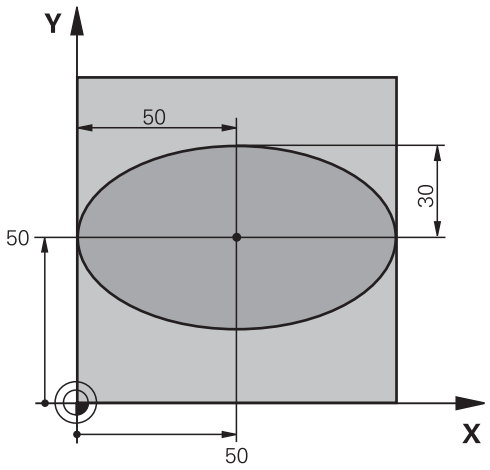
The control uses the **SGN** function to detect whether a number is positive or negative.

%ROUND G71 *	
N10 D00 Q1 P01 +34.789*	First number to be rounded
N20 D00 Q2 P01 +34.345*	Second number to be rounded
N30 D00 Q3 P01 -34.345*	Third number to be rounded
N40 ;	
N50 Q11 = INT (Q1 + 0.5 * SGN Q1)	Add the value 0.5 to Q1, then truncate the decimal places
N60 Q12 = INT (Q2 + 0.5 * SGN Q2)	Add the value 0.5 to Q2, then truncate the decimal places
N70 Q13 = INT (Q3 + 0.5 * SGN Q3)	Subtract the value 0.5 from Q3, then truncate the decimal places
N99999999 %ROUND G71 *	

Example: Ellipse

Program run

- The contour of the ellipse is approximated by many short line segments (defined in **Q7**). The more calculation steps you define for the lines, the smoother the curve becomes.
- The milling direction is determined with the starting angle and end angle in the plane:
  Machining direction is clockwise:
  Starting angle > end angle
  Machining direction is counterclockwise:
  Starting angle < end angle
- The tool radius is not taken into account



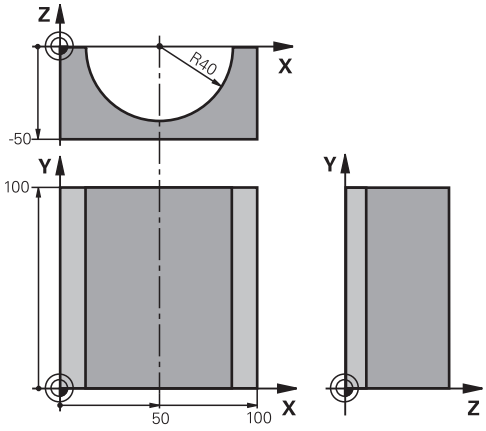
%ELLIPSE G71 *	
N10 D00 Q1 P01 +50*	Center in X axis
N20 D00 Q2 P01 +50*	Center in Y axis
N30 D00 Q3 P01 +50*	Semiaxis in X
N40 D00 Q4 P01 +30*	Semiaxis in Y
N50 D00 Q5 P01 +0*	Starting angle in the plane
N60 D00 Q6 P01 +360*	End angle in the plane
N70 D00 Q7 P01 +40*	Number of calculation steps
N80 D00 Q8 P01 +30*	Rotational position of the ellipse
N90 D00 Q9 P01 +5*	Milling depth
N100 D00 Q10 P01 +100*	Feed rate for plunging
N110 D00 Q11 P01 +350*	Feed rate for milling
N120 D00 Q12 P01 +2*	Set-up clearance for pre-positioning
N130 G30 G17 X+0 Y+0 Z-20*	Workpiece blank definition
N140 G31 G90 X+100 Y+100 Z+0*	
N150 T1 G17 S4000*	Tool call
N160 G00 G40 G90 Z+250*	Retract the tool
N170 L10.0*	Call machining operation
N180 G00 Z+250 M2*	Retract the tool, end program
N190 G98 L10*	Subprogram 10: Machining operation
N200 G54 X+Q1 Y+Q2*	Shift datum to center of ellipse
N210 G73 G90 H+Q8*	Account for rotational position in the plane
N220 Q35 = (Q6 - Q5) / Q7	Calculate angle increment
N230 D00 Q36 P01 +Q5*	Copy starting angle
N240 D00 Q37 P01 +0*	Set counter
N250 Q21 = Q3 * COS Q36	Calculate X coordinate for starting point
N260 Q22 = Q4 * SIN Q36	Calculate Y coordinate for starting point
N270 Q00 G40 X+Q21 Y+Q22 M3*	Move to starting point in the plane

N280 Z+Q12*	Pre-position in spindle axis to set-up clearance
N290 G01 Z-Q9 FQ10*	Move to working depth
N300 G98 L1*	
N310 Q36 = Q36 + Q35	Update the angle
N320 Q37 = Q37 + 1	Update the counter
N330 Q21 = Q3 * COS Q36	Calculate the current X coordinate
N340 Q22 = Q4 * SIN Q36	Calculate the current Y coordinate
N350 G01 X+Q21 Y+Q22 FQ11*	Move to next point
N360 D12 P01 +Q37 P02 +Q7 P03 1*	Unfinished? If not finished, return to LBL 1
N370 G73 G90 H+0*	Reset the rotation
N380 G54 X+0 Y+0*	Reset the datum shift
N390 G00 G40 Z+Q12*	Move to set-up clearance
N400 G98 L0*	End of subprogram
N99999999 %ELLIPSE G71 *	

Example: Concave cylinder machined with ball-nose cutter

Program run

- This NC program works only with a ball-nose cutter. The tool length is measured from the sphere center
- The contour of the cylinder is approximated by many short line segments (defined in **Q13**). The more line segments you define, the smoother the contour becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The milling direction is determined with the starting angle and end angle in space:
  Machining direction clockwise:
  Starting angle > end angle
  Machining direction counterclockwise:
  Starting angle < end angle
- The tool radius is compensated automatically



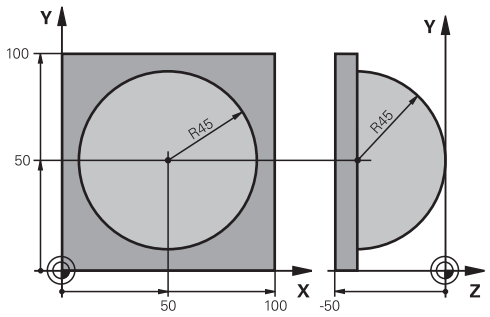
%CYLIN G71 *	
N10 D00 Q1 P01 +50*	Center in X axis
N20 D00 Q2 P01 +0*	Center in Y axis
N30 D00 Q3 P01 +0*	Center in Z axis
N40 D00 Q4 P01 +90*	Starting angle in space (Z/X plane)
N50 D00 Q5 P01 +270*	End angle in space (Z/X plane)
N60 D00 Q6 P01 +40*	Cylinder radius
N70 D00 Q7 P01 +100*	Length of the cylinder
N80 D00 Q8 P01 +0*	Rotational position in the X/Y plane
N90 D00 Q10 P01 +5*	Allowance for cylinder radius
N100 D00 Q11 P01 +250*	Feed rate for plunging
N110 D00 Q12 P01 +400*	Feed rate for milling
N120 D00 Q13 P01 +90*	Number of cuts
N130 G30 G17 X+0 Y+0 Z-50*	Workpiece blank definition
N140 G31 G90 X+100 Y+100 Z+0*	
N150 T1 G17 S4000*	Tool call
N160 G00 G40 G90 Z+250*	Retract the tool
N170 L10.0*	Call machining operation
N180 D00 Q10 P01 +0*	Reset allowance
N190 L10.0*	Call machining operation
N200 G00 G40 Z+250 M2*	Retract the tool, end program
N210 G98 L10*	Subprogram 10: Machining operation
N220 Q16 = Q6 - Q10 - Q108	Account for allowance and tool, based on the cylinder radius
N230 D00 Q20 P01 +1*	Set counter
N240 D00 q24 p01 +Q4*	Copy starting angle in space (Z/X plane)
N250 Q25 = (Q5 - Q4) / Q13	Calculate angle increment
N260 G54 X+Q1 Y+Q2 Z+Q3*	Shift datum to center of cylinder (X axis)

N270 G73 G90 H+Q8*	Account for rotational position in the plane
N280 G00 G40 X+0 Y+0*	Pre-position in the plane to the cylinder center
N290 G01 Z+5 F1000 M3*	Pre-position in the spindle axis
N300 G98 L1*	
N310 I+0 K+0*	Set pole in the Z/X plane
N320 G11 R+Q16 H+Q24 FQ11*	Move to starting position on cylinder, plunge-cutting obliquely into the material
N330 G01 G40 Y+Q7 FQ12*	Longitudinal cut in Y+ direction
N340 D01 Q20 P01 +Q20 P02 +1*	Update the counter
N350 D01 Q24 P01 +Q24 P02 +Q25*	Update solid angle
N360 D11 P01 +Q20 P02 +Q13 P03 99*	Finished? If finished, jump to end
N370 G11 R+Q16 H+Q24 FQ11*	Move on an approximated arc for the next longitudinal cut
N380 G01 G40 Y+0 FQ12*	Longitudinal cut in Y– direction
N390 D01 Q20 P01 +Q20 P02 +1*	Update the counter
N400 D01 Q24 P01 +Q24 P02 +Q25*	Update solid angle
N410 D12 P01 +Q20 P02 +Q13 P03 1*	Unfinished? If not finished, return to LBL 1
N420 G98 L99*	
N430 G73 G90 H+0*	Reset the rotation
N440 G54 X+0 Y+0 Z+0*	Reset the datum shift
N450 G98 L0*	End of subprogram
N99999999 %CYLIN G71 *	

Example: Convex sphere machined with end mill

Program run

- NC program requires an end mill.
- The contour of the sphere is approximated by many short line segments (in the Z/X plane, defined in **Q14**). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in **Q18**).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically



%SPHERE G71 *	
N10 D00 Q1 P01 +50*	Center in X axis
N20 D00 Q2 P01 +50*	Center in Y axis
N30 D00 Q4 P01 +90*	Starting angle in space (Z/X plane)
N40 D00 Q5 P01 +0*	End angle in space (Z/X plane)
N50 D00 Q14 P01 +5*	Angle increment in space
N60 D00 Q6 P01 +45*	Sphere radius
N70 D00 Q8 P01 +0*	Starting angle of rotational position in the X/Y plane
N80 D00 Q9 p01 +360*	End angle of rotational position in the X/Y plane
N90 D00 Q18 P01 +10*	Angle increment in the X/Y plane for roughing
N100 D00 Q10 P01 +5*	Allowance in sphere radius for roughing
N110 D00 Q11 P01 +2*	Set-up clearance for pre-positioning in the spindle axis
N120 D00 Q12 P01 +350*	Feed rate for milling
N130 G30 G17 X+0 Y+0 Z-50*	Workpiece blank definition
N140 G31 G90 X+100 Y+100 Z+0*	
N150 T1 G17 S4000*	Tool call
N160 G00 G40 G90 Z+250*	Retract the tool
N170 L10.0*	Call machining operation
N180 D00 Q10 P01 +0*	Reset allowance
N190 D00 Q18 P01 +5*	Angle increment in the X/Y plane for finishing
N200 L10.0*	Call machining operation
N210 G00 G40 Z+250 M2*	Retract the tool, end program
N220 G98 L10*	Subprogram 10: Machining operation
N230 D01 Q23 P01 +Q11 P02 +Q6*	Calculate Z coordinate for pre-positioning
N240 D00 Q24 P01 +Q4*	Copy starting angle in space (Z/X plane)
N250 D01 Q26 P01 +Q6 P02 +Q108*	Compensate sphere radius for pre-positioning
N260 D00 Q28 P01 +Q8*	Copy rotational position in the plane
N270 D01 Q16 P01 +Q6 P02 -Q10*	Account for allowance in the sphere radius
N280 G54 X+Q1 Y+Q2 Z-Q16*	Shift datum to center of sphere
N290 G73 G90 H+Q8*	Account for starting angle of rotational position in the plane
N300 G98 L1*	Pre-position in the spindle axis

N310 I+0 J+0*	Set pole in the X/Y plane for pre-positioning
N320 G11 G40 R+Q26 H+Q8 FQ12*	Pre-position in the plane
N330 I+Q108 K+0*	Set pole in the Z/X plane, offset by the tool radius
N340 G01 Y+0 Z+0 FQ12*	Move to working depth
N350 G98 L2*	
N360 G11 G40 R+Q6 H+Q24 FQ12*	Move upward on an approximated arc
N370 D02 Q24 P01 +Q24 P02 +Q14*	Update solid angle
N380 D11 P01 +Q24 P02 +Q5 P03 2*	Inquire whether an arc is finished. If not finished, return to LBL 2
N390 G11 R+Q6 H+Q5 FQ12*	Move to the end angle in space
N400 G01 G40 Z+Q23 F1000*	Retract in the spindle axis
N410 G00 G40 X+Q26*	Pre-position for next arc
N420 D01 Q28 P01 +Q28 P02 +Q18*	Update rotational position in the plane
N430 D00 Q24 P01 +Q4*	Reset solid angle
N440 G73 G90 H+Q28*	Activate new rotational position
N450 D12 P01 +Q28 P02 +Q9 P03 1*	Unfinished? If not finished, return to LBL 1
N460 D09 P01 +Q28 P02 +Q9 P03 1*	
N470 G73 G90 H+0*	Reset the rotation
N480 G54 X+0 Y+0 Z+0*	Reset the datum shift
N490 G98 L0*	End of subprogram
N99999999 %SPHERE G71 *	

10

Special Functions

10.1 Overview of special functions

The control provides the following powerful special functions for a large number of applications:

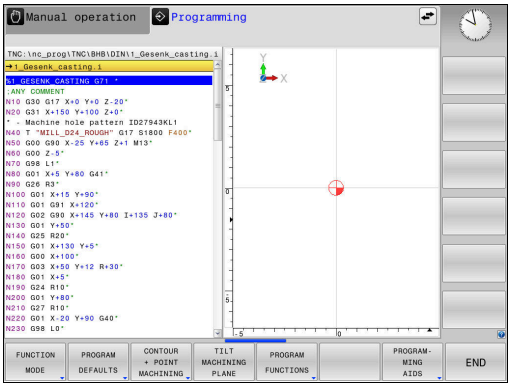
Function	Description
Working with text files	Page 382
Working with freely definable tables	Page 386

Press the **SPEC FCT** key and the corresponding soft keys to access further special functions of the control. The following tables give you an overview of which functions are available.

Main menu for SPEC FCT special functions

SPEC FCT

- Press the **SPEC FCT** key to select the special functions



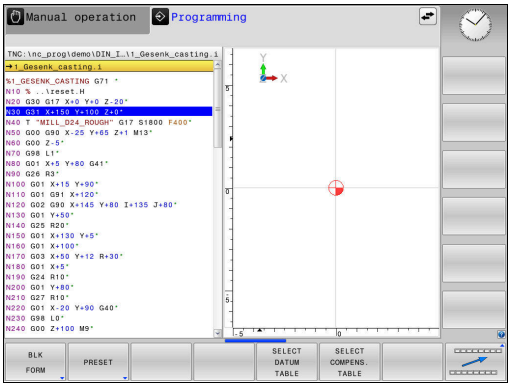
Soft key	Function	Description
FUNCTION MODE	Select machining mode or kinematics	Page 355
PROGRAM DEFAULTS	Define program defaults	Page 353
CONTOUR + POINT MACHINING	Functions for contour and point machining	Page 353
TILT MACHINING PLANE	Define the **PLANE** function	Page 408
PROGRAM FUNCTIONS	Define different ISO functions	Page 354
PROGRAM-MING AIDS	Programming aids	Page 201

After pressing the **SPEC FCT** key, you can open the **smartSelect** selection window with the **GOTO** key. The control displays a structure overview with all available functions. You can rapidly navigate with the cursor or mouse and select functions in the tree diagram. The control displays online help for the selected function in the window on the right.

Program defaults menu

PROGRAM DEFAULTS

- Press the Program Defaults soft key

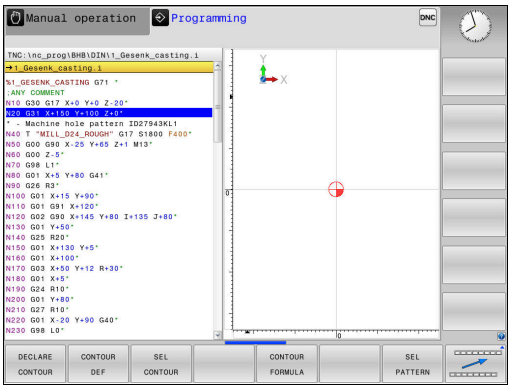


Soft key	Function	Description
BLK FORM	Define workpiece blank	Page 101
PRESET	Modifying the preset	Page 363
SELECT DATUM TABLE	Select datum table	Page 371
SELECT COMPENS. TABLE	Select compensation table	Page 374

Functions for contour and point machining menu

CONTOUR + POINT MACHINING

- Press the soft key for functions for contour and point machining



Soft key	Function
DECLARE CONTOUR	Assign contour description
CONTOUR DEF	Define a simple contour formula
SEL CONTOUR	Select a contour definition
CONTOUR FORMULA	Define a complex contour formula
SEL PATTERN	Select the point file with machining positions

Further information: User's Manual for **Programming of Machining Cycles**

Menu for defining different ISO functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

Soft key	Function	Description
TRANSFORM / CORRDATA	Activate compensation values	Page 374
FUNCTION COUNT	Define the counter	Page 380
STRING FUNCTIONS	Define string functions	Page 323
FUNCTION SPINDLE	Define pulsing spindle speed	Page 394
FUNCTION FEED	Define recurring dwell time	Page 397
FUNCTION DWELL	Define dwell time in seconds or revolutions	Page 399
FUNCTION LIFTOFF	Lift off tool at NC stop	Page 400
DIN/ISO	Define ISO functions	Page 362
INSERT COMMENT	Add comments	Page 205
TABDATA	Write and read table values	Page 376
POLARKIN	Define polar kinematics	Page 356

10.2 Function mode

Program function mode

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

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

Procedure

To switch the kinematic model:

SPEC FCT

- Show the soft-key row for special functions

FUNCTION MODE

- Press the **FUNCTION MODE** soft key

MILL

- Press the **MILL** soft key

SELECT

- Press the **SELECT KINEMATICS** soft key
- Select the desired kinematic model

Function Mode Set

Refer to your machine manual.
This function must be enabled and adapted by the machine manufacturer.
Your machine manufacturer defines the available options in the machine parameter **CfgModeSelect** (no. 132200).

FUNCTION MODE SET allows you to activate settings defined by the machine manufacturer (e.g., changes to the range of traverse) from within the NC program

To select a setting:

SPEC FCT

- Show the soft-key row with special functions

FUNCTION MODE

- Press the **FUNCTION MODE** soft key

SET

- Press the **SET** soft key

SELECT

- Press the **SELECT** soft key, if required
- > The control opens a selection window.
- Select the desired setting

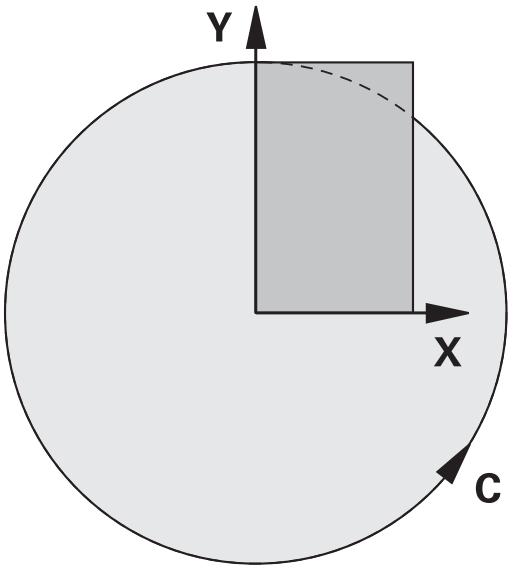
10.3 Machining with polar kinematics

Overview

In a polar kinematic model, the path contours of the working plane are performed by one linear axis and one rotary axis instead of by two linear principal axes. The working plane is defined by the linear principal axis and the rotary axis while the working space is defined by these two axes and the infeed axis.

On turning and grinding machines that have only two linear principal axes, polar kinematics enable milling operations to be performed on the front face.

On milling machines, various linear principal axes can be replaced with suitable rotary axes. For example on large machines, polar kinematics enable you to machine much larger surfaces than with only the principal axes.



Refer to your machine manual.

Your machine must be configured by the machine tool builder so that you can use polar kinematics.

A polar kinematic model consists of two linear axes and one rotary axis. The programmable axes vary depending on the machine.

The polar rotary axis must be installed onto the table side so that it is opposite the selected linear axes and must be configured as a modulo axis. Thus, the linear axes must not be positioned between the rotary axis and the table. The maximum range of traverse of the rotary axis is limited by the software limit switches if necessary.

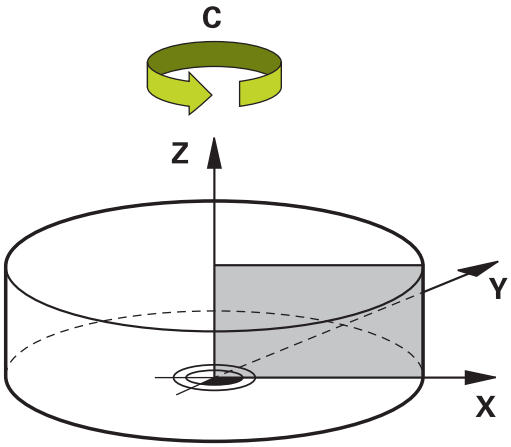
The principal axes X, Y, and Z as well as the possible parallel axes U, V, and W can be used as radial axes or infeed axes.

The control, combined with polar kinematics, provides the following functions:

Soft key	Function	Meaning	Page
POLARKIN AXES	**POLARKIN AXES**	Define and activate polar kinematics	357
POLARKIN OFF	**POLARKIN OFF**	Deactivate polar kinematics	360

Activating FUNCTION POLARKIN

Use the **POLARKIN AXES** function to activate the polar kinematics. The axis data define the radial axis, the infeed axis, and the polar axis. The **MODE** data influence the positioning behavior, whereas the **POLE** data define the machining at the pole. The pole is the center of rotation of the rotary axis in this case.



Notes on the axes to be selected:

- The first linear axis must be radial to the rotary axis.
- The second linear axis defines the infeed axis and must be parallel to the rotary axis.
- The rotary axis defines the polar axis and is defined last.
- Any available modulo axis that is installed at the table opposite to the selected linear axes can be used as the rotary axis.
- The two selected linear axes thus span a plane that also includes the rotary axis.

MODE options:

Syntax	Function
POS	Seen from the center of rotation, the control performs machining in the positive direction of the radial axis. The radial axis must be prepositioned correspondingly.
NEG	Seen from the center of rotation, the control performs machining in the negative direction of the radial axis. The radial axis must be prepositioned correspondingly.
KEEP	The control remains with the radial axis on that side of the center of rotation on which the axis was positioned when the function was activated. If the radial axis is positioned at the center of rotation upon switch-on, **POS** applies.
ANG	The control remains with the radial axis on that side of the center of rotation on which the axis was positioned when the function was activated. If you set **POLE** to **ALLOWED**, positioning through the pole is possible. The pole side is changed and a 180-degree rotation of the rotary axis is prevented.

POLE options:

Syntax	Function
ALLOWED	The control permits machining operations at the pole
SKIPPED	The control prevents machining operations at the pole

> The disabled area corresponds to a circular surface with a radius of 0.001 mm (1 µm) around the pole.

To program this behavior:

SPEC FCT

- Show the soft-key row for special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

POLARKIN

- Press the **POLARKIN** soft key

POLARKIN AXES

- Press the **POLARKIN AXES** soft key
- Define the axes of the polar kinematics
- Select the **MODE** option
- Select the **POLE** option

Example

```
N60 POLARKIN AXES X Z C MODE: KEEP POLE:ALLOWED*
```

If polar kinematics is active, the control displays an icon in the status display.

Icon	Mode
	Polar kinematics is active **i** The **POLARKIN** icon hides the active **PARAXCOMP DISPLAY** icon. The control additionally displays the selected **Principal axes** on the **POS** tab of the additional status display.
No icon	Standard kinematics is active

Notes

Programming notes:

- Before activating the polar kinematics, you must program the function **PARAXCOMP DISPLAY** with at least the main axes X, Y, and Z.

  > In ISO programs, you cannot directly enter the **PARAXCOMP** functions. You program the required functions by calling an external Klartext program.
  >
  > HEIDENHAIN recommends defining all of the available axes within the **PARAXCOMP DISPLAY** function.

- Position the linear axis that will not be included in the polar kinematics to the coordinate of the pole, before the **POLARKIN** function. Otherwise, a non-machinable area with a radius that corresponds to at least the value of the deselected linear axis would result.
- Avoid performing machining operations at the pole or near the pole, because feed-rate variations may occur in this area. For this reason, ideally use the following **POLE** option: **SKIPPED**.
- Polar kinematics cannot be combined with the following functions:
  - Traverses with **M91**
  - Tilting the working plane
- The machine manufacturer uses the optional machine parameter **presetToAlignAxis** (no. 300203) to define for each axis how the control is to interpret offset values. For **FUNCTION POLARKIN**, the machine parameter applies only to the rotary axis that rotates about the tool axis (in most cases **C_OFFS**).

  Further information: User's Manual for **Setup, Testing and Running NC Programs**

  - If the machine parameter axis has not been defined or has been set to **TRUE**, the offset can be used to compensate a misalignment of the workpiece in the plane. The offset affects the orientation of the workpiece coordinate system **W-CS**.

    Further information: "Workpiece coordinate system W-CS", Page 92
  - If the machine parameter axis has been defined with **FALSE**, the offset cannot be used to compensate a misalignment of the workpiece in the plane. The control will not take the offset into account when executing the commands.

Machining information:

The polar kinematics may require continuous motions to be divided into submotions (e.g., a linear motion that is divided into two submotions: a motion for approaching the pole and a motion for departing from the pole). As a result, the distance-to-go display may differ from that of the standard kinematics.

Deactivating FUNCTION POLARKIN

Use the **POLARKIN OFF** function to deactivate the polar kinematics.
Program this as follows:

- Show the soft key row with special functions

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **POLARKIN** soft key

- Press the **POLARKIN OFF** soft key

Example

```
N60 POLARKIN OFF*
```

When the polar kinematics is not active, the control does not display the corresponding icon or entries on the **POS** tab.

Note

The following scenarios lead to deactivation of the polar kinematics:

- Execution of the **POLARKIN OFF** function
- Selection of an NC program
- Reaching the end of the NC program
- Abortion of the NC program
- Selecting a kinematic model
- Restarting the control

Example: SL cycles in the polar kinematics

%POLARKIN_SL G71 *	
N10 G30 G17 X-100 Y-100 Z-30*	
N20 G31 X+100 Y+100 Z+0*	
N30 T2 G17 S2000 F750*	
N40 % PARAXCOMP-DISPLAY_X Y Z.H	; Activate **PARAXCOMP DISPLAY**
N50 G00 G90 X+0 Y+0.0011 Z+10 A+0 C+0 G40 M3*	; Pre-position outside the disabled pole area
N60 POLARKIN AXES Y Z C MODE:KEEP POLE:SKIPPED*	; Activate **POLARKIN**
N70 G54 X+50 Y+50 Z+0*	; Datum shift in polar kinematics
N80 G37 P01 2*	
N90 G120 CONTOUR DATA	
Q1=-10 ;MILLING DEPTH	
Q2=+1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=+2 ;SET-UP CLEARANCE	
Q7=+50 ;CLEARANCE HEIGHT	
Q8=+0 ;ROUNDING RADIUS	
Q9=+1 ;ROTATIONAL DIRECTION*	
N100 G122 ROUGH-OUT	
Q10=-5 ;PLUNGING DEPTH	
Q11=+150 ;FEED RATE FOR PLNGNG	
Q12=+500 ;FEED RATE F. ROUGHNG	
Q18=+0 ;COARSE ROUGHING TOOL	
Q19=+0 ;FEED RATE FOR RECIP.	
Q208=+99999 ;RETRACTION FEED RATE	
Q401=+100 ;FEED RATE FACTOR	
Q404=+0 ;FINE ROUGH STRATEGY*	
N110 M99	
N120 G54 X+0 Y+0 Z+0*	
N130 POLARKIN OFF*	; Deactivate **POLARKIN**
N140 % PARAXCOMP-DISPLAY_OFF_XYZ.H	; Deactivate **PARAXCOMP DISPLAY**
N150 G00 G90 X+0 Y+0 Z+10 A+0 C+0 G40*	
N160 M30*	
N170 G98 L2*	
N180 G01 G90 X-20 Y-20 G42*	
N190 G01 X+0 Y+20*	
N200 G01 X+20 Y-20*	
N210 G01 X-20 Y-20*	
N220 G98 L0*	
N99999999 %POLARKIN_SL G71 *	

10.4 Defining ISO functions

Overview

> If an alphanumeric keyboard is connected via a USB port, you can also enter the ISO functions directly through the alphanumeric keyboard.

The control provides soft keys with the following functions for creating ISO programs:

Soft key	Function
DIN/ISO	Select ISO functions
F	Feed rate
G	Tool movements, cycles and program functions
I	X coordinate of the circle center or pole
J	Y coordinate of the circle center or pole
L	Label call for subprogram and program section repeat
M	Miscellaneous function
N	Block number
T	Tool call
H	Polar coordinate angle
K	Z coordinate of the circle center or pole
R	Polar coordinate radius
S	Spindle speed

10.5 Modifying presets

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

Activating a preset

The **PRESET SELECT** function allows you to use a preset defined in the preset table and activate it as a new preset.

To activate the preset, use the row number or the content in the **DOC** column.

NOTICE

Danger of collision!

Depending on the machine parameter **CfgColumnDescription** (no. 105607), you can define the same content several times in the **DOC** column of the preset table. In this case, if you activate a preset using the **DOC** column, the control selects the preset with the lowest row number. If the control does not select the desired preset there is a risk of collision.

- ▶ Uniquely define the content of the **DOC** column
- ▶ Only activate the preset with the row number

If you program **PRESET SELECT** without optional parameters, then the behavior is identical to Cycle **G247 PRESETTING**.

Use the optional parameters to define the following:

- **KEEP TRANS**: Retain simple transformations
  - Cycle **G53/G54 DATUM SHIFT**
  - Cycle **G28 MIRRORING**
  - Cycle **G73 ROTATION**
  - Cycle **G72 SCALING FACTOR**
- **WP**: Any changes apply to the workpiece preset

Procedure

Proceed as follows for the definition:

- Press the **SPEC FCT** key

- Press the **PROGRAM DEFAULTS** soft key

- Press the **PRESET** soft key

- Press the **PRESET SELECT** soft key
- Define the desired preset number
- Alternatively, define the entry from the **DOC** column
- Retain the transformations where necessary
- If necessary, select the preset to which the change is to apply

Example

N30 PRESET SELECT #3 KEEP TRANS WP*	Select Preset 3 as the workpiece preset, and retain the transformations

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

Copying a preset

The function **PRESET COPY** allows you to copy a preset defined in the preset table and activate the preset copied.

To select the preset to be copied, use the row number or the entry in the **DOC** column.

Use the optional parameters to define the following:

- **SELECT TARGET**: Activate the copied preset
- **KEEP TRANS**: Retain simple transformations

NOTICE

Danger of collision!

Depending on the machine parameter **CfgColumnDescription** (no. 105607), you can define the same content several times in the **DOC** column of the preset table. In this case, if you activate a preset using the **DOC** column, the control selects the preset with the lowest row number. If the control does not select the desired preset there is a risk of collision.

- Uniquely define the content of the **DOC** column
- Only activate the preset with the row number

Procedure

Proceed as follows for the definition:

- Press the **SPEC FCT** key

- Press the **PROGRAM DEFAULTS** soft key

- Press the **PRESET** soft key

- Press the **PRESET COPY** soft key
- Define the preset number to be copied
- Alternatively, define the entry from the **DOC** column
- Define the new preset number
- Activate the copied preset, if necessary
- Retain the transformations where necessary

Example

N130 PRESET COPY #1 TO #3 SELECT TARGET KEEP TRANS*	Copy the preset 1 to line 3, activate the preset 3, and retain the transformations

Correcting a preset

The function **PRESET CORR** allows you to correct the active preset.

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 active coordinate system.

Procedure

Proceed as follows for the definition:

- Show the soft key row with special functions

- Press the **PROGRAM DEFAULTS** soft key

- Press the **PRESET** soft key

- Press the **PRESET CORR** soft key
- Define the desired compensation values

Example

N30 PRESET CORR X+10 SPC+45*	The active preset is corrected by a value of +10 mm in X, and by +45° in SPC

10.6 Datum table

Application

You can save the workpiece-related datums in a datum table. To use a datum table, you must activate it.

Description

Datums from a datum table always reference the current preset. The coordinate values from datum tables are only effective as absolute coordinate values.

Use datum tables for the following purposes:

- Frequent use of the same datum shift
- Frequently recurring machining sequences on the workpiece
- Frequently recurring machining sequences at various locations on the workpiece

The values of the columns **X**, **Y** and **Z** are applied as shifts in the workpiece coordinate system **W-CS**. The values of the columns **A**, **B**, **C**, **U**, **V** and **W** are applied as shifts in the machine coordinate system **M-CS**.

Further information: "Workpiece coordinate system W-CS", Page 92

Further information: "Machine coordinate system M-CS", Page 89

Further information: User's Manual for **Setup, Testing and Running NC Programs**

The datum table contains the following parameters:

Parameter	Meaning	Input
D	Sequential number of the datums	**0...99999999**
X	X coordinate of the datum	**-99999.99999...99999.99999**
Y	Y coordinate of the datum	**-99999.99999...99999.99999**
Z	Z coordinate of the datum	**-99999.99999...99999.99999**
A	Axis angle of the A axis for the datum	**-360.0000000...360.0000000**
B	Axis angle of the B axis for the datum	**-360.0000000...360.0000000**
C	Axis angle of the C axis for the datum	**-360.0000000...360.0000000**
U	Position of the U axis for the datum	**-99999.99999...99999.99999**
V	Position of the V axis for the datum	**-99999.99999...99999.99999**
W	Position of the W axis for the datum	**-99999.99999...99999.99999**
DOC	Comment column	Max. 16 characters

Creating a datum table

To create a new datum table:

- Switch to the **Programming** operating mode

- Press the **PGM MGT** key

- Press the **NEW FILE** soft key
- > The control opens the **New file** window where you can enter the file name.
- Enter the file name with the file type ***.d**

- Confirm with the **ENT** key
- > The control opens the **Select table format** window, if necessary.
- Select a table format, if necessary

- Press the **OK** soft key, if necessary
- Select unit of measure **MM** or **INCH**, if necessary
- > The control opens the datum table.

You can select the table format, if there is at least one prototype of the table type.

The control displays whether the prototype is defined to use mm or inches as unit of measure. If the control displays both units of measure, you can select one of them.

The machine manufacturer defines the prototypes.

The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**).

Opening and editing a datum table

After you have changed a value in a datum table, you must save the change with the **ENT** key. Otherwise, the change will not be taken into account when the NC program is executed.

To open and edit a datum table:

PGM MGT

- Press the **PGM MGT** key
- Select the desired datum table
- > The control opens the datum table.
- Select the row you wish to edit

ENT

- Save your input, e.g. by pressing the **ENT** key.

To delete the value from the input field, press the **CE** key.

The control displays the following functions in the soft-key row:

Soft key	Function
BEGIN	Select the table start
END	Select the table end
PAGE	Go to previous page
PAGE	Go to next page
FIND	Search The control opens a window where you can enter the text or value you are looking for.
RESET TABLE	Reset table
BEGIN LINE	Move the cursor to the beginning of the row
END LINE	Move the cursor to the end of the row
COPY FIELD	Copy the current value
PASTE FIELD	Paste the copied value
APPEND N LINES AT END	Insert the specified number of rows New rows can only be inserted at the end of the table.

Soft key	Function
INSERT LINE	Insert row New rows can only be inserted at the end of the table.
DELETE LINE	Delete row
SORT/ HIDE COLUMNS	Sort/hide columns The control opens the **Column sequence** window with the following options: ■ **Use standard format** ■ Display/hide columns ■ Arrange columns ■ Freeze columns (3 max.)
MORE FUNCTIONS	Additional functions, e.g. Delete
RESET COLUMN	Reset the column
EDIT CURRENT FIELD	Edit the current field
SORT	Sort the datum table A window opens where you can select the sorting order.

If you enter the code number 555343, the control will display the **EDIT FORMAT** soft key. With this soft key, you can change the table properties.

Activating the datum table in your NC program

To activate a workpiece datum table in your NC program:

PGM CALL

- Press the **PGM CALL** key

SELECT DATUM TABLE

- Press the **SELECT DATUM TABLE** soft key

SELECT FILE

- Press the **SELECT FILE** soft key
- A file selection window opens.
- Select the desired datum table

ENT

- Confirm with the **ENT** key

If you enter the datum table name manually, please note the following:

- If the datum table is located in the same directory as the NC program, enter the file name only.
- If the datum table is not located in the same directory as the NC program, enter the complete path.

Program **%:TAB:** before Cycle **G54**.

Activating the datum table manually

If you do not use **%:TAB:**, you must activate the desired datum table prior to the test run.

To activate a datum table for the test run:

- Select the **Test Run** operating mode

PGM MGT

- Press the **PGM MGT** key
- Select the desired datum table
- The control activates the datum table for the test run and marks the file with the **S** status.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

10.7 Compensation table

Application

With the compensation table, you can save compensations in the tool coordinate system (T-CS) or in the working plane coordinate system (WPL-CS).

The compensation table **.tco** is the alternative to compensating with **DL**, **DR** and **DR2** in the T block. As soon as you have activated a compensation table, the control overwrites the compensation value from the T block.

The compensation tables offer the following benefits:

- Values can be changed without adapting the NC program
- Values can be changed during NC program run

If you change a value, then this change does not become active until the compensation is called again.

Types of compensation tables

Via the file name extension, you can determine in which coordinate system the control will perform the compensation.

The control provides the following compensation tables:

- tco (tool correction): Compensation in the tool coordinate system (**T-CS**)
- wco (workpiece correction): Compensation in the working plane coordinate system (**WPL-CS**)

Compensation via the table is an alternative to the compensation in the **T** block. Compensation from the table overwrites an already programmed compensation in the **T** block.

Compensation in the tool coordinate system (T-CS)

Any compensation in the compensation tables with the ***.tco** file name extension applies to the active tool. The table applies to all tool types. Therefore, columns that you may not need for your specific tool type will be displayed during creation.

> Enter only those values that are relevant to your tool. If you compensate for values that are not present with the existing tool, the control issues an error message.

The compensations have the following effects:

- In the case of milling cutters, as an alternative to the delta values in the **TOOL CALL**

If a shift with the ***.tco** compensation table is active, the control displays it on the **TOOL** tab of the additional status display.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Compensation in the working plane coordinate system (WPL-CS)

The values from the compensation tables with the ***.wco** file name extension are applied as shifts in the working plane coordinate system (**WPL-CS**).

If a shift with the ***.wco** compensation table is active, the control displays it, including the path, on the **TRANS** tab of the additional status display.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Creating a compensation table

Before you can work with a compensation table, you must first create the respective table.

You can create a compensation table as follows:

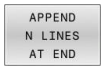
- Switch to the **Programming** operating mode

- Press the **PGM MGT** key

- Press the **NEW FILE** soft key
- Enter a file name with the desired extension (e.g., Corr.tco)

- Confirm by pressing the **ENT** key
- The control opens the **Select table format** window, if necessary.
- Select a table format, if necessary

- Press the **OK** soft key, if necessary
- Select **MM** or **INCH** as the unit of measure, if necessary
- The control opens the compensation table.



- Press the **APPEND N LINES AT END** soft key
- Enter the compensation values

You can select the table format, if there is at least one prototype of the table type.

The control displays whether the prototype is defined to use mm or inches as unit of measure. If the control displays both units of measure, you can select one of them.

The machine manufacturer defines the prototypes.

Activate the compensation table

Selecting a compensation table

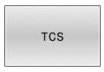
If you are using compensation tables, then use the **SEL CORR-TABLE** function to activate the desired compensation table from within the NC program.

To add a compensation table to the NC program:

- Press the **SPEC FCT** key

- Press the **PROGRAM DEFAULTS** soft key

- Press the **SELECT COMPENS. TABLE** soft key



- Press the soft key of the table type (e.g., **TCS**
- Select the table

If you are working without the **SEL CORR-TABLE** function, then you must activate the desired table prior to the test run or program run.

In all operating modes, proceed as follows:

- Select the desired operating mode
- Select the desired table in the file manager
- In the **Test Run** operating mode, the table receives the status S; in the **Program run, single block** and **Program run, full sequence** operating modes, it receives the status M.

Activating a compensation value

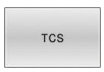
To activate a compensation value in the NC program:

- Press the **SPEC FCT** key

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **TRANSFORM / CORRDATA** soft key

- Press the **FUNCTION CORRDATA** soft key



- Press the soft key of the desired compensation (e.g., **TCS**
- Enter the line number

Duration of active compensation

Activated compensation stays in effect until the end of the program or until a tool change occurs.

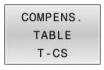
With **FUNCTION CORRDATA RESET**, you can program the compensations to reset.

Editing a compensation table during program run

You can change the values in the active compensation table during program run. As long as the compensation table is not yet active, the control dims the soft key.

Proceed as follows:

- Press the **OPEN COMPENS. TABLES** soft key



- Press the soft key of the desired table (e.g., **COMPENS. TABLE T-CS**)

- Set the **EDIT** soft key to **ON**
- Use the arrow keys to navigate to the desired location
- Edit the value

The changed data do not take effect until after the compensation has been activated again.

10.8 Accessing table values

Application

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.

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.

Reading a table value

The function **TABDATA READ** allows you to read a value from a table and save it to a Q parameter.

Depending on the type of column you want to transfer, you can use **Q**, **QL**, **QR**, or **QS** to save the value. The control automatically converts the table values to the unit of measure used in the NC program.

The control reads from the currently active tool table and preset table. You can read a value from a compensation table only if you have activated the table concerned.

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.

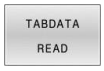
Procedure

Proceed as follows:

- Press the **SPEC FCT** key

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **TABDATA** soft key



- Press the **TABDATA READ** soft key
- Enter the Q parameter for the result

- Confirm with the **ENT** key

- Press the soft key for the desired table (e.g., **CORR-TCS**)
- Enter the column name

- Confirm with the **ENT** key
- Enter the row number of the table

- Press the **ENT** key

Example

N120 SEL CORR-TABLE TCS "TNC:\table\corr.tco"*	Activate the compensation table
N130 TABDATA READ Q1 = CORR-TCS COLUMN "DR" KEY "5"*	Save the value of row 5, column DR, from the compensation table to Q1

Writing a table value

Use the function **TABDATA WRITE** to write a value into a table.

Depending on the type of column you want to write to, you can use **Q**, **QL**, **QR**, or **QS** as a transfer parameter. Alternatively, you can define the value directly in the NC function **TABDATA WRITE**.

In order to write into a compensation table, you need to activate the 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.

Procedure

Proceed as follows:

- Press the **SPEC FCT** key

- Press the **PROGRAM FUNCTIONS** soft key.

- Press the **TABDATA** soft key

- Press the **TABDATA WRITE** soft key

- Press the soft key for the desired table (e.g., **CORR-TCS**)
- Enter the column name

- Confirm with the **ENT** key
- Enter the row number of the table

- Confirm with the **ENT** key
- Enter the number, name or variable

- Confirm with the **ENT** key

Example

N120 SEL CORR-TABLE TCS "TNC:\table\corr.tco"*	Activate the compensation table
N130 TABDATA WRITE CORR-TCS COLUMN "DR" KEY "3" = Q1*	Write the value from Q1 into line 3, column DR, of the compensation table

Adding a table value

Use the **TABDATA ADD** function to add a value to an existing table value.

Depending on the type of column you want to write to, you can use **Q**, **QL**, or **QR** as a transfer parameter. Alternatively, you can define the value directly in the NC function **TABDATA ADD**.

In order to write into a compensation table, you need to activate the table.

You can use the **TABDATA ADD** function to update a tool compensation value after a measurement has been repeated, for example.

Procedure

Proceed as follows:

- Press the **SPEC FCT** key

- Press the **PROGRAM FUNCTIONS** soft key.

- Press the **TABDATA** soft key

- Press the **TABDATA ADDITION** soft key

- Press the soft key for the desired table (e.g., **CORR-TCS**)
- Enter the column name

- Confirm with the **ENT** key
- Enter the row number of the table

- Confirm with the **ENT** key
- Enter the number or variable

- Confirm with the **ENT** key

Example

N120 SEL CORR-TABLE TCS "TNC:\table\corr.tco"*	Activate the compensation table
N130 TABDATA ADD CORR-TCS COLUMN "DR" KEY "3" = Q1*	Add the value from Q1 to line 3, column DR, of the compensation table

10.9 Defining a counter

Application

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

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 up to which the control is to repeat the NC program.

To program this behavior:

SPEC FCT

- Show the soft-key row for special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION COUNT

- Press the **FUNCTION COUNT** soft key

NOTICE

Caution: Data may be lost!

Only one counter can be managed by the control. If you execute an NC program that resets the counter, any counter progress of another NC program will be deleted.

- Please check prior to machining whether a counter is active.
- If necessary, note down the counter value and enter it again via the MOD menu after execution.

You can engrave the current counter reading with Cycle **G225 ENGRAVING**.

Further information: User's Manual for **Programming of Machining Cycles**

Effect in the Test Run operating mode

You can simulate the counter in the **Test Run** operating mode. Only the counter reading you have defined directly in the NC program is active. The counter reading in the MOD menu remains unaffected.

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

The counter reading from the MOD menu is only active in the **Program Run Single Block** and **Program Run Full Sequence** operating modes.

The counter reading remains the same after a restart of the control.

Defining FUNCTION COUNT

The **FUNCTION COUNT** NC function provides the following counter functions:

Soft key	Function
FUNCTION COUNT INC	Increase the counter by 1
FUNCTION COUNT RESET	Reset the counter
FUNCTION COUNT TARGET	Define the target count to be reached Input value: 0 to 9999
FUNCTION COUNT SET	Assign a defined value to the counter Input value: 0 to 9999
FUNCTION COUNT ADD	Increase the counter by a defined value Input value: 0 to 9999
FUNCTION COUNT REPEAT	Repeat the NC program from the label if the defined target count has not been reached yet

Example

N50 FUNCTION COUNT RESET*	Reset the counter reading
N60 FUNCTION COUNT TARGET10*	Enter the target number of parts to be machined
N70 G98 L11*	Enter the jump label
N80 G ...	Machining operation
N510 FUNCTION COUNT INC*	Increment the counter reading
N520 FUNCTION COUNT REPEAT LBL 11*	Repeat the machining operations if more parts are to be machined
N530 M30*	
N540 %COUNT G71*	

10.10 Creating text files

Application

You can use the control's text editor to write and edit texts. Typical applications:

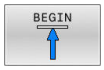
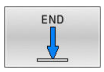
- Recording test results
- Documenting working procedures
- Creating formula collections

Text files have the extension .A (for ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

Opening and exiting a text file

- Operating mode: Press the **Programming** key
- To call the file manager, press the **PGM MGT** key.
- Display type .A files: Press the **SELECT TYPE** soft key and then the **SHOW ALL** soft key
- Select a file and open it with the **SELECT** soft key or **ENT** key, or open a new file by entering the new file name and confirming your entry with the **ENT** key

To leave the text editor, call the file manager and select a file of a different file type, for example an NC program.

Soft key	Cursor movements
	Move cursor one word to the right
	Move cursor one word to the left
	Go to next screen page
	Go to previous screen page
	Cursor at beginning of file
	Cursor at end of file

Editing texts

Above the first line of the text editor, there is an information field showing the file name, location and line information:

File: Name of the text file
Line: Line in which the cursor is presently located
Column: Column in which the cursor is presently located

The text is inserted or overwritten at the location of the cursor. You can move the cursor to any desired position in the text file by pressing the arrow keys.

You can insert a line break with the **RETURN** or **ENT** key.

Deleting and re-inserting characters, words and lines

With the text editor, you can erase words and even lines, and insert them at any desired location in the text.

- Move the cursor to the word or line that you wish to erase and insert at a different place in the text
- Press the **DELETE WORD** or **DELETE LINE** soft key: The text is removed and buffered
- Move the cursor to the location where you wish to insert the text, and press the **INSERT LINE / WORD** soft key

Soft key	Function
DELETE LINE	Delete and temporarily store a line
DELETE WORD	Delete and temporarily store a word
DELETE CHAR	Delete and temporarily store a character
INSERT LINE / WORD	Insert a line or word from temporary storage

Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before any of these actions, you must first select the desired text block:

- To select a text block: Move the cursor to the first character of the text you wish to select.

SELECT BLOCK

- Press the **SELECT BLOCK** soft key
- Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys—the selected text is shown in a different color.

After selecting the desired text block, you can edit the text with the following soft keys:

Soft key	Function
CUT OUT BLOCK	Delete the selected block and store temporarily
COPY BLOCK	Store the selected block temporarily without erasing (copy)

If desired, you can now insert the temporarily stored block at a different location:

- Move the cursor to the location where you want to insert the temporarily stored text block

INSERT BLOCK

- Press the **INSERT BLOCK** soft key: The text block is inserted

You can insert the temporarily stored text block as often as desired

Transferring the selected block to a different file

- Select the text block as described previously

APPEND TO FILE

- Press the **APPEND TO FILE** soft key.
- > The control displays the **Destination file =** dialog prompt.
- Enter the path and the name of the destination file.
- > The control appends the selected text block to the specified file. If no target file with the specified name is found, the control creates a new file with the selected text.

Inserting another file at the cursor position

- Move the cursor to the location in the text where you wish to insert another file

READ FILE

- Press the **READ FILE** soft key.
- > The control displays the **File name =** dialog prompt.
- Enter the path and name of the file you want to insert

Finding text sections

With the text editor, you can search for words or character strings in a text. The control provides the following two options.

Finding the current text

The search function is used for finding the next occurrence of the word in which the cursor is presently located:

- Move the cursor to the desired word.
- Select the search function: Press the **FIND** soft key
- Press the **FIND CURRENT WORD** soft key
- Find a word: Press the **FIND** soft key
- Exit the search function: Press the **END** soft key

Finding any text

- To select the search function, press the **FIND** soft key. The control shows the **Find text :** dialog prompt
- Enter the text that you wish to find
- Find text: Press the **FIND** soft key
- Exit the search function: Press the **END** soft key

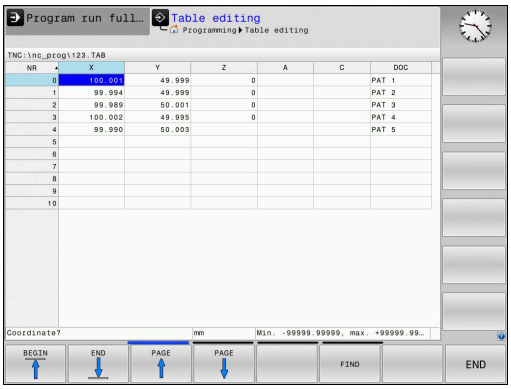
10.11 Freely definable tables

Fundamentals

In freely definable tables you can save and read any information from the NC program. The Q parameter functions **D26** to **D28** are provided for this purpose.

You can change the format of freely definable tables, i.e. the columns and their properties, by using the structure editor. They enable you to make tables that are exactly tailored to your application.

You can also toggle between a table view (standard setting) and form view.



The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**).

Creating a freely definable table

Proceed as follows:

PGM MGT

- Press the **PGM MGT** key
- Enter any desired file name with the extension .TAB

ENT

- Confirm with the **ENT** key

> The TNC displays a pop-up window with permanently saved table formats.

- Use the arrow key to select a table template, e.g. **example.tab**

ENT

- Confirm with the **ENT** key

> The control opens a new table in the predefined format.

- To adapt the table to your requirements you have to edit the table format
  Further information: "Editing the table format", Page 387

Refer to your machine manual.

Machine manufacturers may define their own table templates and save them in the control. When you create a new table, the control opens a pop-up window listing all available table templates.

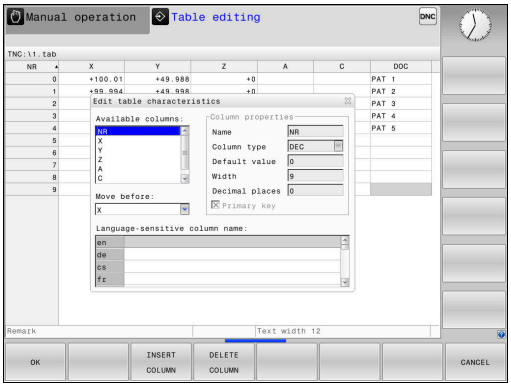
You can also save your own table templates in the TNC. To do so, create a new table, change the table format and save the table in the **TNC:\system\proto** directory. If you then create a new table, the control offers your template in the selection window for table templates.

Editing the table format

Proceed as follows:

EDIT FORMAT

- ▶ Press the **EDIT FORMAT** soft key
- > The control opens a pop-up window displaying the table structure.
- ▶ Adapt the format



The control provides the following options:

Structure command	Meaning
Available columns:	List of all columns contained in the table
Move before:	The entry highlighted in **Available columns** is moved in front of this column
Name	Column name: Displayed in the header
Column type	**TEXT**: Text entry **SIGN**: Algebraic sign + or - **BIN**: Binary digit **DEC**: Decimal, positive, integer number (cardinal number) **HEX**: Hexadecimal number **INT**: Integer number **LENGTH**: Length (converted in programs with inches) **FEED**: Feed rate (mm/min or 0.1 inch/min) **IFEED**: Feed rate (mm/min or inch/min) **FLOAT**: Floating-point number **BOOL**: Logical value **INDEX**: Index **TSTAMP**: Permanently defined format for date and time **UPTEXT**: Text entry in uppercase letters **PATHNAME**: Path name
Default value	Default value for the fields in this column
Width	Maximum number of characters in the column The column width is limited as follows: ■ Columns for alphanumeric entries allow up to 100 characters ■ Columns for numeric entries allow up to 15 characters **i** In addition to those 15 characters, the control can display an algebraic sign and a decimal separator
Primary key	First table column
Language-sensitive column name	Language-sensitive dialogs

Columns with a column type that permits letters, such as **TEXT**, can only be output or written to via QS parameters, even if the content of the cell is a number.

You can use a connected mouse or the navigation keys to move through the form.

Proceed as follows:

- Press the navigation keys to jump to the input fields

- Open the selection menus with the **GOTO** key

- Use the arrow keys to navigate within an input field

In a table that already contains lines you can not change the table properties **Name** and **Column type**. Once you have deleted all lines, you can change these properties. If required, create a backup copy of the table beforehand.

With the **CE** and **ENT** key combination, you can reset invalid values in fields with the **TSTAMP** column type.

Closing the structure editor

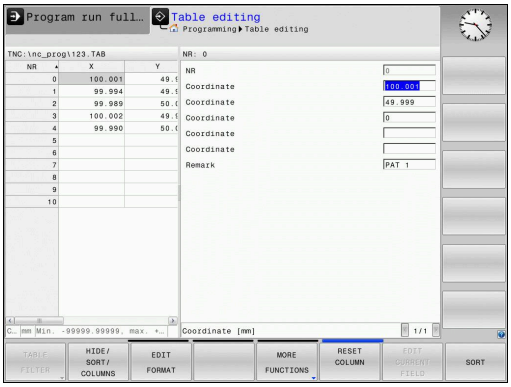
Proceed as follows:

- Press the **OK** soft key
- > The control closes the editing form and applies the changes.

- Alternative: Press the **CANCEL** soft key
- > The control discards all entered changes.

Switching between table and form view

All tables with the **.TAB** extension can be opened in either list view or form view.



Switch the view as follows:

- Press the **Screen layout** key

- Press the soft key with the desired view

In the left half of the form view, the control lists the line numbers with the contents of the first column.

You can change the data as follows in the form view:

- Press the **ENT** key in order to switch to the next input field on the right-hand side

Selecting another row to be edited:

- Press the **Next tab** key
- The cursor jumps to the left window.

- Use the arrow keys to select the desired row

- Press the **Next tab** key to switch back to the input window

D26 Opening a freely definable table

With the **D26** NC function, you open a freely definable table to be written to with **D27** or to be read from with **D28**.

Only one table can be opened in an NC program at any one time. A new NC block with **D26** automatically closes the last opened table.

The table to be opened must have the extension **.TAB**.

```
11 FN 26: TABOPEN TNC:\table \TAB1.TAB
```

; Open table with **FN 26**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 26: TABOPEN	Start of syntax for opening a table
File	Path of the table to be opened Fixed or variable name Selection by means of a selection window

Example: Open the table TAB1.TAB, which is saved in the directory TNC:\DIR1.

```
N560 D26  TNC:\DIR1\TAB1.TAB
```

Use the **SYNTAX** soft key to place paths within quotation marks. The quotation marks define the beginning and the end of the path. This enables the control to identify any special characters as a part of the path.

Further information: "File names", Page 117

If the complete path is enclosed in quotation marks, you can use both **** and **/** to separate the folders and files.

D27 writing to a freely definable table

With the **D27** NC function, you write to the table that you previously opened with **D26**.

Use the **D27** NC function 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. You can previously define the contents to be written to the columns in variables; or you define it directly in the NC function **FN 27**.

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.

If you write values to multiple columns, the control can either write only numbers or only names.

If you define a fixed value in the **FN 27** NC function, the control will write the same value to each defined column.

Input

```
11 FN 27: TABWRITE
   2/“Length,Radius“ = Q2
```
; Write to table with **FN 27**

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 27: TABWRITE	Start of syntax for writing to a table
Number	Row number of the table to be written to Fixed or variable number
Name or **QS**	Column names in the table to be written to Fixed or variable name Use commas to separate multiple column names.
Number, **Name**, or **QS**	Table value Fixed or variable number or name

Example

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.

```
N50 Q5 = 3,75
N60 Q6 = -5
N70 Q7 = 7,5
N80 D27  P01 5/“RADIUS,DEPTH,D“ = Q5
```

D28 reading a freely definable table

With the **D28** NC function, you can read data from the table previously opened with **D26**.

Use the **D28** 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.

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**).

Input

```
11 FN 28: TABREAD Q1 = 2 / "Length"   ; Read table with FN 28
```

The NC function includes the following syntax elements:

Syntax element	Meaning
FN 28: TABREAD	Start of syntax for reading from a table
Q, **QL**, **QR**, or **QS**	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 Fixed or variable number
Name or **QS**	Column name in the table to be read Fixed or variable name Use commas to separate multiple column names.

Example

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.

```
N50 D28  Q10 = 6/“X,Y,D“*
N60 D28  QS1 = 6/“DOC“*
```

Adapting the table format

NOTICE

Caution: Data may be lost!

The **ADAPT NC PGM / TABLE** function changes the format of all tables permanently. The control does not perform an automatic backup of the files prior to a format change. The files will thus be permanently changed and may no longer be usable.

- Only use the function in consultation with the machine manufacturer.

Soft key	Function
ADAPT NC PGM / TABLE	Adapt format of tables present after changing the control software version

The names of tables and table columns must start with a letter and must not contain an arithmetic operator (e.g., **+**).

10.12 Pulsing spindle speed FUNCTION S-PULSE

Program pulsing spindle speed

Application

Refer to your machine manual.
Read and note the functional description of the machine manufacturer.
Follow the safety precautions.

Using the **FUNCTION S-PULSE** function, you can program a pulsing spindle speed to avoid natural oscillations of the machine, for example.

With the **P-TIME** input value, you define the duration of an oscillation (oscillation period), and with the **SCALE** input value, the spindle speed change in percent. The spindle speed changes in a sinusoidal form around the nominal value.

Use **FROM-SPEED** and **TO-SPEED** to define the upper and lower spindle speed limits of a spindle speed range in which the pulsing spindle speed is in effect.. Both input values are optional. If you do not define a parameter, the function applies to the entire speed range.

Input

N30 FUNCTION S-PULSE P-TIME10 SCALE5 FROM-SPEED4800 TO-SPEED5200*	; Spindle speed variation of 5% around the nominal value within 10 seconds (with limit values)

The NC function includes the following syntax elements:

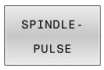
Syntax element	Meaning
FUNCTION S-PULSE	Start of syntax for pulsing spindle speed
P-TIME or **RESET**	Define the duration of an oscillation in seconds, or reset the pulsing spindle speed
SCALE	Spindle speed change in % Only if **P-TIME** has been selected
FROM-SPEED	Lower speed limit from which the pulsing spindle speed will be in effect Only if **P-TIME** has been selected Optional syntax element
TO-SPEED	Upper speed limit up to which the pulsing spindle speed will be in effect Only if **P-TIME** has been selected Optional syntax element

Proceed as follows for the definition:

- Show the soft key row with special functions

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **FUNCTION SPINDLE** soft key



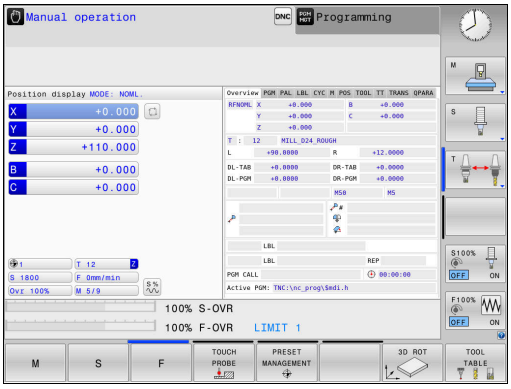
- Press the **SPINDLE-PULSE** soft key
- Define the oscillation period **P-TIME**
- Define the speed change **SCALE**

The control never exceeds a programmed speed limit. The spindle speed is maintained until the sinusoidal curve of the **FUNCTION S-PULSE** falls below the maximum speed once more.

Icons

In the status bar, the icon indicates the condition of the pulsing spindle speed:

Icon	Function
S %	Pulsing spindle speed active



Resetting the pulsing spindle speed

Example

```
N40 FUNCTION S-PULSE RESET*
```

Use the **FUNCTION S-PULSE RESET** to reset the pulsing spindle speed.

Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION SPINDLE

- Press the **FUNCTION SPINDLE** soft key

RESET SPINDLE-PULSE

- Press the **RESET SPINDLE-PULSE** soft key.

10.13 Dwell time FUNCTION FEED DWELL

Programming a dwell time

Application

Refer to your machine manual.
Read and note the functional description of the machine manufacturer.
Follow the safety precautions.

FUNCTION FEED DWELL allows you to program a cyclic dwell time in seconds, such as for forcing chip breaking.

Program **FUNCTION FEED DWELL** immediately prior to the operation you wish to run with chip breaking.

The **FUNCTION FEED DWELL** function is not effective with rapid traverse movements and probing motions.

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

Procedure

Example

```
N30 FUNCTION FEED DWELL D-TIME0.5 F-TIME5*
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION FEED
- Press the **FUNCTION FEED** soft key

FEED DWELL
- Press the **FEED DWELL** soft key
- Define the interval duration **D-TIME** for dwelling
- Define the interval duration **F-TIME** for cutting

Resetting the dwell time

Reset the dwell time immediately following the machining with chip breaking.

Example

```
N40 FUNCTION FEED DWELL RESET*
```

Use **FUNCTION FEED DWELL RESET** to reset the recurring dwell time.

Proceed as follows for the definition:

SPEC FCT

- Show the soft-key row with special functions

PROGRAM FUNCTIONS

- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION FEED

- Press the **FUNCTION FEED** soft key

RESET FEED DWELL

- Press the **RESET FEED DWELL** soft key

You can also reset the dwell time by entering **D-TIME 0**.
The control automatically resets the **FUNCTION FEED DWELL** function at the end of a program.

10.14 Dwell time FUNCTION DWELL

Programming a dwell time

Application

The **FUNCTION DWELL** function allows you to program a dwell time in seconds or define the number of spindle revolutions for dwelling.

Procedure

Example

```
N30 FUNCTION DWELL TIME10*
```

Example

```
N40 FUNCTION DWELL REV5.8*
```

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION DWELL
- Press the **FUNCTION DWELL** soft key

DWELL TIME
- Press the **DWELL TIME** soft key
- Define the duration in seconds

DWELL REVOLUTIONS
- Or press the **DWELL REVOLUTIONS** soft key
- Define the number of spindle revolutions

10.15 Lift-off tool at NC stop: FUNCTION LIFTOFF

Programming liftoff with FUNCTION LIFTOFF

Requirement

> Refer to your machine manual.
> This function must be configured and enabled by your machine manufacturer. In machine parameter **CfgLiftOff** (no. 201400), the machine manufacturer defines the path the tool is supposed to traverse for a **LIFTOFF** command. You can also use machine parameter **CfgLiftOff** to deactivate the function.

In the **LIFTOFF** column of the tool table, set the **Y** parameter for the active tool.

Further information: User's Manual for **Setup, Testing and Running NC Programs**

Application

The **LIFTOFF** function is effective in the following situations:

- In case of an NC stop triggered by you
- In case of an NC stop triggered by the software (e. g., if an error has occurred in the drive system)
- In case of a power interruption

The tool retracts from the contour by up to 2 mm. The control calculates the liftoff direction based on the input in the **FUNCTION LIFTOFF** block.

You have the following options for programming the **LIFTOFF** function:

- **FUNCTION LIFTOFF TCS X Y Z**: Liftoff in the tool coordinate system (**T-CS**) with the vector resulting from **X**, **Y** and **Z**
- **FUNCTION LIFTOFF ANGLE TCS SPB**: Liftoff in the tool coordinate system (**T-CS**) with a defined spatial angle
- Liftoff in the tool axis direction with **M148**

Further information: "Lifting off the tool automatically from the contour at NC stop: M148", Page 252

Programing tool liftoff with a defined vector

Example

```
N40 FUNCTION LIFTOFF TCS X+0 Y+0.5 Z+0.5*
```

LIFTOFF TCS X Y Z allows you to define the liftoff direction as a vector in the tool coordinate system. The control calculates the liftoff height in each axis based on the tool path defined by the machine manufacturer.

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION LIFTOFF
- Press the **FUNCTION LIFTOFF** soft key

LIFTOFF TCS
- Press the **LIFTOFF TCS** soft key
- Enter X, Y, and Z vector components

Programing tool liftoff with a defined angle

Example

```
N40 FUNCTION LIFTOFF ANGLE TCS SPB+20*
```

LIFTOFF ANGLE TCS SPB allows you to define the liftoff direction as a spatial angle in the tool coordinate system.

The SPB angle you enter describes the angle between Z and X. If you enter 0°, the tool lifts off in the tool axis direction Z.

Proceed as follows for the definition:

SPEC FCT
- Show the soft key row with special functions

PROGRAM FUNCTIONS
- Press the **PROGRAM FUNCTIONS** soft key

FUNCTION LIFTOFF
- Press the **FUNCTION LIFTOFF** soft key

LIFTOFF ANGLE TCS
- Press the **LIFTOFF ANGLE TCS** soft key
- Enter the SPB angle

Resetting the lift-off function

Example

```
N40 FUNCTION LIFTOFF RESET*
```

FUNCTION LIFTOFF RESET allows you to reset the lift-off function.

Proceed as follows for the definition:

- Show the soft key row with special functions

- Press the **PROGRAM FUNCTIONS** soft key

- Press the **FUNCTION LIFTOFF** soft key

- Press the **LIFTOFF RESET** soft key

The control uses the **M149** function to deactivate the **FUNCTION LIFTOFF** function without resetting the liftoff direction. If you program **M148**, the control will automatically liftoff the tool in the direction defined by the **FUNCTION LIFTOFF** function.

The control automatically resets the **FUNCTION LIFTOFF** function at the end of a program.

11

Multiple-axis Machining

11.1 Functions for multi-axis machining

This chapter summarizes the control functions for machining with multiple axes:

Control function	Description	Page
PLANE	Define machining in the tilted working plane	405
M116	Feed rate of rotary axes	434
M126	Shortest-path traverse of rotary axes	435
M94	Reduce display value of rotary axes	436
M138	Selection of tilted axes	437

11.2 The PLANE function: Tilting the working plane (option 8)

Introduction

Refer to your machine manual.

The machine manufacturer must enable the functions for tilting the working plane!

You can only use the **PLANE** function in its entirety on machines having at least two rotary axes (table axes, head axes or combined axes). The **PLANE AXIAL** function is an exception. **PLANE AXIAL** can also be used on machines which have only one programmed rotary axis.

The **PLANE** functions provide powerful options to define tilted working planes in various ways.

The parameter definition of the **PLANE** functions is subdivided into two parts:

- The geometric definition of the plane, which is different for each of the available **PLANE** functions.
- The positioning behavior of the **PLANE** function, which is independent of the plane definition and is identical for all **PLANE** functions
  Further information: "Defining the positioning behavior of the PLANE function", Page 424

NOTICE

Danger of collision!

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

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

NOTICE

Danger of collision!

Cycle **28 MIRRORING** can have different effects in conjunction with the **Tilt working plane** function. The programming sequence, the mirrored axes, and the tilting function used are critical in this regard. There is a risk of collision during the tilting operation and subsequent machining!

- Check the sequence and positions using a graphic simulation
- Carefully test the NC program or program section in the **Program run, single block** operating mode

Examples

1 When Cycle **28 MIRRORING** is programmed before the tilting function without rotary axes:
   - The tilt of the **PLANE** function used (except **PLANE AXIAL**) is mirrored
   - Mirroring takes effect after tilting with **PLANE AXIAL** or Cycle **G80**

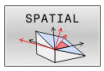
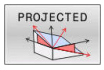
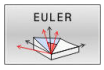
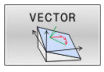
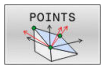
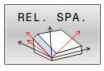
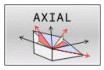
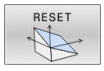
2 When Cycle **28 MIRRORING** is programmed before the tilting function with a rotary axis:
   - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

Operating and programming notes:

- The actual-position-capture function is not possible with an active tilted working plane.
- If you use the **PLANE** function when **M120** is active, the control automatically rescinds the radius compensation, which also rescinds the **M120** function.
- Always reset all **PLANE** functions with **PLANE RESET**. For example, if you define all spatial angles with 0, the control resets only the angles and not the tilting function.
- If you restrict the number of rotary axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine manufacturer decides whether the control takes the angles of deselected axes into account or sets them to 0.
- The control only supports tilting functions if tool axis **Z** is active.

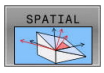
Overview

Most **PLANE** functions (except **PLANE AXIAL**) can be used to describe the desired working plane independently of the rotary axes available on your machine. The following possibilities are available:

Soft key	Function	Required parameters	Page
	**SPATIAL**	Three spatial angles: **SPA**, **SPB**, and **SPC**	410
	**PROJECTED**	Two projection angles: **PROPR** and **PROMIN** and a rotation angle **ROT**	413
	**EULER**	Three Euler angles: precession (**EULPR**), nutation (**EULNU**) and rotation (**EULROT**),	415
	**VECTOR**	Normal vector for defining the plane and base vector for defining the direction of the tilted X axis	417
	**POINTS**	Coordinates of any three points in the plane to be tilted	419
	**RELATIVE**	Single, incrementally effective spatial angle	421
	**AXIAL**	Up to three absolute or incremental axis angles **A**,**B**,**C**	422
	**RESET**	Reset the PLANE function	409

Running an animation

To familiarize yourself with the various definition possibilities of each **PLANE** function, you can start animated sequences via soft key. To do so, first enter animation mode and then select the desired **PLANE** function. While the animation plays, the control highlights the soft key of the selected **PLANE** function with a blue color.

Soft key	Function
	Switch on the animation mode
	Select the desired animation (highlighted in blue)

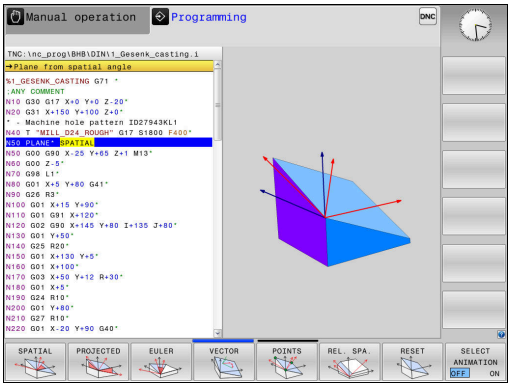
Defining the PLANE function

SPEC FCT

- Show the soft-key row with special functions

TILT MACHINING PLANE

- Press the **TILT MACHINING PLANE** soft key
- The control displays the available **PLANE** functions in the soft-key row.
- Select the **PLANE** function



Selecting functions

- Press the soft key linked to the desired function
- The control continues the dialog and prompts you for the required parameters.

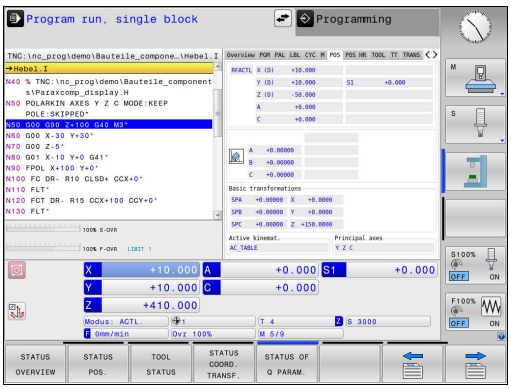
Selecting the function while animation is active

- Press the soft key linked to the desired function
- The control plays the animation.
- To apply the currently active function, press the soft key of that function again or press the **ENT** key

Position display

As soon as a **PLANE** function (except **PLANE AXIAL**) is active, the control shows the calculated spatial angle in the additional status display.

During tilting into position (**MOVE** or **TURN** mode), the control shows, in the rotary axis, the distance to go to the calculated final position of the rotary axis in the distance-to-go display (**ACTDST** and **REFDST**).



Resetting PLANE function

Example

```
N10 PLANE RESET MOVE DIST50 F1000*
```

SPEC FCT

- Show the soft-key row with special functions

TILT MACHINING PLANE

- Press the **TILT MACHINING PLANE** soft key
- The control displays the available **PLANE** functions in the soft-key row

RESET

- Select the reset function

MOVE

- Specify whether the control should automatically move the tilting axes to home position (**MOVE** or **TURN**) or not (**STAY**)
  Further information: "Automatic tilting into position MOVE/TURN/STAY", Page 425

END

- Press the **END** key.

The **PLANE RESET** function resets the active tilt and the angles (**PLANE** function or Cycle **G80**) (angle = 0 and function inactive). It does not need to be defined more than once.

The function does not reset any offset values!

Further information: User's Manual for **Setup, Testing and Running NC Programs**

- Deactivate tilting in the **Manual operation** mode via the 3D-ROT menu.
  Further information: User's Manual for **Setup, Testing and Running NC Programs**
- With the touch probe functions you can save the misalignment of the workpiece as a 3D basic rotation in the preset table (e.g., **Plane (PL)**). In the NC program you must then align the workpiece with a tilting function (e.g., with **PLANE SPATIAL SPA+0 SPB+0 SPC+0 TURN FMAX**). You must not use **PLANE RESET** for machining, because the control does not take into account the 3D basic rotation with this function.
  Further information: "Defining the working plane with spatial angles: PLANE SPATIAL", Page 410

Defining the working plane with spatial angles: PLANE SPATIAL

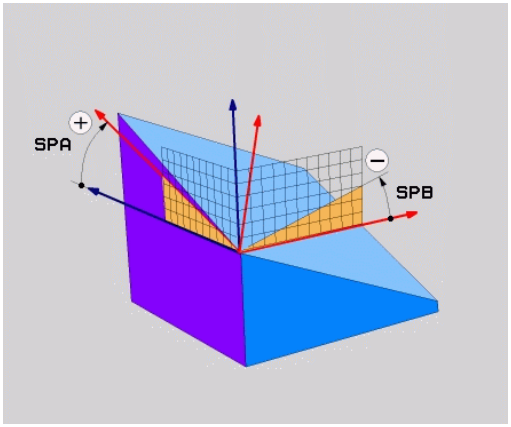
Application

Spatial angles define a working plane through up to three rotations in the non-tilted workpiece coordinate system (**tilting sequence A-B-C**).

Most users assume three successive rotations in reverse order (**tilting sequence C-B-A**).

The result is identical for both perspectives, as the following comparison shows.

Further information: "Comparison of views - Example: chamfer", Page 411



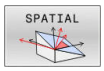
Programming notes:

- You must always define all three spatial angles **SPA**, **SPB** and **SPC**, even if one or more have the value 0.
- Depending on the machine, Cycle **G80** requires you to enter spatial angles or axis angles. If the configuration (machine parameter setting) allows the input of spatial angles, the angle definition is the same in Cycle **G80** and in the **PLANE SPATIAL** function.
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 424

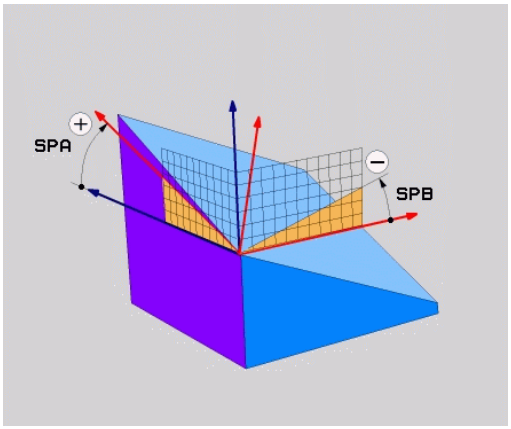
Input parameters

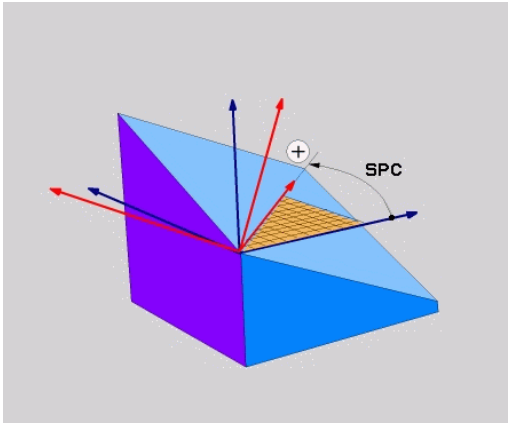
Example

```
N50 PLANE SPATIAL SPA+27 SPB+0 SPC+45 .....*
```



- **Spatial angle A?**: Rotational angle **SPA** about the (non-tilted) X axis. Input range from -359.9999 to +359.9999
- **Spatial angle B?**: Rotational angle **SPB** about the (non-tilted) Y axis. Input range from -359.9999 to +359.9999
- **Spatial angle C?**: Rotational angle **SPC** about the (non-tilted) Z axis. Input range from -359.9999 to +359.9999
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 424



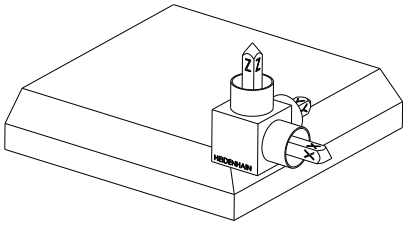


Comparison of views - Example: chamfer

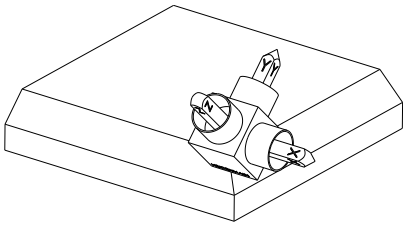
Example

```
N110 PLANE SPATIALSPA+45 SPB+0 SPC+90 TURN MB MAX FMAX
     SYM- TABLE ROT*
```

View A-B-C



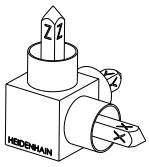
Initial state



SPA+45

Orientation of tool axis **Z**

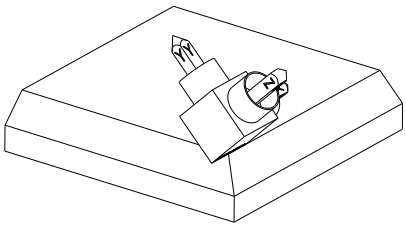
Rotation around the X axis of the non-tilted workpiece coordinate system **W-CS**



SPB+0

Rotation around the Y axis of the non-tilted **W-CS**

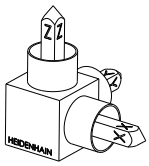
No rotation with value 0



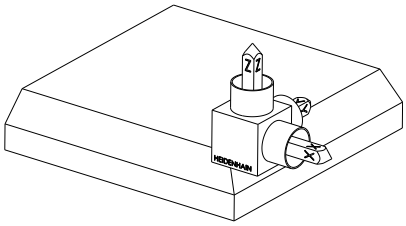
SPC+90

Orientation of main axis **X**

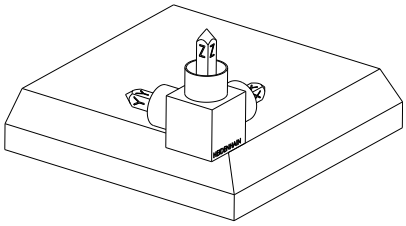
Rotation around the Z axis of the non-tilted **W-CS**



View C-B-A



Initial state



SPC+90

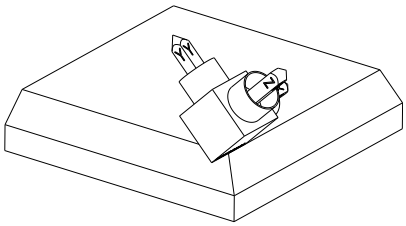
Orientation of main axis **X**

Rotation around the Z axis of the workpiece coordinate system **W-CS**, meaning in the non-tilted working plane

SPB+0

Rotation around the Y axis in the working plane coordinate system **WPL-CS**, meaning in the tilted working plane

No rotation with value 0



SPA+45

Orientation of tool axis **Z**

Rotation around the X axis in **WPL-CS**, meaning in the tilted working plane

Both views have an identical result.

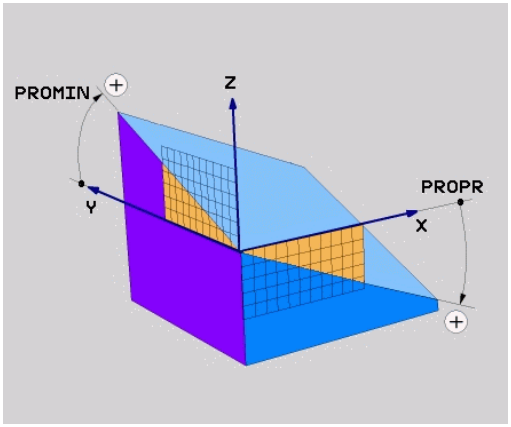
Abbreviations used

Abbreviation	Meaning
SPATIAL	**Spatial** = in space
SPA	**Sp**atial **A**: Rotation about the (non-tilted) X axis
SPB	**Sp**atial **B**: Rotation about the (non-tilted) Y axis
SPC	**Sp**atial **C**: Rotation about the (non-tilted) Z axis

Defining the working plane with projection angles: PLANE PROJECTED

Application

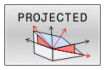
Projection angles define a machining plane through the entry of two angles that you determine by projecting the first coordinate plane (Z/X plane with tool axis Z) and the second coordinate plane (Y/Z with tool axis Z) onto the machining plane to be defined.



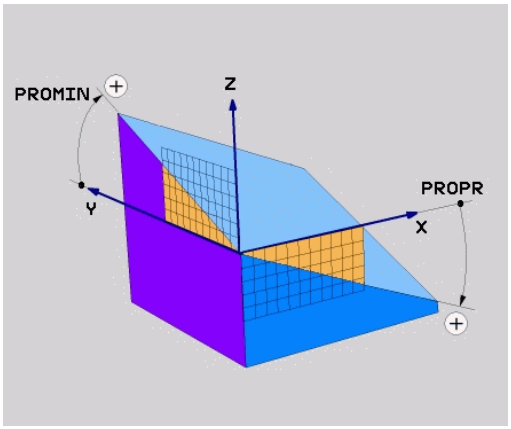
Programming notes:

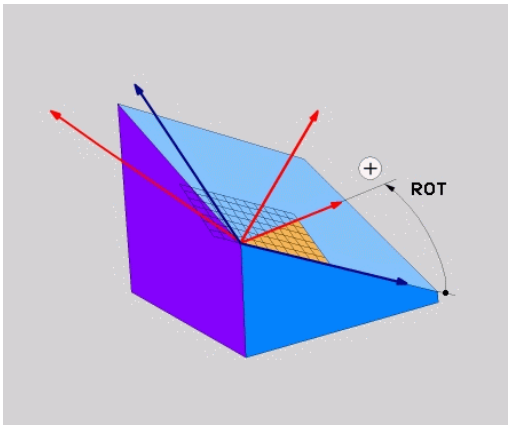
- The projection angles correspond to the angle projections on the planes of a rectangular coordinate system. The angles at the outer faces of the workpiece only are identical to the projection angles if the workpiece is rectangular. Thus, with workpieces that are not rectangular, the angle specifications from the engineering drawing often differ from the actual projection angles.
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 424

Input parameters



- **Projection angle on 1st Coordinate plane?**: Projected angle of the tilted machining plane in the 1st coordinate plane of the untilted coordinate system (Z/X for tool axis Z). Input range: from –89.9999° to +89.9999°. The 0° axis is the principal axis of the active working plane (X for tool axis Z, positive direction)
- **Projection angle on 2nd Coordinate plane?**: Projected angle in the 2nd coordinate plane of the untilted coordinate system (Y/Z for tool axis Z). Input range: from –89.9999° to +89.9999°. The 0° axis is the secondary axis of the active machining plane (Y for tool axis Z)
- **ROT angle of tilted plane?**: Rotation of the tilted coordinate system around the tilted tool axis (corresponds to a rotation with Cycle **G73**). The rotation angle provides an easy way to specify the direction of the principal axis of the working plane (X for tool axis Z, Z for tool axis Y). Input range: –360° to +360°
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 424





Example

```
N50 PLANE  PROJECTED PROPR+24  PROMIN+24 ROT+30  .....*
```

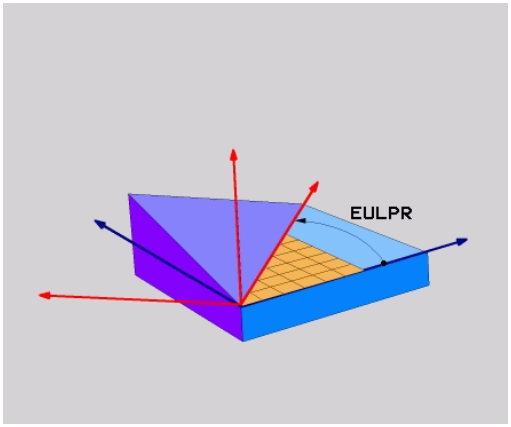
Abbreviations used:

PROJECTED	Projected
PROPR	Principal plane
PROMIN	Minor plane
ROT	Rotation

Defining the working plane with Euler angles: PLANE EULER

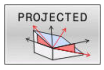
Application

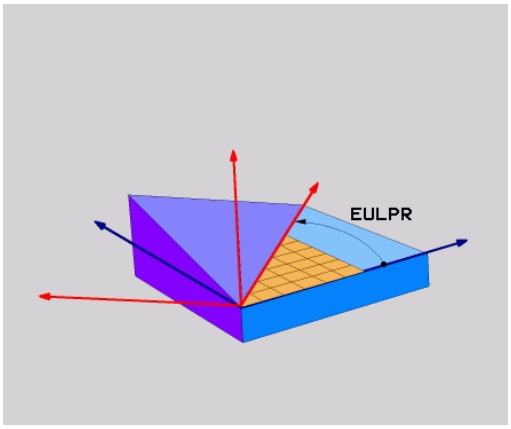
Euler angles define a machining plane through up to three **rotations about the respectively tilted coordinate system**. The Swiss mathematician Leonhard Euler defined these angles.



> You can select the desired positioning behavior.
> **Further information:** "Defining the positioning behavior of the PLANE function", Page 424

Input parameters





- **Rot. angle Main coordinate plane?:** Rotary angle **EULPR** around the Z axis. Please note:
  - Input range: -180.0000° to 180.0000°
  - The 0° axis is the X axis
- **Tilting angle tool axis?**: Tilting angle **EULNUT** of the coordinate system around the X axis shifted by the precession angle. Please note:
  - Input range: 0° to 180.0000°
  - The 0° axis is the Z axis
- **ROT angle of tilted plane?**: **EULROT** rotation of the tilted coordinate system around the tilted Z axis (corresponds to a rotation with Cycle **G73**). Use the rotation angle to easily define the direction of the X axis in the tilted working plane.

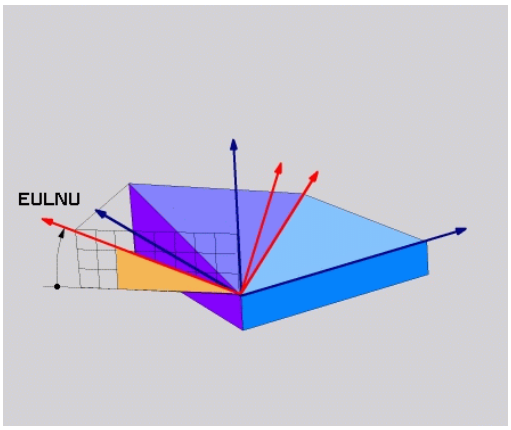
  Please note:
  - Input range: 0° to 360.0000°
  - The 0° axis is the X axis
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 424

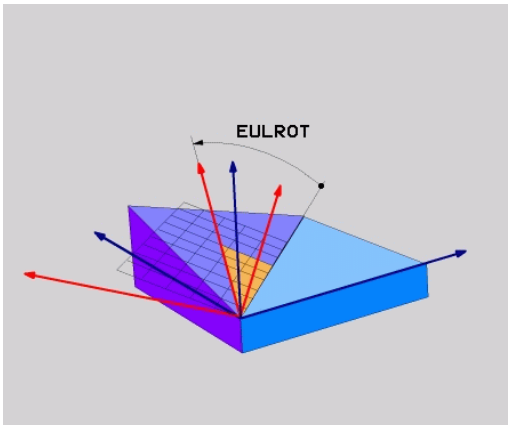
Example

```
N50 PLANE  EULER EULPR45  EULNU20 EULROT22  .....*
```

Abbreviations used

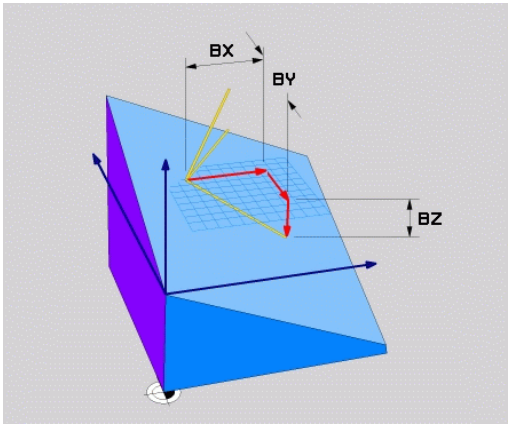
Abbreviation	Meaning
EULER	Swiss mathematician who defined these angles
EULPR	**Pr**ecession angle: angle describing the rotation of the coordinate system around the Z axis
EULNU	**Nu**tation angle: angle describing the rotation of the coordinate system around the X axis shifted by the precession angle
EULROT	**Rot**ation angle: angle describing the rotation of the tilted machining plane around the tilted Z axis





Defining the working plane with two vectors: PLANE VECTOR

Application



You can use the definition of a working plane via **two vectors** if your CAD system can calculate the base vector and normal vector of the tilted machining plane. A normalized input is not necessary. The control internally calculates the normal, so you can enter values between -9.999999 and +9.999999.

The base vector required for the definition of the machining plane is defined by the components **BX, BY** and **BZ.** The normal vector is defined by the components **NX**, **NY** and **NZ**.

Programming notes:

- The control calculates standardized vectors from the values you enter.
- The normal vector defines the slope and the orientation of the working plane. The base vector defines the orientation of the main axis X in the defined working plane. To ensure that the definition of the working plane is unambiguous, you must program the vectors perpendicular to each other. The machine tool builder defines how the control will behave for vectors that are not perpendicular.
- The programmed normal vector must not be too short, e.g. all directional components having a length of 0 or 0.0000001. In this case, the control would not be able to determine the slope. Machining is aborted and an error message is displayed. This behavior is independent of the configuration of the machine parameters.
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 424

Refer to your machine manual.

The machine tool builder configures the behavior of the control with vectors that are not perpendicular.

Alternatively to generating the default error message, the control can correct (or replace) the base vector that is not perpendicular. This correction (or replacement) does not affect the normal vector.

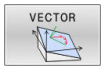
Default correction behavior of the control if the base vector is not perpendicular:

- The base vector is projected along the normal vector onto the working plane (defined by the normal vector).

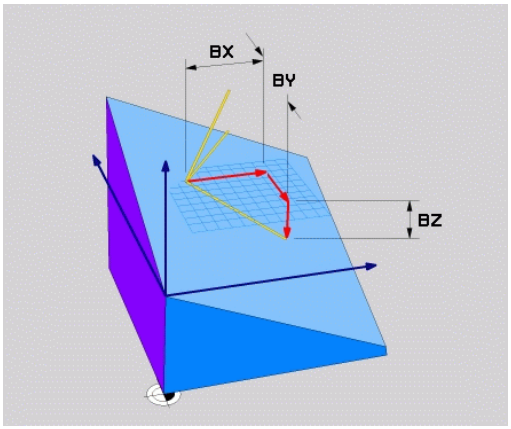
Correction behavior of the control if the base vector is not perpendicular and too short, parallel or antiparallel to the normal vector:

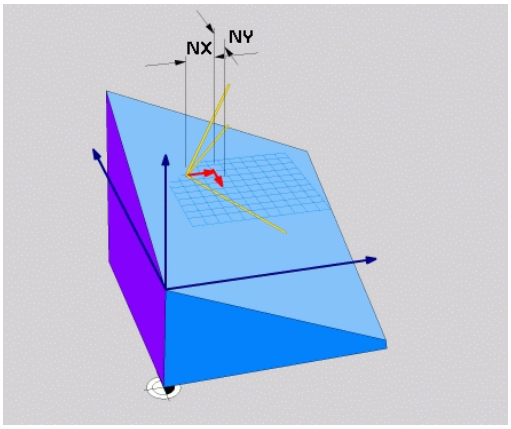
- If the normal vector has no X component, the base vector corresponds to the original X axis
- If the normal vector has no Y component, the base vector corresponds to the original Y axis

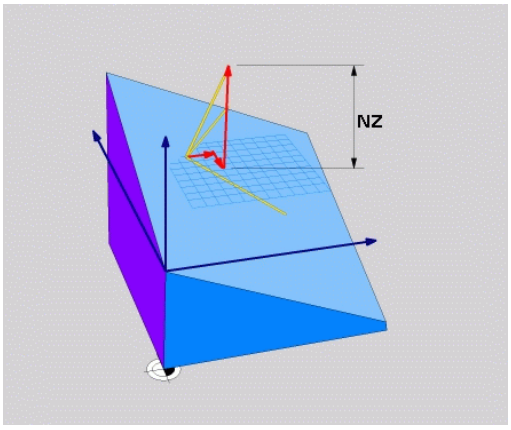
Input parameters



- **X component of base vector?**: X component **BX** of the base vector B; input range: from -9.9999999 to +9.9999999
- **Y component of base vector?**: Y component **BY** of the base vector B; input range: from -9.9999999 to +9.9999999
- **Z component of base vector?**: Z component **BZ** of the base vector B; input range: from -9.9999999 to +9.9999999
- **X component of normal vector?**: X component **NX** of the normal vector N; input range: from -9.9999999 to +9.9999999
- **Y component of normal vector?**: Y component **NY** of the normal vector N; input range: from -9.9999999 to +9.9999999
- **Z component of normal vector?**: Z component **NZ** of the normal vector N; input range: from -9.9999999 to +9.9999999
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 424







Example

```
N50 PLANE  VECTOR BX0.8  BY-0.4 BZ-0.42  NX0.2 NY0.2
    NT0.92 ..*
```

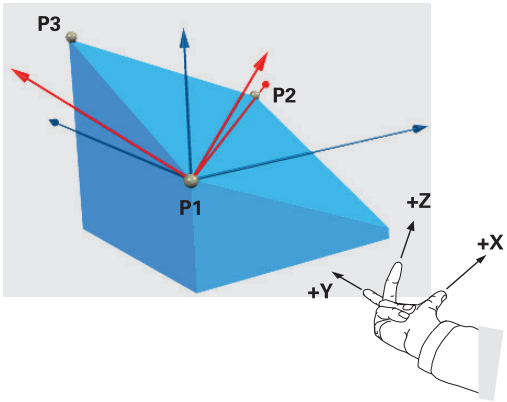
Abbreviations used

Abbreviation	Meaning
VECTOR	Vector
BX, BY, BZ	**B**ase vector : **X**, **Y**, and **Z** components
NX, NY, NZ	**N**ormal vector : **X**, **Y**, and **Z** components

Defining the working plane via three points: PLANE POINTS

Application

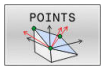
A working plane can be uniquely defined by entering **any three points P1 to P3 in this plane**. This possibility is realized in the **PLANE POINTS** function.



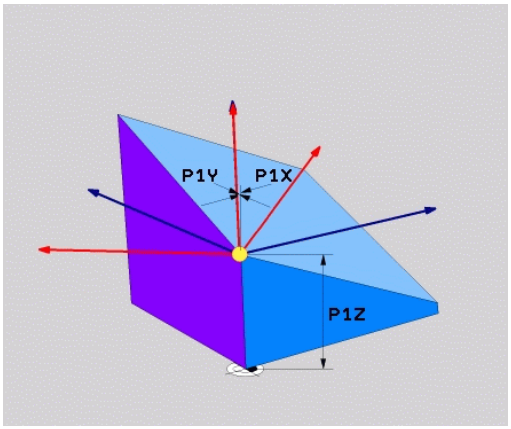
Programming notes:

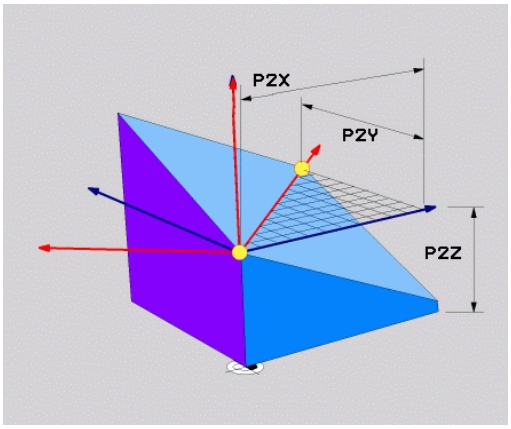
- The three points define the slope and orientation of the plane. The position of the active datum is not changed through **PLANE POINTS**.
- Point 1 and Point 2 determine the orientation of the tilted main axis X (for tool axis Z).
- Point 3 defines the slope of the tilted working plane. In the defined working plane, the Y axis is automatically oriented perpendicularly to the main axis X. The position of Point 3 thus also determines the orientation of the tool axis and consequently the orientation of the working plane. To have the positive tool axis pointing away from the workpiece, Point 3 must be located above the connection line between Point 1 and Point 2 (right-hand rule).
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 424

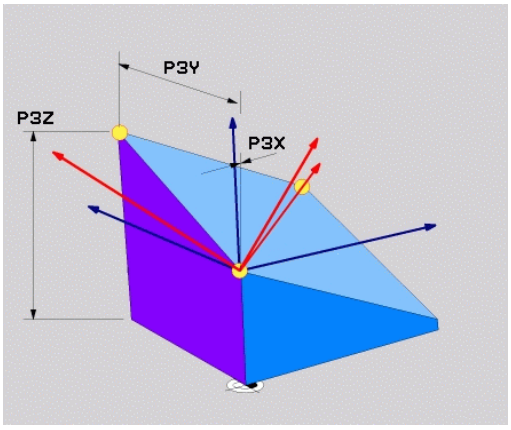
Input parameters



- **X coordinate of 1stplane point?**: X coordinate **P1X** of the 1st plane point
- **Y coordinate of 1stplane point?**: Y coordinate **P1Y** of the 1st plane point
- **Z coordinate of 1stplane point**: Z coordinate **P1Z** of the 1st plane point
- **X coordinate of 2ndplane point?**: X coordinate **P2X** of the 2nd plane point
- **Y coordinate of 2ndplane point?**: Y coordinate **P2Y** of the 2nd plane point
- **Z coordinate of 2ndplane point?**: Z coordinate **P2Z** of the 2nd plane point
- **X coordinate of 3rdplane point?**: X coordinate **P3X** of the 3rd plane point
- **Y coordinate of 3rdplane point?**: Y coordinate **P3Y** of the 3rd plane point
- **Z coordinate of 3rdplane point?**: Z coordinate **P3Z** of the 3rd plane point
- Continue with the positioning properties
  Further information: "Defining the positioning behavior of the PLANE function", Page 424







Example

```
N50 PLANE  POINTS P1X+0  P1Y+0 P1Z+20  P2X+30 P2Y+31  P2Z
    +20 P3X+0 P3Y+41  P3Z+32.5 .....*
```

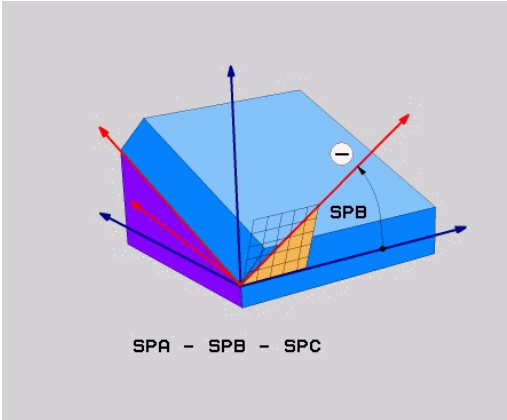
Abbreviations used

Abbreviation	Meaning
POINTS	Points

Defining the working plane via a single incremental spatial angle: PLANE RELATIV

Application

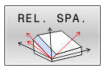
Use a relative spatial angle when an already active tilted working plane is to be tilted by **another rotation**. Example: machining a 45° chamfer on a tilted plane.



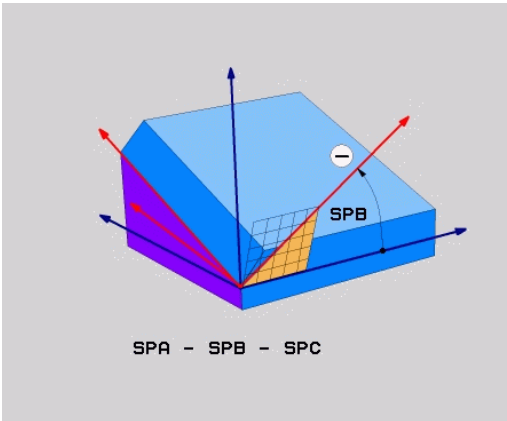
Programming notes:

- The defined angle is always in effect in respect to the active working plane, regardless of the tilting function you used before.
- You can program any number of **PLANE RELATIV** functions in a row.
- If you want to return the working plane to the orientation that was active before the **PLANE RELATIV** function, define the same **PLANE RELATIV** function again but enter the value with the opposite algebraic sign.
- If you use **PLANE RELATIV** without previous tilting, **PLANE RELATIV** will be effective directly in the workpiece coordinate system. In this case, you can tilt the original working plane by entering a defined spatial angle in the **PLANE RELATIV** function.
- You can select the desired positioning behavior. **Further information:** "Defining the positioning behavior of the PLANE function", Page 424

Input parameters



- **Incremental angle?**: Spatial angle by which the active machining plane is to be rotated. Use a soft key to select the axis to be rotated around. Input range: -359.9999° to +359.9999°
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 424



Example

```
N50 PLANE RELATIV SPB-45 .....*
```

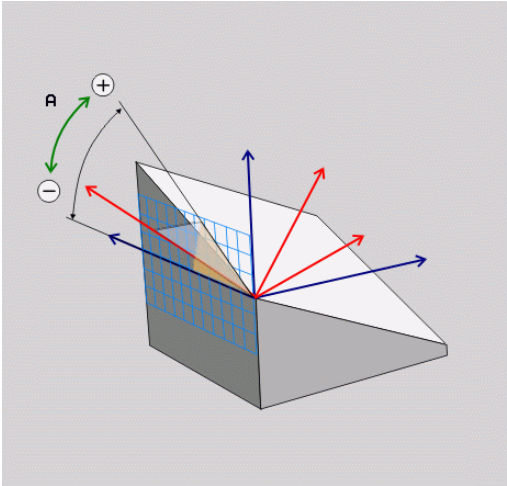
Abbreviations used

Abbreviation	Meaning
RELATIV	Relative to

Tilting the working plane through axis angles: PLANE AXIAL

Application

The **PLANE AXIAL** function defines both the slope and the orientation of the working plane and the nominal coordinates of the rotary axes.



PLANE AXIAL can also be used on machines that have only one rotary axis.

The input of nominal coordinates (axis angle input) is advantageous in that it provides an unambiguously defined tilting situation based on defined axis positions. Spatial angles entered without an additional definition are often mathematically ambiguous. Without the use of a CAM system, entering axis angles, in most cases, only makes sense if the rotary axes are positioned perpendicularly.

Refer to your machine manual.

If your machine allows spatial angle definitions, you can continue your programming with **PLANE RELATIV** after **PLANE AXIAL**.

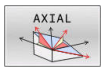
Programming notes:

- The axis angles must correspond to the axes present on the machine. If you try to program axis angles for rotary axes that do not exist on the machine, the control will generate an error message.
- Use **PLANE RESET** to reset the **PLANE AXIAL** function. Entering 0 only resets the axis angle, but does not deactivate the tilting function.
- The axis angles of the **PLANE AXIAL** function are modally effective. If you program an incremental axis angle, the control will add this value to the currently effective axis angle. If you program two different rotary axes in two successive **PLANE AXIAL** functions, the new working plane is derived from the two defined axis angles.
- **SYM** (**SEQ**), **TABLE ROT**, and **COORD ROT** have no function in conjunction with **PLANE AXIAL**.
- The **PLANE AXIAL** function does not take basic rotation into account.

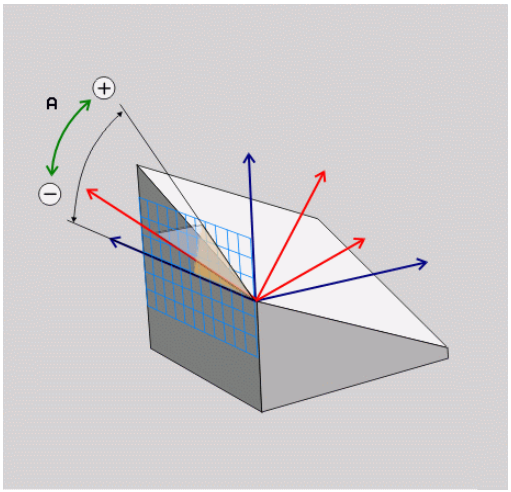
Input parameters

Example

```
N50 PLANE  AXIAL B-45  .....*
```



- **Axis angle A?**: Axis angle **to which** the A axis is to be tilted. If entered incrementally, it is the angle **by which** the A axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- **Axis angle B?**: Axis angle **to which** the B axis is to be tilted. If entered incrementally, it is the angle **by which** the B axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- **Axis angle C?**: Axis angle **to which** the C axis is to be tilted. If entered incrementally, it is the angle **by which** the C axis is to be tilted from its current position. Input range: –99999.9999° to +99999.9999°
- Continue with the positioning properties **Further information:** "Defining the positioning behavior of the PLANE function", Page 424



Abbreviations used

Abbreviation	Meaning
AXIAL	In the axial direction

Defining the positioning behavior of the PLANE function

Overview

Independently of which PLANE function you use to define the tilted machining plane, the following functions are always available for the positioning behavior:

- Automatic positioning
- Selecting alternate tilting options (not for **PLANE AXIAL**)
- Selecting the type of transformation (not for **PLANE AXIAL**)

NOTICE

Danger of collision!

Cycle **28 MIRRORING** can have different effects in conjunction with the **Tilt working plane** function. The programming sequence, the mirrored axes, and the tilting function used are critical in this regard. There is a risk of collision during the tilting operation and subsequent machining!

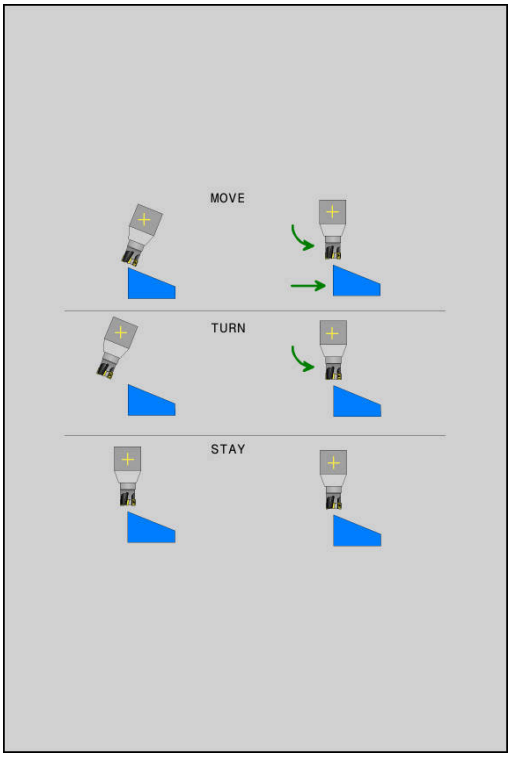
- ▶ Check the sequence and positions using a graphic simulation
- ▶ Carefully test the NC program or program section in the **Program run, single block** operating mode

Examples

1. When Cycle **28 MIRRORING** is programmed before the tilting function without rotary axes:
   - The tilt of the **PLANE** function used (except **PLANE AXIAL**) is mirrored
   - Mirroring takes effect after tilting with **PLANE AXIAL** or Cycle **G80**
2. When Cycle **28 MIRRORING** is programmed before the tilting function with a rotary axis:
   - The mirrored rotary axis has no effect on the tilt specified in the **PLANE** function used, because only the movement of the rotary axis is mirrored

Automatic tilting into position MOVE/TURN/STAY

After you have entered all of the parameters for the plane definition, you must specify how the control is to tilt the rotary axes to the calculated axis value. This entry is mandatory.



The control offers the following ways of tilting the rotary axes to the calculated axis values:

MOVE

- ▶ The PLANE function is to automatically tilt the rotary axes to the calculated axis values, with the relative position between the tool and the workpiece remaining the same.
- > The control carries out a compensating movement in the linear axes.

TURN

- ▶ The PLANE function is to automatically tilt the rotary axes to the calculated axis values, during which only the rotary axes are positioned.
- > The control does **not** carry out a compensating movement in the linear axes.

STAY

- ▶ You tilt the rotary axes into position in a subsequent, separate positioning block

If you have selected the **MOVE** option (**PLANE** function is to automatically tilt into position with a compensation movement), then the two subsequently declared parameters **Dist. tool tip - center of rot.** and **Feed rate? F=** must still be defined.

If you have selected the **TURN** option (**PLANE** function is to automatically tilt into position without compensation movement), then the subsequently declared **Feed rate?** parameter **F=** must still be defined.

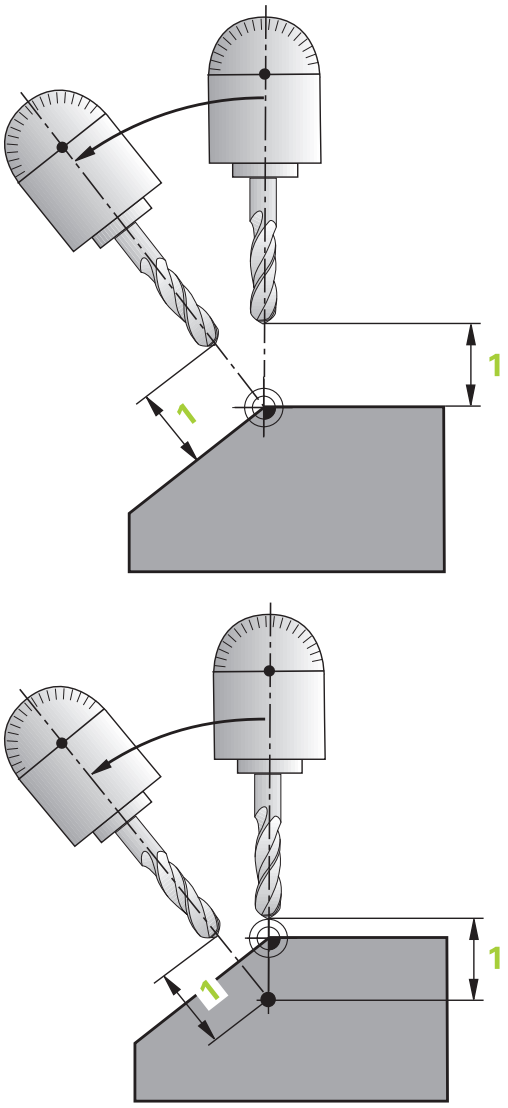
As an alternative to a feed rate **F** defined directly by a numerical value, you can also tilt the axes into position with **FMAX** (rapid traverse) or **FAUTO** (feed rate from the **T** block).

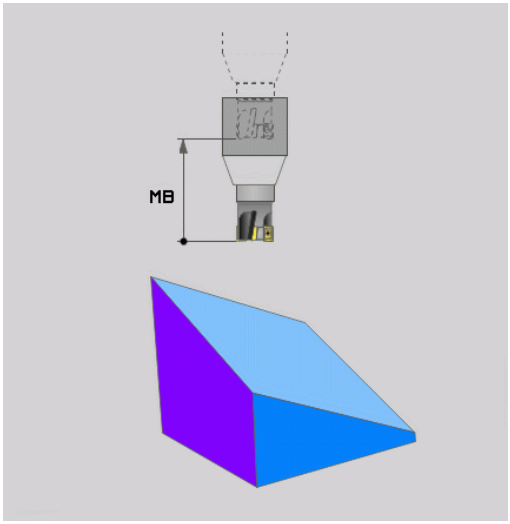
If you use **PLANE** together with **STAY,** you have to position the rotary axes in a separate block after the **PLANE** function.

- **Dist. tool tip - center of rot.** (incremental): The **DIST** parameter shifts the center of rotation of the tilting movement relative to the current position of the tool tip.
  - If the tool is already at the specified distance from the workpiece prior to being tilted into position, then it will be at the same relative position after being tilted into position (see center figure on the right, 1 = DIST)
  - If the tool is not at the specified distance from the workpiece before being tilted into position, then it will be offset relative to the original position after being tilted into position (see lower figure on the right, 1 = DIST)

> The control tilts the tool (or table) relative to the tool tip.

- **Feed rate? F=**: Contour speed at which the tool is to be tilted into position
- **Retraction length in the tool axis?**: The retraction path **MB** takes effect incrementally from the current tool position in the active tool axis direction that the control approaches **before tilting**. **MB MAX** moves the tool to a position just before the software limit switch





Tilting the rotary axes into position in a separate NC block

To tilt the rotary axes into position in a separate positioning block (**STAY** option selected):

NOTICE

Danger of collision!

The control does not automatically check whether collisions can occur between the tool and the workpiece. Incorrect or no pre-positioning before tilting the tool into position can lead to a risk of collision during the tilting movement!

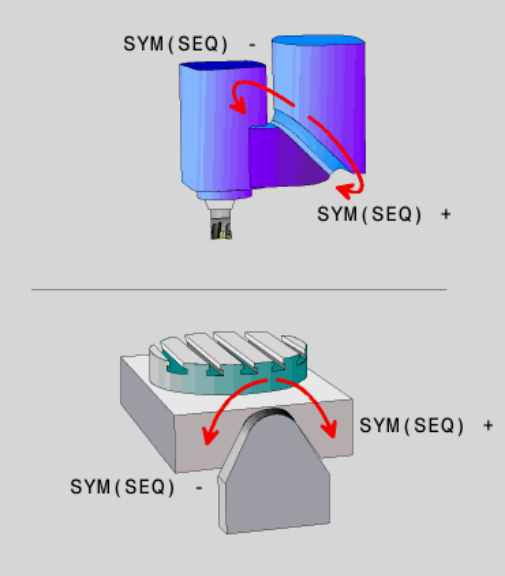
- Program a safe position before the tilting movement
- Carefully test the NC program or program section in the **Program run, single block** operating mode

- Select any **PLANE** function, and define automatic tilting into position with **STAY**. During program run, the control calculates the position values of the rotary axes present on the machine, and stores them in the system parameters **Q120** (A axis), **Q121** (B axis), and **Q122** (C axis)
- Define the positioning block with the angular values calculated by the control

Example: Tilt a machine with a rotary table C and a tilting table A to a spatial angle of B+45

...	
N10 G00 Z+250 G40*	Position at clearance height
N20 PLANE SPATIAL SPA+0 SPB+45 SPC+0 STAY*	Define and activate the PLANE function
N30 G01 A+Q120 C+Q122 F2000*	Position the rotary axis with the values calculated by the control.
...	Define machining in the tilted working plane

Selection of tilting possibilities SYM (SEQ) +/–



Based on the position that you have defined for the working plane, the control must calculate the appropriate position of the rotary axes present on your machine. In general, there are always two possible solutions.

For the selection of one of the possible solutions, the control offers two variants: **SYM** and **SEQ**. You use soft keys to choose the variants. **SYM** is the standard variant.

The entry of **SYM** or **SEQ** is optional.

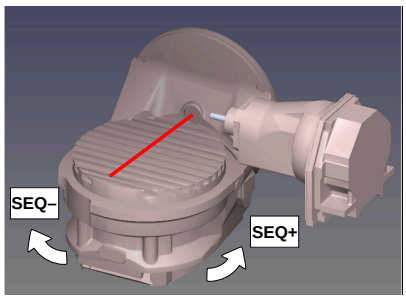
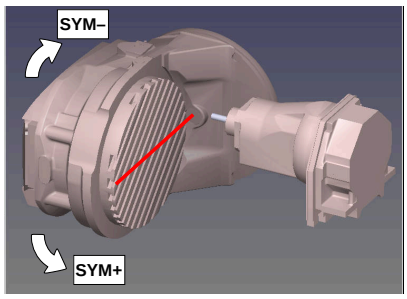
SEQ assumes that the master axis is in its home position (0°). Relative to the tool, the master axis is the first rotary axis, or the last rotary axis relative to the table (depending on the machine configuration). If both possible solutions are in the positive or negative range, then the control automatically uses the closer solution (shorter path). If you need the second possible solution, then you must either pre-position the master axis (in the area of the second possible solution) before tilting the working plane, or work with **SYM**.

As opposed to **SEQ**, **SYM** uses the symmetry point of the master axis as its reference. Every master axis has two symmetry positions, which are 180° apart from each other (sometimes only one symmetry position is in the traverse range).

To determine the symmetry point:

- Perform **PLANE SPATIAL** with any spatial angle and **SYM +**
- Save the axis angle of the master axis in a Q parameter (e.g., –80)
- Repeat the **PLANE SPATIAL** function with **SYM-**
- Save the axis angle of the master axis in a Q parameter (e.g., –100)
- Calculate the average value (e.g., –90)

  The average value corresponds to the symmetry point.

Reference for SEQ	Reference for SYM
	

With the **SYM** function, you select one of the possible solutions relative to the symmetry point of the master axis:

- **SYM+** positions the master axis in the positive half-space relative to the symmetry point
- **SYM-** positions the master axis in the negative half-space relative to the symmetry point

With the **SEQ** function, you select one of the possible solutions relative to the home position of the master axis:

- **SEQ+** positions the master axis in the positive tilting range relative to the home position
- **SEQ-** positions the master axis in the negative tilting range relative to the home position

If the solution you have selected with **SYM** (**SEQ**) is not within the machine's range of traverse, then the control displays the **Entered angle not permitted** error message.

If the **PLANE AXIAL** function is used, the **SYM** (**SEQ**) function has no effect.

If you do not define **SYM** (**SEQ**), then the control determines the solution as follows:

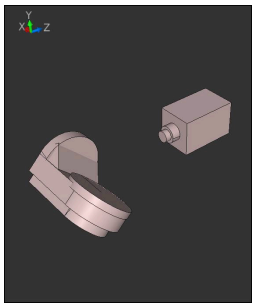
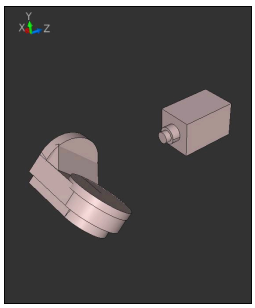
1. Check whether both possible solutions are within the traverse range of the rotary axes
2. Two possible solutions: Based on the current position of the rotary axes, choose the possible solution with the shortest path
3. One possible solution: Choose the only solution
4. No possible solution: Issue the error message **Entered angle not permitted**

Examples

Machine with C rotary axis and A tilting table.
Programmed function: PLANE SPATIAL SPA+0 SPB+45 SPC+0

Limit switch	Start position	SYM = SEQ	Resulting axis position
None	A+0, C+0	Not prog.	A+45, C+90
None	A+0, C+0	+	A+45, C+90
None	A+0, C+0	–	A–45, C–90
None	A+0, C–105	Not prog.	A–45, C–90
None	A+0, C–105	+	A+45, C+90
None	A+0, C–105	–	A–45, C–90
–90 < A < +10	A+0, C+0	Not prog.	A–45, C–90
–90 < A < +10	A+0, C+0	+	Error message
–90 < A < +10	A+0, C+0	–	A–45, C–90

Machine with B rotary axis and A tilting table (limit switches: A +180 and –100). Programmed function: PLANE SPATIAL SPA-45 SPB+0 SPC+0

SYM	SEQ	Resulting axis position	Kinematics view
+		A–45, B+0	
-		Error message	**No solution in limited range**
	+	Error message	**No solution in limited range**
	-	A–45, B+0	

The position of the symmetry point is contingent on the kinematics. If you change the kinematics (such as switching the head), then the position of the symmetry point changes as well.

Depending on the kinematics, the positive direction of rotation of **SYM** may not correspond to the positive direction of rotation of **SEQ**. Therefore, ascertain the position of the symmetry point and the direction of rotation of **SYM** on each machine before programming.

Selection of the transformation type

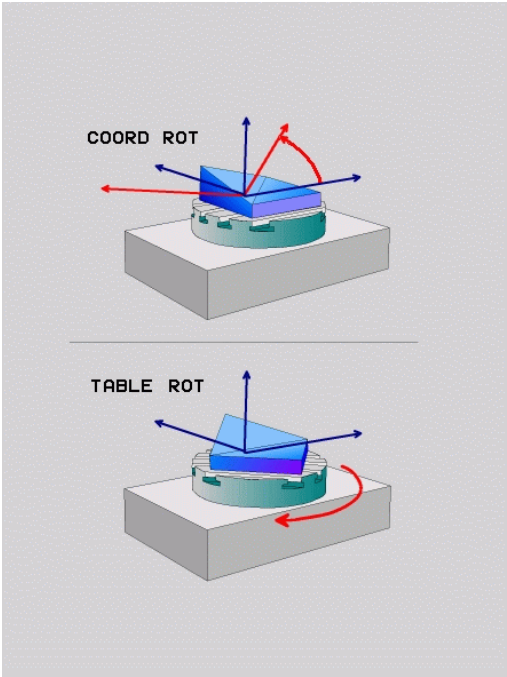
The **COORD ROT** and **TABLE ROT** transformation types influence the orientation of the working plane coordinate system through the axis position of a so-called free rotary axis.

The entry of **COORD ROT** or **TABLE ROT** is optional.

Any rotary axis becomes a free rotary axis with the following configuration:

- The rotary axis has no effect on the tool angle of inclination because the rotary axis and the tool axis are parallel in the tilting situation
- The rotary axis is the first rotary axis in the kinematic chain starting from the workpiece

The effect of the **COORD ROT** and **TABLE ROT** transformation types therefore depends on the programmed spatial angles and the machine kinematics.



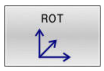
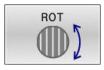
Programming notes:

- If no free rotary axis arises in a tilting situation, then the **COORD ROT** and **TABLE ROT** transformation types have no effect.
- With the **PLANE AXIAL** function, the **COORD ROT** and **TABLE ROT** transformation types have no effect.

Effect with a free rotary axis

Programming notes

- For the positioning behavior with the **COORD ROT** and **TABLE ROT** transformation types, it makes no difference whether the free rotary axis is a table axis or a head axis.
- The resulting axis position of the free rotary axis depends on an active basic rotation, among other factors.
- The orientation of the working plane coordinate system is also dependent on a programmed rotation (e.g., with Cycle **G73 ROTATION**).

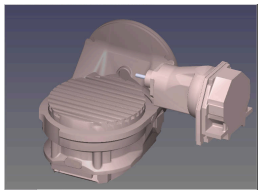
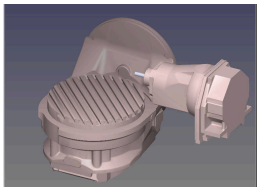
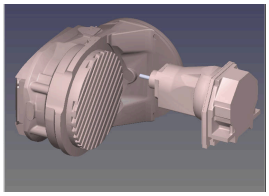
Soft key	Function
	**COORD ROT**: > The control positions the free rotary axis to 0 > The control orients the working plane coordinate system in accordance with the programmed spatial angle
	**TABLE ROT** with: ■ SPA **and** SPB **equal to** 0 ■ SPC **equal or unequal to** 0 > The control orients the free rotary axis in accordance with the programmed spatial angle > The control orients the working plane coordinate system in accordance with the basic coordinate system **TABLE ROT** with: ■ **At least** SPA **or** SPB **unequal to** 0 ■ SPC **equal or unequal to** 0 > The control does not position the free rotary axis. The position prior to tilting the working plane is maintained > Since the workpiece was not positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle

If no transformation type was selected, then the control uses the **COORD ROT** transformation type for the **PLANE** functions

Example

The following example shows the effect of the **TABLE ROT** transformation type in conjunction with a free rotary axis.

...	
N60 G00 B+45 R0*	Pre-position rotary axis
N70 PLANE SPATIAL SPA-90 SPB+20 SPC+0 TURN F5000 TABLE ROT*	Tilt the working plane
...	

Origin	A = 0, B = 45	A = –90, B = 45
		

- The control positions the B axis to the axis angle B+45
- With the programmed tilting situation with SPA–90, the B axis becomes the free rotary axis
- The control does not position the free rotary axis. The position of the B axis prior to the tilting of the working plane is maintained
- Since the workpiece was not also positioned, the control orients the working plane coordinate system in accordance with the programmed spatial angle SPB+20

Tilting the working plane without rotary axes

Refer to your machine manual.

This function must be enabled and adapted by the machine manufacturer.

The machine manufacturer must take the precise angle into account (e.g., the angle of a mounted angle head in the kinematics description).

You can also orient the programmed working plane perpendicularly to the tool without defining rotary axes (e.g., when adapting the working plane for a mounted angle head).

Use the **PLANE SPATIAL** function and the **STAY** positioning behavior to swivel the working plane to the angle specified by the machine manufacturer.

Example of mounted angle head with permanent tool direction **Y**:

Example

```
N110 T 5 G17 S4500*
N120 PLANE SPATIAL SPA+0 SPB-90 SPC+0 STAY*
```

The tilt angle must be precisely adapted to the tool angle, otherwise the control will generate an error message.

11.3 Miscellaneous functions for rotary axes

Feed rate in mm/min on rotary axes A, B, C: M116 (option 8)

Standard behavior

The control interprets the programmed feed rate of a rotary axis in degrees/min (in mm programs and also in inch programs). The feed rate therefore depends on the distance from the tool center to the center of the rotary axis.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116

Refer to your machine manual.

When working with angle heads, keep in mind that the machine geometry is defined by the machine manufacturer in a kinematics description. If you use an angle head during machining, then you must select the correct kinematics description.

Programming notes:

- The **M116** function can be used with table axes and head axes.
- The **M116** function also has an effect if the **Tilt working plane** function is active.

The control interprets the programmed feed rate of a rotary axis in mm/min (or 1/10 inch/min). In this case, the control calculates the feed for the block at the start of each NC block. The feed rate of a rotary axis will not change while the NC block is executed, even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane. Reset **M116** with **M117**. At the end of the program, **M116** is automatically canceled.

M116 becomes effective at the start of the block.

Shorter-path traverse of rotary axes: M126

Standard behavior

M126 has an effect only on modulo axes.

In the case of modulo axes, the axis position begins again at 0° after the modulo length of 0° to 360° has been exceeded. This is the case for rotary axes that are mechanically capable of endless rotation.

In the case of non-modulo axes, the maximum rotation is mechanically limited. The position display of the rotary axis does not switch back to the starting value (e.g., 0° to 540°).

Refer to your machine manual.

The positioning behavior of rotary axes is machine-dependent.

In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether a rotary axis is a modulo axis.

In the optional machine parameter **shortestDistance** (no. 300401), the machine manufacturer defines whether the control by default positions the rotary axis using the shortest traverse path. If the traverse paths in both directions are identical, you can pre-position the rotary axis and thus also influence the direction of rotation. Within the **PLANE** functions, you can also select a tilting solution.

Further information: "Selection of tilting possibilities SYM (SEQ) +/–", Page 428

Behavior without M126:

Without **M126**, the control moves a rotary axis whose position display is reduced to less than 360° along a long path.

Examples:

Actual position	Target position	Traverse distance
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With **M126**, the control moves a rotary axis whose position display is reduced to less than 360° on the shortest path of traverse.

Examples:

Actual position	Target position	Traverse distance
350°	10°	+20°
10°	340°	–30°

Effect

M126 takes effect at the start of the block.

M127 and a program end reset **M126**.

Reducing display of a rotary axis to a value less than 360°: M94

Standard behavior

M94 only affects rollover axes whose actual position display permits values above 360°.

The control moves the tool from the current angular value to the programmed angular value.

Refer to your machine manual.

In the machine parameter **isModulo** (no. 300102) the machine manufacturer defines whether the modulo counting method is used for a rollover axis.

In the optional machine parameter **shortestDistance** (no. 300401), the machine manufacturer defines whether the control by default positions the rotary axis using the shortest traverse path. If the traverse paths in both directions are identical, you can pre-position the rotary axis and thus also influence the direction of rotation. Within the **PLANE** functions, you can also select a tilting solution.

Further information: "Selection of tilting possibilities SYM (SEQ) +/–", Page 428

Example:

Current angular value:	538°
Programmed angular value:	180°
Actual distance of traverse:	-358°

Behavior with M94

At the start of block, the control first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If multiple rotary axes are active, **M94** will reduce the display of all rotary axes. As an alternative, you can specify a rotary axis after **M94**. The control then reduces the display of this axis only.

If you entered a traverse limit or a software limit switch is active, **M94** is ineffective for the corresponding axis.

N210 M94*	; Reduce the display values of all active rotary axes
N210 M94 C*	; Reduce the display value of the C axis
M110 G00 C+180 M94*	; Reduce the display values of all active rotary axes and then move in the C axis to the programmed value

Effect

M94 is effective only in the NC block where it is programmed.

M94 becomes effective at the start of the block.

Selecting tilting axes: M138

Standard behavior

With the function **Tilt working plane**, the control considers those rotary axes that have been specified by the machine manufacturer in the machine parameters.

Behavior with M138

The control performs the above functions only in those tilting axes that you have defined using **M138**.

Refer to your machine manual.

If you restrict the number of rotary axes with the **M138** function, your machine may provide only limited tilting possibilities. The machine manufacturer decides whether the control takes the angles of deselected axes into account or sets them to 0.

Effect

M138 becomes effective at the start of the block.

You can cancel **M138** by reprogramming it without specifying any axes.

Example

Perform the above-mentioned functions only in the tilting axis C.

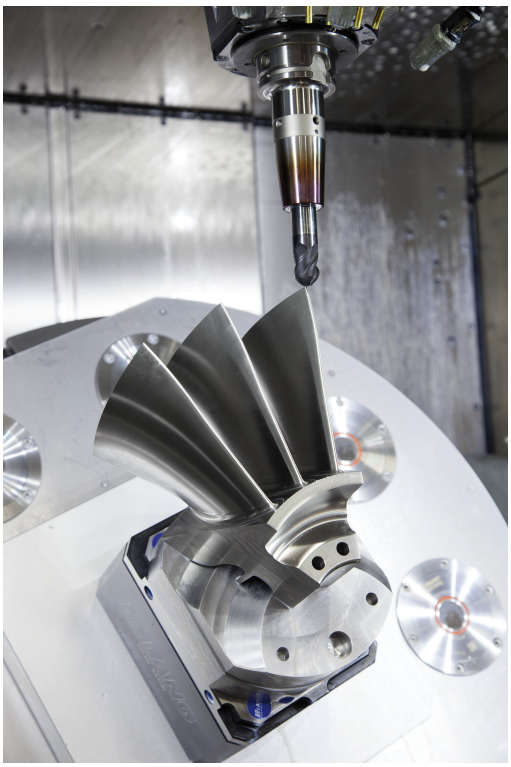
```
N110 G00 Z+100 G40 M138 C*  ; Define that the C axis should be taken into account
```

11.4 Running CAM programs

If you create NC programs externally using a CAM system, you should pay attention to the recommendations detailed below. This will enable you to optimally use the powerful motion control functionality provided by the control and usually create better workpiece surfaces with shorter machining times. Despite high machining speeds, the control still achieves a very high contour accuracy. The basis for this is the HEROS 5 real-time operating system in conjunction with the **ADP** (Advanced Dynamic Prediction) function of the TNC 320. This enables the control to also efficiently process NC programs with high point densities.

From 3-D model to NC program

Here is a simplified description of the process for creating an NC program from a CAD model:



- **CAD: Model creation**
  Construction departments prepare a 3-D model of the workpiece to be machined. Ideally the 3-D model is designed for the center of tolerance.
- **CAM: Path generation, tool compensation**
  The CAM programmer specifies the machining strategies for the areas of the workpiece to be machined. The CAM system uses the surfaces of the CAD model to calculate the paths of the tool movements. These tool paths consist of individual points calculated by the CAM system so that each surface to be machined is approximated as nearly as possible while considering chord errors and tolerances. This way, a machine-neutral NC program is created, known as a CLDATA file (cutter location data). A postprocessor generates a machine- and control-specific NC program, which can be processed by the CNC control. The postprocessor is adapted according to the machine tool and the control. The postprocessor is the link between the CAM system and the CNC control.

  > In the **BLK FORM FILE** syntax, you can integrate 3-D models in STL format as a workpiece blank and a finished part.
  > **Further information:** "Defining the workpiece blank: G30/G31", Page 101

- **Control: Motion control, tolerance monitoring, velocity profile**
  The control uses the points defined in the NC program to calculate the movements of each machine axis as well as the required velocity profiles. Powerful filter functions then process and smooth the contour so that the control does not exceed the maximum permissible path deviation.
- **Mechatronics: Feed control, drive technology, machine tool**
  The motions and velocity profiles calculated by the control are realized as actual tool movements by the machine's drive system.

Considerations required for post processor configuration

Take the following points into account with post processor configuration:

- Always set the data output for axis positions to at least four decimal places. This way you improve the quality of the NC data and avoid rounding errors, which can result in defects visible to the naked eye on the workpiece surface. Output of five decimal places may achieve improved surface quality for optical components as well as components with very large radii (i.e. small curvatures), for example molds for the automotive industry
- Always set the data output for the machining of surface normal vectors (LN blocks, only Klartext conversational programming) to exactly seven decimal places
- Avoid using successive incremental NC blocks because this may lead to the tolerances of the individual NC blocks being added together in the output
- Set the tolerance in Cycle **G62** so that in standard behavior it is at least twice as large as the chord error defined in the CAM system. Also note the information describing the functioning of Cycle **G62**
- If the chord error selected in the CAM program is too large, then, depending on the respective curvature of a contour, large distances between NC blocks can result, each with large changes of direction. During machining this leads to drops in the feed rate at the block transitions. Recurring and equal accelerations (i.e. force excitation), caused by feed-rate drops in the heterogeneous NC program, can lead to undesirable excitation of vibrations in the machine structure.
- You can also use arc blocks instead of linear blocks to connect the path points calculated by the CAM system. The control internally calculates circles more accurately than can be defined via the input format
- Do not output any intermediate points on exactly straight lines. Intermediate points that are not exactly on a straight line can result in defects visible to the naked eye on the workpiece surface
- There should be exactly one NC data point at curvature transitions (corners)
- Avoid sequences of many short block paths. Short paths between blocks are generated in the CAM system when there are large curvature transitions with very small chord errors in effect. Exactly straight lines do not require such short block paths, which are often forced by the continuous output of points from the CAM system
- Avoid a perfectly even distribution of points over surfaces with a uniform curvature, since this could result in patterns on the workpiece surface
- For 5-axis simultaneous programs: avoid the duplicated output of positions if they only differ in the tool's angle of inclination
- Avoid the output of the feed rate in every NC block. This would negatively influence the control's velocity profile

- If a subprogram call and a subprogram definition are separated by multiple NC blocks, program execution might be interrupted due to the calculation effort. Use the following options to avoid problems such as dwell marks due to interruptions:
  - Put subprograms that define retraction positions at the beginning of the program. Thus, the control "knows" where to find the subprogram when it is called later.
  - Use a separate NC program for machining positions or coordinate transformations. This ensures that the control simply needs to call that program when safety positions and coordinate transformations are required in the NC program.

Useful configurations for the machine tool operator:

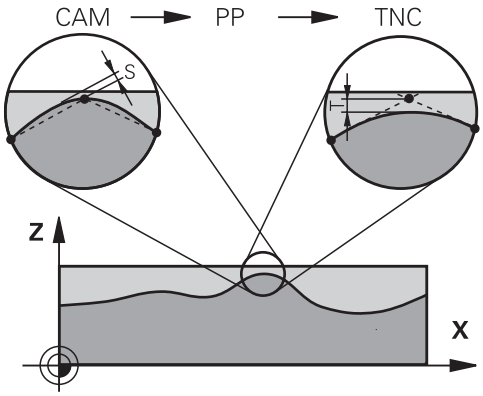
- In order to enable a realistic graphic simulation, use 3-D models in STL format as a workpiece blank and finished part
  Further information: "Defining the workpiece blank: G30/G31", Page 101
- In order to improve the structure of large NC programs, use the control's structuring function
  Further information: "Structuring NC programs", Page 209
- Use the control's commenting function in order to document NC programs
  Further information: "Adding comments", Page 205
- Use the comprehensive cycles of the control available for the machining of holes and simple pocket geometries
  Further information: User's Manual for **Programming of Machining Cycles**
- For fits, output the contours with **RL/RR** tool radius compensation. This makes it easy for the machine operator to make necessary compensations
  Further information: "Tool compensation", Page 145
- Separate feed rates for pre-positioning, machining, and downfeeds, and define them via Q parameters at the beginning of the program

Please note the following for CAM programming

Adapting chord errors

Programming notes:

- For finishing operations, do not set the chord error in the CAM system to a value greater than 5 µm. In Cycle **G62**, use an appropriate tolerance factor **T** of 1.3 to 3.
- For roughing operations, the total of the chord error and the tolerance **T** must be less than the defined machining oversize. In this way you can avoid contour damage.
- The specific values depend upon the dynamics of your machine.



Adapt the chord error in the CAM program, depending on the machining:

- **Roughing with preference for speed:**
  Use higher values for the chord error and the matching tolerance value in Cycle **G62**. Both values depend on the oversize required on the contour. If a special cycle is available on your machine, use the roughing mode. In roughing mode the machine generally moves with high jerk values and high accelerations
  - Normal tolerance in Cycle **G62**: Between 0.05 mm and 0.3 mm
  - Normal chord error in the CAM system: Between 0.004 mm and 0.030 mm
- **Finishing with preference for high accuracy:**
  Use smaller values for the chord error and a matching low tolerance in Cycle **G62**. The data density must be high enough for the control to detect transitions and corners exactly. If a special cycle is available on your machine, use the finishing mode. In finishing mode the machine generally moves with low jerk values and low accelerations
  - Normal tolerance in Cycle **G62**: Between 0.002 mm and 0.006 mm
  - Normal chord error in the CAM system: Between 0.001 mm and 0.004 mm
- **Finishing with preference for high surface quality:**
  Use small values for the chord error and a matching larger tolerance in Cycle **G62**. The control is then able to better smooth the contour. If a special cycle is available on your machine, use the finishing mode. In finishing mode the machine generally moves with low jerk values and low accelerations
  - Normal tolerance in Cycle **G62**: Between 0.010 mm and 0.020 mm
  - Normal chord error in the CAM system: Approx. 0.005 mm

Further adaptations

Take the following points into account with CAM programming:

- For slow machining feed rates or contours with large radii, define the chord error to be only one-third to one-fifth of tolerance **T** in Cycle **G62**. Additionally, define the maximum permissible point spacing to be between 0.25 mm and 0.5 mm. The geometry error or model error should also be specified to be very small (max. 1 µm).
- Even at higher machining feed rates, point spacings of greater than 2.5 mm are not recommended for curved contour areas
- For straight contour elements, one NC point at the beginning of a line and one NC point at the end suffice. Avoid the output of intermediate positions
- In programs with five axes moving simultaneously, avoid large changes in the ratio of path lengths in linear and rotational blocks. Otherwise large reductions in the feed rate could result at the tool reference point (TCP)
- NC programs for 5-axis simultaneous machining with spherical cutters should preferably be output for the center of the sphere. The NC data are then generally more uniform. In Cycle **G62**, you can additionally set a higher rotary axis tolerance **TA** (e.g., between 1° and 3°) for an even more constant feed-rate curve at the tool center point (TCP).
- For NC programs for 5-axis simultaneous machining with toroid cutters or spherical cutters, where the NC output is for the south pole of the sphere, choose a lower rotary axis tolerance. 0.1° is a typical value. However, the maximum permissible contour damage is the decisive factor for the rotational axis tolerance. This contour damage depends on the possible tool tilting, tool radius and engagement depth of the tool.
  With 5-axis hobbing with an end mill, you can calculate the maximum possible contour damage T directly from the cutter engagement length L and permissible contour tolerance TA:
  T ~ K x L x TA with K = 0.0175 [1/°]
  Example: L = 10 mm, TA = 0.1°: T = 0.0175 mm

Possibilities for intervention on the control

Cycle **G62 TOLERANCE** is available for influencing the behavior of CAM programs directly on the control. Please note the information describing the functioning of Cycle **G62**. Also note the interactions with the chord error defined in the CAM system.

Further information: User's Manual for **Programming of Machining Cycles**

Refer to your machine manual.

Some machine manufacturer provide an additional cycle for adapting the behavior of the machine to the respective machining operation (e.g., Cycle **G332** Tuning). Cycle **G332** can be used to modify filter settings, acceleration settings, and jerk settings.

Example

```
N340 G62 T0.05 P01 1 P02 3*
```

ADP motion control

This function must be enabled and adapted by the machine manufacturer.

An insufficient quality of data in NC programs created on CAM systems frequently causes inferior surface quality of the milled workpieces. The **ADP** (Advanced Dynamic Prediction) feature expands the conventional look-ahead of the permissible maximum feed rate profile and optimizes the motion control of the feed axes during milling. This enables clean surfaces with short machining times to be cut, even with a strongly fluctuating distribution of points in adjacent tool paths. This significantly reduces or eliminates the reworking complexity.

These are the most important benefits of ADP:

- Symmetrical feed-rate behavior on forward and backward paths with bidirectional milling
- Uniform feed rate curves with adjacent cutter paths
- Improved reaction to negative effects (e.g. short, step-like contours, coarse chord tolerances, heavily rounded block end-point coordinates) in NC programs generated by CAM systems
- Precise compliance to dynamic characteristics even in difficult conditions

12

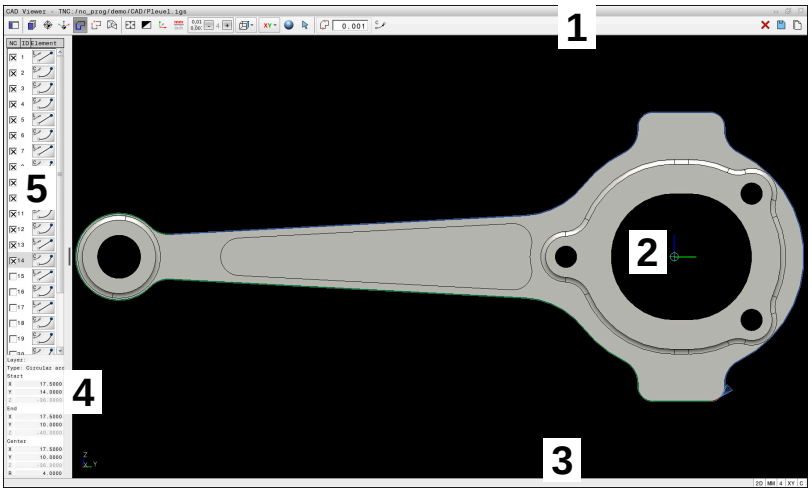
Data Transfer from CAD Files

12.1 Screen layout of CAD Viewer

CAD Viewer fundamentals

Screen display

When you open **CAD Viewer**, the following screen layout is displayed:



1 Menu bar
2 Graphics area
3 Status bar
4 Element information area
5 List View area

File types

CAD Viewer supports the following standard file types that can be opened directly in the control:

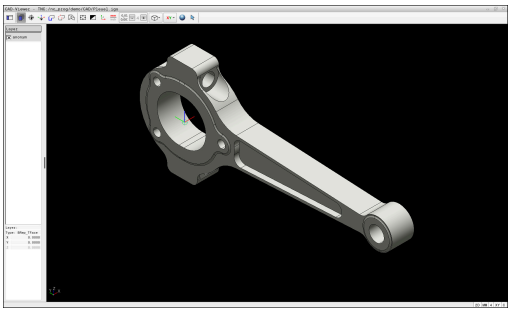
File type	Extension	Format
STEP	*.stp and *.step	■ AP 203 ■ AP 214
IGES	*.igs and *.iges	■ Version 5.3
DXF	*.dxf	■ R10 to 2015 ■ ASCII
STL	*.stl	■ Binary ■ ASCII

CAD Viewer allows you to open CAD files consisting of any number of triangles.

12.2 CAD Import (option 42)

Application

If the control is set to ISO, the extracted contours or machining positions are nevertheless output as Klartext programs in **.H** conversational format.



You can open CAD files directly on the control to extract contours or machining positions from them. You can then store them as Klartext programs or as point files. Klartext programs acquired in this manner can also be run on older HEIDENHAIN controls, since these contour programs by default contain only **L** and **CC**/**C** blocks.

As an alternative to **CC** or **C** blocks, you can configure circular movements to be output as **CR** blocks.

Further information: "Basic settings", Page 449

If you process files in **Programming** mode, then the control generates contour programs with the file extension **.H** and point files with the extension **.PNT** by default. You can select the file type in the Save dialog.

To insert a selected contour or a selected machining position directly into an NC program, use the control's clipboard. Using the clipboard, you can even transfer the contents to additional software tools (e.g., **Leafpad** or **Gnumeric**).

Operating notes:

- Contents from the clipboard can only be inserted into additional software tools as long as **CAD Viewer** is open.
- Before loading the file into the control, ensure that the name of the file contains only permitted characters.
  Further information: "File names", Page 117

Using CAD Viewer

To use **CAD Viewer** , you have to use a mouse or touchpad.

CAD Viewer runs as a separate application on the third desktop of the control. This enables you to use the screen switchover key to switch between the machine operating modes, the programming modes, and **CAD Viewer**. This is particularly useful if you want to add contours or machining positions to a Klartext program using the clipboard.

Opening a CAD file

- Press the **Programming** key

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

- Press the **SELECT TYPE** soft key
- The control displays the selectable file types.

- Press the **SHOW CAD** soft key
- Alternative: Press the **SHOW ALL** soft key

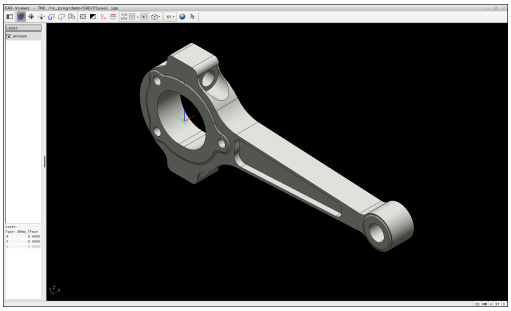
- Select the directory in which the CAD file is saved

- Select the desired CAD file

- Press the **ENT** key
- The control starts **CAD Viewer** and shows the file contents on the screen. The control displays the layers in the List View area and the drawing in the Graphics area.

Basic settings

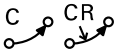
The basic settings specified below can be selected using the icons in the menu bar.



Icon	Setting
	Show sidebar Show, enlarge, or hide the List View area
	Display the layer Display the layer(s) in the List View area **Further information:** "Setting layers", Page 451
	Preset Define the workpiece preset Workpiece preset has been defined Delete the defined workpiece preset **Further information:** "Setting a preset", Page 453
	Datum Set the datum Datum has been set **Further information:** "Setting the datum", Page 455
	Contour Select contour (option 42) **Further information:** "Selecting and saving a contour", Page 459
	Positions Select positions (option 42) **Further information:** "Selecting and saving machining positions", Page 463
	3D mesh Create a 3D mesh (option 152) **Further information:** "Generating STL files with 3D mesh (option 152)", Page 467
	Show all Set the zoom to the largest possible rendering of the entire graphical representation
	Inverted colors Toggle background color (black or white)
	Toggle between 2D and 3D mode. The active mode is highlighted in color
mm inch	Set the unit of measure (**mm** or **inch**) for the file. The control then outputs the contour program and the machining positions in this unit of measure. The active unit of measure is highlighted in red. **CAD Viewer** performs all internal calculations in mm. If you select the inch unit of measure, the **CAD Viewer** converts all values to inches.

Icon	Setting
0.01 0.001	**Number of decimal places** Select the resolution. The resolution defines the number of decimal places and the number of positions for linearization. Default setting: 4 decimal places with **mm**, and 5 decimal places with **inch** as the unit of measure **Further information:** "Selecting and saving a contour", Page 459
	Set perspective Switch between various views of the model (e.g., **Top**)
XY	**Axes** Select the working plane: ■ **XY** ■ **YZ** ■ **ZX** ■ **ZXØ** If you take over a contour or position, the control outputs the NC program in the selected working plane. **Further information:** "Selecting and saving a contour", Page 459
	Toggle a 3D model between a solid model and a wire-frame model.
   	"Select, add, or remove contour elements" mode **i** The icon shows the current mode. Clicking the icon activates the next mode.

The control displays the following icons only in certain modes.

Icon	Setting
	Undoes the most recent step.
	Contour transfer mode: The tolerance specifies how far apart neighboring contour elements may be from each other. You can use the tolerance to compensate for inaccuracies that occurred during drawing creation. The default setting is 0.001 mm.
	Arc mode: You can select whether the control will output circular contours **C** or **CR** in the NC program.
	Point transfer mode: The control hides and displays the tool paths between the positions.

Icon	Setting
	Path optimization mode: The control optimizes the tool traverse movement between the machining positions. When you select the icon again, the control will discard the optimization.
	Machining positions mode: The control opens the **Find circle centers by diameter range** window. You can filter the displayed data by diameter or depth values.

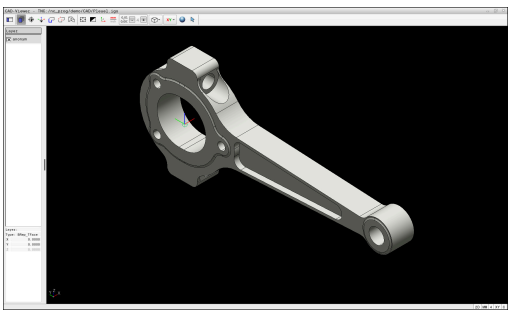
Operating notes:

- Set the correct unit of measure so that **CAD Viewer** shows the correct values.
- When creating NC programs for previous control versions, you must limit the resolution to three decimal places. In addition, the comments that **CAD Viewer** outputs to the contour program need to be removed.
- The control displays the active basic settings in the status bar of the screen.

Setting layers

CAD files usually contain multiple layers. The designer uses these layers to create groups of various types of elements, such as the actual workpiece contour, dimensions, auxiliary and design lines, hatching, and texts.

Hiding unneeded layers makes the graphics easier to read and facilitates the extraction of the required information.



Operating notes:

- The CAD file to be processed must contain at least one layer. The control automatically moves all elements not assigned to a layer to the "anonymous" layer.
- If the name of the layer is not shown completely in the List View area, you can use the **Show sidebar** icon to enlarge this area.
- You can even select a contour if the designer has saved it on different layers.
- If you double-click a layer, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of the contour.

When you open a CAD file in **CAD Viewer**, all available layers are shown.

Hiding a layer

To hide a layer:

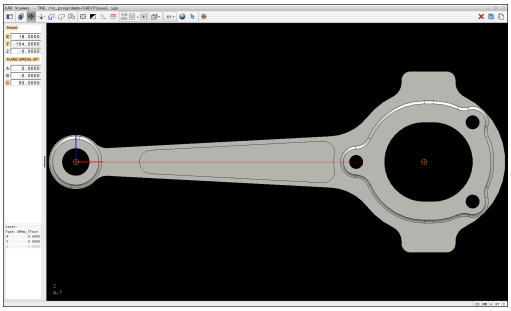
- Select **SET LAYER**
- > In the List View area, the control shows all layers contained in the active CAD file.
- Select the desired layer
- Click the checkbox to deactivate it
- Alternatively, press the spacebar
- > The control hides the selected layer.

Showing a layer

To show a layer:

- Select **SET LAYER**
- > In the List View area, the control shows all layers contained in the active CAD file.
- Select the desired layer
- Click the checkbox to activate it
- Alternatively, press the spacebar
- > The control marks the selected layer in the List View with an ×.
- > The selected layer is shown.

Setting a preset



The datum of the drawing in the CAD file is not always located in a manner that lets you use it as a workpiece preset. Therefore, the control provides a function with which you can shift the workpiece preset to a suitable location by clicking an element. You can also define the orientation of the coordinate system.

You can position the preset at the following locations:

- By direct numerical input in the List View area
- For lines:
  - Starting point
  - Midpoint
  - End point
- For circular arcs:
  - Starting point
  - Midpoint
  - End point
- For full circles:
  - At the quadrant transitions
  - At the center
- At the intersection between:
  - Two lines, even if the point of intersection is actually on the extension of one of the lines
  - Line and circular arc
  - Line and full circle
  - Two circles (regardless of whether a circular arc or a full circle)

Operating note:

You can change the preset even after you have selected the contour. The control does not calculate the actual contour data until you save the selected contour in a contour program.

NC syntax

The preset and optional orientation are inserted in the NC program as a comment starting with **origin**.

```
4 ;orgin = X... Y... Z...
5 ;orgin_plane_spatial = SPA... SPB... SPC...
```

You can save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import (option 42) is not available.

Setting a preset on a single element

To set a preset on a single element:

- ▶ Select the mode for setting a preset
- ▶ Place the mouse pointer on the desired element
- > The control indicates possible locations for the preset on the selected element with stars.
- ▶ Select the star symbol that matches the desired preset position
- ▶ If necessary, use the zoom function
- > The control sets the preset symbol at the selected location.
- ▶ In addition, align the coordinate system as needed **Further information:** "Orienting the working plane", Page 455

Setting a preset at the intersection of two elements

To set a preset at the intersection of two elements:

- ▶ Select the mode for setting a preset
- ▶ Select the first element (line, full circle, or circular arc) using the left mouse button
- > The control highlights the element.
- ▶ Select the second element (line, full circle, or circular arc) using the left mouse button
- > The control sets the preset symbol at the point of intersection.
- ▶ In addition, align the coordinate system as needed **Further information:** "Orienting the working plane", Page 455

Operating notes:

- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Once a preset has been set, the control displays the preset icon with a yellow quadrant .

Use the following icon to delete a preset that has been set .

Orienting the working plane

The following requirements must be met in order to orient the working plane:

- Preset has been defined
- There are elements next to the preset that can be used for the desired orientation

The orientation of the working plane is defined by the orientation of the axes.

To orient the working plane:

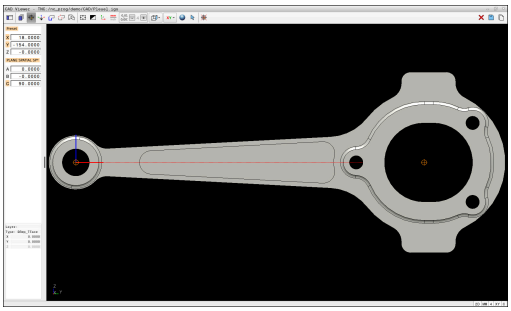
- ▶ Select an element located in the positive X direction using the left mouse button
- > The control orients the X axis.
- > The control changes the angle in C.
- ▶ Select an element located in the positive Y direction using the left mouse button
- > The control orients the Y and Z axes.
- > The control changes the angles in A and C.

For angles not equal to 0, the control shows the List View in orange.

Element information

The control displays the following element information at the left of the area:

- Distance between the defined preset and the drawing datum
- Orientation of the working plane

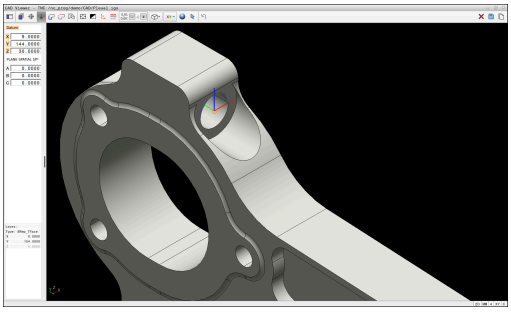


Setting the datum

The workpiece preset is not always located in a manner that lets you machine the entire part. Therefore, the control has a function with which you can define a new datum and a working plane.

The datum with the orientation of the working plane can be set at the same positions as a preset.

Further information: "Setting a preset", Page 453



NC syntax

The datum and its optional orientation can be inserted as NC block or comments in the NC program by using the **TRANS DATUM AXIS** function for the datum and the **PLANE SPATIAL** function for the orientation.

If you define only one datum and its orientation, then the control inserts the functions in the NC program as an NC block.

```
4 TRANS DATUM AXIS X... Y... Z...
5 PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

If you additionally select contours or points, then the control inserts the functions in the NC program as comments.

```
4 ;TRANS DATUM AXIS X... Y... Z...
5 ;PLANE SPATIAL SPA... SPB... SPC... TURN MB MAX FMAX
```

You can save the workpiece preset and workpiece datum information to a file or to the clipboard, even when the software option CAD Import (option 42) is not available.

Setting the datum on a single element

To set the datum on a single element:

- ▶ Select the mode for specifying the datum
- ▶ Place the mouse pointer on the desired element
- > The control indicates possible locations for the datum on the selected element with stars.
- ▶ Select the star symbol that matches the desired datum position
- ▶ If necessary, use the zoom function
- > The control sets the datum icon at the selected location.
- ▶ In addition, align the coordinate system as needed **Further information:** "Adjusting the orientation of the coordinate system", Page 457

Setting a datum at the intersection between two elements

To set a datum at the intersection between two elements:

- ▶ Select the mode for specifying the datum
- ▶ Select the first element (line, full circle, or circular arc) using the left mouse button
- > The control highlights the element.
- ▶ Select the second element (line, circle, or circular arc) using the left mouse button
- > The control sets the datum icon on the point of intersection.
- ▶ In addition, align the coordinate system as needed **Further information:** "Adjusting the orientation of the coordinate system", Page 457

Operating notes:

- If there are several possible intersections, the control selects the intersection nearest the mouse-click on the second element.
- If two elements do not intersect directly, the control automatically calculates the intersection of their extensions.
- If the control cannot calculate an intersection, it deselects the previously selected element.

Once a datum has been set, the control displays the datum icon with a yellow surface: .

Use this icon to delete a datum that has been set: .

Adjusting the orientation of the coordinate system

The following conditions must be met in order to align the coordinate system:

- The datum has been set
- There are elements next to the preset that can be used for the desired alignment

The position of the coordinate system is defined by the orientation of the axes.

To align the coordinate system:

- ▶ Select an element located in the positive X direction using the left mouse button
- > The control aligns the X axis.
- > The control changes the angle in C.
- ▶ Select an element located in the positive Y direction using the left mouse button
- > The control aligns the Y and Z axes.
- > The control changes the angles in A and C.

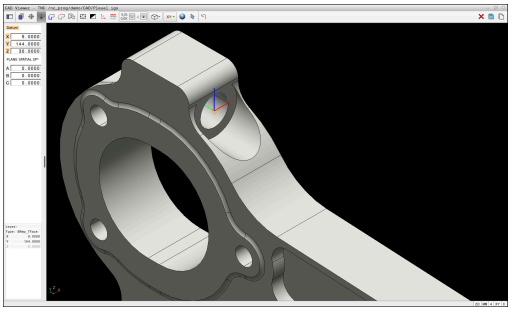
For angles not equal to 0, the control shows the List View in orange.

Element information

In the Element Information area, the control displays how far away the datum you selected is located from the workpiece preset.

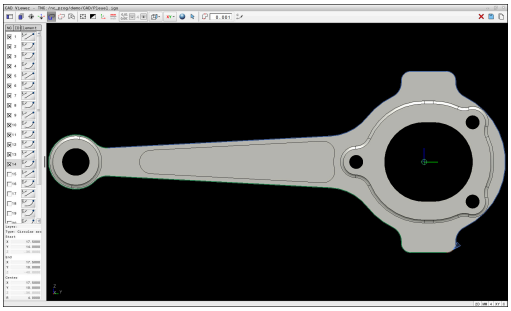
In the Element Information area on the left, the control displays:

- Distance between the datum that has been set and the workpiece preset
- Orientation of the working plane



You can further shift the datum manually after it has been set. To do so, enter the desired axis values into the coordinate field.

Selecting and saving a contour



Operating notes:
- This function is not available if option 42 is not enabled.
- Specify the direction of rotation during contour selection so that it matches the desired machining direction.
- Select the first contour element such that approach without collision is possible.
- If the contour elements are very close to one another, use the zoom function.

The following elements can be selected as a contour:
- Line segment
- Full circle
- Pitch circle
- Polyline
- Any curves (e.g., splines, ellipses)

Linearization

CAD Viewer linearizes all of the contours that are not in the working plane.

During linearization, **CAD Viewer** subdivides a contour into individual segments. From these segments, CAD Import creates straight lines **L** and circular arcs **C** or **CR** that are as long as possible.

Thanks to linearization, it is also possible to import contours with CAD Import that cannot be programmed with the path functions of the control, such as splines.

The higher you define the resolution by specifying decimal places, the lower is the deviation from the imported contour.

Further information: "Basic settings", Page 449

You can prevent the linearization, for example of circles that are not in the working plane. Select the working plane in which the circle has been defined.

Element information

In the Element Information area, the control displays various pieces of information about the contour element selected last in the List View area or in the Graphics area.
- **Layer**: specifies the active plane
- **Type**: specifies the type of element (e.g., line)
- **Coordinates**: specify the starting point and end point of an element, and the circle center and radius where appropriate

Ensure that the unit of measure used in the NC program matches that used in **CAD Viewer**. Elements that have been copied from **CAD Viewer** to the clipboard do not contain any information about the unit of measure.

Select contour

Operating note:
If you double-click a layer in the List View area, the control switches to Contour Transfer mode and selects the first contour element that was drawn. The control highlights the other selectable elements of this contour in green. Especially in case of contours with many short elements, this procedure spares you the effort of running a manual search for the beginning of a contour.

To select a contour using available contour elements:

- Select the contour selection mode
- Place the mouse pointer on the desired element
- > The control displays the suggested direction of rotation as a dashed line.
- If you need to change the direction of rotation, move the mouse pointer towards the opposite end point
- Select the element using the left mouse button
- > The selected contour element turns blue.
- > The control shows the other selectable elements in green.

For branched contours, the control chooses the path with the smallest directional deviation. The control provides an additional mode that allows you to modify the suggested contour path.

Further information: "Creating contour paths independent of available contour elements", Page 462

- Select the last green element of the desired contour using the left mouse button
- > The control changes the color of all selected elements to blue.
- > In the List View, all selected elements are given a check mark in the column **NC**.

Saving a contour

Operating notes:

- The control also transfers two workpiece-blank definitions (**BLK FORM**) to the contour program. The first definition contains the dimensions of the entire CAD file. The second one, which is the active one, contains only the selected contour elements, so that an optimized size of the workpiece blank results.
- The control only saves elements that have been selected (elements highlighted in blue), which means that they have been given a check mark in the List View area.

To save a selected contour:

- ▶ Select the Save icon
- > The control prompts you to select the target directory, a file name, and the file type.
- ▶ Enter this information

ENT

- ▶ Confirm your input
- > The control saves the contour program.

- ▶ Alternative: Copy the selected contour elements to the clipboard

Ensure that the unit of measure used in the NC program matches that used in **CAD Viewer**. Elements that have been copied from **CAD Viewer** to the clipboard do not contain any information about the unit of measure.

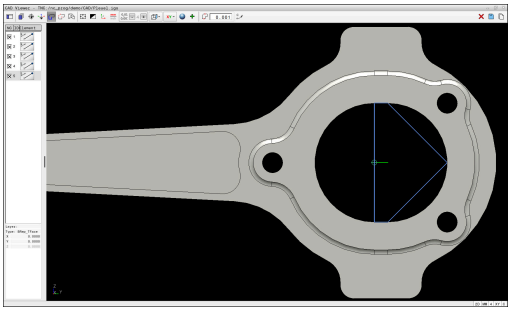
Deselecting the contour

To deselect the selected contour elements:

- ▶ Select the Clear function to deselect all elements
- ▶ Alternative: Select individual elements by clicking them with the left mouse button while holding the **CTRL** key

Creating contour paths independent of available contour elements

To select any contours by using the end point, center, or transition points:



- Select the contour selection mode

- Activate the "Add contour elements" mode
- The control displays the following icon:
  +
- Place the mouse pointer on the contour element
- The control displays selectable points.

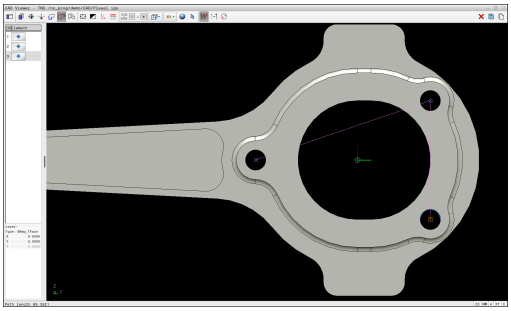
> Selectable points:
> - End point or midpoint of a line or curve
> - Quadrant transitions or center of a circle
> - Points of intersection between existing elements

- Select the starting point as needed
- Select the starting element
- Select the subsequent element
- Alternative: Select any selectable point
- The control creates the desired contour path.

Operating notes:

- The contour paths available depend on the selectable contour elements that are shown in green. Without the green elements, the control will display all solutions available. To remove the proposed contour path, select the first green element by pressing the left mouse button while holding the **CTRL** key down.
  As an alternative, you can switch to the Remove mode:
  –
- If the contour element to be extended or shortened is a straight line, the control will extend or shorten the contour element along the same line. If the contour element to be extended or shortened is a circular arc, the control will extend or shorten the contour element along the same arc.

Selecting and saving machining positions



Operating notes:
- This function is not available if option 42 is not enabled.
- If the contour elements are very close to one another, use the zoom function.
- If required, configure the basic settings so that the control shows the tool paths. **Further information:** "Basic settings", Page 449

Three possibilities are available in the pattern generator for defining machining positions:
- Single selection: You select the desired machining positions by clicking them individually with the mouse
  Further information: "Single selection", Page 464
- Multiple selection by drag box: You select multiple machining positions by dragging a box around them with the mouse
  Further information: "Multiple selection by drag box", Page 464
- Multiple selection by search filter: You select all machining positions within a definable diameter range
  Further information: "Multiple selection by search filter", Page 464

- Machining positions are deselected, deleted, or saved in the same manner as contour elements.
- **CAD Viewer** also recognizes circles as machining positions that consist of two semicircles.

Selecting the file type

The following file types are available:
- Point table (**.PNT**)
- Klartext program (**.H**)

If you save the machining positions to a Klartext program, the control creates a separate linear block with a cycle call for every machining position (**L X... Y... Z... F MAX M99**).

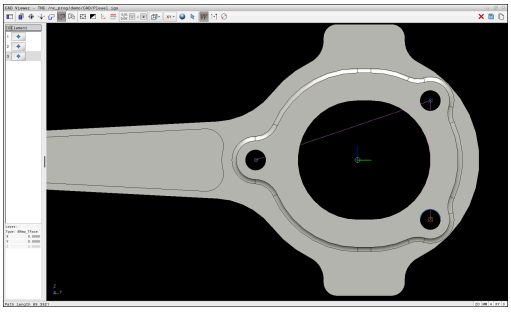
The NC syntax used allows you to export NC programs generated by CAD import to older HEIDENHAIN controls and run them there.

The point tables (**.PNT**) of the TNC 320 and the iTNC 530 are not compatible. Transferring a point table to and running it on the other control model leads to problems and unpredictable behavior.

Single selection

To select individual machining positions:

- Select the mode for choosing a machining position
- Place the mouse pointer on the desired element
- > The control displays the selectable element in orange.
- Select the circle center as machining position
- Alternative: Select the circle or a circle segment
- > The control transfers the selected machining position into the List View area.

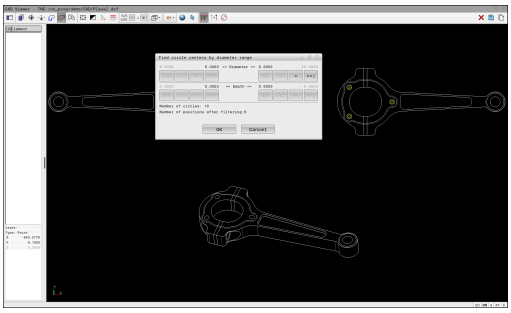


Multiple selection by drag box

To select multiple machining positions by dragging a box around them:

- Select the mode for choosing a machining position

- Activate the Add function
- > The control displays the following icon:
  +
- Drag a box around the desired area while holding down the left mouse button
- > The control opens a pop-up window. The pop-up window shows the identified diameter and depth values.
- Change the filter settings as needed
  Further information: "Filter settings", Page 465
- Confirm with **OK**
- > The control loads all machining positions within the selected diameter or depth range into the List View area.

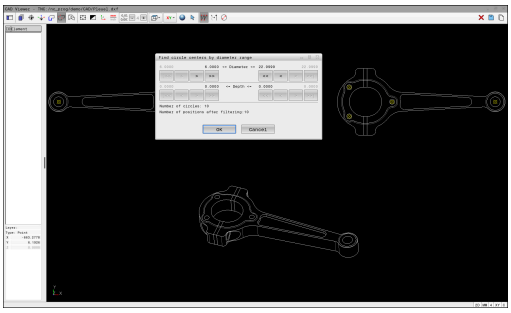


Multiple selection by search filter

To select multiple machining positions by search filter:

- Select the mode for choosing a machining position

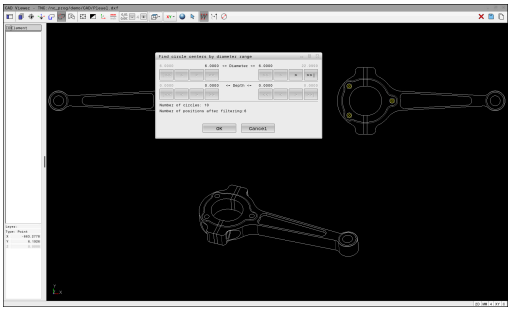
- Activate the search filter
- > The control opens a pop-up window. The pop-up window shows the identified diameter and depth values.
- Change the filter settings as needed
  Further information: "Filter settings", Page 465
- Confirm with **OK**
- > The control loads all machining positions within the selected diameter or depth range into the List View area.

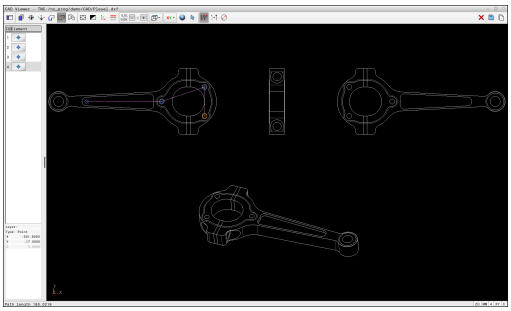


Filter settings

If you use the quick-selection function to mark positions, the **Find circle centers by diameter range** window opens. You can filter the diameter or depth values, referencing the workpiece datum, by means of the buttons below the displayed value. The control will only load the selected diameter or depth values.

The **Find circle centers by diameter range** window provides the following buttons:



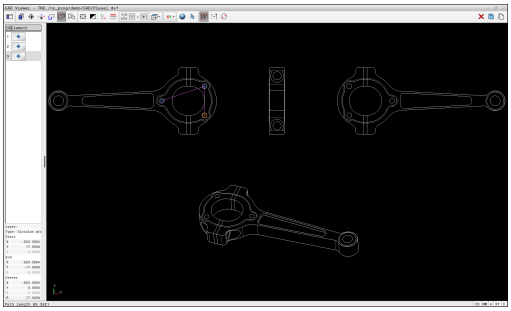


Button	Meaning
| <<	■ The control shows the smallest diameter found. ■ The control shows the smallest depth found. This filter is active by default.
<<	■ The control sets the filter for the largest diameter to the value selected for the smallest diameter. ■ The control sets the filter for the largest depth to the value selected for the smallest depth.
<	■ The control shows the next smaller diameter found. ■ The control shows the next smaller depth found.
>	■ The control shows the next larger diameter found. ■ The control shows the next larger depth found.
>>	■ The control sets the filter for the smallest diameter to the value selected for the largest diameter. ■ The control sets the filter for the smallest depth to the value selected for the largest depth.
>> |	■ The control shows the largest diameter found. ■ The control shows the largest depth found. This filter is active by default.

You can show the tool path by selecting the **SHOW TOOL PATH** icon.

Further information: "Basic settings", Page 449

Element information



In the Element Information area, the control displays the coordinates of the machining position selected most recently.

You can also use the mouse to change the rotation of the graphic. The following functions are available:

- To rotate the graphic, move the mouse while holding down the right mouse button.
- To shift the displayed model, hold down the center mouse button or the mouse wheel (depending on your mouse model), and move the mouse.
- To zoom in on a certain area, select a zoom area by holding the left mouse button down
- To rapidly zoom in or out, rotate the mouse wheel backwards or forwards
- To restore the standard view, double-click with the right mouse button

12.3 Generating STL files with 3D mesh (option 152)

Application

With the **3D mesh** function, you generate STL files from 3D models. This allows you to repair defective fixture and tool holder files, for example, or to position STL files generated from the simulation for another machining operation.

Requirement

- Software option CAD Model Optimizer (option 152)

Description of function

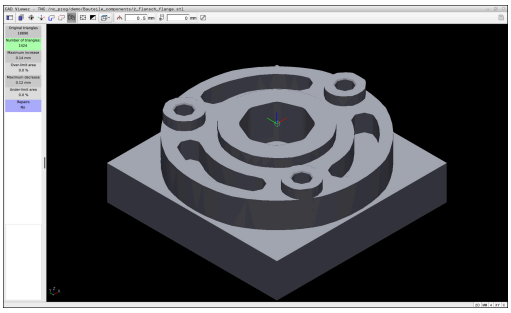
When you select the **3D mesh** icon, the control changes to **3D mesh** mode. The control covers the 3D model displayed in **CAD Viewer** with a mesh of triangles.

The control simplifies the original model and removes errors, such as small holes in a solid or self-intersections of a surface.

You can save the result and use it for various control functions, for example as a workpiece blank with the **BLK FORM FILE** function.

The simplified model or parts of it may be smaller or larger than the original model. The result depends on the quality of the original model and the settings selected in **3D mesh** mode.

The List View area shows the following information:



3D model in **3D mesh** mode

Option	Meaning
Original triangles	Number of triangles in the original model
Number of triangles:	Number of triangles with active settings in the simplified model If this option is highlighted in green, the number of triangles is in the optimum range. You can further reduce the number of triangles using the available functions. **Further information:** "Functions for the simplified model", Page 468
Maximum increase	Maximum increase of the triangle mesh
Over-limit area	Surface increase in percent compared to the original model
Maximum decrease	Maximum decrease of the triangle mesh compared to the original model
Under-limit area	Surface decrease in percent compared to the original model

Option	Meaning
Repairs	Indicates whether the original model has been repaired or not If it has been repaired, the control indicates the type of repair (e.g., **Hole Int Shells**). This indication consists of the following items: ■ **Hole** **CAD Viewer** closed holes in the 3D model. ■ **Int** **CAD Viewer** removed self-intersections. ■ **Shells** **CAD Viewer** joined multiple separate solids.

In order to use STL files for control functions, the saved files must meet the following requirements:

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

The greater the number of triangles in an STL file, the greater the processing power required by the control for simulation.

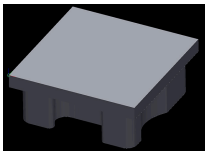
Functions for the simplified model

In order to reduce the number of triangles, you can define further settings for the simplified model.

CAD Viewer provides the following functions:

Icon	Meaning
	Allowed simplification Use this function to simplify the output model by the specified tolerance. The higher the value, the more the surfaces may deviate from the original.
	Remove holes <= diameter Use this function to remove holes and pockets up to the specified diameter from the original model.
	Only optimized mesh shown The control shows the simplified model only.
	Original is displayed The control shows the simplified model, superimposed with the original mesh from the original file. You can use this function to evaluate deviations.
	Save Use this function to save the simplified 3D model with the selected settings as an STL file.

Positioning the 3D model for rear-face machining



To position an STL file for rear-face machining:

- Export of the simulated workpiece as an STL file

  Further information: User's Manual for **Setup, Testing and Running NC Programs**

- Select the **Programming** operating mode

- Press the **PGM MGT** key
- > The control opens the file manager.
- Select the exported STL file
- > The control opens the STL file in **CAD Viewer**.
- Select **Preset**
- > The control displays information on the preset position in the List View area.
- Enter the value of the new preset in the **Preset** area (e.g., **Z-40**)
- Confirm your input
- Orient the coordinate system by specifying values under **PLANE SPATIAL SP***, e.g. **A+180** and **C+90**
- Confirm your input

- Select **3D mesh**
- > The control opens the **3D mesh** mode and simplifies the 3D model using the default settings.
- Further simplify the 3D model using the **3D mesh** mode functions, if required.

  Further information: "Functions for the simplified model", Page 468

- Select **Save**
- > The control opens the **Define file name for 3D mesh** menu.
- Enter the desired name
- Select **Save**
- > The control saves the STL file positioned for rear-face machining.

The resulting file can then be used for rear-face machining with the **BLK FORM FILE** function.

Further information: "Defining the workpiece blank: G30/G31", Page 101

13

Tables and Overviews

13.1 System data

List of D18 functions

The function can be used to read numeric system data and save the value in a Q, QL, or QR parameter (e.g., **FN 18: SYSREAD Q25 = ID210 NR4 IDX3**.)

> The read values of the function **D18** are always output by the control in **metric** units regardless of the NC program's unit of measure.

Further information: "D18 – Reading system data", Page 318

The **SYSSTR** function can be used to read alphanumeric system data and save the value in a QS parameter (e.g., **QS25 = SYSSTR(ID 10950 NR1)**).

Further information: "Reading system data", Page 328

You cannot program any SQL statements in an ISO program. If necessary, you can call a Klartext program with SQL statements from an ISO program.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Program information				
	10	3	-	Number of the active machining cycle
		6	-	Number of the most recently executed touch probe cycle –1 = None
		7	-	Type of calling NC program: –1 = None 0 = Visible NC program 1 = Cycle/macro, main program is visible 2 = Cycle/macro, there is no visible main program
		8	1	Unit of measure of the directly calling NC program (may also be a cycle). Return codes: 0 = mm 1 = inch –1 = there is no corresponding program
			2	Unit of measure of the NC program visible in the block display from which the current cycle was called directly or indirectly. Return codes: 0 = mm 1 = inch –1 = there is no corresponding program
		9	-	Within an M function macro: Number of the M function. Otherwise –1
			-	Within an M function macro: Number of the M function. Otherwise –1
		10	-	Repeat counter: Indicates the number of times the current code has been executed since the current NC program call
		103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
		110	QS parameter number	Is there a file with the name QS(IDX)? 0 = No, 1 = Yes This function resolves relative file paths.
		111	QS parameter number	Is there a directory with the name QS(IDX)? 0 = No, 1 = Yes Only absolute directory paths are possible.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
System jump addresses				
	13	1	-	Label number or label name (string or QS) jumped to during M2/M30 instead of ending the current NC program. Value = 0: M2/M30 have the normal effect
		2	-	Number or name (string or QS) of the label to which the NC program will jump if **FN 14: ERROR** has been programmed with the NC CANCEL reaction, instead of aborting the NC program with an error message. The error number programmed in the **FN 14** command can be read under ID992 NR14. Value = 0: **FN 14** has a normal effect.
		3	-	Label number or label name (string or QS) jumped to in the event of an internal server error (SQL, PLC, CFG) or with erroneous file operations (FUNCTION FILECOPY, FUNCTION FILEMOVE, or FUNCTION FILEDELETE) instead of aborting the NC program with an error message. Value = 0: Error has the normal effect.
Indexed access to Q parameters				
	15	11	Q parameter number	Reads Q(IDX)
		12	QL parameter no.	Reads QL(IDX)
		13	QR parameter no.	Reads QR(IDX)
Machine status				
	20	1	-	Active tool number
		2	-	Prepared tool number
		3	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W
		4	-	Programmed spindle speed
		5	-	Active spindle condition -1 = spindle condition not defined 0 = M3 active 1 = M4 active 2 = M5 active after M3 3 = M5 active after M4
		7	-	Active gear range
		8	-	Active coolant status 0 = off, 1 = on
		9	-	Active feed rate
		10	-	Index of prepared tool

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		11	-	Index of active tool
		14	-	Number of active spindle
		20	-	Programmed cutting speed in turning operation
		21	-	Spindle mode in turning mode: 0 = constant speed 1 = constant cutting speed
		22	-	Coolant status M7: 0 = inactive, 1 = active
		23	-	Coolant status M8: 0 = inactive, 1 = active

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Channel data				
	25	1	-	Channel number
Cycle parameters				
	30	1	-	Set-up clearance
		2	-	Hole depth / milling depth
		3	-	Plunging depth
		4	-	Feed rate for plunging
		5	-	First side length of pocket
		6	-	Second side length of pocket
		7	-	First side length of slot
		8	-	Second side length of slot
		9	-	Radius of circular pocket
		10	-	Feed rate for milling
		11	-	Rotational direction of the milling path
		12	-	Dwell time
		13	-	Thread pitch for Cycles 17 and 18
		14	-	Finishing allowance
		15	-	Roughing angle
		21	-	Probing angle
		22	-	Probing path
		23	-	Probing feed rate
		48	-	Tolerance
		49	-	HSC mode (Cycle 32 Tolerance)
		50	-	Tolerance for rotary axes (Cycle 32 Tolerance)
		52	Q parameter number	Type of transfer parameter for user cycles: –1: Cycle parameter not programmed in CYCL DEF 0: Cycle parameter numerically programmed in CYCL DEF (Q parameter) 1: Cycle parameter programmed as string in CYCL DEF (Q parameter)
		60	-	Clearance height (touch probe cycles 30 to 33)
		61	-	Inspection (touch probe cycles 30 to 33)
		62	-	Cutting edge measurement (touch probe cycles 30 to 33)
		63	-	Q parameter number for the result (touch probe cycles 30 to 33)
		64	-	Q parameter type for the result (touch probe cycles 30 to 33) 1 = Q, 2 = QL, 3 = QR
		70	-	Multiplier for feed rate (cycles 17 and 18)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modal status				
	35	1	-	Dimensions: 0 = absolute (G90) 1 = incremental (G91)
		2	-	Radius compensation: 0 = R0 1 = RR/RL 10 = Face milling 11 = Peripheral milling
Data for SQL tables				
	40	1	-	Result code for the last SQL command. If the last result code was 1 (=error), the error code is transferred as the return code.
Data from the tool table				
	50	1	Tool no.	Tool length L
		2	Tool no.	Tool radius R
		3	Tool no.	Tool radius R2
		4	Tool no.	Oversize for tool length DL
		5	Tool no.	Tool radius oversize DR
		6	Tool no.	Tool radius oversize DR2
		7	Tool no.	Tool locked TL 0 = not locked, 1 = locked
		8	Tool no.	Number of the replacement tool RT
		9	Tool no.	Maximum tool age TIME1
		10	Tool no.	Maximum tool age TIME2
		11	Tool no.	Current tool age CUR.TIME
		12	Tool no.	PLC status
		13	Tool no.	Maximum tooth length LCUTS
		14	Tool no.	Maximum plunge angle ANGLE
		15	Tool no.	TT: Number of tool teeth CUT
		16	Tool no.	TT: Wear tolerance for length, LTOL
		17	Tool no.	TT: Wear tolerance for radius, RTOL
		18	Tool no.	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	Tool no.	TT: Offset in plane R-OFFS R = 99999.9999
		20	Tool no.	TT: Offset in length L-OFFS
		21	Tool no.	TT: Breakage tolerance for length, LBREAK
		22	Tool no.	TT: Breakage tolerance for radius, RBREAK
		28	Tool no.	Maximum speed NMAX
		32	Tool no.	Point angle TANGLE

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		34	Tool no.	LIFTOFF allowed (0 = No, 1 = Yes)
		35	Tool no.	Wear tolerance for radius R2TOL
		36	Tool no.	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	Tool no.	Corresponding line in the touch-probe table
		38	Tool no.	Timestamp of last use
		40	Tool no.	Pitch for thread cycles
		44	Tool no.	Exceeding the tool life
		45	Tool no.	Front-face width of indexable insert (RCUTS)
		46	Tool no.	Usable length of the milling cutter
		47	Tool no.	Neck radius of the milling cutter (RN)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Data from the pocket table				
	51	1	Pocket number	Tool number
		2	Pocket number	0 = no special tool 1 = special tool
		3	Pocket number	0 = no fixed pocket 1 = fixed pocket
		4	Pocket number	0 = pocket not locked 1 = pocket locked
		5	Pocket number	PLC status
Determine the tool pocket				
	52	1	Tool no.	Pocket number
		2	Tool no.	Tool magazine number
File information				
	56	1	-	Number of lines of the tool table
		2	-	Number of lines of the active datum table
		4	-	Number of rows in a freely definable table that has been opened with **FN 26: TABOPEN**
Tool data for T and S strobes				
	57	1	T code	Tool number IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		2	T code	Tool index IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
		5	-	Spindle speed IDX0 = T0 strobe (store tool), IDX1 = T1 strobe (load tool), IDX2 = T2 strobe (prepare tool)
Values programmed in TOOL CALL				
	60	1	-	Tool number T
		2	-	Active tool axis 0 = X 1 = Y 2 = Z 6 = U 7 = V 8 = W
		3	-	Spindle speed S
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Automatic TOOL CALL 0 = Yes, 1 = No
		7	-	Tool radius oversize DR2
		8	-	Tool index
		9	-	Active feed rate

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		10	-	Cutting speed [mm/min]
Values programmed in TOOL DEF				
	61	0	Tool no.	Read the number of the tool change sequence: 0 = Tool already in spindle, 1 = Change between external tools, 2 = Change from internal to external tool, 3 = Change from special tool to external tool, 4 = Load external tool, 5 = Change from external to internal tool, 6 = Change from internal to internal tool, 7 = Change from special tool to internal tool, 8 = Load internal tool, 9 = Change from external tool to special tool, 10 = Change from special tool to internal tool, 11 = Change from special tool to special tool, 12 = Load special tool, 13 = Unload external tool, 14 = Unload internal tool, 15 = Unload special tool
		1	-	Tool number T
		2	-	Length
		3	-	Radius
		4	-	Index
		5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values for LAC and VSC				
	71	0	2	Total inertia determined by the LAC weighing run in [kgm²] (with A/B/C rotary axes) or total mass in [kg] (with X/Y/Z linear axes)
		1	0	Cycle 957 Retraction from thread
Freely available memory area for OEM cycles				
	72	0-39	0 to 30	Freely available memory area for OEM cycles. The values are only reset by the control during a control reboot (= 0). With “Cancel,” the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Freely available memory area for user cycles				
	73	0-39	0 to 30	Freely available memory area for user cycles The values are only reset by the control during a control reboot (= 0). With “Cancel,” the values are not reset to the value that they had at the time of execution. Up to and including 597110-11: only NR 0-9 and IDX 0-9 Starting with 597110-12: NR 0-39 and IDX 0-30
Read minimum and maximum spindle speed				
	90	1	Spindle ID	Minimum spindle speed of the lowest gear stage. If no gear stages are configured, CfgFeedLimits/minFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle
		2	Spindle ID	Maximum spindle speed from the highest gear stage. If no gear stages are configured, CfgFeedLimits/maxFeed of the first parameter set of the spindle is evaluated. Index 99 = active spindle
Tool compensation				
	200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
		2	1 = without oversize 2 = with oversize 3 = with oversize	Active length

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			and oversize from TOOL CALL	
		3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
		6	Tool no.	Tool length Index 0= active tool
Coordinate transformations				
	210	1	-	Basic rotation (manual)
		2	-	Programmed rotation
		3	-	Active mirror axis. Bits 0 to 2 and 6 to 8: Axes X, Y, Z and U, V, W
		4	Axis	Active scaling factor Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		5	Rotary axis	3D-ROT Index: 1 to 3 (A, B, C)
		6	-	Tilt working plane in Program Run operating modes 0 = Not active –1 = Active
		7	-	Tilt working plane in Manual operating modes 0 = Not active –1 = Active
		8	QL parameter no.	Angle of misalignment between spindle and tilted coordinate system. Projects the angle specified in the QL parameter from the input coordinate system to the tool coordinate system. If IDX is omitted, the angle 0 is used for projection.
		10	-	Type of definition of the active tilt: 0 = no tilt—is returned if, both in **Manual Operation** and in the automatic modes, no tilt is active. 1 = axial 2 = spatial angle
		11	-	Coordinate system for manual movements: 0 = Machine coordinate system **M-CS** 1 = Working plane coordinate system **WPL-CS** 2 = Tool coordinate system **T-CS** 4 = Workpiece coordinate system **W-CS**

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		12	Axis	Correction in working plane coordinate system **WPL-CS** (FUNCTION TURNDATA CORR WPL or FUNCTION CORRDATA WPL) Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Active coordinate system				
	211	–	-	1 = input system (default) 2 = REF system 3 = tool change system
Special transformations in turning mode				
	215	1	-	Angle for the precession of the input system in the XY plane in turning mode. To reset the transformation the value 0 must be entered for the angle. This transformation is used in connection with Cycle 800 (parameter Q497).
		3	1-3	Reading out of the spatial angle written with NR2 Index: 1 to 3 (rotA, rotB, rotC)
Current datum shift				
	220	2	Axis	Current datum shift in [mm] Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Read the difference between reference point and preset. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		4	Axis	Read OEM offset values.. Index: 1 to 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
Traverse range				
	230	2	Axis	Negative software limit switches Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		3	Axis	Positive software limit switches Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		5	-	Software limit switch on or off: 0 = on, 1 = off For modulo axes, either both the upper and lower limits or no limit at all must be set.
Read the nominal position in the REF system				
	240	1	Axis	Current nominal position in the REF system
Read the nominal position in the REF system, including offsets (handwheel, etc.)				
	241	1	Axis	Current nominal position in the REF system
Nominal positions of the physical axes in the REF system				
	245	1	Axis	Current nominal positions of the physical axes in the REF system
Read the current position in the active coordinate system				
	270	1	Axis	Current nominal position in the input system When called while tool radius compensation is active, the function supplies the uncompensated positions for the principal axes X, Y, and Z. If the function is called for a rotary axis and tool radius compensation is active, an error message is issued. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read the current position in the active coordinate system, including offsets (handwheel, etc.)				
	271	1	Axis	Current nominal position in the input system
Read information to M128				
	280	1	-	M128 active: –1 = Yes, 0 = No
		3	-	Condition of TCPM after Q No.: Q No. + 0: TCPM active, 0 = no, 1 = yes Q No. + 1: AXIS, 0 = POS, 1 = SPAT Q No. + 2: PATHCTRL, 0 = AXIS, 1 = VECTOR Q No. + 3: Feed rate, 0 = F TCP, 1 = F CONT
Machine kinematics				
	290	5	-	0: Temperature compensation not active 1: Temperature compensation active
		10	-	Index of the machine kinematics from Channels/ChannelSettings/CfgKin-List/kinCompositeModels programmed in FUNCTION MODE MILL or FUNCTION MODE TURN –1 = Not programmed.
Read data of the machine kinematics				
	295	1	QS parameter no.	Read the axis names of the active 3-axis kinematics. The axis names are written according to QS(IDX), QS(IDX+1), and QS(IDX +2). 0 = Operation successful
		2	0	Is FACING HEAD POS function active? 1 = Yes, 0 = No
		4	Rotary axis	Read whether the defined rotary axis partici-pates in the kinematic calculation. 1 = Yes, 0 = No (A rotary axis can be excluded from the kinematics calculating using M138.) Index: 4, 5, 6 (A, B, C)
		5	Secondary axis	Read whether the given secondary axis is used in the kinematics model. –1 = Axis not in the kinematics model 0 = Axis is not included in the kinematics calculation:
		6	Axis	Angle head: Displacement vector in the basic coordinate system B-CS through angle head Index: 1, 2, 3 (X, Y, Z)
		7	Axis	Angle head: Direction vector of the tool in the basic coordinate system B-CS Index: 1, 2, 3 (X, Y, Z)
		10	Axis	Determine programmable axes. Determine the axis ID associated with the specified axis index (index from CfgAxis/axisList). Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		11	Axis ID	Determine programmable axes. Determine the index of the axis (X = 1, Y = 2, ...) for the specified axis ID Index: Axis ID (index from CfgAxis/axisList)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Modify the geometrical behavior				
	310	20	Axis	Diameter programming: –1 = on, 0 = off
		126	-	M126: –1 = on, 0 = off
Current system time				
	320	1	0	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (real time).
			1	System time in seconds that have elapsed since 01.01.1970, 00:00:00 (look-ahead calculation).
		3	-	Read the processing time of the current NC program.
Formatting of system time				
	321	0	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm:ss
		1	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm:ss
		2	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY h:mm
		3	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY h:mm
		4	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm:ss

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm:ss
		5	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD hh:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD hh:mm
		6	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD h:mm
		7	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD h:mm
		8	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY
		9	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YYYY
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YYYY
		10	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: D.MM.YY

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: D.MM.YY
		11	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YYYY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YYYY-MM-DD
		12	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: YY-MM-DD
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: YY-MM-DD
		13	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: hh:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: hh:mm:ss
		14	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm:ss
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm:ss
		15	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: h:mm
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: h:mm
		16	0	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (real time) Format: DD.MM.YYYY hh:mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
			1	Formatting of: System time in seconds that have elapsed since 00:00:00 UTC on January 1, 1970 (look-ahead calculation) Format: DD.MM.YYYY hh:mm
		20	0	The current calendar week number according to ISO 8601 (real time)
			1	The current calendar week number according to ISO 8601 (look-ahead calculation)
Global Program Settings (GPS): Global activation status				
	330	0	-	0 = No Global Program Settings active 1 = Any GPS settings active
Global Program Settings (GPS): Individual activation status				
	331	0	-	0 = No Global Program Settings active 1 = Any GPS settings active
		1	-	GPS: Basic rotation 0 = Off, 1 = On
		3	Axis	GPS: Mirroring 0 = Off, 1 = On Index: 1 - 6 (X, Y, Z, A, B, C)
		4	-	GPS: Shift in the modified workpiece system 0 = Off, 1 = On
		5	-	GPS: Rotation in input system 0 = Off, 1 = On
		6	-	GPS: Feed rate factor 0 = Off, 1 = On
		8	-	GPS: Handwheel superimpositioning 0 = Off, 1 = On
		10	-	GPS: Virtual tool axis VT 0 = Off, 1 = On
		15	-	GPS: Selection of the handwheel coordinate system 0 = Machine coordinate system M-CS 1 = Workpiece coordinate system W-CS 2 = Modified workpiece coordinate system mW-CS 3 = Working plane coordinate system WPL-CS
		16	-	GPS: Shift in the workpiece system 0 = Off, 1 = On
		17	-	GPS: Axis offset 0 = Off, 1 = On

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Global Program Settings (GPS)				
	332	1	-	GPS: Angle of a basic rotation
		3	Axis	GPS: Mirroring 0 = Not mirrored, 1 = Mirrored Index: 1 to 6 (X, Y, Z, A, B, C)
		4	Axis	GPS: Shift in the modified workpiece coordinate system mW-CS Index: 1 to 6 (X, Y, Z, A, B, C)
		5	-	GPS: Angle of rotation in input coordinate system I-CS
		6	-	GPS: Feed rate factor
		8	Axis	GPS: Handwheel superimpositioning Maximum value Index: 1 to 10 (X, Y, Z, A, B, C, U, V, W, VT)
		9	Axis	GPS: Value for handwheel superimpositioning Index: 1 to 10 (X, Y, Z, A, B, C, U, V, W, VT)
		16	Axis	GPS: Shift in the workpiece coordinate system W-CS Index: 1 to 3 (X, Y, Z)
		17	Axis	GPS: Axis offset Index: 4 to 6 (A, B, C)
TS touch trigger probe				
	350	50	1	Touch probe type: 0: TS120, 1: TS220, 2: TS440, 3: TS630, 4: TS632, 5: TS640, 6: TS444, 7: TS740
			2	Line in the touch-probe table
		51	-	Effective length
		52	1	Effective radius of the stylus tip
			2	Rounding radius
		53	1	Center offset (reference axis)
			2	Center offset (minor axis)
		54	-	Spindle-orientation angle in degrees (center offset)
		55	1	Rapid traverse
			2	Measuring feed rate
			3	Feed rate for pre-positioning: FMAX_PROBE or FMAX_MACHINE
		56	1	Maximum measuring range
			2	Set-up clearance
		57	1	Spindle orientation possible 0=No, 1=Yes
			2	Angle of spindle orientation in degrees

Group name	Group number ID...	System data number NO....	Index IDX...	Description
TT tool touch probe for tool measurement				
	350	70	1	TT: Touch probe type
			2	TT: Line in the tool touch probe table
			3	TT: Designation of the active line in the touch-probe table
			4	TT: Touch probe input
		71	1/2/3	TT: Touch probe center (REF system)
		72	-	TT: Touch probe radius
		75	1	TT: Rapid traverse
			2	TT: Measuring feed rate with stationary spindle
			3	TT: Measuring feed rate with rotating spindle
		76	1	TT: Maximum probing path
			2	TT: Safety clearance for linear measurement
			3	TT: Safety clearance for radius measurement
			4	TT: Distance from the lower edge of the cutter to the upper edge of the stylus
		77	-	TT: Spindle speed
		78	-	TT: Probing direction
		79	-	TT: Activate radio transmission
			-	TT: Stop probing movement upon stylus deflection
		100	-	Distance after which the probe is deflected during touch probe simulation

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Preset from touch probe cycle (probing results)				
	360	1	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (input coordinate system). Compensations: length, radius, and center offset
		2	Axis	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (machine coordinate system, only axes from the active 3D kinematics are allowed as index). Compensation: only center offset
		3	Coordinate	Result of measurement in the input system of touch probe Cycles 0 and 1. The measurement result is read out in the form of coordinates. Compensation: only center offset
		4	Coordinate	Last preset of a manual touch probe cycle, or last touch point from Cycle 0 (workpiece coordinate system). The measurement result is read in the form of coordinates. Compensation: only center offset
		5	Axis	Axis values, not compensated
Preset from the touch probe cycle (probing results)				
	360	6	Coordinate / axis	Readout of the measurement results in the form of coordinates / axis values in the input system from probing operations. Compensation: only length
Preset from touch probe cycle (probing results)				
	360	10	-	Oriented spindle stop
		11	-	Error status of probing: 0: Probing was successful –1: Touch point not reached –2: Touch probe already deflected at the start of the probing process
Settings for touch probe cycles				
	370	2	-	Rapid traverse for measurement
		3	-	Machine rapid traverse as rapid traverse for measurement
		5	-	Angle tracking on/off
		6	-	Automatic measuring cycles: interruption with info about on/off
Settings for touch-probe cycles				
	370	7	-	Reaction when the automatic 14xx measuring cycle does not reach the probing point: 0 = Cancellation 1 = Warning 2 = No message

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				In case of values 1 and 2, the measurement result must be evaluated, and a corresponding reaction is required.
Read values from or write values to the active datum table				
	500	Row number	Column	Read values
Read values from or write values to the preset table (basic transformation)				
	507	Row number	1-6	Read values
Read axis offsets from or write axis offsets to the preset table				
	508	Row number	1-9	Read values
Data for pallet machining				
	510	1	-	Active line
		2	-	Current pallet number. Read value of the NAME column of the last PAL-type entry. If the column is empty or does not contain a numerical value, a value of –1 is returned.
		3	-	Active row of the pallet table.
		4	-	Last line of the NC program for the current pallet.
		5	Axis	Tool-oriented editing: Clearance height is programmed: 0 = No, 1 = Yes Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		6	Axis	Tool-oriented editing: Clearance height The value is invalid if ID510 NR5 returns the value 0 with the corresponding IDX. Index: 1 to 9 (X, Y, Z, A, B, C, U, V, W)
		10	-	Row number up to which the pallet table is to be searched during block scan.
		20	-	Type of pallet editing? 0 = Workpiece-oriented 1 = Tool oriented
		21	-	Automatic continuation after NC error: 0 = Locked 1 = Active 10 = Abort continuation 11 = Continuation with the rows in the pallet table that would have been executed next if not for the NC error 12 = Continuation with the row in the pallet table in which the NC error arose 13 = Continuation with the next pallet

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read data from the point table				
	520	Row number	10	Read value from active point table.
			11	Read value from active point table.
			1-3 X/Y/Z	Read value from active point table.
Read or write the active preset				
	530	1	-	Number of the active preset in the active preset table.
Active pallet preset				
	540	1	-	Number of the active pallet preset. Returns the number of the active preset. If no pallet preset is active, then the function returns the value -1.
		2	-	Number of the active pallet preset. Same as NO1.
Values for the basic transformation of the pallet preset				
	547	Row number	Axis	Read the basic transformation values from the pallet-preset table.. Index: 1 to 6 (X, Y, Z, SPA, SPB, SPC)
Axis offsets from the pallet preset table				
	548	Row number	Offset	Read the axis-offset values from the pallet preset table.. Index: 1 to 9 (X_OFFS, Y_OFFS, Z_OFFS,...)
OEM offset				
	558	Row number	Offset	Read values for OEM offset.. Index: 4 to 9 (A_OFFS, B_OFFS, C_OFFS,...)
Read and write the machine status				
	590	2	1-30	Freely available; not deleted during program selection.
		3	1-30	Freely available; not deleted during a power failure (persistent storage).
Read/write look-ahead parameter of a single axis (at machine level)				
	610	1	-	Minimum feed rate (**MP_minPathFeed**) in mm/min
		2	-	Minimum feed rate at corners (**MP_min-CornerFeed**) in mm/min
		3	-	Feed-rate limit for high speeds (**MP_maxG1Feed**) in mm/min
		4	-	Max. jerk at low speeds (**MP_maxPathJerk**) in m/s^3
		5	-	Max. jerk at high speeds (**MP_maxPath-JerkHi**) in m/s^3
		6	-	Tolerance at low speeds (**MP_pathTolerance**) in mm

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		7	-	Tolerance at high speeds (**MP_pathToleranceHi**) in mm
		8	-	Max. derivative of jerk (**MP_maxPathYank**) in m/s^4
		9	-	Tolerance factor for curve machining (**MP_curveTolFactor**)
		10	-	Factor for max. permissible jerk at curvature changes (**MP_curveJerkFactor**)
		11	-	Maximum jerk with probing movements (**MP_pathMeasJerk**)
		12	-	Angle tolerance for machining feed rate (**MP_angleTolerance**)
		13	-	Angle tolerance for rapid traverse (**MP_angleToleranceHi**)
		18	-	Radial acceleration with machining feed rate (**MP_maxTransAcc**)
		19	-	Radial acceleration with rapid traverse (**MP_maxTransAccHi**)
		20	Index of physical axis	Max. feed rate (**MP_maxFeed**) in mm/min
		21	Index of physical axis	Max. acceleration (**MP_maxAcceleration**) in m/s^2
		22	Index of physical axis	Maximum transition jerk of the axis in rapid traverse (**MP_axTransJerkHi**) in m/s^2
		23	Index of physical axis	Maximum transition jerk of the axis during machining free rate (**MP_axTransJerk**) in m/s^3
		24	Index of physical axis	Acceleration feedforward control (**MP_compAcc**)
		25	Index of physical axis	Axis-specific jerk at low speeds (**MP_axPathJerk**) in m/s^3
		26	Index of physical axis	Axis-specific jerk at high speeds (**MP_axPathJerkHi**) in m/s^3
		27	Index of physical axis	More precise tolerance examination in corners (**MP_reduceCornerFeed**) 0 = deactivated, 1 = activated
		28	Index of physical axis	DCM: Maximum tolerance for linear axes in mm (**MP_maxLinearTolerance**)
		29	Index of physical axis	DCM: Maximum angle tolerance in [°] (**MP_maxAngleTolerance**)
		30	Index of physical axis	Tolerance monitoring for successive threads (**MP_threadTolerance**)
		31	Index of physical axis	Form (**MP_shape**) of the **axisCutterLoc** filter 0: Off 1: Average

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				2: Triangle 3: HSC 4: Advanced HSC
		32	Index of physical axis	Frequency (**MP_frequency**) of the **axisCutterLoc** filter in Hz
		33	Index of physical axis	Form (**MP_shape**) of the **axisPosition** filter 0: Off 1: Average 2: Triangle 3: HSC 4: Advanced HSC
		34	Index of physical axis	Frequency (**MP_frequency**) of the **axisPosition** filter in Hz
		35	Index of physical axis	Order of the filter for **Manual** operating mode (**MP_manualFilterOrder**)
		36	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisCutterLoc** filter
		37	Index of physical axis	HSC mode (**MP_hscMode**) of the **axisPosition** filter
		38	Index of physical axis	Axis-specific jerk for probing movements (**MP_axMeasJerk**)
		39	Index of physical axis	Weighting of the filter error for calculating filter deviation (**MP_axFilterErrWeight**)
		40	Index of physical axis	Maximum filter length of position filter (**MP_maxHscOrder**)
		41	Index of physical axis	Maximum filter length of CLP filter (**MP_maxHscOrder**)
		42	-	Maximum feed rate of the axis at machining feed rate (**MP_maxWorkFeed**)
		43	-	Maximum path acceleration at machining feed rate (**MP_maxPathAcc**)
		44	-	Maximum path acceleration at rapid traverse (**MP_maxPathAccHi**)
		45	-	Shape of the smoothing filter (**CfgSmoothingFilter/shape**) 0 = Off 1 = Average 2 = Triangle
		46	-	Order of smoothing filter (only odd-numbered values) (**CfgSmoothingFilter/order**)
		47	-	Type of acceleration profile (**CfgLaPath/profileType**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		48	-	Type of acceleration profile for rapid traverse (**CfgLaPath/profileTypeHi**) 0 = Bellshaped 1 = Trapezoidal 2 = Advanced Trapezoidal
		49	-	Filter reduction mode (**CfgPositionFilter/timeGainAtStop**) 0 = Off 1 = NoOvershoot 2 = FullReduction
		51	Index of physical axis	Compensation of following error in the jerk phase (**MP_lpcJerkFact**)
		52	Index of physical axis	kv factor of the position controller in 1/s (**MP_kvFactor**)
		53	Index of physical axis	Radial jerk, normal feed rate (**MP_maxTransJerk**)
		54	Index of physical axis	Radial jerk, high feed rate (**MP_maxTransJerkHi**)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Read or write look-ahead parameters of a single axis (at cycle level)				
	613	see ID610	see ID610	Same as ID610 but is only in effect at the cycle level. Overwrite values from the machine configuration and values at the machine level. **Further information:** "FN functions ID610, ID611, ID613", Page
Measure the maximum utilization of an axis				
	621	0	Index of physical axis	Conclude measurement of the dynamic load and save the result in the specified Q parameter.
Read SIK contents				
	630	0	Option no.	You can explicitly determine whether the SIK option given under **IDX** has been set or not. 1 = option is enabled 0 = option is not enabled
		1	-	You can determine whether a Feature Content Level (for upgrade functions) is set, and which one. –1 = No FCL is set <No.> = FCL that is set
		2	-	Read serial number of the SIK -1 = No valid SIK in the system
		3	-	Read the SIK type (generation) 1 = SIK1 or no SIK 2 = SIK2
		4	Option number (4 digits)	Read the status of a software option (only available with SIK2) 0 = Not enabled 1 or higher = Number of enabled options
		10	-	Define the type of control: 0 = iTNC 530 1 = NCK-based control (TNC7, TNC 640, TNC 620, TNC 320, TNC 128, PNC 610, ...)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Counter				
	920	1	-	Planned workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		2	-	Already machined workpieces. In **Test Run** operating mode the counter generally generates the value 0.
		12	-	Workpieces still to be machined. In **Test Run** operating mode the counter generally generates the value 0.
Read and write data of current tool				
	950	1	-	Tool length L
		2	-	Tool radius R
		3	-	Tool radius R2
		4	-	Oversize for tool length DL
		5	-	Tool radius oversize DR
		6	-	Tool radius oversize DR2
		7	-	Tool locked TL 0 = not locked, 1 = locked
		8	-	Number of the replacement tool RT
		9	-	Maximum tool age TIME1
		10	-	Maximum tool age TIME2 at TOOL CALL
		11	-	Current tool age CUR.TIME
		12	-	PLC status
		13	-	Tooth length in the tool axis LCUTS
		14	-	Maximum plunge angle ANGLE
		15	-	TT: Number of tool teeth CUT
		16	-	TT: Wear tolerance for length LTOL
		17	-	TT: Wear tolerance for radius RTOL
		18	-	TT: Direction of rotation DIRECT 0 = positive, –1 = negative
		19	-	TT: Offset in plane R-OFFS R = 99999.9999
		20	-	TT: Offset in length L-OFFS
		21	-	TT: Break tolerance for length LBREAK
		22	-	TT: Break tolerance for radius RBREAK
		28	-	Maximum spindle speed [rpm] NMAX
		32	-	Point angle TANGLE
		34	-	LIFTOFF allowed (0 = No, 1 = Yes)
		35	-	Wear tolerance for radius R2TOL

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		36	-	Tool type TYPE (miller = 0, grinder = 1, ... touch probe = 21)
		37	-	Corresponding line in the touch-probe table
		38	-	Timestamp of last use
		39	-	ACC
		40	-	Pitch for thread cycles
		44	-	Exceeding the tool life
		45	-	Front-face width of indexable insert (RCUTS)
		46	-	Usable length of the milling cutter
		47	-	Neck radius of the milling cutter (RN)
		48	-	Radius at the tool tip (R_TIP)

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Tool usage and tooling				
	975	1	-	Tool usage test for the current NC program: Result –2: Test not possible, function disabled in the configuration Result –1: Test not possible, tool usage file missing Result 0: Test OK, all tools available Result 1: Test not OK
		2	Line	Check availability of the tools required in the pallet from line IDX in the current pallet table. –3 = No pallet is defined in row IDX, or function was called outside of pallet editing –2 / –1 / 0 / 1 see NO1
Touch probe cycles and coordinate transformations				
	990	1	-	Approach behavior: 0 = Standard behavior 1 = Approach probing position without compensation. Effective radius, set-up clearance is zero
		2	16	Automatic / Manual machine operating modes
		4	-	0 = Stylus not deflected 1 = Stylus deflected
		6	-	TT tool touch probe active? 1 = Yes 0 = No
		8	-	Momentary spindle angle in [°]
		10	QS parameter no.	Determine the tool number from the tool name. The return value depends on the rules configured for the search of the replacement tool. If there are multiple tools with the same name, the first tool from the tool table will be selected. If the tool selected by these rules is locked, a replacement tool will be returned. –1: No tool with the specified name found in the tool table or all qualifying tools are locked.
		16	0	0 = Transfer control over the channel spindle to the PLC, 1 = Assume control over the channel spindle
			1	0 = Pass tool spindle control to the PLC, 1 = Take control of the tool spindle
		19	-	Suppress touch prove movement in cycles: 0 = Movement will be suppressed (CfgMachineSimul/simMode parameter not equal to FullOperation or **Test Run** operating mode is active) 1 = Movement will be performed (CfgMa-

Group name	Group number ID...	System data number NO....	Index IDX...	Description
				chineSimul/simMode parameter = FullOperation, can be programmed for testing purposes)
		28	-	Read inclination angle of the current tool spindle

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Status of execution				
	992	10	-	Block scan active 1 = yes, 0 = no
		11	-	Block scan—information on block scan: 0 = NC program started without block scan 1 = Iniprog system cycle is run before block scan 2 = Block scan is running 3 = Functions are being updated –1 = Iniprog cycle was canceled before block scan –2 = Cancellation during block scan –3 = Cancellation of the block scan after the search phase, before or during the update of functions –99 = Implicit cancellation
		12	-	Type of canceling for interrogation within the OEM_CANCEL macro: 0 = No cancellation 1 = Cancellation due to error or emergency stop 2 = Explicit cancellation with internal stop after stop in the middle of the block 3 = Explicit cancellation with internal stop after stop at the end of a block
		14	-	Number of the last **FN 14** error
		16	-	Real execution active? 1 = execution, 0 = simulation
		17	-	2D graphics during programming active? 1 = Yes 0 = No
		18	-	Live programming graphics (**AUTO DRAW** soft key) active? 1 = Yes 0 = No
		20	-	Information on combined milling/turning mode of operation: 0 = Milling (after **FUNCTION MODE MILL**) 1 = Turning (after **FUNCTION MODE TURN**) 10 = Execute the operations for the turning-to-milling transition 11 = Execute the operations for the milling-to-turning transition
		30	-	Interpolation of multiple axes permitted? 0 = No (e.g. for straight cut control) 1 = yes
		31	-	R+/R– possible/permitted in MDI mode? 0 = No 1 = Yes

Group name	Group number ID...	System data number NO....	Index IDX...	Description
		32	Cycle number	Single cycle enabled: 0 = No 1 = Yes
		33	-	Write-access enabled for DNC (Python scripts) for executed entries in the pallet table: 0 = No 1 = Yes
		40	-	Copy tables in **Test Run** operating mode? Value 1 will be set when a program is selected and when the **RESET+START** soft key is pressed. The **iniprog.h** system cycle will then copy the tables and reset the system datum. 0 = no 1 = yes
		101	-	M101 active (visible condition)? 0 = no 1 = yes
		136	-	M136 active? 0 = no 1 = yes

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Activate machine parameter subfile				
	1020	13	QS parameter no.	Has a machine parameter subfile with path from QS number (IDX) been loaded? 1 = Yes 0 = No
Configuration settings for cycles				
	1030	1	-	Display the **Spindle is not rotating** error message (CfgGeoCycle/**displaySpindleErr**) 0 = No, 1 = Yes
		2	-	Display the **Check the depth sign** error message (CfgGeoCycle/**displayDepthErr**) 0 = No, 1 = Yes
Data transfer between HEIDENHAIN cycles and OEM macros				
	1031	1	0	Component monitoring: counter of the measurement. Cycle 238 Measure machine data automatically increments this counter.
			1	Component monitoring: Type of measurement –1 = No measurement 0 = Circular interpolation test 1 = Waterfall chart test 2 = Frequency response 3 = Envelope curve spectrum 4 = Advanced frequency response
			2	Component monitoring: Index of the axis from CfgAxes**axisList**
			3 – 9	Component monitoring: further arguments depend on the measurement
		2	3 – 9	Component monitoring: further arguments depend on the measurement
		3	0	KinematicsOpt: Read the current cycle number (450-453)
		100	-	Component monitoring: optional names of the monitoring tasks, as specified in **System \Monitoring\CfgMonComponent**. After completion of the measurement, the monitoring tasks stated here are executed consecutively. When assigning the input parameters, remember to separate the listed monitoring tasks by commas.

Group name	Group number ID...	System data number NO....	Index IDX...	Description
User settings for the user interface				
	1070	1	-	Feed rate limit of soft key FMAX; 0 = FMAX is inactive
Bit test				
	2300	Number	Bit number	This function checks whether a bit has been set in a number. The number to be checked is transferred as NR, the bit to be searched for as IDX, with IDX0 designating the least significant bit. To call this function for large numbers, make sure to transfer NR as a Q parameter. 0 = Bit not set 1 = Bit set
Read program information (system string)				
	10010	1	-	Path of the current main program or pallet program.
		2	-	Path of the NC program shown in the block display.
		3	-	Path of the cycle selected with **SEL CYCLE** or **CYCLE DEF 12 PGM CALL**, or path of the currently active cycle
		10	-	Path of the NC program selected with **SEL PGM "..."**.
Indexed access to QS parameters				
	10015	20	QS parameter no.	Reads QS(IDX)
		30	QS parameter no.	Returns the string that you obtain if you replace anything except for letters and digits in QS(IDX) by '_'.
Read channel data (system string)				
	10025	1	-	Name of machining channel (key)
Read data for SQL tables (system string)				
	10040	1	-	Symbolic name of the preset table.
		2	-	Symbolic name of the datum table.
		3	-	Symbolic name of the pallet preset table.
		10	-	Symbolic name of the tool table.
		11	-	Symbolic name of the pocket table.
		12	-	Symbolic name of the turning tool table
		13	-	Symbolic name of the grinding tool table
		14	-	Symbolic name of the dressing tool table
		21	-	Symbolic name of the compensation table in the T-CS tool coordinate system
		22	-	Symbolic name of the compensation table in the WPL-CS working plane coordinate system

Group name	Group number ID...	System data number NO....	Index IDX...	Description
Values programmed in the tool call (system string)				
	10060	1	-	Tool name
Read machine kinematics (system strings)				
	10290	10	-	Symbolic name of the machine kinematics from Channels/ChannelSettings/CfgKinList/kinCompositeModels programmed in **FUNCTION MODE MILL** or **FUNCTION MODE TURN**.
Traverse range switchover (system string)				
	10300	1	-	Key name of the last active range of traverse
Read current system time (system string)				
	10321	0 - 16, 20	-	1: DD.MM.YYYY hh:mm:ss 2: D.MM.YYYY h:mm 3: DD.MM.YY hh:mm 4: YYYY-MM-DD hh:mm:ss 5: YYYY-MM-DD hh:mm 6: YYYY-MM-DD h:mm 7: YY-MM-DD h:mm 8: DD.MM.YYYY 9: D.MM.YYYY 10: D.MM.YY 11: YYYY-MM-DD 12: YY-MM-DD 13: hh:mm:ss 14: h:mm:ss 15: h:mm 16: DD.MM.YYYY hh:mm 20: Calender week as per ISO 8601 As an alternative, you can use **DAT** in **SYSSTR(...)** to specify a system time in seconds that is to be used for formatting.
Read data of touch probes (TS, TT) (system string)				
	10350	50	-	Type of TS probe from TYPE column of the touch probe table (**tchprobe.tp**)
		51	-	Shape of stylus from column STYLUS in the touch probe table (**tchprobe.tp**).
		70	-	Type of TT tool touch probe from CfgTT/type.
		73	-	Key name of the active tool touch probe TT from **CfgProbes/activeTT**.
		74	-	Serial number of the active tool touch probe TT from **CfgProbes/activeTT**.
Read the data for pallet machining (system string)				
	10510	1	-	Pallet name
		2	-	Path of the selected pallet table.
Read version ID of the NC software (system string)				

Group name	Group number ID...	System data number NO....	Index IDX...	Description
	10630	10	-	The string corresponds to the format of the version ID shown (e.g., **340590 09** or **817601 05 SP1**)
Read information on unbalance cycle (system string)				
	10855	1	-	Path of the unbalance calibration table belonging to the active kinematics
Read data of the current tool (system string)				
	10950	1	-	Current tool name
		2	-	Entry from the DOC column of the active tool
		3	-	AFC control setting
		4	-	Tool-carrier kinematics
		5	-	Entry from the DR2TABLE column – file name of the compensation value table for 3D-ToolComp
Read current tool data (system string)				
	10950	6	-	Entry from the TSHAPE column - file name of the 3D tool shape (*.stl)
Read information from OEM macros and HEIDENHAIN cycles (system string)				
	11031	10	-	Returns the selection of the FUNCTION MODE SET <OEM mode> macro as a string.
		100	-	Cycle 238: list of key names for component monitoring
		101	-	Cycle 238: file names for log file

Comparison: D18 functions

The following table lists the D18 functions from previous controls, which were not implemented in this manner in the TNC 320.

In most cases, this function has been replaced by another function.

No.	IDX	Contents	Replacement function
ID 10 Program information			
1	-	mm/inch condition	Q113
2	-	Overlap factor for pocket milling	CfgRead
4	-	Number of the active fixed cycle	ID 10 no. 3
ID 20 Machine status			
15	Log. axis	Assignment between logic and geometric axes	
16	-	Feed rate for transition arcs	
17	-	Currently selected range of traverse	SYSTRING 10300
19	-	Maximum spindle speed for current gear stage and spindle	Maximum gear range: ID 90 No. 2
ID 50 Data from the tool table			
23	Tool no.	PLC value	[1]

No.	IDX	Contents	Replacement function
24	Tool no.	Probe center offset in reference axis (CAL-OF1)	ID 350 NR 53 IDX 1
25	Tool no.	Probe center offset in minor axis (CALOF-2)	ID 350 NR 53 IDX 2
26	Tool no.	Spindle angle during calibration (CAL-ANG)	ID 350 NR 54
27	Tool no.	Tool type for pocket table (PTYP)	2)
29	Tool no.	Position P1	1)
30	Tool no.	Position P2	1)
31	Tool no.	Position P3	1)
33	Tool no.	Thread pitch (Pitch)	ID 50 NR 40
ID 51 Data from the pocket table			
6	Pocket no.	Tool type	2)
7	Pocket no.	P1	2)
8	Pocket no.	P2	2)
9	Pocket no.	P3	2)
10	Pocket no.	P4	2)
11	Pocket no.	P5	2)
12	Pocket no.	Pocket reserved 0 = No, 1 = Yes	2)
13	Pocket no.	Box magazine: Pocket above occupied: 0 = No, 1 = Yes	2)
14	Pocket no.	Box magazine: Pocket below occupied: 0 = No, 1 = Yes	2)
15	Pocket no.	Box magazine: Pocket to the left occupied: 0 = No, 1 = Yes	2)
16	Pocket no.	Box magazine: Pocket to the right occupied: 0 = No, 1 = Yes	2)
ID 56 File information			
1	-	Number of lines of the tool table	
2	-	Number of lines of the active datum table	
3	Q parameters	Number of active axes that are programmed in the active datum table	
4	-	Number of lines in a freely definable table that has been opened with D26	
ID 214 Current contour data			
1	-	Contour transition mode	
2	-	Max. linearization error	
3	-	Mode for M112	
4	-	Character mode	
5	-	Mode for M124	1)
6	-	Specification for contour pocket machining	
7	-	Filter for control loop	

No.	IDX	Contents	Replacement function
8	-	Tolerance programmed with Cycle G62	ID 30 no. 48
ID 240 Nominal positions in the REF system			
8	-	ACTUAL position in the REF system	
ID 280 Information on M128			
2	-	Feed rate that was programmed with M128	ID 280 NR 3
ID 290 Switch the kinematics			
1	-	Line of the active kinematics table	SYSSTRING 10290
2	Bit no.	Interrogate the bits in MP7500	Cfgread
3	-	Status of collision monitoring (old)	Can be activated and deactivated in the NC program
4	-	Status of collision monitoring (new)	Can be activated and deactivated in the NC program
ID 310 Modifications of geometrical behavior			
116	-	M116: –1 = On, 0 = Off	
126	-	M126: –1 = On, 0 = Off	
ID 350 Touch-probe data			
10	-	TS: Touch-probe axis	ID 20 NR 3
11	-	TS: Effective ball radius	ID 350 NR 52
12	-	TS: Effective length	ID 350 NR 51
13	-	TS: Ring gauge radius	
14	1/2	TS: Center offset in reference/minor axis	ID 350 NR 53
15	-	TS: Direction of center offset relative to 0° position	ID 350 NR 54
20	1/2/3	TT: Center point X/Y/Z	ID 350 NR 71
21	-	TT: Plate radius	ID 350 NR 72
22	1/2/3	TT: 1st probing position X/Y/Z	Cfgread
23	1/2/3	TT: 2nd probing position X/Y/Z	Cfgread
24	1/2/3	TT: 3rd probing position X/Y/Z	Cfgread
25	1/2/3	TT: 4th probing position X/Y/Z	Cfgread
ID 370 Touch probe cycle settings			
1	-	Do not move to set-up clearance in Cycle 0.0 (as with ID990 NR1)	ID 990 NR 1
2	-	MP 6150 Rapid traverse for measurement	ID 350 NR 55 IDX 1
3	-	MP 6151 Machine rapid traverse as rapid traverse for measurement	ID 350 NR 55 IDX 3
4	-	MP 6120 Feed rate for measurement	ID 350 NR 55 IDX 2
5	-	MP 6165 Angle tracking on/off	ID 350 NR 57
ID 501 Datum table (REF system)			
Line	Column	Value in datum table	Preset table
ID 502 Preset table			

No.	IDX	Contents	Replacement function
Line	Column	Read the value from preset table, taking into account the active machining system	
ID 503 Preset table			
Line	Column	Read the value directly from the preset table	ID 507
ID 504 Preset table			
Line	Column	Read the basic rotation from the preset table	ID 507 IDX 4-6
ID 505 Datum table			
1	-	0 = No datum table selected 1 = Datum table selected	
ID 510 Data for pallet machining			
7	-	Test the insertion of a fixture from the PAL line	
ID 530 Active preset			
2	Line	Write-protect the line in the active preset table: 0 = No, 1 = Yes	D26 and D28: read out the Locked column
ID 990 Approach behavior			
2	10	0 = No execution in block scan 1 = Execution in block scan	ID 992 NR 10 / NR 11
3	Q parameters	Number of axes that are programmed in the selected datum table	
ID 1000 Machine parameter			
MP number	MP index	Value of the machine parameter	CfgRead
ID 1010 Machine parameter is defined			
MP number	MP index	0 = Machine parameter does not exist 1 = Machine parameter exists	CfgRead

1) Function or table column no longer exists
2) Use D26 and D28 to read out the table cell

13.2 Overview tables

Miscellaneous functions

M	Effect	Effective at block	Start	End	Page
M0	Program STOP/Spindle STOP/Coolant OFF			■	237
M1	Optional program STOP/Spindle STOP/Coolant OFF			■	237
M2	Program STOP/Spindle STOP/Coolant OFF/ CLEAR status display (depending on machine parameter)/Return jump to block 0			■	237
M3 M4 M5	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP		■ ■	 ■	237
M8 M9	Coolant ON Coolant OFF		■	 ■	237
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON		■ ■		237
M30	Same function as M2			■	237
M89	Cycle call, modally effective		■	 ■	Cycles Manual
M91	Within the positioning block: Coordinates are referenced to machine datum		■		238
M92	Within the positioning block: Coordinates are referenced to a position defined by machine manufacturer, e.g. tool change position		■		238
M94	Reduce the rotary axis display to a value below 360°		■		436
M97	Machine small contour steps			■	241
M98	Machine open contours completely			■	242
M99	Blockwise cycle call			■	Cycles Manual
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101			■ ■	142
M103	Feed rate factor for plunging movements		■		243
M107 M108	Suppress error message for replacement tools with oversize Reset M107			■ ■	142
M109 **M110** M111	Constant contouring speed at cutting edge (feed rate increase and reduction) Constant contouring speed at cutting edge (only feed rate reduction) Reset M109/M110		■ ■	 ■	245
M116 M117	Feed rate in mm/min on rotary axes Reset M116		■	 ■	434
M118	Superimpose handwheel positioning during program run		■		248
M120	Pre-calculate the radius-compensated contour (LOOK AHEAD)		■		246
M126 M127	Shorter-path traverse of rotary axes Reset M126		■	 ■	435
M130	Within the positioning block: Points are referenced to the untilted coordinate system		■		240
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136		■		244

M	Effect	Effective at block	Start	End	Page
M138	Selection of tilted axes		■		437
M140	Retraction from the contour in the tool-axis direction		■		249
M141	Suppress touch probe monitoring		■		251
M143	Delete basic rotation		■		251
M148	Automatically retract tool from the contour at an NC stop		■		252
M149	Reset M148			■	
M197	Corner rounding		■	■	253

User functions

User functions	Standard	Option	Meaning
Short description	✓		Basic version: 3 axes plus closed-loop spindle
		0	Additional axis for 4 axes plus closed-loop spindle
		1	Additional axis for 5 axes plus closed-loop spindle
Program entry	✓		In HEIDENHAIN Klartext format and ISO (G codes)
Position entry	✓		Nominal positions for lines and arcs in Cartesian coordinates or polar coordinates
	✓		Incremental or absolute dimensions
	✓		Display and entry in mm or inches
Tool compensation	✓		Tool radius in the working plane and tool length
	✓		Radius-compensated contour look-ahead for up to 99 NC blocks (M120)
Tool tables	✓		Multiple tool tables with any number of tools
Constant contour speed	✓		With respect to the path of the tool center
	✓		With respect to the cutting edge
Parallel operation	✓		Creating an NC program with graphical support while another NC program is being run
Rotary table machining (Advanced Function Set 1)		8	Programming of cylindrical contours as if in two axes
		8	Feed rate in distance per minute

User functions	Standard	Option	Meaning
Contour elements	✓		Straight line
	✓		Chamfer
	✓		Circular path
	✓		Circle center
	✓		Circle radius
	✓		Tangentially connected arc
	✓		Rounded corners
Approaching and departing the contour	✓		Via straight line: tangential or perpendicular
	✓		Via circular arc
FK free contour programming	✓		FK free contour programming in HEIDENHAIN Klartext format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	✓		Subprograms
	✓		Program section repeats
	✓		Calling any NC program
Machining cycles	✓		Cycles for drilling, and conventional and rigid tapping
	✓		Cycles for pecking, reaming, boring, and counterboring
	✓		Cycles for milling internal and external threads
	✓		Roughing and finishing rectangular and circular pockets
	✓		Roughing and finishing rectangular and circular studs
	✓		Point patterns for circles, lines, and DataMatrix codes
	✓		Cycles for clearing level and inclined surfaces
	✓		Cycles for milling linear and circular slots
	✓		Engraving
	✓		Contour pocket
	✓		Contour train
	✓		OEM cycles (special machining cycles developed by the machine manufacturer) can also be integrated
Coordinate transformation	✓		Datum shift, rotation, mirroring
	✓		Scaling factor (axis-specific)
		8	Tilting the working plane (Advanced Function Set 1)

User functions	Standard	Option	Meaning
Q parameters Programming with variables	✓		Mathematical functions: =, +, –, *, sin α, cos α, root
	✓		Logical operations (=, ≠, <, >)
	✓		Calculating with parentheses
	✓		tan α, arc sin, arc cos, arc tan, a^n, e^n, ln, log, absolute value of a number, constant π, negation, truncation of digits before or after the decimal point
	✓		Functions for calculation of circles
	✓		Functions for text processing
Programming aids	✓		Calculator
	✓		Color highlighting of syntax elements
	✓		Complete list of all current error messages
	✓		Context-sensitive help function
	✓		Graphic support for the programming of cycles
	✓		Comment and structure blocks in the NC program
Teach-In	✓		Actual positions can be transferred directly to the NC program
Test graphics Display modes	✓		Graphic simulation before a program run, even while another NC program is being run
	✓		Plan view / projection in 3 planes / 3D view / 3D line graphic
	✓		Detail enlargement
Programming graphics	✓		In the Programming mode, the contours of the NC blocks are drawn on screen while they are being entered (2D pencil-trace graphics), even while another NC program is being run
Program-run graphics Display modes	✓		Graphic simulation of real-time machining in plan view / projection in 3 planes / 3D view
Machining time	✓		Calculation of machining time in the **Test Run** operating mode
	✓		Display of the current machining time in the Program Run operating modes
Preset management	✓		For saving any datums
Returning to the contour	✓		Block scan in any NC block in the NC program, returning the tool to the calculated nominal position to continue machining
	✓		NC program interruption, contour departure and return
Datum tables	✓		Multiple datum tables for storing workpiece-specific datums
Touch probe cycles	✓		Calibrating the touch probe
	✓		Compensation of workpiece misalignment, manual or automatic
	✓		Presetting, manual or automatic
	✓		Automatically measuring workpieces
	✓		Cycles for automatic tool measurement

For a detailed overview of the user functions, see the brochure for the TNC 320. You can find the brochures related to the product range of CNC controls in the download area of the HEIDENHAIN website.

13.3 ISO function overview TNC 320

G codes

Tool movements	
G00	**Cartesian line in rapid traverse**
G01	**Cartesian line at feed rate**
G02	**Cartesian circle clockwise**
G03	**Cartesian circle CCW**
G05	**Cartesian circle**
G06	**Cartesian circle, tang. transit.**
G07	**Cartesian line, paraxial**
G10	**Polar line in rapid traverse**
G11	**Polar line at feed rate**
G12	**Polar circle clockwise**
G13	**Polar circle counterclockwise**
G15	**Polar circle**
G16	**Polar circle, tang. transition**
Approach or depart from chamfer/rounding/contour	
G24	**Chamfer with length R** with chamfer length R
G25	**Corner rounding with radius R** with radius R
G26	**Tangential approach** of a contour with radius R
G27	**Tangential departure** from a contour with radius R
Tool definition	
G99	**Tool definition** with tool number T, length L, and radius R
Tool radius compensation	
G40	**Path of tool center** without tool radius compensation
G41	**Radius compensation left of path**
G42	**Radius compens. right of path**
G43	**Radius compensation: extend path** for G07
G44	**Radius compens.: shorten path** for G07
Blank form definition for graphics	
G30	**Workpiece blank def.: MIN point** (G17/G18/G19)
G31	**Workpiece blank def.: MAX point** (G90/G91)
Cycles for drilling, tapping and thread milling	
G200	**DRILLING**
G201	**REAMING**
G202	**BORING**
G203	**UNIVERSAL DRILLING**

Cycles for drilling, tapping and thread milling	
G204	**BACK BORING**
G205	**UNIVERSAL PECKING**
G206	**TAPPING** with floating tap holder
G207	**RIGID TAPPING** without floating tap holder
G208	**BORE MILLING**
G209	**TAPPING W/ CHIP BRKG**
G240	**CENTERING**
G241	**SINGLE-LIP D.H.DRLNG**
G262	**THREAD MILLING**
G263	**THREAD MLLNG/CNTSNKG**
G265	**HEL. THREAD DRLG/MLG**
G267	**OUTSIDE THREAD MLLNG**

Cycles for milling pockets, studs, and slots	
G233	**FACE MILLING**
G251	**RECTANGULAR POCKET**
G252	**CIRCULAR POCKET**
G253	**SLOT MILLING**
G254	**CIRCULAR SLOT**
G256	**RECTANGULAR STUD**
G257	**CIRCULAR STUD**
G258	**POLYGON STUD**

Coordinate transformation	
G28	**MIRRORING**
G53	**DATUM SHIFT**
G54	**DATUM SHIFT**
G72	**SCALING FACTOR**
G73	**ROTATION**
G80	**WORKING PLANE**
G247	**PRESETTING**

SL cycles	
G37	**CONTOUR**
G120	**CONTOUR DATA**
G121	**PILOT DRILLING**
G122	**ROUGH-OUT**
G123	**FLOOR FINISHING**
G124	**SIDE FINISHING**
G125	**CONTOUR TRAIN**

SL cycles	
G127	**CYLINDER SURFACE**
G128	**CYLINDER SURFACE**
G129	**CYL SURFACE RIDGE**
G139	**CYL. SURFACE CONTOUR**
G270	**CONTOUR TRAIN DATA**
G271	**OCM CONTOUR DATA**
G272	**OCM ROUGHING**
G273	**OCM FINISHING FLOOR**
G274	**OCM FINISHING SIDE**
G275	**TROCHOIDAL SLOT**
G276	**THREE-D CONT. TRAIN**
Cycles for creating point patterns	
G220	**POLAR PATTERN**
G221	**CARTESIAN PATTERN**
G224	**DATAMATRIX CODE PATTERN**
Cycles for turning	
G37	**CONTOUR**
G800	**ADJUST XZ SYSTEM**
G801	**RESET ROTARY COORDINATE SYSTEM**
G810	**TURN CONTOUR LONG.**
G811	**SHOULDER, LONGITDNL.**
G812	**SHOULDER, LONG. EXT.**
G813	**TURN PLUNGE CONTOUR LONGITUDINAL**
G814	**TURN PLUNGE LONGITUDINAL EXT.**
G815	**CONTOUR-PAR. TURNING**
G820	**TURN CONTOUR TRANSV.**
G821	**SHOULDER, FACE**
G822	**SHOULDER, FACE. EXT.**
G823	**TURN TRANSVERSE PLUNGE**
G824	**TURN PLUNGE TRANSVERSE EXT.**
G830	**THREAD CONTOUR-PARALLEL**
G831	**THREAD LONGITUDINAL**
G832	**THREAD EXTENDED**
G840	**RECESS TURNG, RADIAL**
G841	**SIMPLE REC. TURNG., RADIAL DIR.**
G842	**ENH.REC.TURNNG, RAD.**
G850	**RECESS TURNG, AXIAL**
G851	**SIMPLE REC TURNG, AX**

Cycles for turning	
G852	ENH.REC.TURNING, AX.
G860	CONT. RECESS, RADIAL
G861	SIMPLE RECESS, RADL.
G862	EXPND. RECESS, RADL.
G870	CONT. RECESS, AXIAL
G871	SIMPLE RECESS, AXIAL
G872	EXPND. RECESS, AXIAL
G883	TURNING SIMULTANEOUS FINISHING
G892	CHECK UNBALANCE

Special cycles	
G4	DWELL TIME
G36	ORIENTATION
G39	PGM CALL
G62	TOLERANCE
G86	THREAD CUTTING
G225	ENGRAVING
G232	FACE MILLING
G238	MEASURE MACHINE STATUS
G239	ASCERTAIN THE LOAD
G291	COUPLG.TURNG.INTERP.
G292	CONTOUR.TURNG.INTRP.

Cycles for grinding	
G1000	DEFINE RECIP. STROKE
G1001	START RECIP. STROKE
G1002	STOP RECIP. STROKE
G1010	DRESSING DIAMETER
G1015	PROFILE DRESSING
G1030	ACTIVATE WHEEL EDGE
G1032	GRINDING WHL LENGTH COMPENSATION
G1033	GRINDING WHL RADIUS COMPENSATION

Touch probe cycles for measuring workpiece misalignment	
G400	BASIC ROTATION
G401	ROT OF 2 HOLES
G402	ROT OF 2 STUDS
G403	ROT IN ROTARY AXIS
G404	SET BASIC ROTATION
G405	ROT IN C AXIS

Touch probe cycles for measuring workpiece misalignment	
G1410	**PROBING ON EDGE**
G1411	**PROBING TWO CIRCLES**
G1420	**PROBING IN PLANE**

Touch probe system cycles for setting datum	
G408	**SLOT CENTER PRESET**
G409	**RIDGE CENTER PRESET**
G410	**PRESET INSIDE RECTAN**
G411	**PRESET OUTS. RECTAN**
G412	**PRESET INSIDE CIRCLE**
G413	**PRESET OUTS. CIRCLE**
G414	**PRESET OUTS. CORNER**
G415	**PRESET INSIDE CORNER**
G416	**PRESET CIRCLE CENTER**
G417	**PRESET IN TS AXIS**
G418	**PRESET FROM 4 HOLES**
G419	**PRESET IN ONE AXIS**

Touch probe cycles for workpiece measurement	
G55	**REF. PLANE**
G420	**MEASURE ANGLE**
G421	**MEASURE HOLE**
G422	**MEAS. CIRCLE OUTSIDE**
G423	**MEAS. RECTAN. INSIDE**
G424	**MEAS. RECTAN. OUTS.**
G425	**MEASURE INSIDE WIDTH**
G426	**MEASURE RIDGE WIDTH**
G427	**MEASURE COORDINATE**
G430	**MEAS. BOLT HOLE CIRC**
G431	**MEASURE PLANE**

Special cycles	
G441	**FAST PROBING**
G444	**PROBING IN 3-D**
G600	**GLOBAL WORKING SPACE**
G601	**LOCAL WORKING SPACE**

Touch probe cycles for touch probe calibration	
G460	**TS CALIBRATION OF TOOL LENGTH**
G461	**CALIBRATION OF A TS IN A RING**
G462	**TS CALIBRATION ON STUD**

Touch probe cycles for touch probe calibration	
G463	**CALIBRATION OF TS ON A SPHERE**
Touch probe cycles for kinematics measurement	
G450	**SAVE KINEMATICS**
G451	**MEASURE KINEMATICS**
G452	**PRESET COMPENSATION**
G453	**KINEMATICS GRID**
Touch probe cycles for tool measurement	
G480	**CALIBRATE TT**
G481	**CAL. TOOL LENGTH**
G482	**CAL. TOOL RADIUS**
G483	**MEASURE TOOL**
G484	**CALIBRATE IR TT**
Define the working plane	
G17	**Spindle axis Z - plane XY**
G18	**Spindle axis Y - plane ZX**
G19	**Spindle axis X - plane YZ**
Dimensions	
G70	Unit of measure: Inch
G71	Unit of measure: mm
G90	**Absolute dimension**
G91	Incremental dimension
Other G codes	
G29	**Load current position**
G38	**Stop program run**
G51	**Prepare tool changer**
G79	Cycle call
G98	**Set label**

Addresses

Addresses	
%	■ Program start ■ Program call
#	Datum number with G53
A	Rotation about X axis
B	Rotation about Y axis
C	Rotation about Z axis
D	Q parameter definitions
DL	Wear compensation for length with T
DR	Wear compensation for radius with T
E	Tolerance ■ M112 ■ M124
F	■ Feed rate ■ Dwell time with G04 ■ Scaling factor with G72 ■ Factor for feed-rate reduction F with M103
G	G codes
H	■ Polar coordinate angle ■ Rotation angle with G73 ■ Max. permissible angle with M112
I	X coordinate of the circle center/pole
Y	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	■ Set a label number with G98 ■ Jump to a label number ■ Tool length with G99
M	M functions
N	Block number
P	■ Cycle parameters in machining cycles ■ Value or Q parameter in Q parameter definition
Q	Q parameter
R	■ Polar coordinate radius ■ Radius with G02/G03/G05 ■ Rounding radius with G25/G26/G27 ■ Tool radius with G99
S	■ Spindle speed ■ Spindle orientation with G36
T	■ Tool definition with G99 ■ Tool call ■ Next tool with G51

Addresses	
U	Axis parallel to X axis
V	Axis parallel to Y axis
W	Axis parallel to Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

Contour cycles

Program structure with machining with multiple tools	
List of contour programs	G37 P01 ...
Defining **contour data**	G120 Q1 ...
Drill define/call Contour cycle: Pilot drilling Cycle call	G121 Q10 ...
Roughing mill define/call Contour cycle: Rough-out Cycle call	G122 Q10 ...
Finishing mill define/call Contour cycle: Floor finishing Cycle call	G123 Q11 ...
Finishing mill define/call Contour cycle: Side finishing Cycle call	G124 Q11 ...
End of main program, return	**M02**
Contour subprograms	G98 ... G98 L0

Radius compensation of the contour subprograms

Contour	**Programming sequence of the contour elements**	**Radius compensation**
Inside (pocket)	clockwise (CW) counterclockwise (CCW)	G42 (RR) G41 (RL)
Outside (island)	clockwise (CW) counterclockwise (CCW)	G41 (RL) G42 (RR)

Coordinate transformations

Coordinate transformation	**Activate**	**Rescind**
Datum shift	G54 X+20 Y+30 Z+10	G54 X0 Y0 Z0
Mirroring	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F 0.8	G72 F1 ...
Machining plane	G80 A+10 B+10 C+15	G80
Machining plane	PLANE ...	PLANE RESET

Q parameter definitions

D	Function
00	Assignment
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Square root
06	Sine
07	Cosine
08	Root of sum of squares c = $\sqrt{(a^2+b^2)}$
09	If equal, go to label number
10	If not equal, go to label number
11	If greater than, go to label number
12	If less than, go to label number
13	Angle with ARCTAN
14	Output error messages
15	External output
16	Formatted output of texts or Q parameter values
18	Read system data
19	Send values to the PLC
20	NC and PLC synchronization
26	Opening a freely definable table
27	Write to a freely definable table
28	Read from a freely definable table
29	Transfer up to eight values to the PLC
37	Export local Q parameters or QS parameters into a calling NC program
38	Send information from the NC program

Index

3

3D mesh ... 467

A

About this manual ... 26
Actual position capture ... 109
Adding comments ... 204, **205**
Additional axes ... 96
Additional axes for rotary axes ... 434
ADP ... 443
Align tool axis ... 433
ASCII files ... 382

B

Block ... 111
  Delete ... 111
  Inserting and modifying ... 111

C

CAD Import ... 447
CAD viewer ... 447
  Basic settings ... 449
  defining the plane ... 455
  Filter for drilling positions ... 465
  Presetting ... 453
  Selecting a contour ... 459
  Selecting a machining position ... 463
  Setting layers ... 451
Calculating with parentheses ... 294
Calculation of circles ... 290
Calculator ... 211
Calling a program
  Calling any NC program ... 260
CAM programming ... 438
Cartesian coordinates
  Circular arc around circle center CC ... 171
  Circular arc with specified radius ... 173
  Circular arc with tangential transition ... 175
  Linear motion superimposed on circular contour ... 176
  Straight line ... 167
Chamfer ... 168
Circle ... 182
Circle center ... 170
Circular arc
  around circle center CC ... 171
  With fixed radius ... 173
  with tangential transition ... 175
Circular contour
  Superimposed linear motion ... 176
Circular path
  Around pole ... 182
Compensation table
  Creating ... 373
  Type ... 372
Context-sensitive help ... 228
Contour
  Approaching ... 155
  Departing ... 155
  Selecting from DXF file ... 459
Copying program sections ... 113
Counter ... 380

D

D14: error message output ... 302
D16: F-PRINT: formatted output of text ... 308
D18: reading system data ... 318
D19: Transfer values to the PLC 318
D20: NC and PLC synchronization ... 319
D23: CIRCLE DATA: Calculate a circle from 3 points ... 290
D24: CIRCLE DATA: Calculate a circle from 4 points ... 290
D26: TABOPEN:Table, freely definable, opening ... 389
D27: TABWRITE: Table, freely definable, writing ... 390
D28: TABREAD: Table, freely definable, reading ... 392
D29: Transfer values to PLC ... 320
D37 EXPORT ... 320
D38: Information ... 321
Data output
  Displaying ... 316
  To server ... 317
Datum table ... 367
  Columns ... 367
  Creating ... 368
  Selecting ... 371
Defining local Q parameters ... 282
Defining nonvolatile Q parameters ... 282
Defining the workpiece blank ... 106
Dialog ... 107
DIN/ISO ... 107
Directory ... 118, 123
  Copy ... 126
  Create ... 123
  Delete ... 127
Display of the NC program ... 204
Display screen ... 81
DNC
  Information from NC program ... 321
Drilling position, selecting
  Drag box ... 464
  Icon ... 464
  Single selection ... 464
Drilling positions
  Filter for CAD data transfer ... 465
Dwell time
  Cyclic ... 397
  Once ... 399
  resetting ... 398

E

Error message ... 221
  deleting ... 224
  filtering ... 223
  help with ... 221
  Output ... 302

F

Feed rate
  On rotary axes, M116 ... 434
Feed rate factor for plunging movements M103 ... 243
Feed rate in millimeters per spindle revolution M136 ... 244
File
  Copying ... 123
  create ... 123
  Overwriting ... 124
  protecting ... 130
  Sorting ... 129
File management
  Copying a table ... 125
  External file types ... 118
File manager
  Calling ... 120
  Delete file ... 127
  Directories
    Copy ... 126
    Create ... 123
  Directory ... 118
  File type ... 116
  Function overview ... 119
  Hidden files ... 131
  Rename file ... 129
  Selecting files ... 121
Files
  Tagging ... 128
File status ... 120
FK programming ... 187
  Auxiliary points ... 196
  Circle data ... 194
  Circular paths ... 192
  Closed contours ... 195
  Dialog initiation ... 190
  Direction and length of contour elements ... 193
  End point ... 193
  Fundamentals ... 187
  Graphics ... 189
  Relative data ... 197

Straight lines... 191
Working plane... 188
Fluctuating spindle speed... 394
Form view... 389
Full circle... 171
FUNCTION COUNT... 380
FUNCTION DWELL... 399
FUNCTION FEED DWELL... 397
Fundamentals... 87

G

GOTO... 202
Graphics
With programming... 218
Magnification of details... 220

H

Hard disk... 116
Helical interpolation... 183
Helix... 183
Help file, downloading... 232
Help system... 228
Help with error message... 221
Hidden files... 131

I

Import
Table from iTNC 530... 393
iTNC 530... 80

J

Jump conditions... 291
Jumping
with GOTO... 202

L

Liftoff... **400**
Lift-off... 252
Log, writing to... 321
Look ahead... 246

M

M91, M92... 238
Message
Screen output... 316
Message, printing... 317
Miscellaneous functions... 236
entering... 236
For coordinate entries... 238
For path behavior... 241
For program run inspection... 237
For spindle and coolant... 237
Modes of Operation... 84
motion control... 443
Multi-axis machining... 404

N

NC and PLC synchronization... 319
NC block... 111
NC error message... 221
NC program... 100
Editing... 110
structuring... 209
Nesting... 269

O

Open contour corners M98... 242
Operating panel... 82
Option... 30

P

Part families... 283
Path... 118
Path contours... 166
Cartesian coordinates... 166
Overview... 166
Polar coordinates... 180
Circular path with tangential connection... 182
Overview... 180
Straight line... 181
Path functions
Fundamentals... 150
Circles and circular arcs... 153
Pre-positioning... 154
PLANE function... 405
Automatic tilting into position... 425
Axis angle definition... 422
Euler angle definition... 415
Incremental definition... 421
Overview... 407
Point definition... 419
positioning behavior... 424
Projection angle definition... 413
Resetting... 409
Selection of possible solutions... 428
Spatial angle definition... 410
Transformation type... 431
Vector definition... 417
PLC and NC synchronization... 319
Point tables... 265
Polar coordinates... 97
Circular path around pole CC 182
Fundamentals... 97
Programming... 180
Polar kinematics... 356
Position, selecting from CAD files... 463
Positioning
with tilted working plane... 240
Post processor... 439
Preset
Selecting... 99
Principal axes... 96
Process chain... 438
Program... 100
Opening a new program... 106
structuring... 209
Program defaults... 353
Programm
Structure... 100
Programming graphics... 189
Programming tool movement... 107
Program-section repeat... 258
Pulsing spindle speed... 394

Q

Q parameter programming
Additional functions... 301
Calculation of circles... 290
If-then decision... 291
Mathematical functions... 284
Q-parameter programming
Programming notes... 281
Trigonometric functions... 288
Q parameters... 278, 279
checking... 299
Export... 320
Formatted output... 308
Local parameters Q... 278
Local parameters QL... 279
Preassigned... 335
programming... 278, 323
Residual parameters QR 278, 279
String parameters QS... 323
Transfer values to PLC... 320
Q parameters
Transfer values to PLC... 318

R

Radius compensation... 146
Entering... 147
Outside corner, inside corner 148
Rapid traverse... 134
Reading out machine parameters... 333
Reading system data... **318**, 328
Reference system... 88, 96
Basic... 91
Input... 95
Machine... 89
Tool... 96
Working plane... 94
Workpiece... 92
Replacing texts... 115
Resonance vibration... 394
Retraction from the contour... 249
Rotary axes... 434
Rotary axis
Reduce display M94... 436
Shorter-path traverse: M126... 435
Rounded corners... 169

Rounding corners M197 ... 253
Rounding of values ... 343

S

Screen keypad ... 203, 203
Screen layout ... 82
  CAD viewer ... 446
Search function ... 114
Selecting the unit of measure ... 106
SEL TABLE ... 371
Service files, saving ... 227
Software option ... 30
SPEC FCT ... 352
Special functions ... 352
Spindle speed
  Entering ... 140
STL file
  Optimizing ... 467
Straight line ... **167**, 181
String parameter
  Converting ... 329
  Copying a substring ... 327
  Determine length ... 331
  Testing ... 330
String parameters ... 323
  Assign ... 324
  Chain-linking ... 325
  Reading system data ... 328
Structuring NC programs ... 209
Subprogram ... 257
Superimposing handwheel positioning M118 ... 248
Surface normal vector ... 417
System data
  list ... 472

T

TABDATA ... 376
Table, freely definable
  Opening ... 389
  Reading ... 392
  Writing ... 390
Table access
  TABDATA ... 376
  TABWRITE ... 390
Teach In ... **109**, 167
Text editor ... 207
Text file ... 382
  Creating ... 308
  Delete functions ... 383
  Finding text sections ... 385
  Formatted output ... 308
  Opening and exiting ... 382
Text variables ... 323
Tilting
  Resetting ... 409
  Working plane ... 405
Tilting without rotary axes ... 433
Tilt working plane
  Programmed ... 405
TNCguide ... 228
Tool change ... 142
Tool compensation ... 145
  Length ... 145
  Radius ... 146
  Table ... 372
Tool data ... 136
  Calling ... 140
  Delta values ... 139
  Entering into the program ... 139
  Replacing ... 125
Tool length ... 137
Tool name ... 136
Tool number ... 136
Tool radius ... 139
Touch probe monitoring ... 251
Trigonometric functions ... 288
Trigonometry ... 288

V

Vector ... 417

W

Workpiece positions ... 98

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

Original document
1096983-25 · Ver05 · SW18 · 10/2023 · F&W · Printed in Germany