



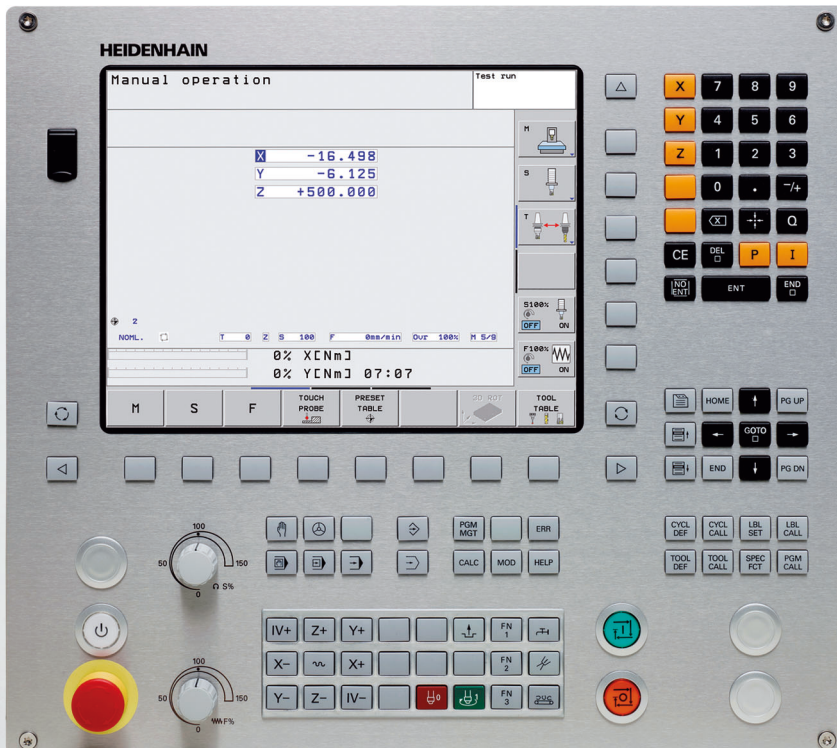
# HEIDENHAIN

User's Manual  
HEIDENHAIN  
Conversational  
Programming

## TNC 128





NC Software  
771841-01  
771844-01

English (en)  
4/2013








# Controls of the TNC



## Keys on visual display unit

Key	Function
	Split screen layout
	Toggle the display between machining and programming modes
	Soft keys for selecting functions on screen
	Shift between 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 and Editing
	Test Run

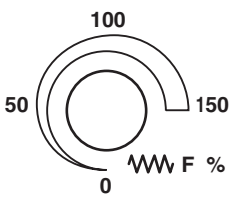
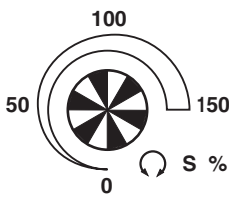
## Program/file management, TNC functions

Key	Function
	Select or delete programs and files, external data transfer
	Define program call, select datum and point tables
	Select MOD functions
	Display help text for NC error messages, call TNCguide
	Display all current error messages
	Show calculator






## Navigation keys

Key	Function
	Move highlight
	Go directly to blocks, cycles and parameter functions

## Potentiometer for feed rate and spindle speed



Feed rate	Spindle speed
	

## Cycles, subprograms and program section repeats




Key	Function
 	Define and call cycles
 	Enter and call labels for subprogramming and program section repeats
	Enter program stop in a program

















**Tool functions**

Key	Function
	Define tool data in the program
	Call tool data

**Special functions**

Key	Function
	Show special functions
	Select the next tab in forms
	Up/down one dialog box or button

**Coordinate axes and numbers: Entering and editing**

Key	Function
 . . . 	Select coordinate axes or enter them into the program
 . . . 	Numbers
 	Decimal point / Reverse algebraic sign
	Enter incremental dimensions
	Q parameter programming / Q parameter status
	Save actual position or values from calculator
	Skip dialog questions, delete words
	Confirm entry and resume dialog
	Conclude block and exit entry
	Clear numerical entry or TNC error message
	Abort dialog, delete program section







# About this manual

The symbols used in this manual are described below.



This symbol indicates that important notes about the function described must be regarded.



This symbol indicates that there is one or more of the following risks when using the described function:

- Danger to workpiece
- Danger to fixtures
- Danger to tool
- Danger to machine
- Danger to operator



This symbol indicates that the described function must be adapted by the machine tool builder. The function described may therefore vary depending on the machine.



This symbol indicates that you can find detailed information about a function in another manual.

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

We are continuously striving to improve our documentation for you. Please help us by sending your requests to the following e-mail address: **[tnc-userdoc@heidenhain.de](mailto:tnc-userdoc@heidenhain.de)**.



# TNC model, software and features

This manual describes functions and features provided by TNCs as of the following NC software numbers.

TNC model	NC software number
TNC 128	771841-01
TNC 128 Programming Station	771844-01

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

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

- Probing functions for the 3-D touch probe

Please contact your machine tool builder to become familiar with the features of your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.



# Software options

The TNC 128 features various software options that can be enabled by your machine tool builder. Each option is to be enabled separately and contains the following respective functions:

## Touch probe function (option number 17)

### Touch probe cycles

- Datum setting
- Workpiece measurement

## HEIDENHAIN DNC (option number 18)

Communication with external PC applications over COM component

## Additional conversational language (option number 41)

Function for enabling the conversational languages Slovenian, Slovak, Norwegian, Latvian, Estonian, Korean, Turkish, Romanian, Lithuanian.



## Feature content level (upgrade functions)

Along with software options, significant further improvements of the TNC software are managed via the Feature Content Level (**FCL**) upgrade functions. Functions subject to the FCL are not available simply by updating the software on your TNC.



All upgrade functions are available to you without surcharge when you receive a new machine.

Upgrade functions are identified in the manual with **FCL n**, where **n** indicates the sequential number of the feature content level.

You can purchase a code number in order to permanently enable the FCL functions. For more information, contact your machine tool builder or HEIDENHAIN.

## Intended place of operation

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

## Legal information

This product uses open source software. Further information is available on the control under

- Programming and Editing operating mode
- MOD function
- LICENSE INFO soft key

# Contents

First steps with the TNC 128	1
Introduction	2
Programming: Fundamentals, file management	3
Programming: Programming aids	4
Programming: Tools	5
Programming: Tool movements	6
Programming: Subprograms and program section repeats	7
Programming: Q parameters	8
Programming: Miscellaneous functions	9
Programming: Special functions	10
Manual operation and setup	11
Positioning with manual data input	12
Test run and program run	13
MOD functions	14
Fundamentals of cycles	15
Drilling, boring and thread cycles	16
Fixed cycles: Pocket milling / stud milling / slot milling	17
Cycles: Coordinate transformations	18
Cycles: Special functions	19
Touch probe cycles	20
Tables and overviews	21



## 1 First steps with the TNC 128 ..... 35

- 1.1 Overview ..... 36
- 1.2 Machine switch-on ..... 37
  - Acknowledging the power interruption and moving to the reference points ..... 37
- 1.3 Programming the first part ..... 38
  - Selecting the correct operating mode ..... 38
  - The most important TNC keys ..... 38
  - Creating a new program/file management ..... 39
  - Defining a workpiece blank ..... 40
  - Program layout ..... 41
  - Programming a simple contour ..... 42
  - Creating a cycle program ..... 45
- 1.4 Graphically testing the first part ..... 48
  - Selecting the correct operating mode ..... 48
  - Selecting the tool table for the test run ..... 48
  - Choosing the program you want to test ..... 49
  - Selecting the screen layout and the view ..... 49
  - Starting the test run ..... 50
- 1.5 Tool setup ..... 51
  - Selecting the correct operating mode ..... 51
  - Preparing and measuring tools ..... 51
  - The tool table TOOL.T ..... 51
- 1.6 Workpiece setup ..... 52
  - Selecting the correct operating mode ..... 52
  - Clamp the workpiece ..... 52
  - Datum setting with a 3-D touch probe ..... 53
- 1.7 Running the first program ..... 54
  - Selecting the correct operating mode ..... 54
  - Choosing the program you want to run ..... 54
  - Starting the program ..... 54



## 2 Introduction ..... 55

- 2.1 The TNC 128 ..... 56
  - Programming: HEIDENHAIN conversational format ..... 56
  - Compatibility ..... 56
- 2.2 Visual display unit and keyboard ..... 57
  - Visual display unit ..... 57
  - Setting the screen layout ..... 58
  - Operating panel ..... 59
- 2.3 Operating modes ..... 60
  - Manual Operation and El. Handwheel ..... 60
  - Positioning with Manual Data Input ..... 60
  - Programming and Editing ..... 61
  - Test Run ..... 61
  - Program Run, Full Sequence and Program Run, Single Block ..... 62
- 2.4 Status displays ..... 63
  - "General" status display ..... 63
  - Additional status displays ..... 64
- 2.5 Window manager ..... 71
  - Task bar ..... 72
- 2.6 Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels ..... 73
  - 3-D touch probes ..... 73
  - HR electronic handwheels ..... 73





3.1 Fundamentals .....	76
Position encoders and reference marks .....	76
Reference system .....	76
Reference system on milling machines .....	77
Designation of the axes on milling machines .....	77
Absolute and incremental workpiece positions .....	78
Setting the datum .....	79
3.2 Creating and writing programs .....	80
Organization of an NC program in HEIDENHAIN conversational format .....	80
Defining the workpiece blank: BLK FORM .....	80
Creating a new part program .....	81
Programming tool movements in conversational format .....	83
Actual position capture .....	85
Editing a program .....	86
The TNC search function .....	90
3.3 File management: Fundamentals .....	92
Files .....	92
Showing externally created files on the TNC .....	94
Data backup .....	94
3.4 Working with the file manager .....	95
Directories .....	95
Paths .....	95
Overview: Functions of the file manager .....	96
Calling the file manager .....	97
Selecting drives, directories and files .....	98
Creating a new directory .....	100
Creating a new file .....	100
Copying a single file .....	101
Copying files into another directory .....	102
Copying a table .....	103
Copying a directory .....	103
Choosing one of the last files selected .....	104
Deleting a file .....	104
Deleting a directory .....	105
Tagging files .....	106
Renaming a file .....	107
File sorting .....	107
Additional functions .....	108
Additional tools for management of external file types .....	109
Data transfer to or from an external data medium .....	114
The TNC in a network .....	116
USB devices on the TNC .....	117



## 4 Programming: Programming aids ..... 119

- 4.1 Screen keyboard ..... 120
  - Entering text with the screen keyboard ..... 120
- 4.2 Adding comments ..... 121
  - Application ..... 121
  - Adding comments ..... 121
  - Functions for editing of the comment ..... 121
- 4.3 Structuring programs ..... 122
  - Definition and applications ..... 122
  - Displaying the program structure window / Changing the active window ..... 122
  - Inserting a structuring block in the (left) program window ..... 122
  - Selecting blocks in the program structure window ..... 122
- 4.4 Integrated calculator ..... 123
  - Operation ..... 123
- 4.5 Programming graphics ..... 125
  - Generating / not generating graphics during programming ..... 125
  - Generating a graphic for an existing program ..... 125
  - Block number display ON/OFF ..... 126
  - Erasing the graphic ..... 126
  - Showing grid lines ..... 126
  - Magnifying or reducing a detail ..... 126
- 4.6 Error messages ..... 127
  - Display of errors ..... 127
  - Opening the error window ..... 127
  - Closing the error window ..... 127
  - Detailed error messages ..... 128
  - INTERNAL INFO soft key ..... 128
  - Clearing errors ..... 129
  - Error log ..... 129
  - Keystroke log ..... 130
  - Informational texts ..... 131
  - Saving service files ..... 131
  - Calling the TNCguide help system ..... 131
- 4.7 Context-sensitive help system ..... 132
  - Application ..... 132
  - Working with the TNCguide ..... 133
  - Downloading current help files ..... 137



## 5 Programming: Tools ..... 139

- 5.1 Entering tool-related data ..... 140
  - Feed rate F ..... 140
  - Spindle speed S ..... 141
- 5.2 Tool data ..... 142
  - Requirements for tool compensation ..... 142
  - Tool numbers and tool names ..... 142
  - Tool length L ..... 142
  - Tool radius R ..... 142
  - Delta values for lengths and radii ..... 143
  - Entering tool data into the program ..... 143
  - Entering tool data in the table ..... 144
  - Calling tool data ..... 151
  - Tool change ..... 152
- 5.3 Tool compensation ..... 155
  - Introduction ..... 155
  - Tool length compensation ..... 155
  - Tool radius compensation for paraxial positioning blocks ..... 156



## **6 Programming: Tool movements ..... 159**

### 6.1 Fundamentals ..... 160

Tool movements in the program ..... 160

Radius compensation ..... 160

Miscellaneous functions M ..... 160

Subprograms and program section repeats ..... 161

Programming with Q parameters ..... 161

### 6.2 Programming tool movements ..... 162

Programming tool movements for workpiece machining ..... 162

## 7 Programming: Subprograms and program section repeats ..... 165

- 7.1 Labeling subprograms and program section repeats ..... 166
  - Labels ..... 166
- 7.2 Subprograms ..... 167
  - Operating sequence ..... 167
  - Programming notes ..... 167
  - Programming a subprogram ..... 167
  - Calling a subprogram ..... 167
- 7.3 Program section repeats ..... 168
  - Label LBL ..... 168
  - Operating sequence ..... 168
  - Programming notes ..... 168
  - Programming a program section repeat ..... 168
  - Calling a program section repeat ..... 168
- 7.4 Separate program as subprogram ..... 169
  - Operating sequence ..... 169
  - Programming notes ..... 169
  - Calling any program as a subprogram ..... 170
- 7.5 Nesting ..... 171
  - Types of nesting ..... 171
  - Nesting depth ..... 171
  - Subprogram within a subprogram ..... 172
  - Repeating program section repeats ..... 173
  - Repeating a subprogram ..... 174
- 7.6 Programming examples ..... 175



## 8 Programming: Q parameters ..... 179

- 8.1 Principle and overview ..... 180
  - Programming notes ..... 181
  - Calling Q-parameter functions ..... 182
- 8.2 Part families—Q parameters in place of numerical values ..... 183
  - Application ..... 183
- 8.3 Describing contours through mathematical operations ..... 184
  - Application ..... 184
  - Overview ..... 184
  - Programming fundamental operations ..... 185
- 8.4 Trigonometric functions ..... 186
  - Definitions ..... 186
  - Programming trigonometric functions ..... 187
- 8.5 Circle calculations ..... 188
  - Application ..... 188
- 8.6 If-then decisions with Q parameters ..... 189
  - Application ..... 189
  - Unconditional jumps ..... 189
  - Programming if-then decisions ..... 190
  - Abbreviations used: ..... 190
- 8.7 Checking and changing Q parameters ..... 191
  - Procedure ..... 191
- 8.8 Additional functions ..... 192
  - Overview ..... 192
  - FN 14: ERROR: Display error messages ..... 193
  - FN 16: F-PRINT: Formatted output of text and Q-parameter values ..... 198
  - FN 18: SYS-DATUM READ ..... 202
  - FN 19: PLC: Transfer values to the PLC ..... 211
  - FN 20: WAIT FOR: NC and PLC synchronization ..... 211
  - FN 29: PLC: Transfer values to the PLC ..... 213
  - FN 37: EXPORT ..... 214
- 8.9 Accessing tables with SQL commands ..... 215
  - Introduction ..... 215
  - A transaction ..... 216
  - Programming SQL commands ..... 218
  - Overview of the soft keys ..... 218
  - SQL BIND ..... 219
  - SQL SELECT ..... 220
  - SQL FETCH ..... 223
  - SQL UPDATE ..... 224
  - SQL INSERT ..... 224
  - SQL COMMIT ..... 225
  - SQL ROLLBACK ..... 225



8.10 Entering formulas directly .....	226
Entering formulas .....	226
Rules for formulas .....	228
Programming example .....	229
8.11 String parameters .....	230
String processing functions .....	230
Assigning string parameters .....	231
Chain-linking string parameters .....	232
Converting a numerical value to a string parameter .....	233
Copying a substring from a string parameter .....	234
Converting a string parameter to a numerical value .....	235
Checking a string parameter .....	236
Finding the length of a string parameter .....	237
Comparing alphabetic priority .....	238
Reading machine parameters .....	239
8.12 Preassigned Q parameters .....	242
Values from the PLC: Q100 to Q107 .....	242
Active tool radius: Q108 .....	242
Tool axis: Q109 .....	243
Spindle status: Q110 .....	243
Coolant on/off: Q111 .....	243
Overlap factor: Q112 .....	243
Unit of measurement for dimensions in the program: Q113 .....	244
Tool length: Q114 .....	244
Coordinates after probing during program run .....	244
Deviation between actual value and nominal value during automatic tool measurement with the TT 130 .....	245



## 9 Programming: Miscellaneous functions ..... 247

- 9.1 Entering miscellaneous functions M and STOP ..... 248
  - Fundamentals ..... 248
- 9.2 Miscellaneous functions for program run control, spindle and coolant ..... 249
  - Overview ..... 249
- 9.3 Miscellaneous functions for coordinate data ..... 250
  - Programming machine-referenced coordinates: M91/M92 ..... 250
  - Reducing display of a rotary axis to a value less than 360°: M94 ..... 252
- 9.4 Miscellaneous functions for contouring behavior ..... 253
  - Feed rate factor for plunging movements: M103 ..... 253
  - Feed rate in millimeters per spindle revolution: M136 ..... 254
  - Retraction from the contour in the tool-axis direction: M140 ..... 255
  - Suppressing touch probe monitoring: M141 ..... 256





## 10 Programming: Special functions ..... 257

- 10.1 Overview of special functions ..... 258
  - Main menu for SPEC FCT special functions ..... 258
  - Program defaults menu ..... 259
  - Functions for contour and point machining menu ..... 259
  - Menu for defining various plain-language functions ..... 260
- 10.2 Freely definable tables ..... 261
  - Fundamentals ..... 261
  - Creating a freely definable table ..... 261
  - Editing the table format ..... 262
  - Switching between table and form view ..... 264
  - FN26: TABOPEN: Opening a freely definable table ..... 265
  - FN 27: TABWRITE: Writing to a freely definable table ..... 266
  - FN28: TABREAD: Reading a freely definable table ..... 267
- 10.3 File functions ..... 268
  - Application ..... 268
  - Defining file functions ..... 268
- 10.4 Defining coordinate transformations ..... 269
  - Overview ..... 269
  - TRANS DATUM AXIS ..... 269
  - TRANS DATUM TABLE ..... 270
  - TRANS DATUM RESET ..... 270
- 10.5 Creating text files ..... 271
  - Application ..... 271
  - Opening and exiting text files ..... 271
  - Editing texts ..... 272
  - Deleting and re-inserting characters, words and lines ..... 272
  - Editing text blocks ..... 273
  - Finding text sections ..... 274



## 11 Manual operation and setup ..... 275

- 11.1 Switch-on, switch-off ..... 276
  - Switch-on ..... 276
  - Switch-off ..... 277
- 11.2 Moving the machine axes ..... 278
  - Note ..... 278
  - Moving the axis using the machine axis direction buttons ..... 278
  - Incremental jog positioning ..... 279
  - Traversing with the HR 410 electronic handwheel ..... 280
- 11.3 Spindle speed S, feed rate F and miscellaneous functions M ..... 281
  - Function ..... 281
  - Entering values ..... 281
  - Changing the spindle speed and feed rate ..... 282
  - Activating feed-rate limitation ..... 283
- 11.4 Datum setting without a 3-D touch probe ..... 284
  - Note ..... 284
  - Preparation ..... 284
  - Workpiece presetting with axis keys ..... 285
  - Datum management with the preset table ..... 286
- 11.5 Using the 3-D touch probe ..... 291
  - Overview ..... 291
  - Functions in touch probe cycles ..... 292
  - Selecting touch probe cycles ..... 293
  - Recording measured values from the touch-probe cycles ..... 294
  - Writing the measured values from touch probe cycles to datum tables ..... 295
  - Writing the measured values from touch probe cycles to the preset table ..... 296
- 11.6 Calibrating a 3-D touch probe ..... 297
  - Introduction ..... 297
  - Calibrating the effective length ..... 298
  - Calibrating the effective radius and compensating center offset ..... 299
  - Displaying calibration values ..... 302
- 11.7 Datum setting with a 3-D touch probe ..... 303
  - Overview ..... 303
  - Datum setting in any axis ..... 303
  - Circle center as datum ..... 304
  - Center line as datum ..... 307
  - Measuring workpieces with a 3-D touch probe ..... 308
  - Using touch probe functions with mechanical probes or dial gauges ..... 310

## **12 Positioning with manual data input ..... 311**

- 12.1 Programming and executing simple machining operations ..... 312
  - Positioning with manual data input (MDI) ..... 312
  - Protecting and erasing programs in \$MDI ..... 314



## 13 Test run and program run ..... 315

- 13.1 Graphics ..... 316
  - Application ..... 316
  - Setting the speed of the test run ..... 317
  - Overview of display modes ..... 318
  - Plan view ..... 318
  - Projection in 3 planes ..... 319
  - 3-D view ..... 320
  - Magnifying details ..... 322
  - Repeating graphic simulation ..... 323
  - Displaying the tool ..... 323
  - Measuring the machining time ..... 324
- 13.2 Showing the workpiece blank in the working space ..... 325
  - Application ..... 325
- 13.3 Functions for program display ..... 326
  - Overview ..... 326
- 13.4 Test Run ..... 327
  - Application ..... 327
- 13.5 Program run ..... 330
  - Application ..... 330
  - Running a part program ..... 331
  - Interrupting machining ..... 332
  - Moving the machine axes during an interruption ..... 333
  - Resuming program run after an interruption ..... 334
  - Mid-program startup (block scan) ..... 335
  - Returning to the contour ..... 337
- 13.6 Optional block skip ..... 338
  - Application ..... 338
  - Inserting the "/" character ..... 338
  - Erasing the "/" character ..... 338
- 13.7 Optional program-run interruption ..... 339
  - Application ..... 339



## 14 MOD functions ..... 341

- 14.1 Selecting MOD functions ..... 342
  - Selecting the MOD functions ..... 342
  - Changing the settings ..... 342
  - Exiting the MOD functions ..... 342
  - Overview of MOD functions ..... 343
- 14.2 Software numbers ..... 344
  - Application ..... 344
- 14.3 Entering code numbers ..... 345
  - Application ..... 345
- 14.4 Setting the data interfaces ..... 346
  - Serial interfaces on the TNC 128 ..... 346
  - Application ..... 346
  - Setting the RS-232 interface ..... 346
  - Setting the baud rate (baudRate) ..... 346
  - Setting the protocol (protocol) ..... 346
  - Setting the data bits (dataBits) ..... 347
  - Parity check (parity) ..... 347
  - Setting the stop bits (stopBits) ..... 347
  - Setting the handshake (flowControl) ..... 347
  - Settings for data transfer with the TNCserver PC software ..... 348
  - Setting the operating mode of the external device (fileSystem) ..... 348
  - Software for data transfer ..... 349
- 14.5 Ethernet interface ..... 351
  - Introduction ..... 351
  - Connection possibilities ..... 351
  - Connecting the control to the network ..... 352
- 14.6 Position display types ..... 357
  - Application ..... 357
- 14.7 Unit of measurement ..... 358
  - Application ..... 358
- 14.8 Displaying operating times ..... 359
  - Application ..... 359



## 15 Fundamentals of cycles ..... 361

- 15.1 Introduction ..... 362
- 15.2 Available cycle groups ..... 363
  - Overview of fixed cycles ..... 363
- 15.3 Working with fixed cycles ..... 364
  - Machine-specific cycles ..... 364
  - Defining a cycle using soft keys ..... 365
  - Defining a cycle using the GOTO function ..... 365
  - Calling cycles ..... 366
- 15.4 PATTERN DEF pattern definition ..... 368
  - Application ..... 368
  - Entering PATTERN DEF ..... 369
  - Using PATTERN DEF ..... 369
  - Defining individual machining positions ..... 370
  - Defining a single row ..... 371
  - Defining a single pattern ..... 372
  - Defining individual frames ..... 373
  - Defining a full circle ..... 374
  - Defining a pitch circle ..... 375
- 15.5 POLAR PATTERN (Cycle 220) ..... 376
  - Cycle run ..... 376
  - Please note while programming: ..... 376
  - Cycle parameters ..... 377
- 15.6 CARTESIAN PATTERN (Cycle 221) ..... 379
  - Cycle run ..... 379
  - Please note while programming: ..... 379
  - Cycle parameters ..... 380
- 15.7 Point tables ..... 381
  - Application ..... 381
  - Creating a point table ..... 381
  - Hiding single points from the machining process ..... 382
  - Selecting a point table in the program ..... 382
  - Calling a cycle in connection with point tables ..... 383



## 16 Drilling, boring and thread cycles ..... 385

- 16.1 Fundamentals ..... 386
  - Overview ..... 386
- 16.2 CENTERING (Cycle 240) ..... 387
  - Cycle run ..... 387
  - Please note while programming: ..... 387
  - Cycle parameters ..... 388
- 16.3 DRILLING (Cycle 200) ..... 389
  - Cycle run ..... 389
  - Please note while programming: ..... 389
  - Cycle parameters ..... 390
- 16.4 REAMING (Cycle 201) ..... 391
  - Cycle run ..... 391
  - Please note while programming: ..... 391
  - Cycle parameters ..... 392
- 16.5 BORING (Cycle 202) ..... 393
  - Cycle run ..... 393
  - Please note while programming: ..... 394
  - Cycle parameters ..... 395
- 16.6 UNIVERSAL DRILLING (Cycle 203) ..... 397
  - Cycle run ..... 397
  - Please note while programming: ..... 398
  - Cycle parameters ..... 399
- 16.7 BACK BORING (Cycle 204) ..... 401
  - Cycle run ..... 401
  - Please note while programming: ..... 402
  - Cycle parameters ..... 403
- 16.8 UNIVERSAL PECKING (Cycle 205) ..... 405
  - Cycle run ..... 405
  - Please note while programming: ..... 406
  - Cycle parameters ..... 407
- 16.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241) ..... 409
  - Cycle run ..... 409
  - Please note while programming: ..... 409
  - Cycle parameters ..... 410



16.10 Programming examples .....	412
16.11 TAPPING NEW with floating tap holder (Cycle 206) .....	416
Cycle run .....	416
Please note while programming: .....	416
Cycle parameters .....	417
16.12 RIGID TAPPING without a floating tap holder NEW (Cycle 207) .....	418
Cycle run .....	418
Please note while programming: .....	419
Cycle parameters .....	420
16.13 Programming example .....	421





## 17 Fixed cycles: Pocket milling / stud milling / slot milling ..... 425

- 17.1 Fundamentals ..... 426
  - Overview ..... 426
- 17.2 RECTANGULAR POCKET (Cycle 251) ..... 427
  - Cycle run ..... 427
  - Please note while programming: ..... 428
  - Cycle parameters ..... 429
- 17.3 RECTANGULAR STUD (Cycle 256) ..... 432
  - Cycle run ..... 432
  - Please note while programming: ..... 433
  - Cycle parameters ..... 434
- 17.4 Programming examples ..... 437



## 18 Cycles: Coordinate transformations ..... 441

- 18.1 Fundamentals ..... 442
  - Overview ..... 442
  - Effect of coordinate transformations ..... 442
- 18.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54) ..... 443
  - Effect ..... 443
  - Cycle parameters ..... 443
- 18.3 DATUM SHIFT with datum tables (Cycle 7) ..... 444
  - Effect ..... 444
  - Please note while programming: ..... 445
  - Cycle parameters ..... 446
  - Selecting a datum table in the part program ..... 446
  - Editing the datum table in the Programming and Editing mode of operation ..... 447
  - Configuring the datum table ..... 448
  - Exiting a datum table ..... 448
  - Status displays ..... 448
- 18.4 DATUM SETTING (Cycle 247) ..... 449
  - Effect ..... 449
  - Please note before programming: ..... 449
  - Cycle parameters ..... 449
  - Status displays ..... 449
- 18.5 MIRROR IMAGE (Cycle 8) ..... 450
  - Effect ..... 450
  - Please note while programming: ..... 450
  - Cycle parameters ..... 451
- 18.6 SCALING FACTOR (Cycle 11) ..... 452
  - Effect ..... 452
  - Cycle parameters ..... 452
- 18.7 AXIS-SPECIFIC SCALING (Cycle 26) ..... 453
  - Effect ..... 453
  - Please note while programming: ..... 453
  - Cycle parameters ..... 454
- 18.8 Programming examples ..... 455



## 19 Cycles: Special functions ..... 457

- 19.1 Fundamentals ..... 458
  - Overview ..... 458
- 19.2 DWELL TIME (Cycle 9) ..... 459
  - Function ..... 459
  - Cycle parameters ..... 459
- 19.3 PROGRAM CALL (Cycle 12) ..... 460
  - Cycle function ..... 460
  - Please note while programming: ..... 460
  - Cycle parameters ..... 461
- 19.4 ORIENTED SPINDLE STOP (Cycle 13) ..... 462
  - Cycle function ..... 462
  - Please note while programming: ..... 462
  - Cycle parameters ..... 462



## 20 Touch probe cycles ..... 463

- 20.1 General information about touch probe cycles ..... 464
  - Method of function ..... 464
  - Touch probe cycles in the Manual Operation and El. Handwheel modes ..... 464
- 20.2 Before you start working with touch probe cycles ..... 465
  - Maximum traverse to touch point: DIST in touch probe table ..... 465
  - Set-up clearance to touch point: SET\_UP in touch probe table ..... 465
  - Orient the infrared touch probe to the programmed probe direction: TRACK in touch probe table ..... 465
  - Touch trigger probe, probing feed rate: F in touch probe table ..... 466
  - Touch trigger probe, rapid traverse for positioning: FMAX ..... 466
  - Touch trigger probe, rapid traverse for positioning: F\_PREPOS in touch probe table ..... 466
  - Executing touch probe cycles ..... 466
- 20.3 Touch probe table ..... 467
  - General information ..... 467
  - Editing touch probe tables ..... 467
  - Touch probe data ..... 468
- 20.4 Fundamentals of tool measurement ..... 469
  - Overview ..... 469
  - Setting the machine parameters ..... 471
  - Entries in the tool table TOOL.T ..... 472
- 20.5 Calibrating the TT (Cycle 480) ..... 474
  - Cycle run ..... 474
  - Please note while programming: ..... 474
  - Cycle parameters ..... 474
- 20.6 Calibrating the wireless TT 449 (Cycle 484) ..... 475
  - Fundamentals ..... 475
  - Cycle run ..... 475
  - Please note while programming: ..... 475
  - Cycle parameters ..... 475
- 20.7 Measuring the tool length (Cycle 481) ..... 476
  - Cycle run ..... 476
  - Please note while programming: ..... 476
  - Cycle parameters ..... 477
- 20.8 Measuring the tool radius (Cycle 482) ..... 478
  - Cycle run ..... 478
  - Please note while programming: ..... 478
  - Cycle parameters ..... 479
- 20.9 Measuring the tool length and radius (Cycle 483) ..... 480
  - Cycle run ..... 480
  - Please note while programming: ..... 480
  - Cycle parameters ..... 481



## 21 Tables and overviews ..... 483

- 21.1 Machine-specific user parameters ..... 484
  - Application ..... 484
- 21.2 Pin layouts and connecting cables for the data interfaces ..... 492
  - RS-232-C/V.24 interface for HEIDENHAIN devices ..... 492
  - Non-HEIDENHAIN devices ..... 493
  - Ethernet interface RJ45 socket ..... 493
- 21.3 Technical information ..... 494





# 1

**First steps with the  
TNC 128**



## 1.1 Overview

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

The following topics are included in this chapter:

- Machine switch-on
- Programming the first part
- Graphically testing the first part
- Tool setup
- Workpiece setup
- Running the first program





## 1.2 Machine switch-on

### Acknowledging the power interruption and moving to the reference points



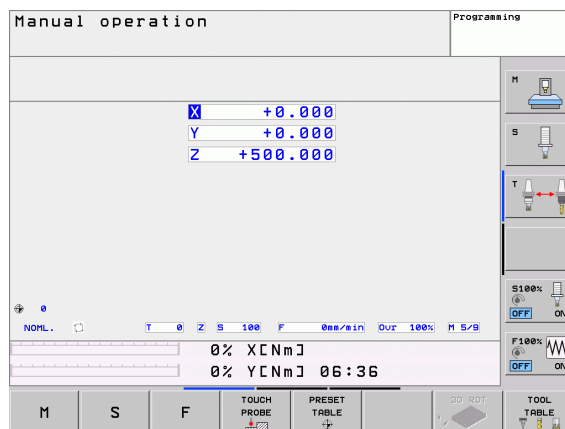
Switch-on and crossing the reference points can vary depending on the machine tool. Your machine manual provides more detailed information.

- Switch on the power supply for control and machine. The TNC starts the operating system. This process may take several minutes. Then the TNC will display the message "Power interrupted" in the screen header
- CE** ► Press the CE key: The TNC compiles the PLC program
- I** ► Switch on the control voltage: The TNC checks operation of the emergency stop circuit and goes into the reference run mode
- I** ► Cross the reference points manually in the displayed sequence: For each axis press the machine START button. If you have absolute linear and angle encoders on your machine there is no need for a reference run

The TNC is now ready for operation in the **Manual Operation** mode.

#### Further information on this topic

- Traversing the reference marks: See "Switch-on" on page 276
- Operating modes: See "Programming and Editing" on page 61



## 1.3 Programming the first part

### Selecting the correct operating mode

You can write programs only in the Programming and Editing mode:








- Press the operating modes key: The TNC goes into the **Programming and Editing** mode

#### Further information on this topic

- Operating modes: See "Programming and Editing" on page 61

### The most important TNC keys

Functions for conversational guidance	Key
Confirm entry and activate the next dialog prompt	
Ignore the dialog question	
End the dialog immediately	
Abort dialog, discard entries	
Soft keys on the screen with which you select functions appropriate to the active state	

#### Further information on this topic

- Writing and editing programs: See "Editing a program" on page 86
- Overview of keys: See "Controls of the TNC" on page 2

## Creating a new program/file management

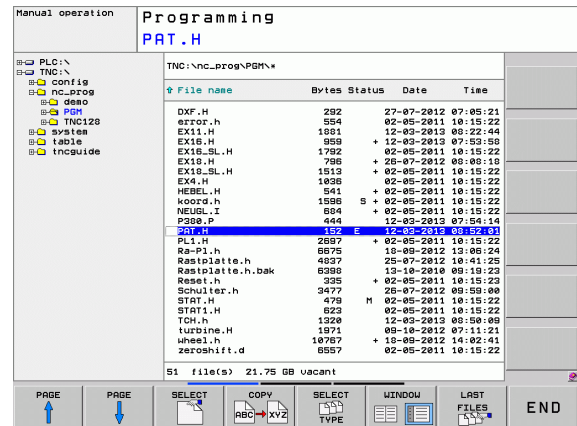
PGM  
MGT

- Press the PGM MGT key: The TNC opens the file manager. The file management of the TNC is arranged much like the file management on a PC with the Windows Explorer. The file management enables you to manipulate data on the TNC hard disk
- Use the arrow keys to select the folder in which you want to open the new file
- Enter a file name with the extension **.H**: The TNC then automatically opens a program and asks for the unit of measure for the new program
- To select the unit of measure, press the MM or INCH soft key: The TNC automatically starts the workpiece blank definition (see "Defining a workpiece blank" on page 40)

The TNC automatically generates the first and last blocks of the program. Afterwards you can no longer change these blocks.

### Further information on this topic

- File management: See "Working with the file manager" on page 95
- Creating a new program: See "Creating and writing programs" on page 80



## Defining a workpiece blank

Immediately after you have created a new program, the TNC starts the dialog for entering the workpiece blank definition. Always define the workpiece blank as a cuboid by entering the MIN and MAX points, each with reference to the selected reference point.

After you have created a new program, the TNC automatically initiates the workpiece blank definition and asks for the required data:

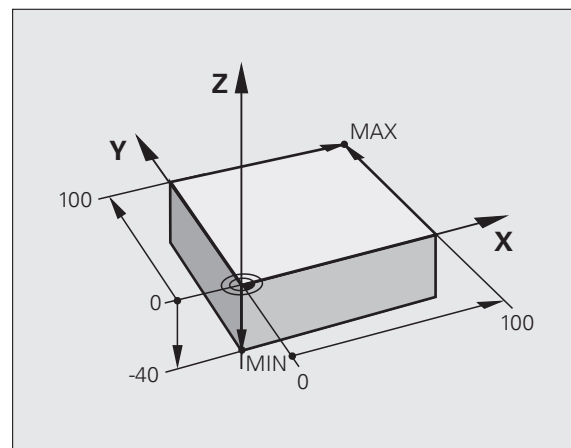
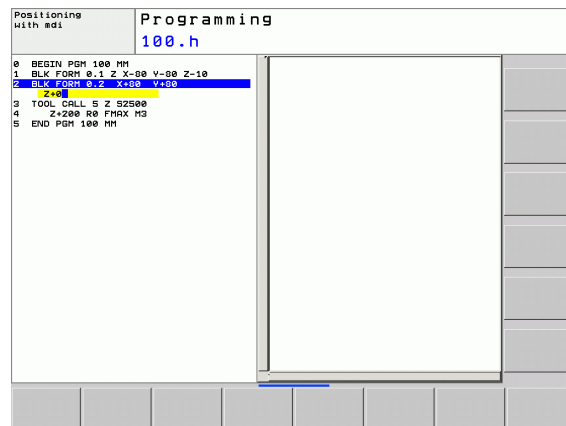
- ▶ **Working plane in graphic: XY?**: Enter the active spindle axis. Z is saved as default setting. Accept with the ENT key
- ▶ **Workpiece blank def.: minimum X**: Enter the smallest X coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the ENT key
- ▶ **Workpiece blank def.: minimum Y**: Smallest Y coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the ENT key
- ▶ **Workpiece blank def.: minimum Z**: Smallest Z coordinate of the workpiece blank with respect to the reference point, e.g. -40. Confirm with the ENT key
- ▶ **Workpiece blank def.: maximum X**: Largest X coordinate of the workpiece blank with respect to the reference point, e.g. 100. Confirm with the ENT key
- ▶ **Workpiece blank def.: maximum Y**: Largest Y coordinate of the workpiece blank with respect to the reference point, e.g. 100. Confirm with the ENT key
- ▶ **Workpiece blank def.: maximum Z**: Largest Z coordinate of the workpiece blank with respect to the reference point, e.g. 0. Confirm with the ENT key. The TNC concludes the dialog

### Example NC blocks

```
0 BEGIN PGM NEW MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 END PGM NEW MM
```

### Further information on this topic

- Defining the workpiece blank: (see page 81)



# 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

- 1 Call tool, define tool axis
- 2 Retract the tool
- 3 Pre-position the first axis in the working plane in front of the contour starting point
- 4 Pre-position the second axis in the working plane next to the contour starting point
- 5 In the tool axis, position the tool above the workpiece, or preposition immediately to workpiece depth. If required, switch on the spindle/coolant
- 6 Approach the contour
- 7 Machine the contour
- 8 Depart the contour
- 9 Retract the tool, end the program

Further information on this topic:

- Contour programming: See "Fundamentals" on page 160

## Recommended program layout for simple cycle programs

- 1 Call the tool, define the tool axis
- 2 Retract the tool
- 3 Define the machining positions
- 4 Define the fixed cycle
- 5 Call the cycle, switch on the spindle/coolant
- 6 Retract the tool, end the program

Further information on this topic:

- Cycle programming: See "Working with fixed cycles" on page 364.

### Example: Layout of contour machining programs

```

0 BEGIN PGM BSPCONT MM
1 BLK FORM 0.1 Z X... Y... Z...
2 BLK FORM 0.2 X... Y... Z...
3 TOOL CALL 5 Z S5000
4   Z+250 R0 FMAX
5   X... R0 FMAX
6   Y... R0 FMAX
7   Z+10 R0 F3000 M13
...
16   X... R0 FMAX
17   Z+250 R0 FMAX M2
18 END PGM BSPCONT MM

```

### Example: Cycle program layout

```

0 BEGIN PGM BSBCYC MM
1 BLK FORM 0.1 Z X... Y... Z...
2 BLK FORM 0.2 X... Y... Z...
3 TOOL CALL 5 Z S5000
4   Z+250 R0 FMAX
5 PATTERN DEF POS1( X... Y... Z... ) ...
6 CYCL DEF...
7 CYCL CALL PAT FMAX M13
8   Z+250 R0 FMAX M2
9 END PGM BSBCYC MM

```



## Programming a simple contour

The contour shown to the right is to be milled once to a depth of 5 mm. You have already defined the workpiece blank. After you have initiated a dialog through a function key, enter all the data requested by the TNC in the screen header.

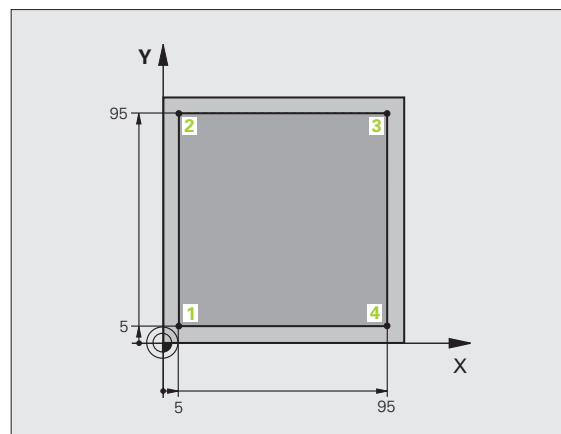
TOOL  
CALL

Z

X

Y

- ▶ Call the tool: Enter the tool data. Confirm each of your entries with the ENT key. Do not forget the tool axis
- ▶ Retract the tool: Press the orange Z axis key in order to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- ▶ Confirm **Radius comp.: R+/R-/no comp?** by pressing the ENT key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (**FMAX**)
- ▶ Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block
- ▶ Preposition the tool in the working plane: Press the orange X axis key and enter the value for the position to be approached, e.g. -20
- ▶ Confirm **Radius comp.: R+/R-/no comp?** by pressing the ENT key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (**FMAX**)
- ▶ Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block
- ▶ Press the orange Y axis key and enter the value for the position to be approached, e.g. -20. Confirm with the ENT key
- ▶ Confirm **Radius comp.: R+/R-/no comp?** by pressing the ENT key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (**FMAX**)
- ▶ Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block



Z

- ▶ Move the tool to workpiece depth: Press the orange Z axis key and enter the value for the position to be approached, e.g. -5. Confirm with the ENT key
- ▶ Confirm **Radius comp.: R+/R-/no comp?** by pressing the ENT key: Do not activate the radius compensation
- ▶ **Feed rate F=?** Enter the positioning feed rate, e.g. 3000 mm/min and confirm with the ENT key
- ▶ **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**. Confirm with the END key: The TNC saves the entered positioning block

X

- ▶ Approach contour point **1**: Press the orange X axis key and enter the value 5 for the position to be approached
- ▶ **Tool radius comp: R+/R-/no comp?** Select the R- soft key: The traverse path is decreased by the tool radius
- ▶ **Feed rate F=?** Enter the machining feed rate, e.g. 700 mm/min, and confirm your entry with the END key

Y

- ▶ Approach contour point **2**: Press the orange Y axis key and enter the value 95 for the position to be approached
- ▶ **Tool radius comp: R+/R-/no comp?** Select the R+ soft key: The traverse path is increased by the tool radius. Confirm your entry with the END soft key

X

- ▶ Approach contour point **3**: Press the orange X axis key and enter the value 95 for the position to be approached
- ▶ **Tool radius comp: R+/R-/no comp?** Select the R+ soft key: The traverse path is increased by the tool radius. Confirm your entry with the END soft key

Y

- ▶ Approach contour point **4**: Press the orange Y axis key and enter the value 5 for the position to be approached
- ▶ **Tool radius comp: R+/R-/no comp?** Select the R+ soft key: The traverse path is increased by the tool radius. Confirm your entry with the END soft key



X

► Approach contour point **1** and retract the tool: Press the orange X axis key and enter the value 0 for the position to be approached

Z

► **Tool radius comp: R+/R-/no comp?** Select the R+ soft key: The traverse path is increased by the tool radius. Confirm your entry with the END soft key

► Retract the tool: Press the orange Z axis key in order to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key

► Confirm **Radius comp.: R+/R-/no comp?** by pressing the ENT key: Do not activate the radius compensation

► Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (**FMAX**)

► **Miscellaneous function M?** Enter **M2** to end the program and confirm with the END key: The TNC saves the entered positioning block

## Further information on this topic

- Creating a new program: See "Creating and writing programs" on page 80
- Programmable feed rates: See "Possible feed rate input" on page 84
- Tool radius compensation: See "Tool radius compensation for paraxial positioning blocks" on page 156
- Miscellaneous functions (M): See "Miscellaneous functions for program run control, spindle and coolant" on page 249



## Creating a cycle program

The holes (depth of 20 mm) shown in the figure at right are to be drilled with a standard drilling cycle. You have already defined the workpiece blank.

TOOL  
CALL

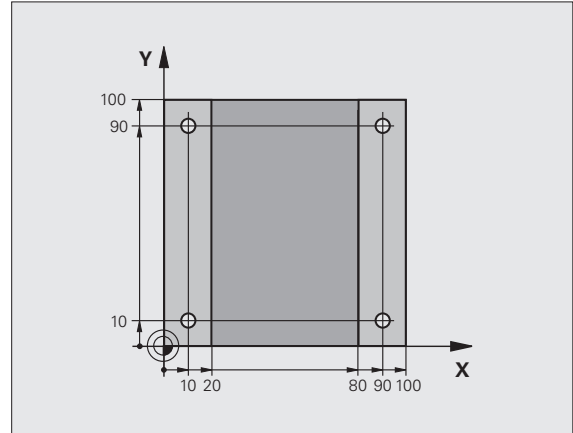
Z

CYCL  
DEF

DRILLING/  
THREAD

200

- Call the tool: Enter the tool data. Confirm each of your entries with the ENT key. Do not forget the tool axis
- Retract the tool: Press the orange Z axis key in order to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- Confirm **Radius comp.:** R+/R-/no comp? by pressing the ENT key: Do not activate the radius compensation
- Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (FMAX)
- Confirm the **Miscellaneous function M?** with the END key: The TNC saves the entered positioning block
- Call the cycle menu
- Display the drilling cycles
- Select the standard drilling cycle 200: The TNC starts the dialog for cycle definition. Enter all parameters requested by the TNC step by step and conclude each entry with the ENT key. In the screen to the right, the TNC also displays a graphic showing the respective cycle parameter



Manual operation	Programming
	<b>Depth?</b>
0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X=0 V=0 Z=20	
2 BLK FORM 0.2 X=100 V=100 Z=0	
3 TOOL CALL S 2 S2222	
4 X=20 R0 FMAX M3	
5 V=5 R0 FMAX	
6 Z=5 R0 FMAX	
7 X=90 R+ F200	
8 V=50 R+	
9 X=100	
10 CYCL DEF 200 DRILLING	
Q200=+2 ;SET-UP CLEARANCE	
Q201=+20 ;DEPTH	
Q202=+150 ;FEED RATE FOR PLUNGING	
Q203=+5 ;PLUNGING DEPTH	
Q210=+0 ;DUELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=+50 ;2ND SET-UP CLEARANCE	
Q211=+0 ;DUELL TIME AT DEPTH	
11 SEL PATTERN	
"TNC-NC-PT09TNC128-1.pnt"	
12 CYCL CALL PAT FMAX M3	
13 END PGM 1 MM	

SPEC  
FCT

CONTOUR  
+ POINT  
MACHINING

PATTERN  
DEF

POINT  
+

CYCL  
CALL

CYCLE  
CALL  
PAT

Z

- ▶ Call the menu for special functions
- ▶ Display the functions for point machining
- ▶ Select the pattern definition
- ▶ Select point entry: Enter the coordinates of the 4 points and confirm each with the ENT key. After entering the fourth point, save the block with the END key
- ▶ Display the menu for defining the cycle call
- ▶ Run the drilling cycle on the defined pattern:
- ▶ Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (**FMAX**)
- ▶ **Miscellaneous function M?** Switch on the spindle and coolant, e.g. **M13**. Confirm with the END key: The TNC saves the entered positioning block
- ▶ Retract the tool: Press the orange Z axis key in order to retract in the tool axis, and enter the value for the position to be approached, e.g. 250. Confirm with the ENT key
- ▶ Confirm **Radius comp.: RL/RR/no comp?** by pressing the ENT key: Do not activate the radius compensation
- ▶ Confirm **Feed rate F=?** with the ENT key: Move at rapid traverse (**FMAX**)
- ▶ **Miscellaneous function M?** Enter **M2** to end the program and confirm with the END key: The TNC saves the entered positioning block

**Example NC blocks**

0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 5 Z S4500	Tool call
4 Z+250 R0 FMAX	Retract the tool
5 PATTERN DEF POS1 (X+10 Y+10 Z+0) POS2 (X+10 Y+90 Z+0) POS3 (X+90 Y+90 Z+0) POS4 (X+90 Y+10 Z+0)	Define the machining positions
6 CYCL DEF 200 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	
7 CYCL CALL PAT FMAX M13	Spindle and coolant on, call the cycle
8 Z+250 R0 FMAX M2	Retract the tool, end program
9 END PGM C200 MM	

**Further information on this topic**

- Creating a new program: See "Creating and writing programs" on page 80
- Cycle programming: See User's Manual for Cycles



## 1.4 Graphically testing the first part

### Selecting the correct operating mode

You can test programs only in the Test Run mode:



- ▶ Press the **Test Run** operating mode key: the TNC switches to that mode

#### Further information on this topic

- Operating modes of the TNC: See "Operating modes" on page 60
- Testing programs: See "Test Run" on page 327

### Selecting the tool table for the test run

You only need to execute this step if you have not activated a tool table in the Test Run mode.



- ▶ Press the PGM MGT key: The TNC opens the file manager



- ▶ Press the SELECT TYPE soft key: The TNC shows a soft-key menu for selection of the file type to be displayed



- ▶ Press the SHOW ALL soft key: The TNC shows all saved files in the right window



- ▶ Move the highlight to the left onto the directories



- ▶ Move the highlight to the **TNC:\table** directory



- ▶ Move the highlight to the right onto the files



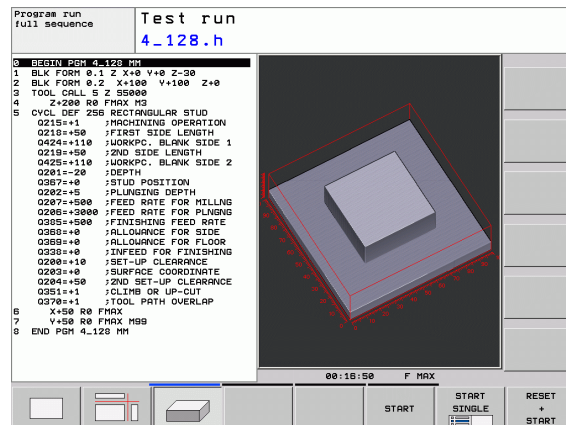
- ▶ Move the highlight to the file TOOL.T (active tool table) and load with the ENT key: TOOL.T receives the status **S** and is therefore active for the test run



- ▶ Press the END key: Exit the file manager

#### Further information on this topic

- Tool management: See "Entering tool data in the table" on page 144
- Testing programs: See "Test Run" on page 327



## Choosing the program you want to test



- ▶ Press the PGM MGT key: The TNC opens the file manager



- ▶ Press the LAST FILES soft key: The TNC opens a pop-up window with the most recently selected files
- ▶ Use the arrow keys to select the program that you want to test. Load with the ENT key

### Further information on this topic

- Selecting a program: See "Working with the file manager" on page 95

## Selecting the screen layout and the view



- ▶ Press the key for selecting the screen layout. The TNC shows all available alternatives in the soft-key row



- ▶ Press the PROGRAM + GRAPHICS soft key: In the left half of the screen the TNC shows the program; in the right half it shows the workpiece blank

- ▶ Select the desired view via soft key



- ▶ Plan view



- ▶ Projection in three planes



- ▶ 3-D view

### Further information on this topic

- Graphic functions: See "Graphics" on page 316
- Running a test run: See "Test Run" on page 327



## Starting the test run

RESET  
+  
START

▶ Press the RESET + START soft key: The TNC simulates the active program up to a programmed break or to the program end

▶ While the simulation is running, you can use the soft keys to change views

STOP

▶ Press the STOP soft key: The TNC interrupts the test run

START

▶ Press the START soft key: The TNC resumes the test run after a break

### Further information on this topic

- Running a test run: See "Test Run" on page 327
- Graphic functions: See "Graphics" on page 316
- Adjusting the test speed: See "Setting the speed of the test run" on page 317

## 1.5 Tool setup

### Selecting the correct operating mode

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



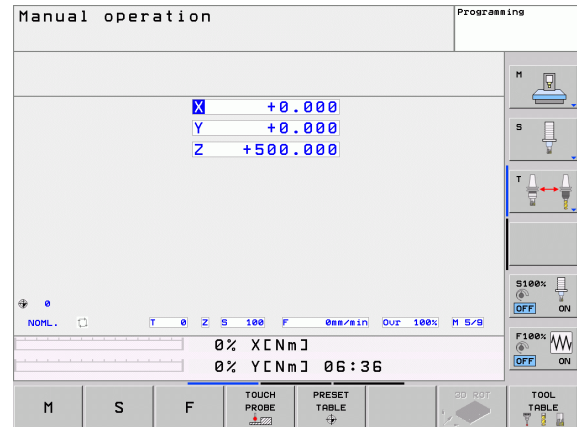
- ▶ Press the **Manual Operation** operating mode key: the TNC switches to that mode

#### Further information on this topic

- Operating modes of the TNC: See "Operating modes" on page 60

### Preparing and measuring tools

- ▶ Clamp the required tools in their chucks
- ▶ When measuring with an external tool presetter: Measure the tools, note down the length and radius, or transfer them directly to the machine through a transfer program
- ▶ When measuring on the machine: Insert the tool



### The tool table TOOL.T

In the tool table TOOL.T (permanently saved under **TNC:\TABLE\**), save the tool data such as length and radius, but also further tool-specific information that the TNC needs to perform its functions.

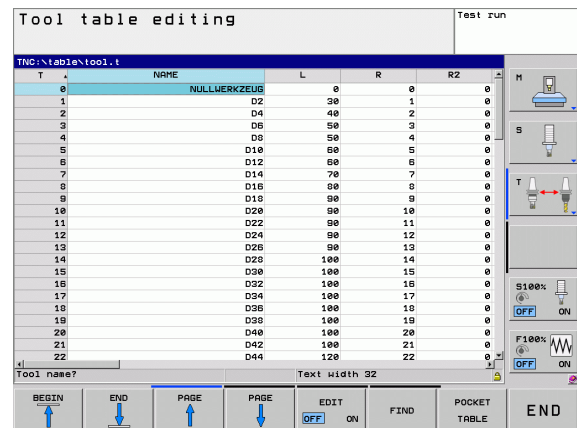
To enter tool data in the tool table TOOL.T, proceed as follows:



- ▶ Display the tool table



- ▶ Edit the tool table: Set the EDITING soft key to ON
- ▶ With the upward or downward arrow keys you can select the tool number that you want to edit
- ▶ With the rightward or leftward arrow keys you can select the tool data that you want to edit
- ▶ To exit the tool table, press the END key



#### Further information on this topic

- Operating modes of the TNC: See "Operating modes" on page 60
- Working with the tool table: See "Entering tool data in the table" on page 144



## 1.6 Workpiece setup

### Selecting the correct operating mode

Workpieces are set up in the **Manual Operation** or **EL. Handwheel** mode



- ▶ Press the **Manual Operation** operating mode key: the TNC switches to that mode

#### Further information on this topic

- Manual Operation mode: See "Moving the machine axes" on page 278

### Clamp the workpiece

Mount the workpiece with a fixture on the machine table so that it is fixed with its edges parallel to the machine axes.



## Datum setting with a 3-D touch probe

- ▶ Insert the 3-D touch probe: In the MDI mode, run a **TOOL CALL** block containing the tool axis and then return to the **Manual Operation** mode



- ▶ Select the probing functions: The TNC displays the available functions in the soft-key row



- ▶ Select the function for setting the datum, e.g. Probe Position
- ▶ Position the touch probe near the first touch point on the first workpiece edge
- ▶ Select the probing direction via soft key, e.g. +X
- ▶ Press NC start: The touch probe moves in the defined direction until it contacts the workpiece and then automatically returns to its starting point
- ▶ Then the TNC displays the coordinates of the measured position



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

- ▶ Repeat this procedure for all axes, in which you want to set the datum

### Further information on this topic

- Datum setting: See "Datum setting with a 3-D touch probe" on page 303

## 1.7 Running the first program

### Selecting the correct operating mode

You can run programs either in the Single Block or the Full Sequence mode:



- ▶ Press the operating mode key: The TNC goes into the **Program Run, Single Block** mode and the TNC executes the program block by block. You have to confirm each block with the NC start key



- ▶ Press the **Program Run, Full Sequence** operating mode key: The TNC switches to that mode and runs the program after NC start up to a program interruption or to the end of the program

#### Further information on this topic

- Operating modes of the TNC: See "Operating modes" on page 60
- Running programs: See "Program run" on page 330

### Choosing the program you want to run



- ▶ Press the PGM MGT key: The TNC opens the file manager



- ▶ Press the LAST FILES soft key: The TNC opens a pop-up window with the most recently selected files
- ▶ If desired, use the arrow keys to select the program that you want to run. Load with the ENT key

#### Further information on this topic

- File management: See "Working with the file manager" on page 95

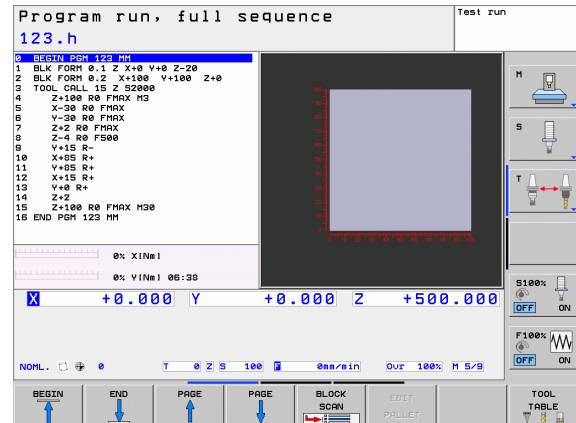
### Starting the program



- ▶ Press the NC start key: The TNC runs the active program

#### Further information on this topic

- Running programs: See "Program run" on page 330



# 2

**Introduction**



## 2.1 The TNC 128

The TNC 128 is a workshop-oriented straight cut control that enables you to program conventional machining operations right at the machine in an easy-to-use conversational programming language. They are designed for milling, drilling and boring machines with up to 3 axes. You can also change the angular position of the spindle under program control.

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

### Programming: HEIDENHAIN conversational format

The HEIDENHAIN conversational programming format is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. Workpiece machining can be graphically simulated either during or before actual machining.

You can also enter and test one program while the control is running another.

### Compatibility

Machining programs created on the HEIDENHAIN TNC 124 straight cut control may not always run on the TNC 128. If NC blocks contain invalid elements, the TNC will mark them as ERROR blocks when the file is opened.



## 2.2 Visual display unit and keyboard

### Visual display unit

The TNC is shipped with a 12.1-inch TFT flat-panel display.

#### 1 Header

When the TNC is on, the selected operating modes are shown in the screen header: the machining mode at the left and the programming mode at right. The currently active operating mode is displayed in the larger box, where the dialog prompts and TNC messages also appear (unless the TNC is showing only graphics).

#### 2 Soft keys

In the footer the TNC 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 arrow keys to the right and left. The bar representing the active soft-key row is highlighted.

#### 3 Soft-key selection keys

#### 4 Shifting between soft-key rows

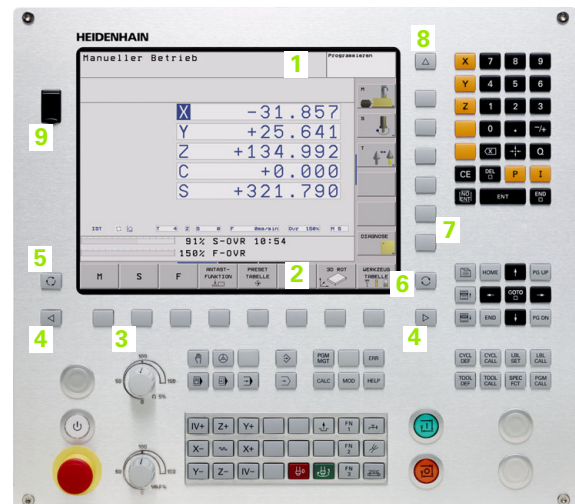
#### 5 Setting the screen layout

#### 6 Shift key for switchover between machining and programming modes

#### 7 Soft-key selection keys for machine tool builders

#### 8 Switching the soft-key rows for machine tool builders

#### 9 USB connection



## Setting the screen layout

You select the screen layout yourself: In the PROGRAMMING AND EDITING mode of operation, for example, you can have the TNC show program blocks in the left window while the right window displays programming graphics. As an alternative, the program can be displayed in a large window. The available screen windows depend on the selected operating mode.

To change the screen layout:



Press the screen layout key: The soft-key row shows the available layout options (see "Operating modes", page 60)



Select the desired screen layout



## Operating panel

The TNC 128 is delivered with an integrated keyboard. The figure at right shows the controls and displays of the keyboard:

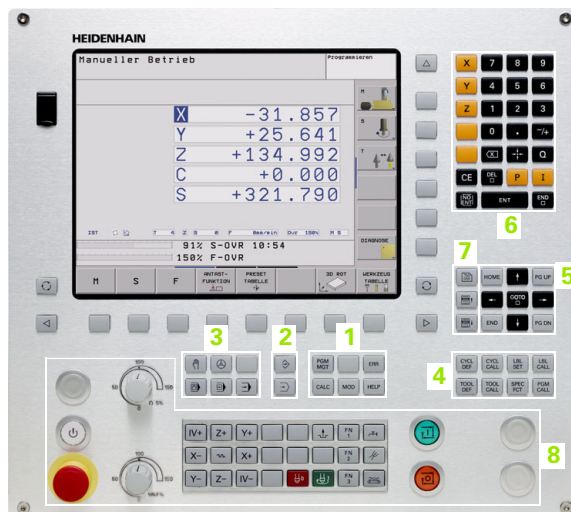
- 1 ■ File management
- Calculator
- MOD function
- HELP function
- 2 Programming modes
- 3 Machine operating modes
- 4 Initiation of programming dialog
- 5 Arrow keys and GOTO jump command
- 6 Numerical input and axis selection
- 7 Navigation keys
- 8 Machine operating panel

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



The keys on the operating panel are assigned to machine-dependent functions. Refer to your machine manual.

Machine panel buttons, e.g. NC START or NC STOP, are described in the manual for your machine tool.



## 2.3 Operating modes

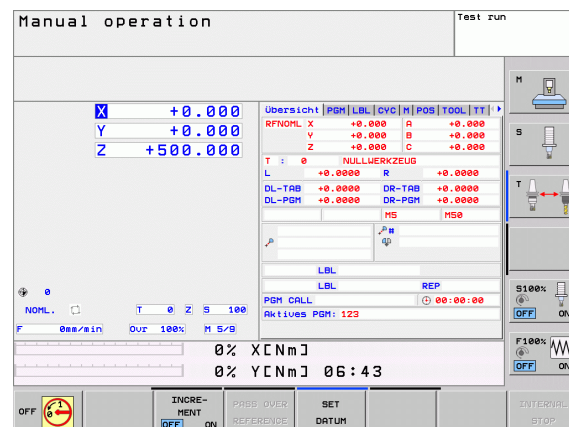
### Manual Operation and El. Handwheel

The Manual Operation mode is required for setting up the machine tool. In this operating mode, you can position the machine axes manually or by increments and set the datums.

The El. Handwheel mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

**Soft keys for selecting the screen layout** (select as described previously)

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

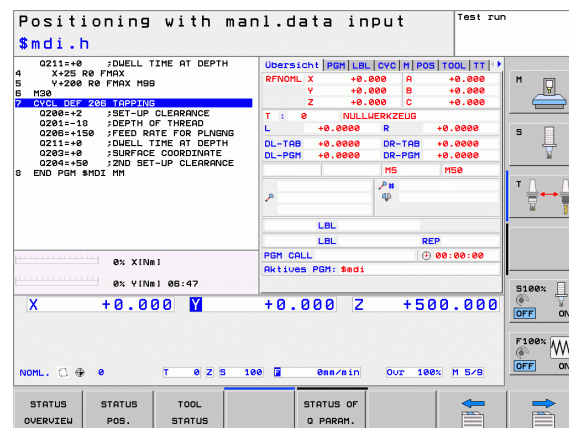


### Positioning with Manual Data Input

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

**Soft keys for selecting the screen layout**

Window	Soft key
Program	PGM
Left: program blocks, right: status display	PROGRAM + STATUS





# Programming and Editing

In this mode of operation you can write your part programs. The various cycles and Q-parameter functions help you with programming and adding necessary information. If desired, you can have the programming graphics show the programmed paths of traverse.

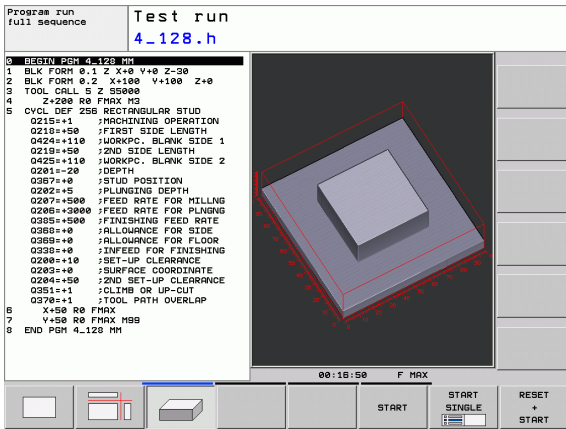
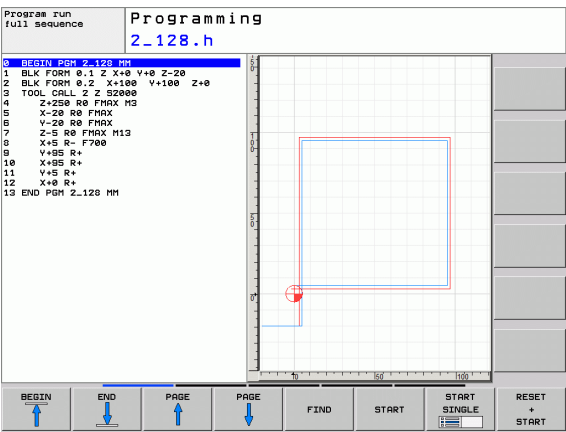
## Soft keys for selecting the screen layout

Window	Soft key
Program	PGM
Left: program, right: program structure	PROGRAM + SECTS
Left: program blocks, right: graphics	PROGRAM + GRAPHICS

# Test Run

In the Test Run mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the working space. This simulation is supported graphically in different display modes.

Soft keys for selecting the screen layout: see "Program Run, Full Sequence and Program Run, Single Block", page 62.



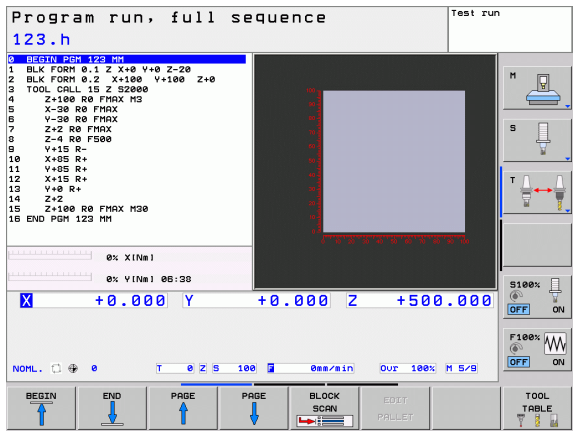
## Program Run, Full Sequence and Program Run, Single Block

In the Program Run, Full Sequence mode of operation the TNC executes a part 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 mode of operation you execute each block separately by pressing the machine START button.

### Soft keys for selecting the screen layout

Window	Soft key
Program	PGM
Left: program, right: program structure	PROGRAM + SECTS
Left: program, right: status	PROGRAM + STATUS
Left: program, right: graphics	PROGRAM + GRAPHICS
Graphics	GRAPHICS



## 2.4 Status displays

### "General" status display

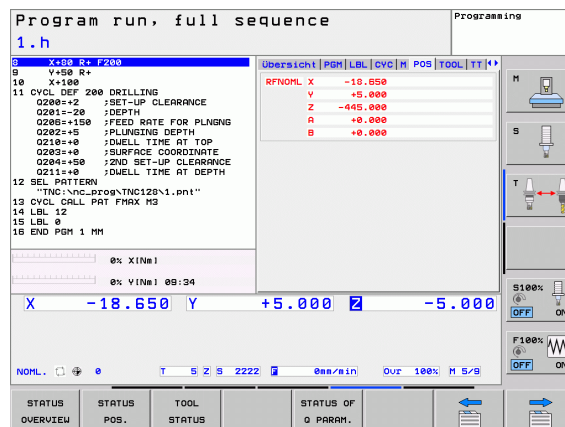
The status display in the lower part of the screen informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:

- Program Run, Single Block and Program Run, Full Sequence, except if the screen layout is set to display graphics only, and
- Positioning with Manual Data Input (MDI).

In the Manual Operation and EI. Handwheel modes the status display appears in the large window.

#### Information in the status display

Symbol	Meaning
<b>ACTL.</b>	Actual or nominal coordinates of the current position
<b>XYZ</b>	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
<b>FSM</b>	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
	Axis is clamped
	Axis can be moved with the handwheel
	No active program
	Program run has started
	Program run is stopped
	Program run is being aborted



## Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes except for the Programming and Editing mode of operation.

### To switch on the additional status display:



Call the soft-key row for screen layout



Select the screen layout with additional status display: In the right half of the screen, the TNC shows the **Overview** status form

### To select an additional status display:



Switch the soft-key rows until the STATUS soft keys appear



Either select the additional status display directly by soft key, e.g. positions and coordinates, or



use the switch-over soft keys to select the desired view

The available status displays described below can be selected either directly by soft key or with the switch-over soft keys.



Please note that some of the status information described below is not available unless the associated software option is enabled on your TNC.

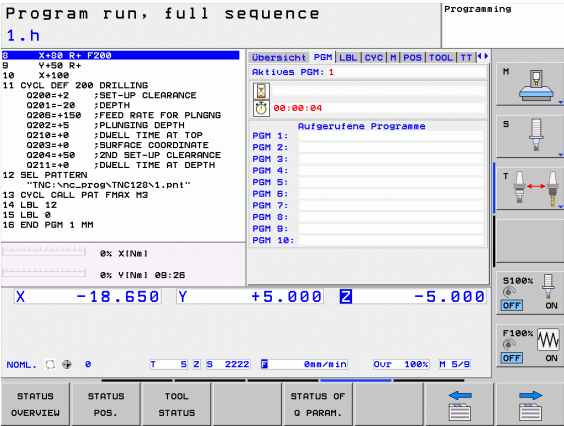
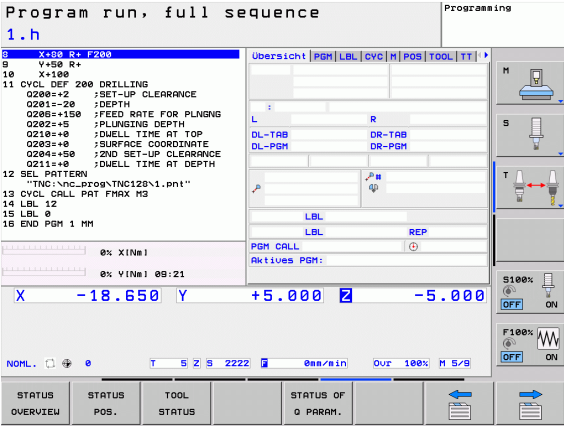
# Overview

After switch-on, the TNC displays the **Overview** status form, provided that you have selected the PROGRAM+STATUS screen layout (or POSITION + STATUS). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.

Soft key	Meaning
STATUS OVERVIEW	Position display
	Tool information
	Active M functions
	Active coordinate transformations
	Active subprogram
	Active program section repeat
	Program called with <b>PGM CALL</b>
	Current machining time
	Name of the active main program

## General program information (PGM tab)

Soft key	Meaning
No direct selection possible	Name of the active main program
	Dwell time counter
	Machining time when the program was completely simulated in the <b>Test Run</b> operating mode
	Current time
	Active programs

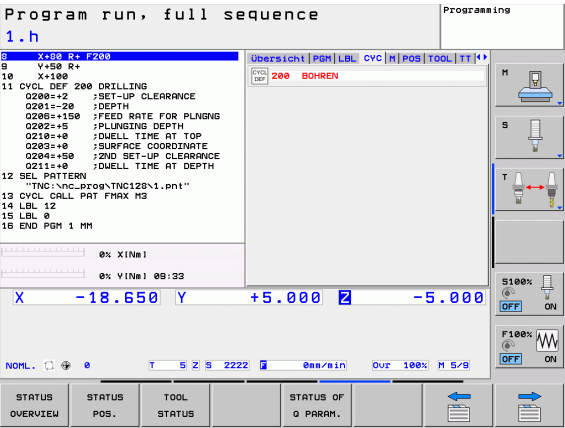
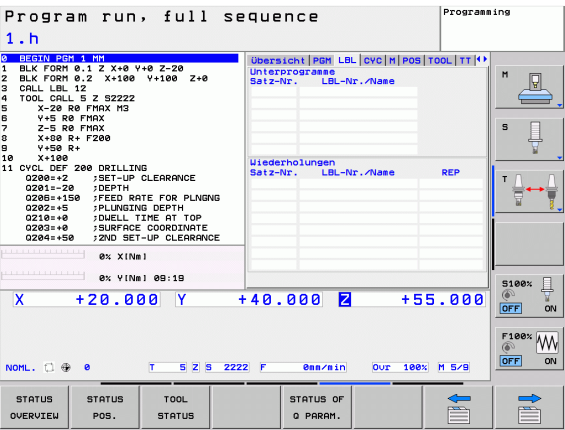


## Program section repeat/Subprograms (LBL tab)

Soft key	Meaning
No direct selection possible	Active program section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
	Active subprogram numbers with block number in which the subprogram was called and the label number that was called

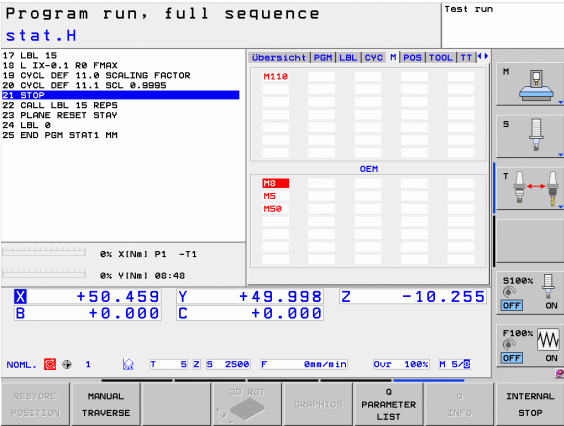
## Information on standard cycles (CYC tab)

Soft key	Meaning
No direct selection possible	Active machining cycle



Active miscellaneous functions M (M tab)

Soft key	Meaning
No direct selection possible	List of the active M functions with fixed meaning
	List of the active M functions that are adapted by your machine manufacturer

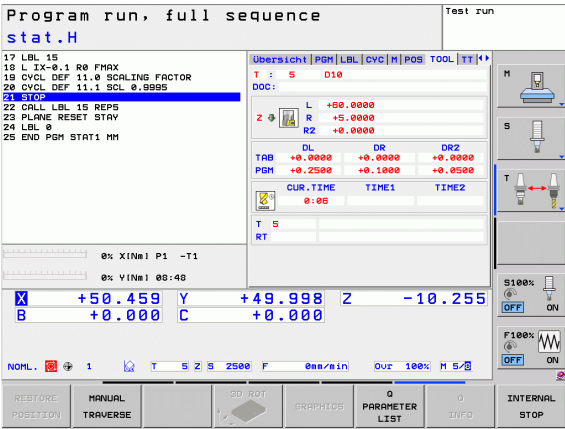
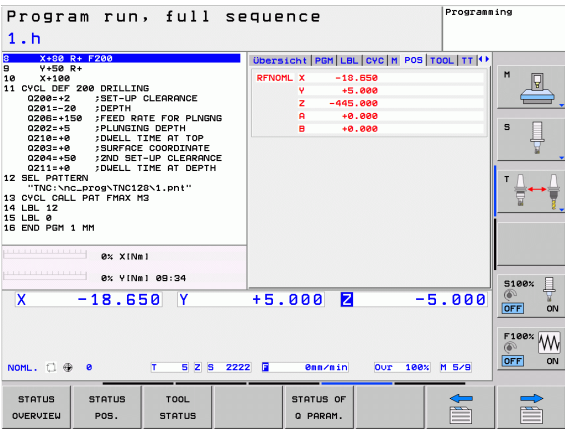


### Positions and coordinates (POS tab)

Soft key	Meaning
STATUS POS.	Type of position display, e.g. actual position

### Information on tools (TOOL tab)

Soft key	Meaning
TOOL STATUS	<ul style="list-style-type: none"> <li>T: Tool number and name</li> <li>RT: Number and name of a replacement tool</li> </ul>
	Tool axis
	Tool length and radii
	Oversizes (delta values) from the tool table (TAB) and the <b>TOOL CALL</b> (PGM)
	Tool life, maximum tool life (TIME 1) and maximum tool life for <b>TOOL CALL</b> (TIME 2)
	Display of the active tool and the (next) replacement tool





## Tool measurement (TT tab)



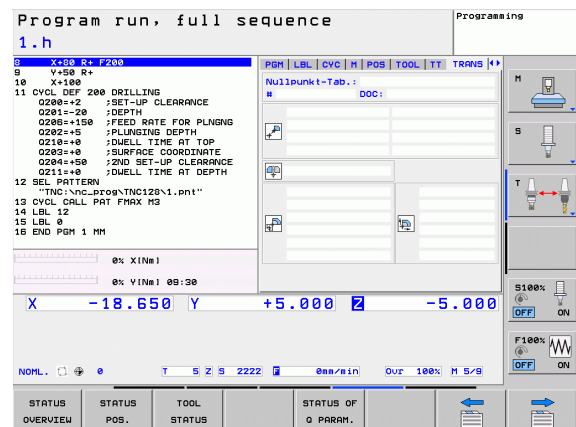
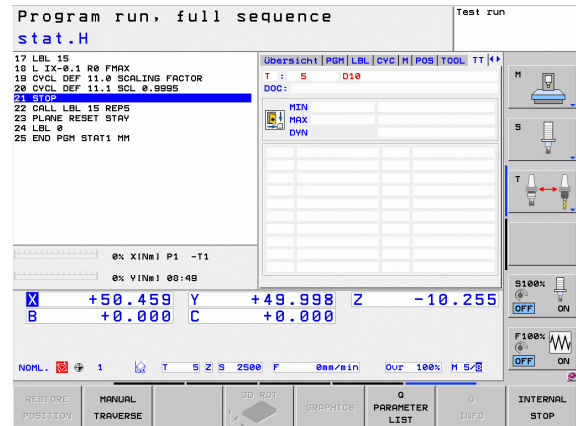
The TNC displays the TT tab only if the function is active on your machine.

Soft key	Meaning
No direct selection possible	Number of the tool to be measured
	Display whether the tool radius or the tool length is being measured
	MIN and MAX values of the individual cutting edges and the result of measuring the rotating tool (DYN = dynamic measurement)
	Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the permissible tolerance in the tool table was exceeded

## Coordinate transformations (TRANS tab)

Soft key	Meaning
STATUS COORD. TRANSF.	Name of the active datum table
	Active datum number (#), comment from the active line of the active datum number (DOC) from Cycle 7
	Active datum shift (Cycle 7); The TNC displays an active datum shift in up to 8 axes
	Mirrored axes (Cycle 8)
	Active scaling factor/factors (Cycles 11 / 26); The TNC displays an active scaling factor in up to 6 axes
	Scaling datum

For further information, refer to the User's Manual for Cycles, "Coordinate Transformation Cycles."

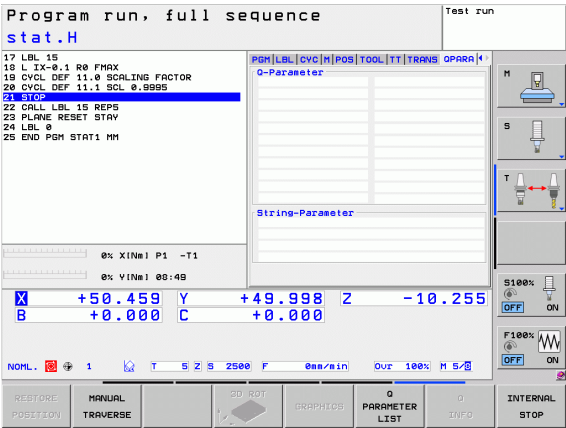


## Displaying Q parameters (QPARA tab)

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



Press the Q PARAMETER LIST soft key. The TNC opens a pop-up window in which you can enter the desired range for display of the Q parameters or string parameters. Multiple Q parameters are entered separated by commas (e.g. Q 1,2,3,4). To define display ranges, enter a hyphen (e.g. Q 10-14).



## 2.5 Window manager



The machine tool builder determines the scope of function and behavior of the window manager. The machine manual provides further information.

The TNC features the Xfce window manager. Xfce is a standard application for UNIX-based operating systems, and is used to manage graphical user interfaces. The following functions are possible with the window manager:

- Display a task bar for switching between various applications (user interfaces).
- Manage an additional desktop, on which special applications from your machine tool builder can run.
- Control the focus between NC-software applications and those of the machine tool builder.
- The size and position of pop-up windows can be changed. It is also possible to close, minimize and restore the pop-up windows.



The TNC shows a star in the upper left of the screen if an application of the window manager or the window manager itself has caused an error. In this case, switch to the window manager and correct the problem. If required, refer to your machine manual.



## Task bar

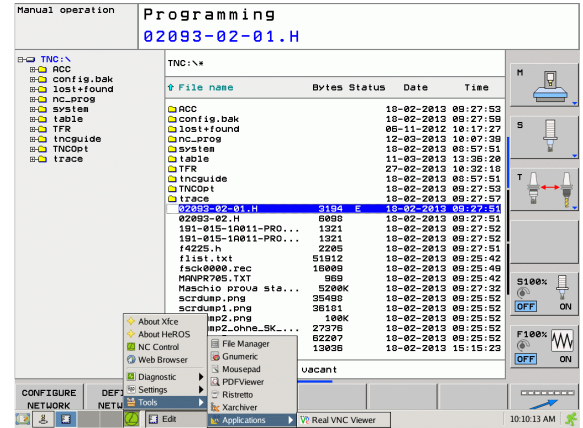
In the task bar you can choose different workspaces by mouse click. The TNC provides the following workspaces:

- Workspace 1: Active mode of operation
- Workspace 2: Active programming mode
- Workspace 3: Manufacturer's applications (optionally available)

In the task bar you can also select other applications that you have started together with the TNC (switch for example to the **PDF viewer** or **TNCguide**).

Click the green HEIDENHAIN symbol to open a menu in which you can get information, make settings or start applications. The following functions are available:

- **About Xfce:** Information on the Windows manager Xfce
- **About HEROS:** Information about the operating system of the TNC
- **NC Control:** Start and stop the TNC software. Only permitted for diagnostic purposes
- **Web Browser:** Start Mozilla Firefox
- **Diagnostics:** Available only to authorized specialists to start diagnostic functions
- **Settings:** Configuration of miscellaneous settings
  - **Date/Time:** Set the date and time
  - **Language:** Language setting for the system dialogs. During startup the TNC overwrites this setting with the language setting of MP 7230
  - **Network:** Network setting
  - **Reset WM-Conf:** Restore basic settings of the Windows Manager. May also reset settings implemented by your machine manufacturer
  - **Screensaver:** Settings for the screen saver; several are available
  - **Shares:** Configure network connections
- **Tools:** Only for authorized users. The applications available under tools can be started directly by selecting the pertaining file type in the file management of the TNC (see "File management: Fundamentals" on page 92)



## 2.6 Accessories: HEIDENHAIN 3-D touch probes and electronic handwheels

### 3-D touch probes

The various HEIDENHAIN 3-D touch probes enable you to:

- Quickly and precisely set datums
- Measure the workpiece

#### TS 220 and KT 130 touch trigger probes

These touch probes are particularly effective for datum setting and workpiece measurement. The TS 220 and KT 130 touch probes transmit the triggering signals to the TNC via cable.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear resisting optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the control, which stores the current position of the stylus as the actual value.



### HR electronic handwheels

Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 panel-mounted handwheels, HEIDENHAIN also offers the HR 410 portable handwheel.





# 3

**Programming:  
Fundamentals,  
file management**



## 3.1 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 TNC 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 association, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From that signal the TNC can re-establish the assignment of displayed positions to machine positions. 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°.

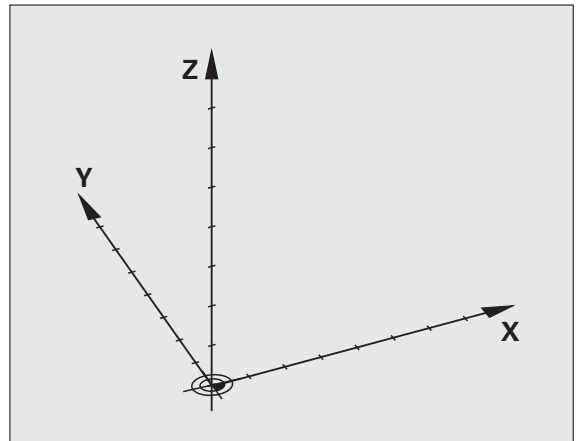
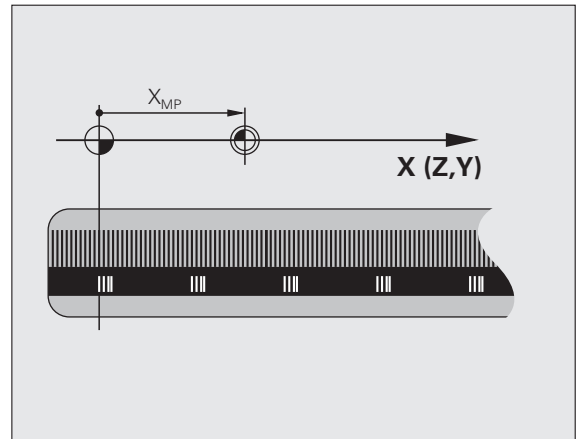
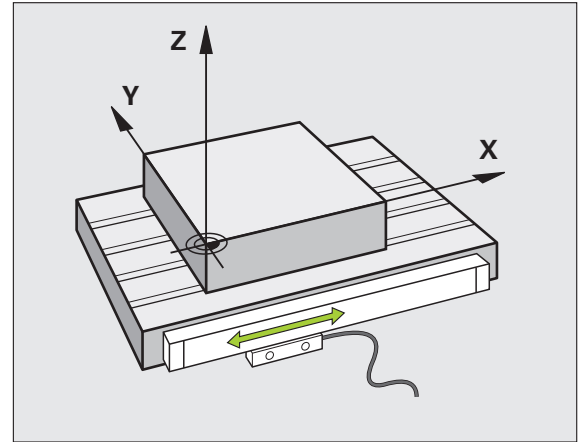
With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.

### Reference system

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.





## Reference system on milling machines

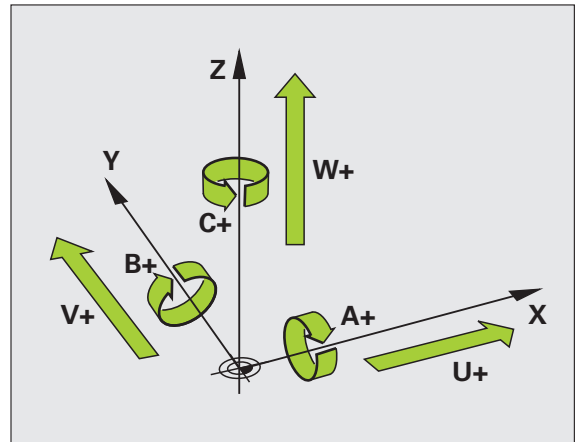
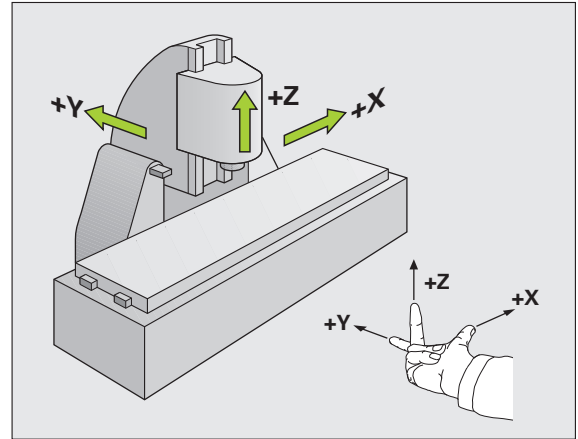
When using a milling machine, you orient tool movements to the Cartesian coordinate system. The illustration at right shows how the Cartesian coordinate system describes the machine axes. The figure illustrates the right-hand rule for remembering the three axis directions: the middle finger points in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb points in the positive X direction, and the index finger in the positive Y direction.

The TNC 128 can control up to 4 axes optionally. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The illustration at lower right shows the assignment of secondary axes and rotary axes to the main axes.

## 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



## 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 uniquely defined by its absolute coordinates.

Example 1: Holes dimensioned in absolute coordinates

Hole <b>1</b>	Hole <b>2</b>	Hole <b>3</b>
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 prefix "I" before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

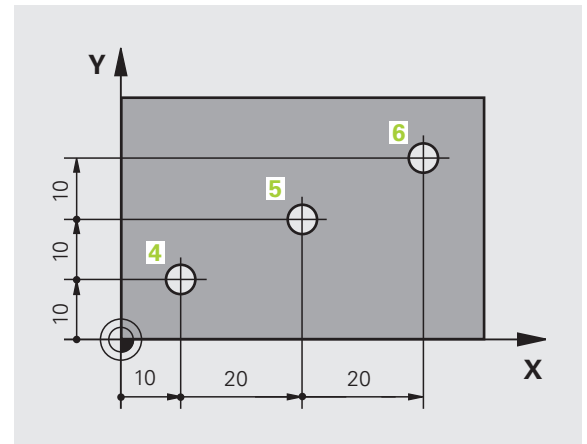
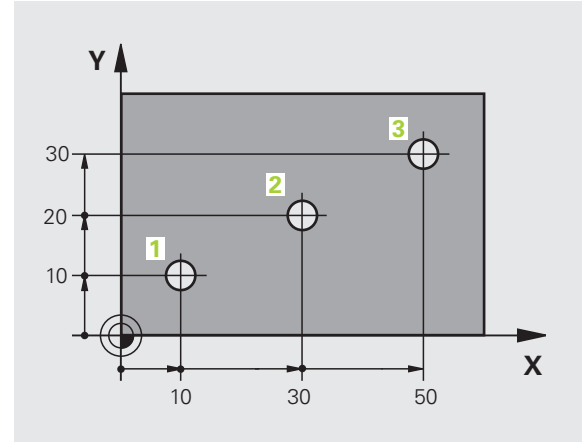
X = 10 mm  
Y = 10 mm

Hole **5**, with respect to **4**

X = 20 mm  
Y = 10 mm

Hole **6**, with respect to **5**

X = 20 mm  
Y = 10 mm



## Setting the datum

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. When setting the datum, you first align the workpiece along the machine axes, and then move the tool in each axis to a defined position relative to the workpiece. Set the display of the TNC either to zero or to a known position value for each position. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.

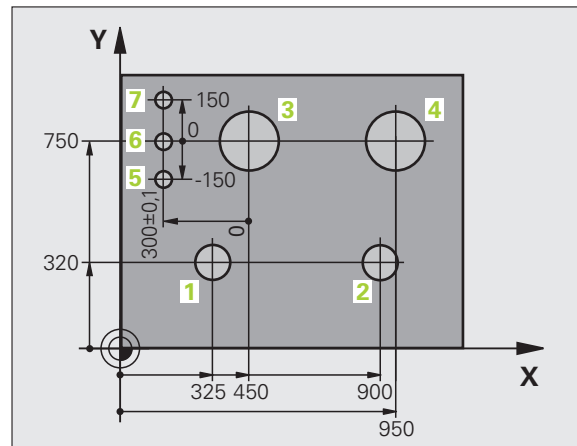
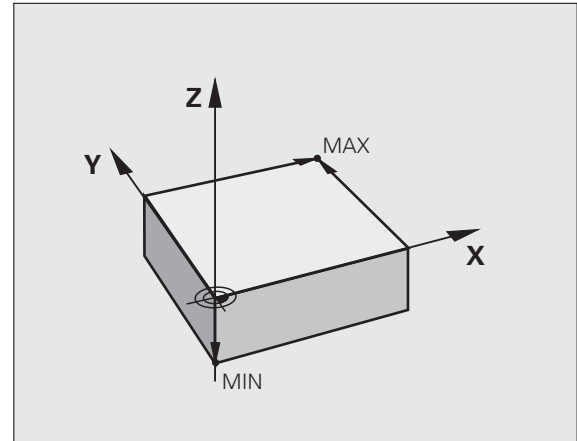
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles (see User's Manual for Cycles, Cycles for Coordinate Transformation).

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece from which the dimensions of the remaining workpiece positions can be most easily measured.

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. See "Setting the Datum with a 3-D Touch Probe" in the Touch Probe Cycles User's Manual.

### Example

The workpiece drawing shows holes (1 to 4) whose dimensions are shown with respect to an absolute datum with the coordinates  $X=0$   $Y=0$ . Holes 5 to 7 are dimensioned with respect to a relative datum with the absolute coordinates  $X=450$ ,  $Y=750$ . With the **DATUM SHIFT** cycle you can temporarily set the datum to the position  $X=450$ ,  $Y=750$ , to be able to program holes 5 to 7 without further calculations.



## 3.2 Creating and writing programs

### Organization of an NC program in HEIDENHAIN conversational format

A part program consists of a series of program blocks. The figure at right illustrates the elements of a block.

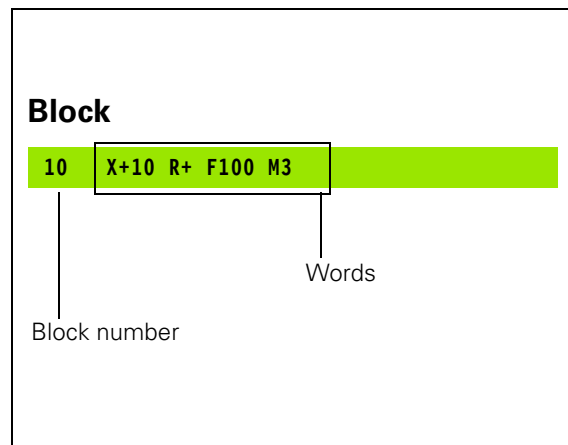
The TNC numbers the blocks in ascending sequence.

The first block of a program is identified by **BEGIN PGM**, the program name and the active unit of measure.

The subsequent 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 **END PGM**, the program name and the active unit of measure.



After each tool call, HEIDENHAIN recommends always traversing to a safe position from which the TNC can position the tool for machining without causing a collision!

### Defining the workpiece blank: BLK FORM

Immediately after initiating a new program, you define a cuboid workpiece blank. If you wish to define the blank at a later stage, press the SPEC FCT key, the PROGRAM DEFAULTS soft key, and then the BLK FORM soft key. This definition is needed for the TNC's graphic simulation feature. The sides of the workpiece blank lie parallel to the X, Y and Z axes and can be up to 100 000 mm long. The workpiece blank is defined by two of its corner points:

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



You only need to define the workpiece blank if you wish to run a graphic test for the program!

## Creating a new part program


You always enter a part program in the **Programming and Editing** mode of operation. An example of program initiation:


 Select the **Programming and Editing** operating mode

 Press the PGM MGT key to call the file manager


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

**FILE NAME = ALT.H**


 Enter the new program name and confirm your entry with the ENT key

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


**WORKING PLANE IN GRAPHIC: XY**

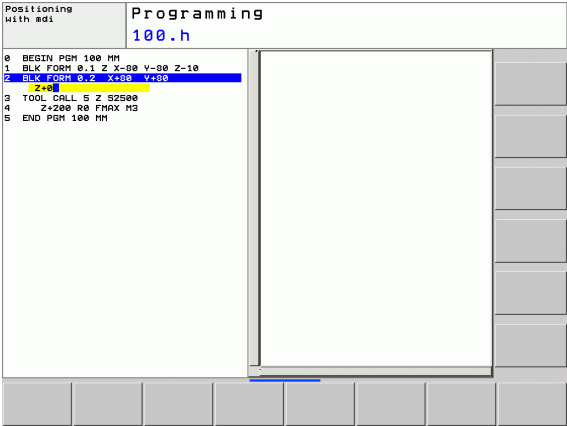
 Enter spindle axis, e.g. Z

**WORKPIECE BLANK DEF.: MINIMUM**

 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**

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



## Example: Display the BLK form in the NC program

0 BEGIN PGM NEW MM	Program begin, name, unit of measure
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEW MM	Program end, name, unit of measure

The TNC generates the block numbers as well as the **BEGIN** and **END** blocks automatically.



If you do not wish to define a blank form, cancel the dialog at **Working plane in graphic: XY** by pressing the DEL key.

The TNC can display the graphics only if the shortest side is at least 50 µm long and the longest side is no longer than 99 999.999 mm.



Programming tool movements in conversational format

To program a block, initiate the dialog by pressing an axis key. In the screen header, the TNC then asks you for all the information necessary to program the desired function.

Example of a positioning block

X

Open

COORDINATES?

10

ENT

Enter the target coordinate for the X axis

TOOL RADIUS COMP: R+/R-/NO COMP?

ENT

Enter "No radius compensation" and go to the next question with ENT

FEED RATE F=? / F MAX = ENT

100

ENT

Enter a feed rate of 100 mm/min for this path contour; go to the next question with ENT

MISCELLANEOUS FUNCTION M?

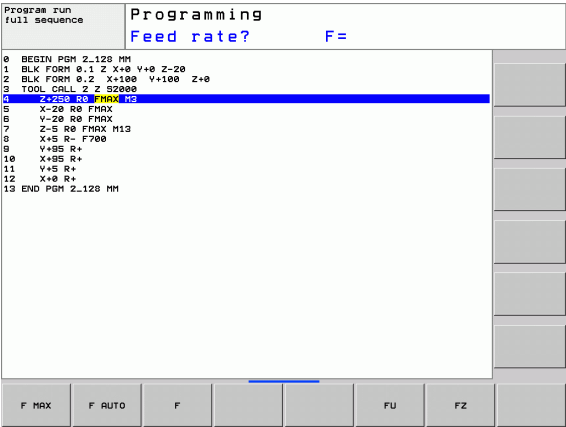
3

ENT






Enter the miscellaneous function **M3** "spindle ON." Pressing the ENT key terminates this dialog




The program-block window displays the following line:

3 X+10 R0 F100 M3



## Possible feed rate input

Functions for setting the feed rate	Soft key
Rapid traverse, non-modal. Exception.	
Traverse feed rate automatically calculated in <b>TOOL CALL</b>	
Move at the programmed feed rate (unit of measure is mm/min or 1/10 inch/min). With rotary axes, the TNC interprets the feed rate in degrees/min, regardless of whether the program is written in mm or inches	
Define the feed per revolution (units in mm/rev or inch/rev). Caution: In inch-programs, FU cannot be combined with M136	
Define the tooth feed (units in mm/tooth or inch/tooth). The number of teeth must be defined in the tool table in the <b>CUT.</b> column	

Functions for conversational guidance	Key
Ignore the dialog question	
End the dialog immediately	
Abort the dialog and erase the block	





## Actual position capture

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

- Positioning-block programming
- Cycle programming

To transfer the correct position values, proceed as follows:

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



- ▶ Select the actual-position-capture function: In the soft-key row the TNC displays the axes whose positions can be transferred



- ▶ Select the axis: The TNC writes the current position of the selected axis into the active input box



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

In the tool axis the TNC always captures the coordinates of the tool tip and thus always takes the active tool length compensation into account.

The TNC keeps the soft-key row for axis selection active until you deactivate it by pressing the actual-position-capture key again. This behavior remains in effect even if you save the current block and open a new one with a path function key. If you select a block element in which you must choose an input alternative via soft key (e.g. for radius compensation), then the TNC also closes the soft-key row for axis selection.














## Editing a program










You cannot edit a program while it is being run by the TNC in a machine operating mode.

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

Function	Soft key/Keys
Go to previous page	
Go to next page	
Go to beginning of program	
Go to end of program	
Change the position of the current block on the screen. Press this soft key to display additional program blocks that are programmed before the current block	
Change the position of the current block on the screen. Press this soft key to display additional program blocks that are programmed after the current block	
Move from one block to the next	 
Select individual words in a block	 
To select a certain block, press the GOTO key, enter the desired block number, and confirm with the ENT key. Or: Enter the block number step and press the N LINES soft key to jump over the entered number of lines upward or downward	



Function	Soft key/Key
Set the selected word to zero	
Erase an incorrect number	
Clear a (non-blinking) error message	
Delete the selected word	
Delete the selected block	
Erase cycles and program sections	
Insert the block that you last edited or deleted	

## Inserting blocks at any desired location

- Select the block after which you want to insert a new block and initiate the dialog

## Editing and inserting words

- Select a word in a block and overwrite it with the new one. The plain-language 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 blocks

To use this function, set the AUTO DRAW soft key to OFF.



Select a word in a block: Press the arrow keys repeatedly until the highlight is on the desired word



Select a block with the arrow keys

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



If you have started a search in a very long program, the TNC shows a progress display window. You then have the option of canceling the search via soft key.

### Finding any text

- ▶ Select the search function: Press the FIND soft key. The TNC displays the **Find text:** dialog prompt
- ▶ Enter the text that you wish to find
- ▶ To find the text, press the FIND soft key

### Marking, copying, deleting and inserting program sections

The TNC provides certain functions for copying program sections within an NC program or into another NC program—see the table below.

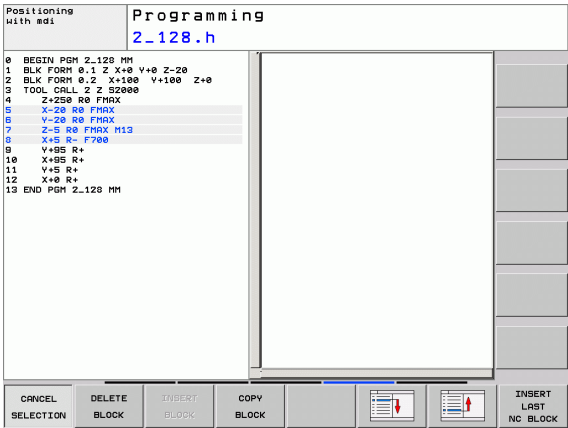
To copy a program section, proceed as follows:

- ▶ Select the soft-key row containing the marking functions
- ▶ Select the first (last) block of the section you wish to copy
- ▶ To mark the first (last) block, press the SELECT BLOCK soft key. The TNC then highlights the first character of the block and the CANCEL SELECTION soft key appears
- ▶ Move the highlight to the last (first) block of the program section you wish to copy or delete. The TNC shows the marked blocks in a different color. You can end the marking function at any time by pressing the CANCEL SELECTION soft key
- ▶ To copy the selected program section, press the COPY BLOCK soft key. To delete the selected section, press the DELETE BLOCK soft key. The TNC stores the selected block
- ▶ Using the arrow keys, select the block after which you wish to insert the copied (deleted) program section



To insert the section into another program, select the corresponding program using the file manager and then mark the block after which you wish to insert the copied block.

- ▶ To insert the block, press the INSERT BLOCK soft key
- ▶ To end the marking function, press the CANCEL SELECTION soft key



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



## The TNC search function

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

### Finding any text

- ▶ If required, select the block containing the word you wish to find
  - ▶ Select the search function: The TNC superimposes the search window and displays the available search functions in the soft-key row (see table of search functions)
- ▶ Enter the text to be searched for. Please note that the search is case-sensitive
- ▶ Start the search process: The TNC moves to the next block containing the text you are searching for
- ▶ Repeat the search process: The TNC moves to the next block containing the text you are searching for
- ▶ End the search function

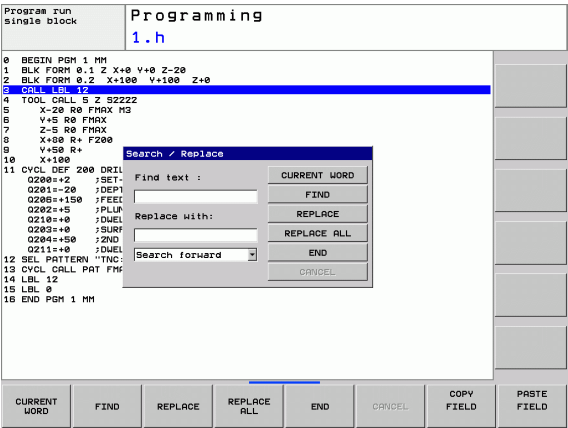
FIND

X +40

FIND

FIND

END



## Finding/Replacing any text



The find/replace function is not possible if

- a program is protected
- the program is currently being run by the TNC

When using the REPLACE ALL function, ensure that you do not accidentally replace text that you do not want to change. Once replaced, such text cannot be restored.

► If required, select the block containing the word you wish to find

FIND

► Select the Search function: The TNC superimposes the search window and displays the available search functions in the soft-key row



► Enter the text to be searched for. Please note that the search is case-sensitive. Then confirm with the ENT key



► Enter the text to be inserted. Please note that the entry is case-sensitive

FIND

► Start the search process: The TNC 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. To replace all text occurrences, press the REPLACE ALL soft key. To skip the text and move to its next occurrence press the FIND soft key

FIND

► End the search function



### 3.3 File management: Fundamentals

#### Files

Files in the TNC	Type
<b>Programs</b>	
In HEIDENHAIN format	.H
In DIN/ISO format	.I
<b>Tables for</b>	
Tools	.T
Tool changers	.TCH
Pallets	.P
Datums	.D
Points	.PNT
Presets	.PR
Touch probes	.TP
Backup files	.BAK
Dependent data (such as structure items)	.DEP
<b>Texts as</b>	
ASCII files	.A
Log files	.TXT
Help files	.CHM

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

The TNC 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.

You can create folder structures in the TNC so that you can manage your files. The memory available in the control is **2 GB**.



Depending on the setting, the TNC generates a backup file (\*.bak) after editing and saving of NC programs. This can reduce the memory space available to you.





### File names

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

PROG20	.H
File name	File type

File names should not exceed 25 characters, otherwise the TNC cannot display the entire file name.

File names on the TNC must comply with this standard: The Open Group Base Specifications Issue 6 IEEE Std 1003.1, 2004 Edition (Posix-Standard). Accordingly, the file names may include the characters below:

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 . \_ -

You should not use any other characters in file names in order to prevent any file transfer problems.



The maximum limit for the path and file name together is 82 characters (see "Paths" on page 95).



## Showing externally created files on the TNC

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

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

For further information about displaying and editing the listed file types: See "Additional tools for management of external file types" on page 109.

## Data backup

We recommend saving newly written programs and files on a PC at regular intervals.

The TNCremoNT data transmission freeware from HEIDENHAIN is a simple and convenient method for backing up data stored on the TNC.

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Ask your machine manufacturer for assistance, if necessary.



Saving the contents of the entire hard disk can take up to several hours. In this case, it is a good idea to save the data outside of work hours, e.g. during the night.

Take the time occasionally to delete any unneeded files so that the TNC always has enough hard-disk space for system files (such as the tool table).



Depending on operating conditions (e.g., vibration load), hard disks generally have a higher failure rate after three to five years of service. HEIDENHAIN therefore recommends having the hard disk inspected after three to five years.

## 3.4 Working with the file manager

### Directories

To ensure that you can easily find your files, we recommend that you organize your hard disk into directories. 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 path, including all drive characters, directory and the file name, including the extension, must not exceed 82 characters!

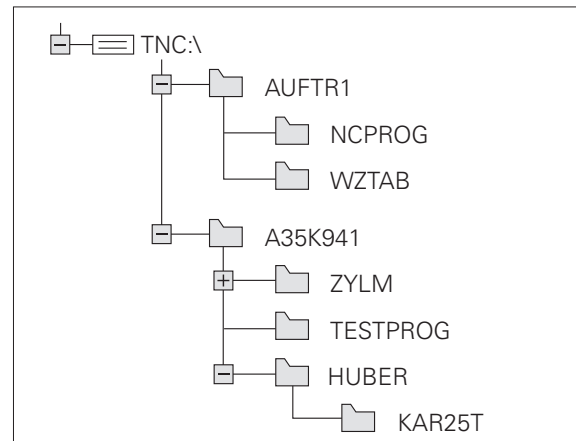
Drive designations must not include more than 8 uppercase letters.

### Example



















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

**`TNC:\AUFTR1\NCPROG\PROG1.H`**

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



## Overview: Functions of the file manager

Function	Soft key	Page
Copy a file		Page 101
Display a specific file type		Page 98
Create new file		Page 100
Display the last 10 files that were selected		Page 104
Delete a file or directory		Page 104
Tag a file		Page 106
Rename a file		Page 107
Protect a file against editing and erasure		Page 108
Cancel file protection		Page 108
Import tool table		Page 150
Manage network drives		Page 116
Select the editor		Page 108
Sort files by properties		Page 107
Copy a directory		Page 103
Delete directory with all its subdirectories		
Display all the directories of a particular drive		
Rename a directory		
Create a new directory		



# Calling the file manager

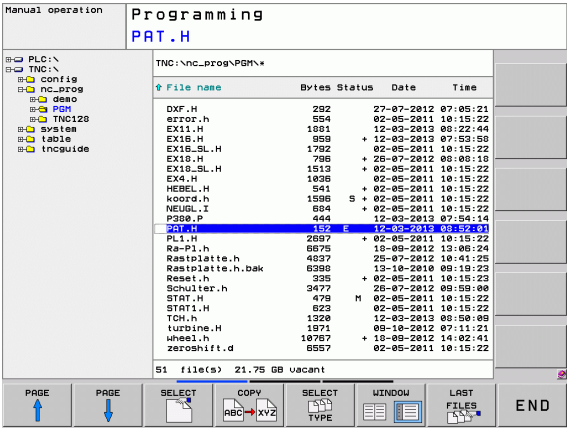


Press the PGM MGT key: The TNC displays the file management window (see figure for default setting. If the TNC displays a different screen layout, press the WINDOW soft key.)

The narrow window on the left shows the available drives and directories. Drives designate devices with which data are stored or transferred. One drive is the hard disk of the TNC. Other drives are the interfaces (RS232, Ethernet), which can be used, for example, for connecting a personal computer. 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. A triangle in front of the folder symbol indicates that there are further subdirectories, which can be shown with the -/+ or ENT keys.

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	Name with max. 25 characters
Type	File type
Bytes	File size in bytes
Status	File properties:
E	Program is selected in the Programming mode of operation
S	Program is selected in the Test Run mode of operation
M	Program is selected in a Program Run mode of operation
	File is protected against erasing and editing
	File is protected against erasing and editing, because it is being run
Date	Date that the file was last edited
Time	Time that the file was last edited



## Selecting drives, directories and files



Call the file manager

Use the arrow keys or the soft keys to move the highlight to the desired position on the screen:



Moves the highlight from the left to the right window, and vice versa



Moves the highlight up and down within a window



Moves the highlight one page up or down within a window

### Step 1: Select drive

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



To select a drive, press the SELECT soft key, or



Press the ENT key

### Step 2: Select a directory

Move the highlight to the desired directory in the left-hand window—the right-hand window automatically shows all files stored in the highlighted directory

## Step 3: Select a file



Press the SELECT TYPE soft key



Press the soft key for the desired file type, or



Press the SHOW ALL soft key to display all files, or

Move the highlight to the desired file in the right window:



Press the SELECT soft key, or



Press the ENT key

The TNC opens the selected file in the operating mode from which you called the file manager



## Creating a new directory

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

NEW  Enter the new file name, and confirm with ENT

CREATE \NEW DIRECTORY?

YES

Press the YES soft key to confirm, or

NO

Abort with the NO soft key

## Creating a new file

Select the directory in which you wish to create the new file

NEW  Enter the new file name with the file extension, and confirm with ENT



Open the dialog box for creating a new file

NEW  Enter the new file name with the file extension, and confirm with ENT



## Copying a single file

- Move the highlight to the file you wish to copy



- Press the COPY soft key to select the copy function. The TNC opens a pop-up window



- Enter the name of the target file and confirm your entry with the ENT key or OK soft key: the TNC copies the file to the active directory or to the selected target directory. The original file is retained, or



- Press the Target Directory soft key to call a pop-up window in which you select the target directory by pressing the ENT key or the OK soft key: the TNC copies the file to the selected directory. The original file is retained



When the copying process has been started with ENT or the OK soft key, the TNC displays a pop-up window with a progress indicator.



## Copying files into another directory

- ▶ Select a screen layout with two equally sized windows
- ▶ To display directories in both windows, press the PATH soft key

In the right window

- ▶ Move the highlight 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

- ▶ Select the directory containing the files that you wish to copy and press ENT to display them



- ▶ Call the file tagging functions



- ▶ Move the highlight to the file you want to copy and tag it. You can tag several files in this way, if desired



- ▶ Copy the tagged files into the target directory

Additional tagging functions: see "Tagging files", page 106.

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

### Overwriting files

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

- ▶ To overwrite all files ("Existing files" check box selected), press the OK soft key, or
- ▶ Press the CANCEL soft key if no file is to be overwritten

If you wish to overwrite a protected file, you need to select the "Protected files" check box or cancel the copying 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 already exist
- The file to be copied must only contain the lines you want to replace
- Both tables must have the same file extension



The **REPLACE FIELDS** function is used to overwrite lines in the target table. To avoid losing data, create a backup copy of the original table.

### 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).

- ▶ Copy this table from the external data medium to any directory
- ▶ Copy the externally created table to the existing table using the TNC file management. The TNC asks if you wish to overwrite the existing **TOOL.T** tool table:
- ▶ If you press the **YES** soft key, the TNC will completely overwrite the current **TOOL.T** tool table. After the copying process the new **TOOL.T** table consists of 10 lines.
- ▶ Or press the **REPLACE FIELDS** soft key for the TNC to overwrite 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.


- ▶ 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 TNC opens the window for selecting the target directory
- ▶ Select the target directory and confirm with **ENT** or the **OK** soft key: The TNC copies the selected directory and all its subdirectories to the selected target directory



## Choosing one of the last files selected





Call the file manager




Display the last 10 files selected: Press the LAST FILES soft key


Use the arrow keys to move the highlight to the file you wish to select:

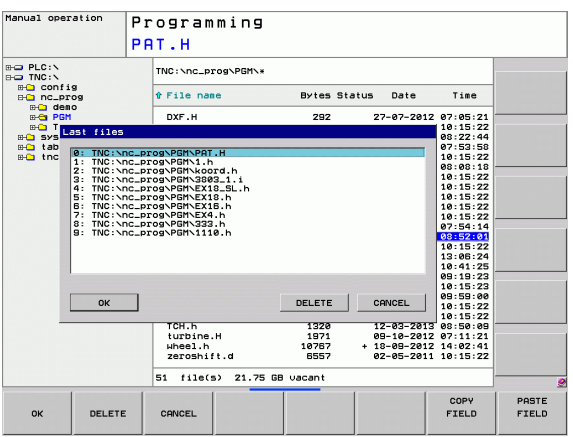
Moves the highlight up and down within a window




To select the file, press the OK soft key, or



Press the ENT key




## Deleting a file



**Caution: Data may be lost!**

Once you delete files they cannot be restored!

► Move the highlight to the file you want to delete



- To select the erasing function, press the DELETE soft key. The TNC asks whether you really want to delete the file
- To confirm, press the OK soft key, or
- To cancel deletion, press the CANCEL soft key



## Deleting a directory

**Caution: Data may be lost!**

Once you delete files they cannot be restored!

- Move the highlight to the directory you want to delete



- To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to delete the directory and all its subdirectories and files
- To confirm, press the OK soft key, or
- To cancel deletion, press the CANCEL soft key



Tagging files

Tagging function	Soft key
Tag a single file	<div>TAG FILE</div>
Tag all files in the directory	<div>TAG ALL FILES</div>
Untag a single file	<div>UNTAG FILE</div>
Untag all files	<div>UNTAG ALL FILES</div>
Copy all tagged files	<div>COPY TAG</div>

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, proceed as follows:

Move the highlight 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 highlight to the next file you wish to tag:  
Only works via soft keys. Do not use the arrow keys!

TAG  
FILE

To tag further files, press the TAG FILE soft key, etc.

COPY TAG

To copy the tagged files, press the COPY TAG soft key, or

END

DELETE

Delete the tagged files by pressing the END soft key to end the tagging functions, and then the DELETE soft key to delete the tagged files



## Renaming a file

- ▶ Move the highlight to the file you wish to rename



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

## File sorting

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



- ▶ Select the SORT soft key
- ▶ Select the soft key with the corresponding display criterion



## Additional functions

### Protecting a file / Canceling file protection

- ▶ Move the highlight to the file you want to protect
  - ▶ To select the additional functions, press the MORE FUNCTIONS soft key
- ▶ To activate file protection, press the PROTECT soft key. The file now has status P
  - ▶ To cancel file protection, press the UNPROTECT soft key



### Selecting the editor

- ▶ Move the highlight in the right window onto the file you want to open
  - ▶ To select the additional functions, press the MORE FUNCTIONS soft key
  - ▶ To select the editor with which to open the selected file, press the SELECT EDITOR soft key
  - ▶ Mark the desired editor
  - ▶ Press the OK soft key to open the file



### Connecting/removing a USB device

- ▶ Move the highlight to the left window
  - ▶ To select the additional functions, press the MORE FUNCTIONS soft key
  - ▶ Shift the soft-key row
  - ▶ Search for a USB device
  - ▶ In order to remove the USB device, move the highlight to the USB device
  - ▶ Remove the USB device



For more information: See "USB devices on the TNC" on page 117.



# Additional tools for management of external file types

The additional tools enable you to display or edit various externally created file types on the TNC.

File types	Description
PDF files (pdf)	Page 109
Excel spreadsheets (xls, csv)	Page 110
Internet files (htm, html)	Page 110
ZIP archives (zip)	Page 111
Text files (ASCII files, e.g. txt, ini)	Page 112
Graphics files (bmp, jpg, gif, png)	Page 113



If you transfer files from a PC to the control by means of TNCremoNT, you must have entered the file name extensions pdf, xls, zip, bmp, gif, jpg and png in the list of the file types for binary transmission (menu item **>Extras >Configuration >Mode** in TNCremoNT).

## Displaying PDF files

To open PDF files directly on the TNC, proceed as follows:

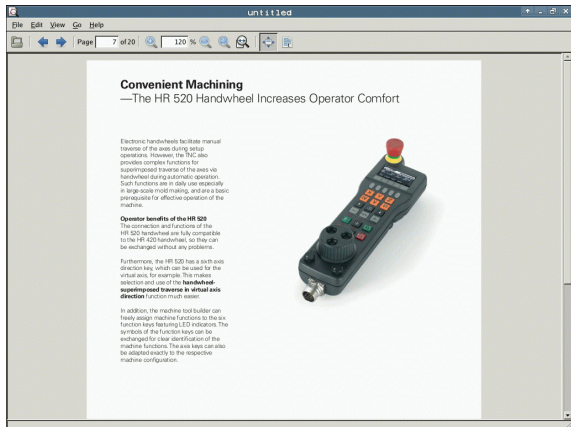
- PGM MGT**
  - ▶ Call the file manager
  - ▶ Select the directory in which the PDF file is saved
  - ▶ Move the highlight to the PDF file
- ENT**
  - ▶ Press ENT: The TNC opens the PDF file in its own application using the **PDF viewer** additional tool

With the key combination ALT+TAB you can always return to the TNC user interface while leaving the PDF file open. Alternatively, you can also click the corresponding symbol in the taskbar to switch back to the TNC interface.

If you position the mouse pointer over a button, a brief tooltip explaining the function of this button will be displayed. More information on how to use the **PDF viewer** is provided under **He1p**.

To exit the **PDF viewer**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Close**: The TNC returns to the file manager



## Displaying and editing Excel files

Proceed as follows to open and edit Excel files with the extension **xls** or **csv** directly on the TNC:

- PGM MGT**
  - ▶ Call the file manager
  - ▶ Select the directory in which the Excel file is saved
  - ▶ Move the highlight to the Excel file
- ENT**
  - ▶ Press ENT: The TNC opens the Excel file in its own application using the **Gnumeric** additional tool

With the key combination ALT+TAB you can always return to the TNC user interface while leaving the Excel file open. Alternatively, you can also click the corresponding symbol in the taskbar to switch back to the TNC interface.

If you position the mouse pointer over a button, a brief tooltip explaining the function of this button will be displayed. More information on how to use the **Gnumeric** function is provided under **Help**.

To exit **Gnumeric**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager

## Displaying Internet files

To open Internet files with the extension **htm** or **html** directly on the TNC, proceed as follows:

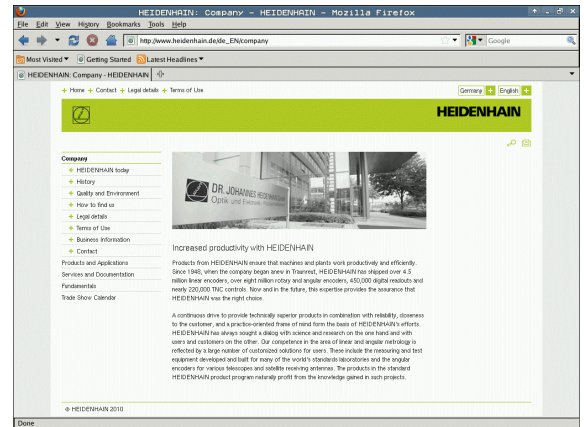
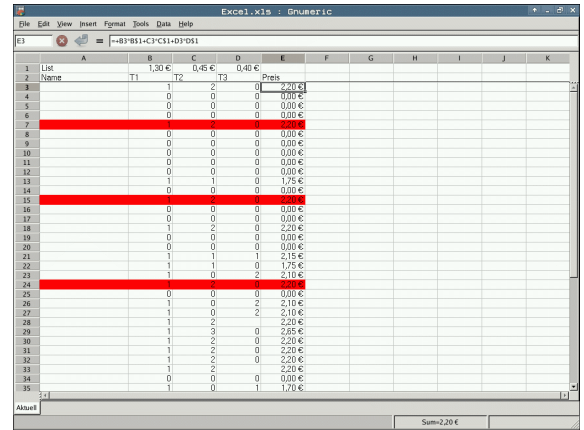
- PGM MGT**
  - ▶ Call the file manager
  - ▶ Select the directory in which the Internet file is saved
  - ▶ Move the highlight to the Internet file
- ENT**
  - ▶ Press ENT: The TNC opens the Internet file in its own application using the **Mozilla Firefox** additional tool

With the key combination ALT+TAB you can always return to the TNC user interface while leaving the PDF file open. Alternatively, you can also click the corresponding symbol in the taskbar to switch back to the TNC interface.

If you position the mouse pointer over a button, a brief tooltip explaining the function of this button will be displayed. More information on how to use **Mozilla Firefox** is provided under **Help**.

To exit **Mozilla Firefox**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager



### Working with ZIP archives

To open ZIP archives with the extension **zip** directly on the TNC, proceed as follows:

- PGM MGT

ENT
- ▶ Call the file manager
  - ▶ Select the directory in which the archive file is saved
  - ▶ Move the highlight to the archive file
  - ▶ Press ENT: The TNC opens the archive file in its own application using the **Xarchiver** additional tool

With the key combination ALT+TAB you can always return to the TNC user interface while leaving the archive file open. Alternatively, you can also click the corresponding symbol in the taskbar to switch back to the TNC interface.

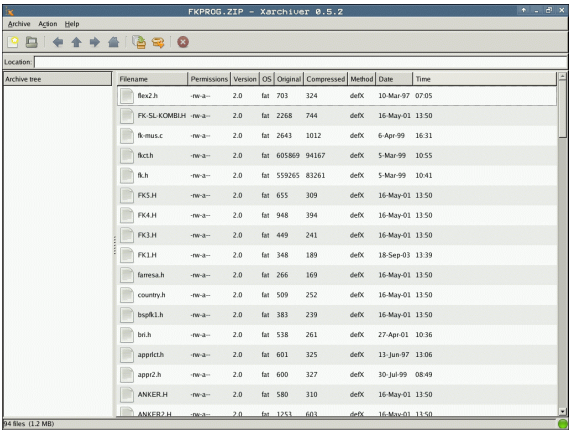
If you position the mouse pointer over a button, a brief tooltip explaining the function of this button will be displayed. More information on how to use the **Xarchiver** function is provided under **Help**.



Please note that the TNC does not carry out any binary-to-ASCII conversion or vice versa when compressing or decompressing NC programs and NC tables. When such files are transferred to TNC controls using other software versions, the TNC may not be able to read them.

To exit **Xarchiver**, proceed as follows:

- ▶ Use the mouse to select the **Archive** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager



## Displaying or editing text files

To open and edit text files (ASCII files, e.g. with the extension **txt** or **ini**), proceed as follows:

PGM  
MGT

- ▶ Call the file manager
- ▶ Select the drive and the directory in which the text file is saved
- ▶ Move the highlight to the text file
- ▶ Press the ENT key: The TNC displays a window for selection of the editor
- ▶ Press ENT to select the **Mousepad** application. Alternatively, you can also open the TXT files with the TNC's internal text editor
- ▶ The TNC opens the text file in its own application using the **Mousepad** additional tool

ENT



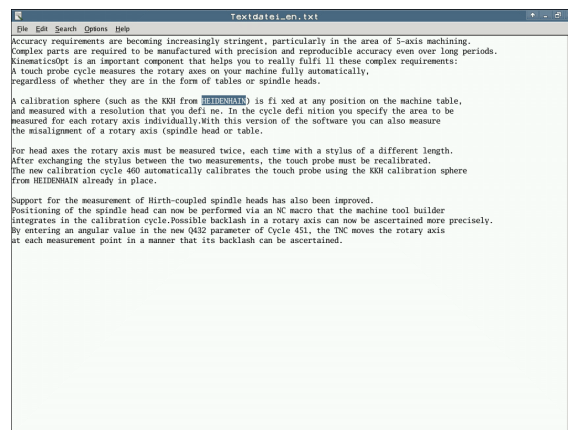
If you open an H or I file on an external drive and save it on the TNC drive using **Mousepad**, the programs are not converted automatically to the internal control format. Programs that are saved in this way cannot be run or opened with the TNC editor.

With the key combination ALT+TAB you can always return to the TNC user interface while leaving the text file open. Alternatively, you can also click the corresponding symbol in the taskbar to switch back to the TNC interface.

The shortcuts you are familiar with from Windows, which you can use to edit texts quickly (CTRL+C, CTRL+V,...), are available within Mousepad.

To exit **Mousepad**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager



## Displaying graphics files

To open graphics files with the extension bmp, gif, jpg or png directly on the TNC, proceed as follows:

- ▶ **PGM MGT**
- ▶ Call the file manager
- ▶ Select the directory in which the graphics file is saved
- ▶ Move the highlight to the graphics file
- ▶ Press **ENT**: The TNC opens the graphics file in its own application using the **ristretto** additional tool

With the key combination ALT+TAB you can always return to the TNC user interface while leaving the graphics file open. Alternatively, you can also click the corresponding symbol in the taskbar to switch back to the TNC interface.

More information on how to use the **ristretto** function is provided under **Help**.

To exit **ristretto**, proceed as follows:

- ▶ Use the mouse to select the **File** menu item
- ▶ Select the menu item **Quit**: The TNC returns to the file manager



Data transfer to or from an external data medium



Before you can transfer data to an external data medium, you must set up the data interface (see "Setting the data interfaces" on page 346).

Depending on the data transfer software you use, problems can occur occasionally when you transmit data over a serial interface. They can be overcome by repeating the transmission.



Call the file manager



Select the screen layout for data transfer: press the WINDOW soft key. In the left half of the screen the TNC shows all files in the current directory. In the right half of the screen it shows all files saved in the root directory (TNC:\)

Use the arrow keys to highlight the file(s) that you want to transfer:



Moves the highlight up and down within a window



Moves the highlight from the left to the right window, and vice versa

If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

Manual operation		Programming	
		PAT.H	
TNC:\nc_prog\PGM\*		TNC:\*	
File name	Bytes Status	File name	Bytes Status
DXF.H	292	config	
error.h	554	nc_prog	
EXIT.H	1801	sysies	
EXIS.H	959	table	
EXIS.SL.H	1792	incguide	
EXIS.H	786	userlog.xml	17260
EXIS.SL.H	1513		
EX4.H	1836		
HEREL.H	541		
koord.h	1586		
NEUG.T	884		
P300.P	444		
PLI.H	2697		
Ra-P1.h	6675		
Rastplatte.h	4937		
Rastplatte.h.bak	6390		
Reset.h	235		
Schulter.h	3477		
STAT.H	479		
STAT.H	623		
TCH.h	1320		
turbine.H	1871		
wheel.h	10767		
zeroshift.d	6557		
51 file(s) 21.75 GB vacant		6 file(s) 21.75 GB vacant	
PAGE	PAGE	SELECT	COPY
↑	↓	ABC	XVZ
		SELECT	WINDOW
		TYPE	
		SHOW	END
		TREE	



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



To select another drive or directory: Press the soft key for choosing the directory. The TNC opens a pop-up window. Select the desired directory in the pop-up window by using the arrow keys and the ENT key



Transfer a single file: Press the COPY soft key, or



To transfer several files, press the TAG soft key (in the second soft-key row, see "Tagging files", page 106)

Confirm with the OK soft key or with the ENT key. A status window appears on the TNC, informing about the copying progress, or



To end data transfer, move the highlight into the left window and then press the WINDOW soft key. The standard file manager window is displayed again



To select another directory in the split-screen display, press the SHOW TREE soft key. If you press the SHOW FILES soft key, the TNC shows the content of the selected directory!



## The TNC in a network



To connect the Ethernet card to your network, see "Ethernet interface", page 351.

The TNC logs error messages during network operation, see "Ethernet interface", page 351.

If the TNC is connected to a network, the directory window displays additional drives (see figure). All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been granted the corresponding rights.

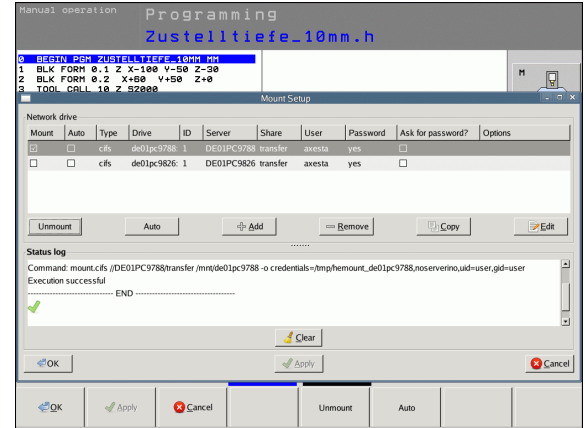
### Connecting and disconnecting a network drive

PGM  
MGT

- ▶ To select the program management: Press the PGM MGT key. If necessary, press the WINDOW soft key to set up the screen as it is shown at the upper right

NET

- ▶ To select the network settings: Press the NETWORK soft key (second soft key row).
- ▶ To manage the network drives: Press the DEFINE NETWORK CONNECTN. soft key. The TNC displays possible network drives to which you have access in a window. The soft keys described below are used to define the connection for each drive



Function	Soft key
Establish the network connection. If the connection is active, the TNC marks the <b>Mount</b> column.	<b>Connect</b>
Disconnect the network connection	<b>Unmount</b>
Automatically establish network connection whenever the TNC is switched on. The TNC marks the <b>Auto</b> column if the connection is established automatically	<b>Auto</b>
Set up new network connection	<b>Add</b>
Delete existing network connection	<b>Remove</b>
Copy network connection	<b>Copy</b>
Edit network connection	<b>Edit</b>
Clear status window	<b>Clear</b>



## USB devices on the TNC

Backing up data from or loading onto the TNC is especially easy with USB devices. The TNC supports the following USB block devices:

- Floppy disk drives with FAT/VFAT file system
- Memory sticks with the FAT/VFAT file system
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The TNC automatically detects these types of USB devices when connected. The TNC does not support USB devices with other file systems (such as NTFS). The TNC displays the **USB: TNC does not support device** error message when such a device is connected.



The TNC also displays the **USB: TNC does not support device** error message if you connect a USB hub. In this case, simply acknowledge the message with the CE key.

In theory, you should be able to connect all USB devices with the file systems mentioned above to the TNC. It may happen that a USB device is not identified correctly by the control. In such cases, use another USB device.

The USB devices appear as separate drives in the directory tree, so you can use the file-management functions described in the earlier chapters correspondingly.



Your machine tool builder can assign permanent names for USB devices. Refer to your machine manual.



To remove a USB device, proceed as follows:



- ▶ Call the file manager: Press the PGM MGT key



- ▶ Select the left window with the arrow key



- ▶ Use the arrow keys to select the USB device to be removed



- ▶ Scroll through the soft-key row



- ▶ Select additional functions



- ▶ Select the function for removing USB devices. The TNC removes the USB device from the directory tree



- ▶ Exit the file manager

In order to re-establish a connection with a USB device that has been removed, press the following soft key:



- ▶ Select the function for reconnection of USB devices

# 4

**Programming:  
Programming aids**



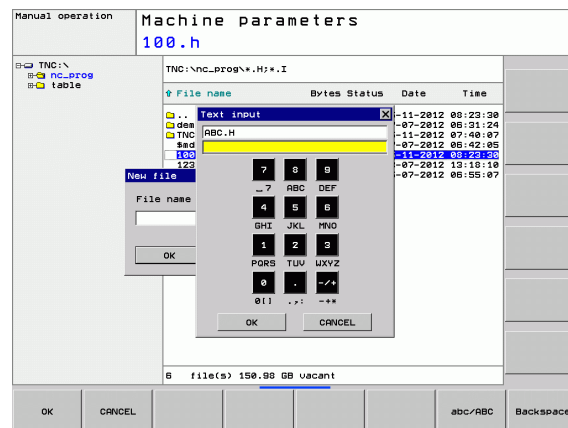
## 4.1 Screen keyboard

You can enter letters and special characters with the screen keypad or (if available) with a PC keyboard connected over the USB port.

### Entering text with the screen keyboard

- ▶ Press the GOTO key if you want to enter a text, for example a program name or directory name, using the screen keyboard
- ▶ The TNC opens a window in which the numeric entry field of the TNC is displayed with the corresponding letters assigned
- ▶ You can move the cursor to the desired character by repeatedly pressing the respective key
- ▶ Wait until the selected character is transferred to the input field before you enter the next character
- ▶ Use the OK soft key to load the text into the open dialog field

Use the **abc/ABC** soft key to select upper or lower case. If your machine tool builder has defined additional special characters, you can call them with the SPECIAL CHARACTER soft key and insert them. To delete individual characters, use the BACKSPACE soft key.



# 4.2 Adding comments

## Application

You can add comments to a part program to explain program steps or make general notes.



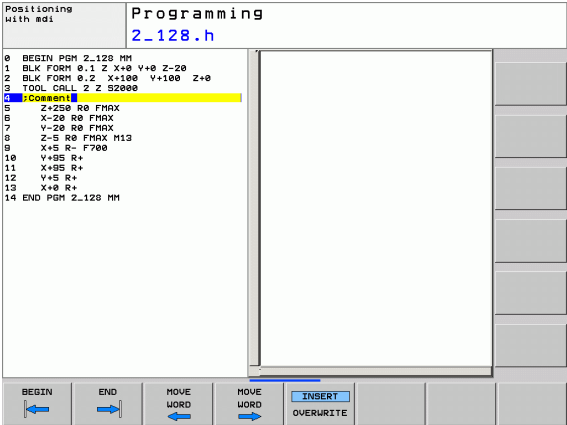
If the TNC cannot show the entire comment on the screen, the >> sign is displayed.  
The last character in a comment block must not have any tilde (~).

## Adding comments

- ▶ Select the block after which the comment is to be inserted
- ▶ Press the SPEC FCT key
- ▶ Press the PROGRAM FUNCTIONS soft key
- ▶ Shift the soft-key row and select the INSERT COMMENT soft key

## Functions for editing of the comment

Function	Soft key
Jump to beginning of comment	
Jump to end of comment	
Jump to the beginning of a word. Words must be separated by a space	
Jump to the end of a word. Words must be separated by a space	
Switch between insert mode and overwrite mode	



## 4.3 Structuring programs

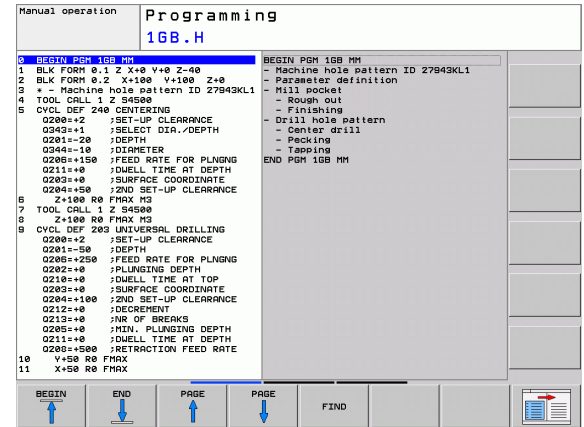
### Definition and applications

This TNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 37 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 programs in a clear and comprehensible manner.

This function is particularly convenient if you want to change the program later. Structuring blocks can be inserted into the part program at any point. They can also be displayed in a separate window, and edited or added to, as desired.



The inserted structure items are managed by the TNC in a separate file (extension: .SEC.DEF). This speeds navigation in the program structure window.



### Displaying the program structure window / Changing the active window

-  ▶ To display the program structure window, select the screen display PROGRAM+SECTS
-  ▶ To change the active window, press the "Change window" soft key

### Inserting a structuring block in the (left) program window

- ▶ Select the block after which the structuring block is to be inserted
  -  ▶ Press the INSERT SECTION soft key
  - ▶ Enter the structuring text
  -  ▶ If necessary, change the structure depth with the soft key

### Selecting blocks in the program structure window

If you are scrolling through the program structure window block by block, the TNC at the same time automatically moves the corresponding NC blocks in the program window. This way you can quickly skip large program sections.

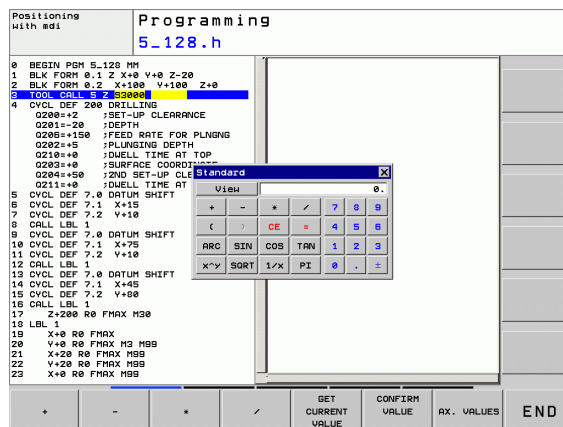
## 4.4 Integrated calculator

### Operation

The TNC features an integrated calculator with the basic mathematical functions.

- Use the CALC key to show and hide the on-line calculator
- The calculator is operated with short commands through the alphabetic keyboard. The commands are shown in a special color in the calculator window:

Mathematical function	Command (key)
Addition	+
Subtraction	−
Multiplication	*
Division	/
Calculations in parentheses	( )
Arc cosine	ARC
Sine	SIN
Cosine	COS
Tangent	TAN
Powers of values	X^Y
Square root	SQRT
Inversion	1/x
pi (3.14159265359)	PI
Add value to buffer memory	M+
Save the value to buffer memory	MS
Recall from buffer memory	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check the algebraic sign	SGN
Form the absolute value	ABS






Mathematical function	Command (key)
Truncate decimal places	INT
Truncate integers	FRAC
Modulus operator	MOD
Select view	View
Delete value	CE
Unit of measure	MM or INCH
Display mode for angle values	DEG (degree) or RAD (radian measure)
Display mode of the numerical value	DEC (decimal) or HEX (hexadecimal)

### Transferring the calculated value into the program

- ▶ Use the arrow keys to select the word into which the calculated value is to be transferred
- ▶ Superimpose the on-line calculator by pressing the CALC key and perform the desired calculation
- ▶ Press the actual-position-capture key for the TNC to superimpose a soft-key row
- ▶ Press the CALC soft key for the TNC to transfer the value into the active input box and to close the calculator

### Adjusting the position of the calculator

Press the ADDITIONAL FUNCTIONS soft key to get to the settings for shifting the calculator:

Function	Soft key
Move calculator in the direction of the arrow	
Adjust the increment for movement	
Position the calculator in the center	





# 4.5 Programming graphics

## Generating / not generating graphics during programming

While you are writing the part program, you can have the TNC generate a 2-D pencil-trace graphic of the programmed contour.

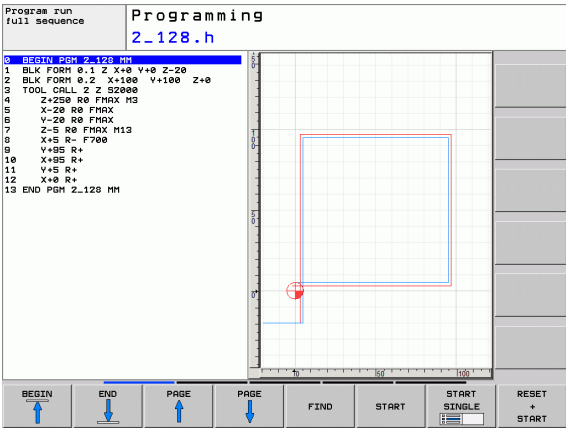
- ▶ To switch the screen layout to displaying program blocks to the left and graphics to the right, press the SPLIT SCREEN key and PROGRAM + GRAPHICS soft key



- ▶ Set the AUTO DRAW soft key to ON. While you are entering the program lines, the TNC generates each path contour you program in the graphics window in the right screen half

If you do not wish to have the TNC generate graphics during programming, set the AUTO DRAW soft key to OFF.

Even when AUTO DRAW ON is active, graphics are not generated for program section repeats.



## Generating a graphic for an existing program

- ▶ Use the arrow keys to select the block up to which you want the graphic to be generated, or press GOTO and enter the desired block number



- ▶ To generate graphics, press the RESET + START soft key

Additional functions:

Function	Soft key
Generate a complete graphic	RESET + START
Generate programming graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Stop the programming graphics. This soft key only appears while the TNC is generating the interactive graphics	STOP



## Block number display ON/OFF



► Shift the soft-key row: see figure



► Show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW

► Hide block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT

## Erasing the graphic



► Shift the soft-key row: see figure



► Erase graphic: Press CLEAR GRAPHICS soft key

## Showing grid lines



► Shift the soft-key row: see figure



► Show grid lines: Press the "Show grid lines" soft key

## Magnifying or reducing a detail

You can select the graphics display by selecting a detail with the frame overlay. You can now magnify or reduce the selected detail.

► Select the soft-key row for detail magnification/reduction (second row, see figure)

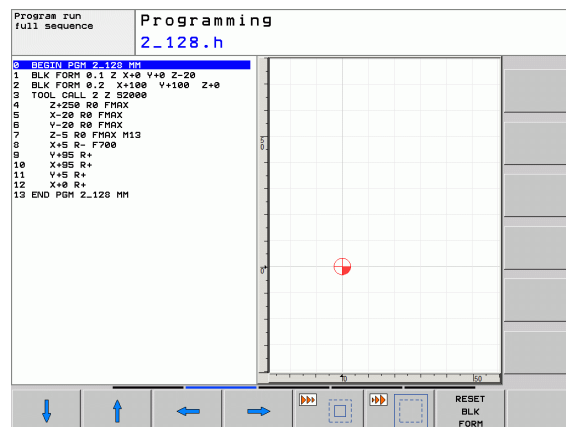
The following functions are available:

Function	Soft key
Show and move the frame overlay. Press and hold the desired soft key to move the frame overlay	
Reduce the frame overlay—press and hold the soft key to reduce the detail	
Enlarge the frame overlay—press and hold the soft key to magnify the detail	



► Confirm the selected area with the WINDOW DETAIL soft key

The RESET WORKPIECE BLANK soft key is used to restore the original section.



## 4.6 Error messages

### Display of errors

The TNC generates error messages when it detects problems such as:

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine
- Incorrect use of touch probes

When an error occurs, it is displayed in red type in the header. Long and multi-line error messages are displayed in abbreviated form. If an error occurs in the background mode, the word "Error" is displayed in red type. Complete information on all pending errors is shown in the error window.

If a rare "processor check error" should occur, the TNC automatically opens the error window. You cannot remove such an error. Shut down the system and restart the TNC.

The error message is displayed in the header until it is cleared or replaced by a higher-priority error.

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block.

### Opening the error window



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

### Closing the error window



- ▶ Press the END soft key—or



- ▶ Press the ERR key. The TNC closes the error window.



## Detailed error messages

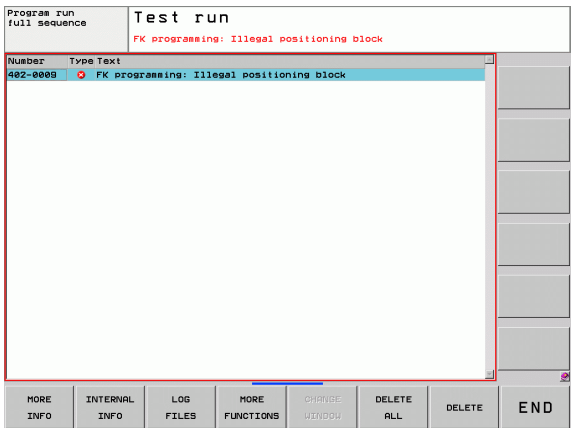
The TNC displays possible causes of the error and suggestions for solving the problem:

- Open the error window
  - Information on the error cause and corrective action: Position the highlight on the error message and press the MORE INFO soft key. The TNC opens a window with information on the error cause and corrective action
  - Exit Info: Press the MORE INFO soft key again

## 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
  - Detailed information about the error message: Position the highlight on the error message and press the INTERNAL INFO soft key. The TNC opens a window with internal information about the error
  - To exit Details, press the INTERNAL INFO soft key again



## Clearing errors

### Clearing errors outside of the error window:



- Clear the error/message in the header: Press the CE key



In some operating modes (such as the Editing mode), the CE button cannot be used to clear the error, since the button is reserved for other functions.

### Clearing more than one error:

- Open the error window

DELETE

- Clear individual errors: Position the highlight on the error message and press the DELETE soft key

DELETE  
ALL

- Clear all errors: Press the DELETE ALL soft key



If the cause of the error has not been removed, the error message cannot be deleted. In this case, the error message remains in the window.

## Error log

The TNC stores errors and important events (e.g. system startup) in an error log. The capacity of the error log is limited. If the log is full, the TNC uses a second file. If this is also full, the first error log is deleted and written to again, and so on. To view the error history, switch between CURRENT FILE and PREVIOUS FILE.

- Open the error window

LOG  
FILES

- Press the LOG FILES soft key

ERROR  
LOG

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

PREVIOUS  
FILE

- If you need the previous log file: Press the PREVIOUS FILE soft key

CURRENT  
FILE

- If you need the current log file: Press the CURRENT FILE soft key

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



Keystroke log

The TNC stores keystrokes and important events (e.g. system startup) in a keystroke log. The capacity of the keystroke log is limited. If the keystroke log is full, the control switches to a second keystroke log. If this second file becomes full, the first keystroke log is cleared and written to again, and so on. To view the keystroke history, switch between CURRENT FILE and PREVIOUS FILE.

- LOG FILES

▶ Press the LOG FILES soft key
- KEYSTROKE LOG








▶ Open the keystroke log file: Press the KEYSTROKE LOG FILE soft key
- PREVIOUS FILE

▶ If you need the previous log file: Press the PREVIOUS FILE soft key
- CURRENT FILE

▶ If you need the current log file: Press the CURRENT FILE soft key

The TNC 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 buttons and soft keys for viewing the log files:

Function	Soft key/Keys
Go to beginning of log file	
Go to end of log file	
Current log file	
Previous log file	
Up/down one line	 
Return to main menu	



## Informational texts

After a faulty operation, such as pressing a key without function or entering a value outside of the valid range, the TNC displays a (green) text in the header, informing you that the operation was not correct. The TNC clears this note upon the next valid input.

## Saving service files

If necessary, you can save the "Current status of the TNC," and make it available to a service technician for evaluation. A group of service files is saved (error and keystroke log files, as well as other files that contain information about the current status of the machine and the machining).

If you repeat the "Save service files" function with the same file name, the previously saved group of service data files is overwritten. To avoid this, use another file name when you repeat the function.

### Saving service files:

- Open the error window



- Press the LOG FILES soft key



- Press the SAVE SERVICE FILES soft key: The TNC opens a pop-up window in which you can enter a name for the service file



- Save the service files: Press the OK soft key

## Calling the TNCguide help system

You can call the TNC's help system via soft key. Immediately the help system shows you the same error explanation that you receive by pressing the HELP soft key.



If your machine manufacturer also provides a help system, the TNC shows an additional MACHINE MANUFACTURER soft key with which you can call this separate help system. There you will find further, more detailed information on the error message concerned.



- Call the help for HEIDENHAIN error messages



- Call the help for HEIDENHAIN error messages, if available

## 4.7 Context-sensitive help system

### Application



Before you can use the TNCguide, you need to download the help files from the HEIDENHAIN home page (see "Downloading current help files" on page 137).

The **TNCguide** context-sensitive help system includes the user documentation in HTML format. The TNCguide is called with the HELP key, and the TNC often immediately displays the information specific to the condition from which the help was called (context-sensitive call). Even if you are editing an NC block and press the HELP key, you are usually brought to the exact place in the documentation that describes the corresponding function.



The TNC always tries to start the TNCguide in the language that you have selected as the conversational language on your TNC. If the files with this language are not yet available on your TNC, it automatically opens the English version.

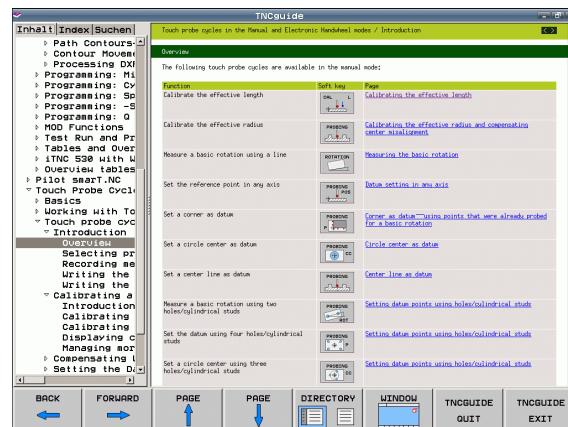
The following user documentation is available in the TNCguide:

- Conversational Programming User's Manual (**BHBKlartext.chm**)
- DIN/ISO User's Manual (**BHBIso.chm**)
- User's Manual for Cycle Programming (**BHBtchprobe.chm**)
- List of All Error Messages (**errors.chm**)

In addition, the **main.chm** "book" file is available, with the contents of all existing .chm files.



As an option, your machine tool builder can embed machine-specific documentation in the **TNCguide**. These documents then appear as a separate book in the **main.chm** file.





## Working with the TNCguide

### Calling the TNCguide

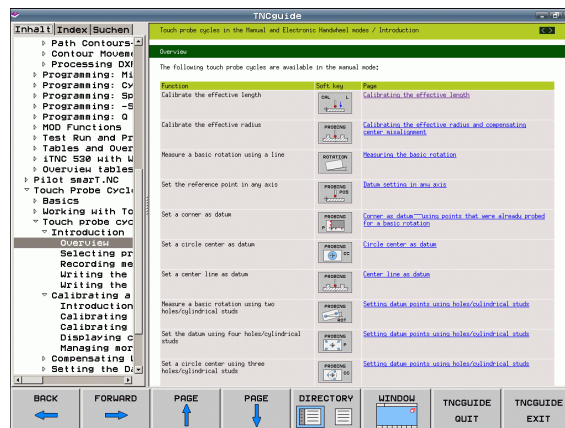
There are several ways to start the TNCguide:

- ▶ Press the HELP key if the TNC is not already showing an error message
- ▶ Click the help symbol at the lower right of the screen beforehand, then click the appropriate soft keys
- ▶ Use the file manager to open a help file (.chm file). The TNC can open any .chm file, even if it is not saved on the TNC's hard disk



If one or more error messages are waiting for your attention, the TNC shows the help directly associated with the error messages. To start the **TNCguide**, you first have to acknowledge all error messages.

When the help system is called on the programming station, the TNC starts the internally defined standard browser (usually the Internet Explorer), or otherwise a browser adapted by HEIDENHAIN.



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 TNC 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, and click: The TNC opens the TNCguide. If no specific part of the help is assigned to the selected soft key, the TNC opens the book file **main.chm**, in which you can use the search function or the navigation to find the desired explanation manually

Even if you are editing an NC block, context-sensitive help is available:

- ▶ Select any NC block
- ▶ Use the arrow keys to move the cursor to the block
- ▶ Press the HELP key: The TNC starts the help system and shows a description for the active function (does not apply to miscellaneous functions or cycles that were integrated by your machine tool builder)



### Navigating in the TNCguide







It's easiest to use the mouse to navigate in the TNCguide. A table of contents appears on the left side of the screen. By clicking the rightward pointing triangle you open subordinate sections, and by clicking the respective entry you open the individual pages. It is operated in the same manner as the Windows Explorer.

Linked text positions (cross references) are shown underlined and in blue. Clicking the link opens the associated page.

Of course you can also operate the TNCguide through keys and soft keys. The following table contains an overview of the corresponding key functions.

Function	Soft key
<ul style="list-style-type: none"> <li>■ If the table of contents at left is active: Select the entry above it or below it</li> <li>■ If the text window at right is active: Move the page downward or upward if texts or graphics are not shown completely</li> </ul>	<div> <div>↓</div> <div>↑</div> </div>
<ul style="list-style-type: none"> <li>■ If the table of contents at left is active: Open a branch of the table of contents. If the branch is at its end, jump into the window at right</li> <li>■ If the text window at right is active: No function</li> </ul>	<div>→</div>
<ul style="list-style-type: none"> <li>■ If the table of contents at left is active: Close a branch of the table of contents</li> <li>■ If the text window at right is active: No function</li> </ul>	<div>←</div>
<ul style="list-style-type: none"> <li>■ If the table of contents at left is active: Use the cursor key to show the selected page</li> <li>■ If the text window at right is active: If the cursor is on a link, jump to the linked page</li> </ul>	<div>ENT</div>
<ul style="list-style-type: none"> <li>■ If the table of contents at left is active: Switch the tab between the display of the table of contents, display of the subject index, and the full-text search function and switching to the screen half at right</li> <li>■ If the text window at right is active: Jump back to the window at left</li> </ul>	<div> <div>☰</div> </div>
<ul style="list-style-type: none"> <li>■ If the table of contents at left is active: Select the entry above it or below it</li> <li>■ If the text window at right is active: Jump to the next link</li> </ul>	<div> <div>☰↓</div> <div>☰↑</div> </div>
Select the page last shown	<div>BACK</div> <div>←</div>
Page forward if you have used the "select page last shown" function	<div>FORWARD</div> <div>→</div>



Function	Soft key
Move up by one page	
Move down by one page	
Display or hide table of contents	
Switch between full-screen display and reduced display. With the reduced display you can see some of the rest of the TNC window	
The focus is switched internally to the TNC application so that you can operate the control when the TNCguide is open. If the full screen is active, the TNC reduces the window size automatically before the change of focus	
Close the 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 cursor keys.

The left side is active.



- ▶ Select the **Index** tab
- ▶ Activate the **Keyword** input field
- ▶ Enter the word for the desired subject and the TNC synchronizes the index and creates a list in which you can find the subject more easily, or
- ▶ Use the arrow key to highlight the desired keyword
- ▶ Use the ENT key to call the information on the selected keyword



You can enter the search word only with a keyboard connected via USB.

## Full-text search

In the **Find** tab you can search the entire TNCguide for a specific word.

The left side is active.



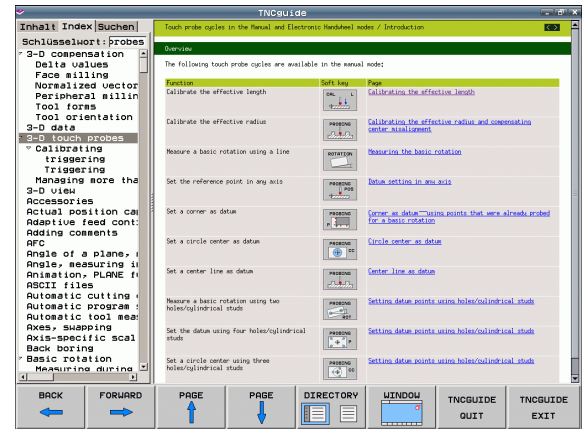
- ▶ Select the **Find** tab
- ▶ Activate the **Find:** input field
- ▶ Enter the desired word and confirm with the ENT key:  
The TNC lists all sources containing the word
- ▶ Use the arrow key to highlight the desired source
- ▶ Press the ENT key to go to the selected source



You can enter the search word only with a keyboard connected via USB.

The full-text search only works for single words.

If you activate the **Search only in titles** function (by mouse or by using the cursor and the space key), the TNC searches only through headings and ignores the body text.



## Downloading current help files

You'll find the help files for your TNC software on the HEIDENHAIN homepage **www.heidenhain.de** under:

- ▶ Documentation / Information
- ▶ User Documentation
- ▶ TNCguide
- ▶ Select the desired language
- ▶ TNC Controls
- ▶ Series, e.g. TNC 600 Series
- ▶ Desired NC software number, e.g. TNC 640 (34059x-01)
- ▶ Select the desired language version from the **TNCguide online help** table
- ▶ Download the ZIP file and unzip it
- ▶ Move the unzipped CHM files to the TNC in the **TNC:\tncguide\en** directory or into the respective language subdirectory (see also the following table)



If you want to use TNCremoNT to transfer the CHM files to the TNC, then in the **Extras>Configuration>Mode>Transfer in binary format** menu item you have to enter the extension **.CHM**.

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 (software option)	TNC:\tncguide\s1
Norwegian	TNC:\tncguide\no
Slovak	TNC:\tncguide\sk
Latvian	TNC:\tncguide\lv
Korean	TNC:\tncguide\kr
Estonian	TNC:\tncguide\et
Turkish	TNC:\tncguide\tr
Romanian	TNC:\tncguide\ro
Lithuanian	TNC:\tncguide\lt



# 5

**Programming: Tools**



## 5.1 Entering tool-related data

### Feed rate F

The feed rate **F** is the speed (in millimeters per minute or inches per minute) 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 **TOOL CALL** block and in every positioning block (see "Creating the program blocks with the axis keys" on page 162). In millimeter-programs you enter the feed rate in mm/min, and in inch-programs, for reasons of resolution, in 1/10 inch/min.

#### Rapid traverse

If you wish to program rapid traverse, enter **F MAX**. To enter **FMAX**, press the ENT key or the FMAX soft key when the dialog question **FEED RATE F = ?** appears on the control's screen.



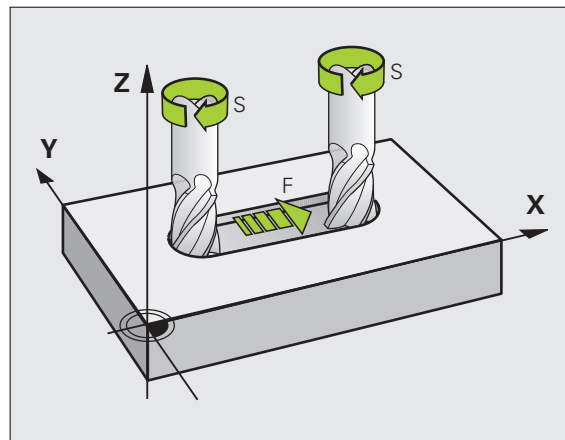
To move your machine at rapid traverse, you can also program the corresponding numerical value, e.g. **F30000**. Unlike **FMAX**, this rapid traverse remains in effect not only in the individual block but in all blocks until you program a new feed rate.

#### Duration of effect

A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. **FMAX** is only effective in the block in which it is programmed. After the block with **FMAX** 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 program run with the feed-rate override knob F.





## Spindle speed S

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

### Programmed change

In the part program, you can change the spindle speed in a **TOOL CALL** block by entering the spindle speed only:

TOOL  
CALL

- ▶ Program a tool call: Press the TOOL CALL key
- ▶ Ignore the dialog question for **Tool number ?** with the NO ENT key
- ▶ Ignore the dialog question for **Working spindle axis X/Y/Z ?** with the NO ENT key
- ▶ Enter the new spindle speed for the dialog question **Spindle speed S= ?**, and confirm with END key, or press the VC soft key to switch to entry of the cutting speed

### Changing during program run

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



## 5.2 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 TNC 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.

Tool data can be entered either directly in the part program with **TOOL DEF** or separately in a tool table. In a tool table, you can also enter additional data for the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

### Tool numbers and tool names

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 16 characters.

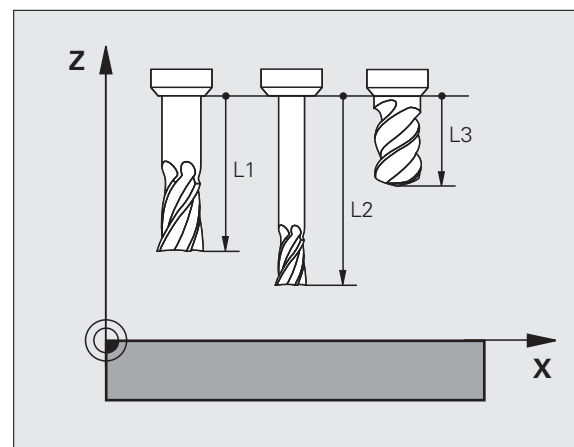
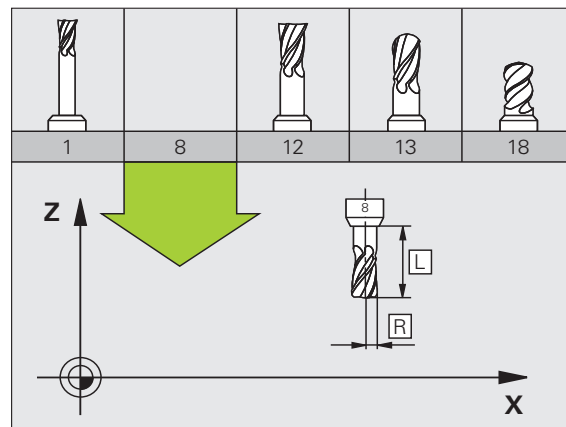
The tool number 0 is automatically defined as the zero tool with the length  $L=0$  and the radius  $R=0$ . In tool tables, tool T0 should also be defined with  $L=0$  and  $R=0$ .

### Tool length L

You should always enter the tool length  $L$  as an absolute value based on the tool reference point. The entire tool length is essential for the TNC in order to perform numerous functions involving multi-axis machining.

### 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 describes a tool oversize (**DL**, **DR**, **DR2**>0). If you are programming the machining data with an allowance, enter the oversize value in the **TOOL CALL** block of the part program.

A negative delta value describes a tool undersize (**DL**, **DR**, **DR2**<0). An undersize is entered in the tool table for wear.

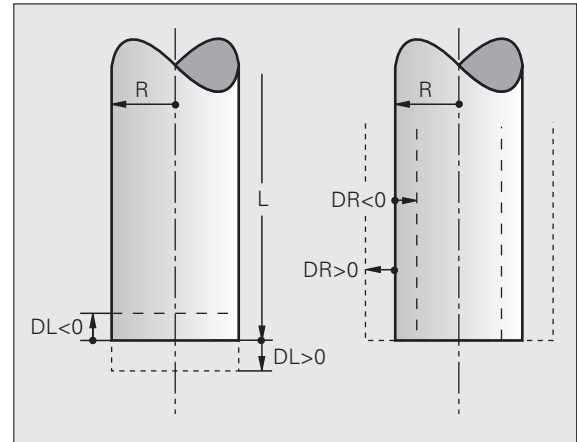
Delta values are usually entered as numerical values. In a **TOOL CALL** block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to  $\pm 99.999$  mm.



Delta values from the tool table influence the graphical representation of the **tool**. The representation of the **workpiece** remains the same in the simulation.

Delta values from the **TOOL CALL** block change the represented size of the **workpiece** during the simulation. The simulated **tool size** remains the same.



## Entering tool data into the program

The number, length and radius of a specific tool is defined in the **TOOL DEF** block of the part program.

► Select the tool definition: Press the TOOL DEF key



- **Tool number**: Each tool is uniquely identified by its tool number
- **Tool length**: Compensation value for the tool length
- **Tool radius**: Compensation value for the tool radius



In the programming dialog, you can transfer the value for tool length and tool radius directly into the input line by pressing the desired axis soft key.

### Example

```
4 TOOL DEF 5 L+10 R+5
```

## Entering tool data in the table

You can define and store up to 9999 tools and their tool data in a tool table. Also see the Editing Functions later in this Chapter. In order to be able to assign various compensation data to a tool (indexing tool number), insert a line and extend the tool number by a dot and a number from 1 to 9 (e.g. **T 5.2**).

You must use tool tables if

- you wish to use indexed tools such as stepped drills with more than one length compensation value
- your machine tool has an automatic tool changer
- you want to work with Cycles 25x



If you create or manage further tool tables, the file name has to start with a letter.

You can select either list view or form view for tables via the "Screen layout" key.

### Tool table: Standard tool data

Abbr.	Inputs	Dialog
<b>T</b>	Number by which the tool is called in the program (e.g. 5, indexed: 5.2)	-
<b>NAME</b>	Name by which the tool is called in the program (no more than 16 characters, all capitals, no spaces)	<b>Tool name?</b>
<b>L</b>	Compensation value for tool length L	<b>Tool length?</b>
<b>R</b>	Compensation value for the tool radius R	<b>Tool radius R?</b>
<b>R2</b>	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical or toroid cutters)	<b>Tool radius R2?</b>
<b>DL</b>	Delta value for tool length L	<b>Tool length oversize?</b>
<b>DR</b>	Delta value for tool radius R	<b>Tool radius oversize?</b>
<b>DR2</b>	Delta value for tool radius R2	<b>Tool radius oversize R2?</b>
<b>LCUTS</b>	Tooth length of the tool for Cycle 22	<b>Tooth length in the tool axis?</b>
<b>ANGLE</b>	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	<b>Maximum plunge angle?</b>
<b>TL</b>	Set tool lock ( <b>TL:</b> for Tool Locked)	<b>Tool locked?</b> Yes = ENT / No = NO ENT
<b>RT</b>	Number of a replacement tool, if available ( <b>RT:</b> for Replacement Tool; see also <b>TIME2</b> )	<b>Replacement tool?</b>
<b>TIME1</b>	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information	<b>Maximum tool age?</b>



Abbr.	Inputs	Dialog
<b>TIME2</b>	Maximum tool life in minutes during <b>TOOL CALL</b> : If the current tool life reaches or exceeds this value, the TNC changes the tool during the next <b>TOOL CALL</b> (see also <b>CUR_TIME</b> ).	<b>Maximum tool age for TOOL CALL?</b>
<b>CUR_TIME</b>	Current age of the tool in minutes: The TNC automatically counts the current tool life ( <b>CUR_TIME</b> ). A starting value can be entered for used tools	<b>Current tool age?</b>
<b>TYPE</b>	Tool type: Press the SELECT TYPE (3rd soft-key row); the TNC superimposes a window where you can select the type of tool you want. You can assign tool types to specify the display filter settings such that only the selected type is visible in the table	<b>Tool type?</b>
<b>DOC</b>	Comment on tool (up to 16 characters)	<b>Tool comment?</b>
<b>PLC</b>	Information on this tool that is to be sent to the PLC	<b>PLC status?</b>
<b>PTYP</b>	Tool type for evaluation in the pocket table	<b>Tool type for pocket table?</b>
<b>TP_NO</b>	Reference to the number of the touch probe in the touch-probe table	<b>Number of the touch probe</b>
<b>T_ANGLE</b>	Point angle of the tool. Is used by the Centering cycle (Cycle 240) in order to calculate the centering depth from the diameter entry	<b>Point angle?</b>
<b>LAST_USE</b>	Date and time that the tool was last inserted via <b>TOOL CALL</b> <b>Input range:</b> Max. 16 characters, format specified internally: Date = yyyy.mm.dd, time = hh.mm	<b>LAST_USE</b>



## Tool table: Tool data required for automatic tool measurement



For a description of the cycles for automatic tool measurement, see the User's Manual for Cycle Programming.

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	Number of teeth?
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: length?
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: radius?
R2TOL	Permissible deviation from tool radius R2 for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: Radius 2?
DIRECT.	Cutting direction of the tool for measuring the tool during rotation	Cutting direction (M3 = -)?
R_OFFS	Tool radius measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	Tool offset: radius?
L_OFFS	Tool length measurement: Tool offset in addition to <b>offsetToolAxis</b> (114104) between upper surface of stylus and lower surface of tool. Default: 0	Tool offset: length?
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: length?
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: radius?



## Editing tool tables

The tool table that is active during execution of the part program is designated TOOL.T and must be saved in the **TNC:\table** directory.

Other tool tables that are to be archived or used for test runs are given any other names with the extension T. By default, for Test Run and Programming modes the TNC uses the "simtool.t" table, which is also stored in the "table" directory. In the Test Run mode, press the TOOL TABLE soft key to edit it.

To open the tool table TOOL.T:

- Select any machine operating mode



- Select the tool table: Press the TOOL TABLE soft key



- Set the EDIT soft key to ON

Tool table editing						Test run
TNC:\table\tool.t						
T	NAME	L	R	R2		
0	NULLWERKZEUG	0	0	0		H
1	D2	20	1	0		S
2	D4	40	2	0		T
3	D6	60	3	0		
4	D8	80	4	0		
5	D10	100	5	0		
6	D12	120	6	0		
7	D14	140	7	0		
8	D16	160	8	0		
9	D18	180	9	0		
10	D20	200	10	0		
11	D22	220	11	0		
12	D24	240	12	0		
13	D26	260	13	0		
14	D28	280	14	0		
15	D30	300	15	0		
16	D32	320	16	0		
17	D34	340	17	0		
18	D36	360	18	0		
19	D38	380	19	0		
20	D40	400	20	0		
21	D42	420	21	0		
22	D44	440	22	0		

## Displaying only specific tool types (filter setting)

- Press the TABLE FILTER soft key (fourth soft-key row)
- Select the tool type by pressing a soft key: The TNC only shows tools of the type selected
- Cancel filter: Press the previously selected tool type again or select another tool type



The machine tool builder adapts the features of the filter function to the requirements of your machine. The machine manual provides further information.

## Opening any other tool table

- ▶ Select the Programming and Editing mode of operation
  - ▶ Call the file manager
  - ▶ Press the SELECT TYPE soft key to select the file type
  - ▶ Show type .T files: Press the SHOW .T soft key
  - ▶ Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key






When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys. You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table below.

If the TNC cannot show all positions in the tool table in one screen page, the highlight bar at the top of the table will display the >> or << symbols.

Editing functions for tool tables	Soft key
Select beginning of table	<div>BEGIN ↑</div>
Select end of table	<div>END ↓</div>
Select previous page in table	<div>PAGE ↑</div>
Select next page in table	<div>PAGE ↓</div>
Find the text or number	<div>FIND</div>
Move to beginning of line	<div>BEGIN LINE ←</div>
Move to end of line	<div>END LINE →</div>
Copy highlighted field	<div>COPY FIELD</div>
Insert copied field	<div>PASTE FIELD</div>
Add the entered number of lines (tools) at the end of the table	<div>APPEND N LINES</div>
Insert a line with definable tool number	<div>INSERT LINE</div>
Delete current line (tool)	<div>DELETE LINE</div>





Editing functions for tool tables	Soft key
Sort the tools according to the content of a column	
Show all drills in the tool table	
Show all cutters in the tool table	
Show all taps/thread cutters in the tool table	
Show all touch probes in the tool table	

### Exiting the tool table

- Call the file manager and select a file of a different type, such as a part program



### Importing tool tables



The machine manufacturer can adapt the IMPORT TABLE function. The machine manual provides further information.

If you export a tool table from an iTNC 530 and import it into a TNC 128, you have to adapt its format and content before you can use the tool table. On the TNC 128, you can adapt the tool table conveniently with the IMPORT TABLE function. The TNC converts the contents of the imported tool table to a format valid for the TNC 128 and saves the changes to the selected file. Follow this procedure:

- ▶ Save the tool table of the iTNC 530 to the **TNC:\table** directory
- ▶ Select the Programming mode of operation
- ▶ Call the file manager: Press the PGM MGT key
- ▶ Move the highlight to the tool table you want to import
- ▶ Press the MORE FUNCTIONS soft key
- ▶ Select the IMPORT TABLE soft key: The TNC inquires whether you really want to overwrite the selected tool table
- ▶ Press the CANCEL soft key if you do not want to overwrite the file, or
- ▶ Press the ADAPT TABLE FORMAT soft key to overwrite the file
- ▶ Open the converted table and check its contents



The following characters are permitted in the **Name** column of the tool table:  
"ABCDEFGHIJKLMNOPQRSTUVWXYZ0123456789#\$&-. \_". The TNC changes a comma in the tool name to a period during import.

The TNC overwrites the selected tool table when running the IMPORT TABLE function. The TNC also creates a backup file with the extension **.t.bak**. To avoid losing data, be sure to make a backup copy of your original tool table before importing it!

The procedure for copying tool tables using the TNC file manager is described in the section on file management (see "Copying a table" on page 103).

When importing a tool table, the TNC 128 saves all tool types that are not available (**TYPE** column) as milling tools (type: **MILL**).

When tool tables of the iTNC 530 are imported, the TYPE column is not imported.



## Calling tool data

A TOOL CALL block in the part program is defined with the following data:

- Select the tool call function with the TOOL CALL key



- **Tool number:** Enter the number or name of the tool. The tool must already be defined in a **TOOL DEF** block or in the tool table. Press the TOOL NAME soft key to enter the name. The TNC automatically places the tool name in quotation marks. The tool name always refers to the entry in the active tool table TOOL.T. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point. There is a SELECT soft key for calling a window from which you can select a tool defined in the tool table TOOL.T directly without having to enter the number or name.
- **Working spindle axis X/Y/Z:** Enter the tool axis
- **Spindle speed S:** Enter the spindle speed in rpm. Alternatively, you can define the cutting speed Vc in m/min. Press the VC soft key
- **Feed rate F:** F [mm/min or 0.1 inch/min] is effective until you program a new feed rate in a positioning or **TOOL CALL** 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 the tool radius 2

### Example: 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 is to be programmed with an oversize of 0.2 mm, the tool radius 2 with an oversize of 0.05 mm, and the tool radius with an undersize of 1 mm.

```
20 TOOL CALL 5.2 Z S2500 F350 DL+0.2 DR-1 DR2+0.05
```

The character **D** preceding **L** and **R** designates a delta value.

### Tool preselection with tool tables

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



## Tool change



The tool change function can vary depending on the individual machine tool. The machine manual provides further information.

### Tool change position

The tool change position must be approachable without collision. Use the miscellaneous functions **M91** and **M92** to enter machine-based (rather than workpiece-based) coordinates for the tool change position. If **TOOL CALL 0** is programmed before the first tool call, the TNC moves the tool spindle in the tool axis to a position that is independent of the tool length.

### Manual tool change

To change the tool manually, stop the spindle and move the tool to the tool change position:

- ▶ Move to the tool change position under program control
- ▶ Interrupt program run (see "Interrupting machining", page 332)
- ▶ Change the tool
- ▶ Resume program run (see "Resuming program run after an interruption", page 334)

## Tool usage test



The tool usage test function must be enabled by your machine manufacturer. Refer to your machine manual.

In order to run a tool usage test, the complete plain-language program must have been simulated in the **Test Run** mode.

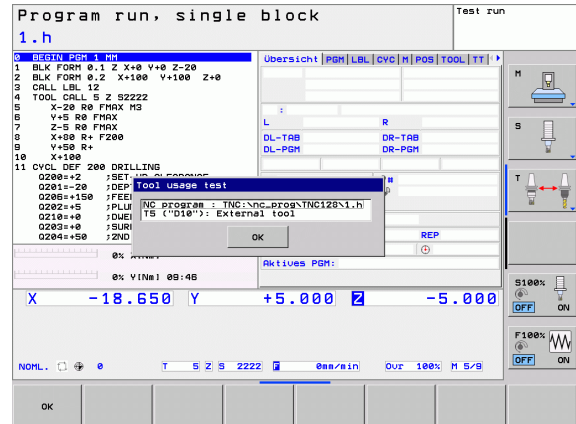
### Applying the tool usage test

Before starting a program in the Program Run mode of operation, you can use the **TOOL USAGE** and **TOOL USAGE TEST** soft keys to check whether the tools being used in the selected program are available and have sufficient remaining service life. The TNC then compares the actual service-life values in the tool table with the nominal values from the tool usage file.

After you have pressed the **TOOL USAGE TEST** soft key, the TNC displays the result of the tool usage test in a pop-up window. To close the pop-up window, press the **ENT** key.

The TNC saves the tool usage times in a separate file with the extension **pgmname.H.T.DEP**. The generated tool usage file contains the following information:

Column	Meaning
<b>TOKEN</b>	<ul style="list-style-type: none"> <li>■ <b>TOOL</b>: Tool usage time per <b>TOOL CALL</b>. The entries are listed in chronological order.</li> <li>■ <b>TTOTAL</b>: Total usage time of a tool</li> <li>■ <b>STOTAL</b>: Call of a subprogram; the entries are listed in chronological order</li> <li>■ <b>TIMETOTAL</b>: The total machining time of the NC program is entered in the <b>WTIME</b> column. In the <b>PATH</b> column the TNC saves the path name of the corresponding NC programs. The <b>TIME</b> column shows the sum of all <b>TIME</b> entries (without rapid traverse). The TNC sets all other columns to 0</li> <li>■ <b>TOOLFILE</b>: In the <b>PATH</b> column, the TNC saves the path name of the tool table with which you conducted the Test Run. This enables the TNC during the actual tool usage test to detect whether you performed the test run with the <b>TOOL.T</b></li> </ul>
<b>TNR</b>	Tool number (–1: No tool inserted yet)
<b>IDX</b>	Tool index
<b>NAME</b>	Tool name from the tool table
<b>TIME</b>	Tool-usage time in seconds (feed time)
<b>WTIME</b>	Tool-usage time in seconds (total usage time between tool changes)



Column	Meaning
<b>RAD</b>	<b>Tool radius R + Oversize of tool radius DR</b> from the tool table. (in mm)
<b>BLOCK</b>	Block number in which the <b>TOOL CALL</b> block was programmed
<b>PATH</b>	<ul style="list-style-type: none"> <li>■ <b>TOKEN = TOOL</b>: Path name of the active main program or subprogram</li> <li>■ <b>TOKEN = STOTAL</b>: Path name of the subprogram</li> </ul>
<b>T</b>	Tool number with tool index
<b>OVRMAX</b>	Maximum feed rate override that occurred during machining. During Test Run the TNC enters the value 100 (%)
<b>OVRMIN</b>	Minimum feed rate override that occurred during machining. During Test Run the TNC enters the value -1
<b>NAMEPROG</b>	<ul style="list-style-type: none"> <li>■ <b>0</b>: The tool number is programmed</li> <li>■ <b>1</b>: The tool name is programmed</li> </ul>

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

- The highlight is on a pallet entry in the pallet file:  
The TNC runs the tool usage test for the entire pallet
- The highlight is on a program entry in the pallet file:  
The TNC runs the tool usage test for the selected program

## 5.3 Tool compensation

### Introduction

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

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

### Tool length compensation

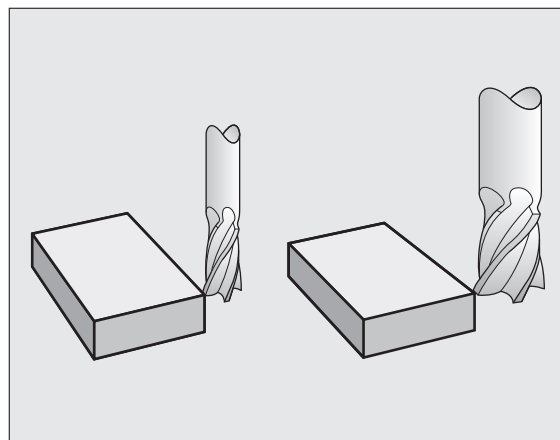
Length compensation becomes effective automatically as soon as a tool is called. To cancel length compensation, call a tool with the length  $L=0$ .



#### Caution: Danger of collision!

If you cancel a positive length compensation with **TOOL CALL 0**, the distance between tool and workpiece will be reduced.

After **TOOL CALL**, the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.



For tool length compensation, the TNC takes the delta values from both the **TOOL CALL** block and the tool table into account:

Compensation value =  $L + DL_{\text{TOOL CALL}} + DL_{\text{TAB}}$  where

**L:** Tool length **L** from the **TOOL DEF** block or tool table  
**DL<sub>TOOL CALL</sub>:** Oversize for length **DL** in the **TOOL CALL 0** block  
**DL<sub>TAB</sub>:** Oversize for length **DL** in the tool table

## Tool radius compensation for paraxial positioning blocks

The TNC can compensate the tool radius in the working plane with the aid of paraxial positioning blocks. You can enter the dimensions directly from the drawing without first having to convert the positions. The TNC extends or shortens the traverse path by the tool radius.

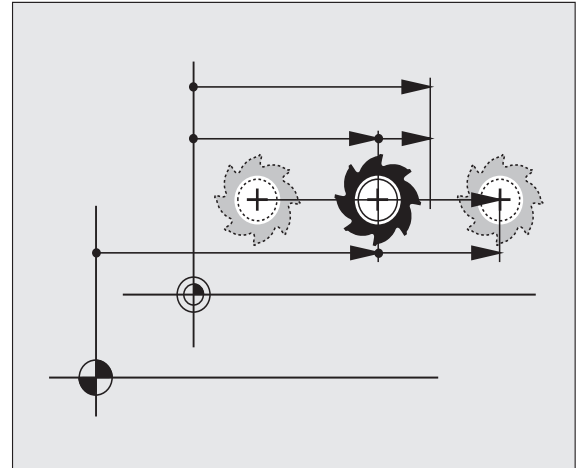
- **R+** extends the tool path by the tool radius
- **R-** shortens the tool path by the tool radius
- **R0** positions the tool using the tool center

Radius compensation becomes effective as soon as a tool is called and is moved with a straight line block in the working plane.



Radius compensation is not effective for positioning movements in the spindle axis.

The last selected radius compensation remains active in a positioning block that does not contain any information about radius compensation.



For radius compensation, the TNC takes the delta values from both the **TOOL CALL** block and the tool table into account:

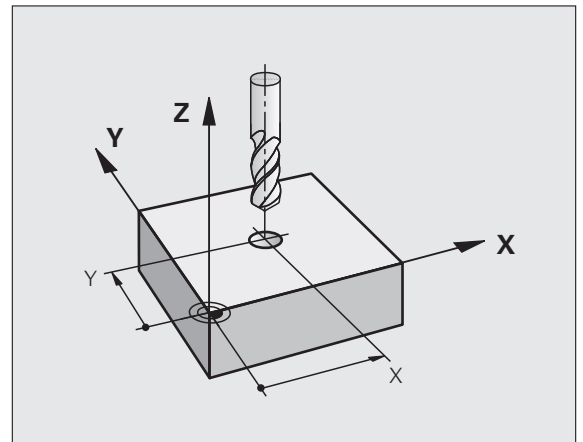
Compensation value = **R** + **DR**<sub>TOOL CALL</sub> + **DR**<sub>TAB</sub> where

- R**: Tool radius **R** from the **TOOL DEF** block or tool table
- DR**<sub>TOOL CALL</sub>: Oversize for radius **DR** in the **TOOL CALL** block
- DR**<sub>TAB</sub>: Oversize for radius **DR** in the tool table

### Contouring without radius compensation: R0

The tool center moves in the working plane along the programmed path or to the programmed coordinates.

Applications: Drilling and boring, pre-positioning





## Entering radius compensation

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

**TOOL RADIUS COMP: R+/R-/NO COMP?**

R +

The TNC extends the traverse path of the tool by the tool radius

R -

The TNC shortens the traverse path of the tool by the tool radius

ENT

Select tool movement without radius compensation or cancel radius compensation: Press the ENT key

END



Terminate the block: Press the END key



# 6

**Programming:  
Tool movements**



## 6.1 Fundamentals

### Tool movements in the program

The orange axis keys initiate the plain language dialog for a paraxial positioning block. The TNC asks you successively for all the necessary information and inserts the program block into the part program.



- ▶ **Coordinates** of the end point of the straight line, if necessary
- ▶ **Radius compensation** RL/RR/RO
- ▶ **Feed rate** F
- ▶ **Miscellaneous function** M

#### Example NC block

```
6 X+45 R+ F200 M3
```

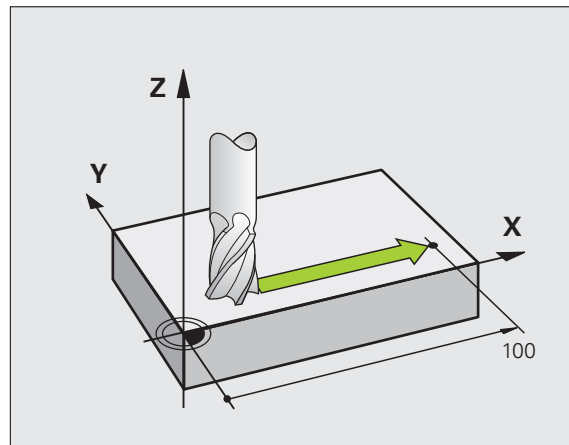
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool were moving and the workpiece remaining stationary.

The TNC moves the tool in a straight line from its current position to the straight-line end point. If you program several positioning blocks in succession in a program, the starting point of a block is the end point of the preceding block.



#### Danger of collision!

Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.



### Radius compensation

The TNC can compensate the tool radius automatically. In paraxial positioning blocks you can select whether the TNC is to extend (R+) or shorten (R-) the traverse path by the value of the tool radius. See "Tool radius compensation for paraxial positioning blocks" on page 156.

### Miscellaneous functions M

With the TNC'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 program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a separate program for execution.

Programming with subprograms and program section repeats is described in Chapter 7.

## Programming with Q parameters

Instead of programming numerical values in a part program, you enter markers called Q parameters. You assign the values to the Q parameters separately with the Q parameter functions. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

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

Programming with Q parameters is described in Chapter 8.



## 6.2 Programming tool movements

### Programming tool movements for workpiece machining

#### Creating the program blocks with the axis keys

Use the orange axis keys to initiate the plain-language dialog. The TNC asks you successively for all the necessary information and inserts the program block into the part program.

Example—programming a straight line:

**X** Select the axis key you want to use for the positioning movement, e.g. X

**COORDINATE?**

**10** **ENT** Enter the coordinate of the straight-line end point, e.g. 10, and confirm with the ENT key

**TOOL RADIUS COMP: R+/R-/NO COMP?**

**R0** Select the radius compensation (here, press the R0 soft key—the tool moves without compensation)

**FEED RATE F=? / F MAX = ENT**

**100** **ENT** Enter the feed rate (here, 100 mm/min), and confirm your entry with ENT. For programming in inches, enter 100 for a feed rate of 10 ipm

**F MAX** Move at rapid traverse: Press the F MAX soft key, or

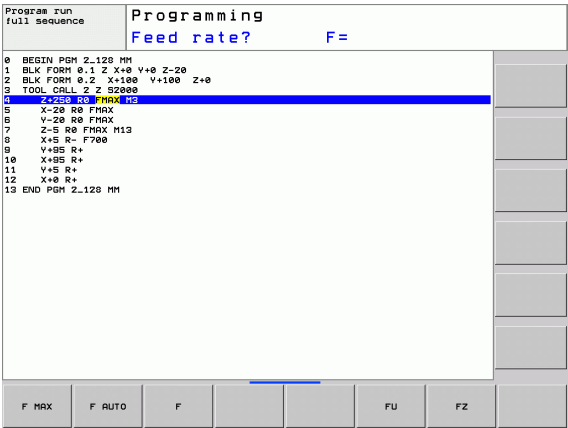
**F AUTO** Traverse with the feed rate defined in the **TOOL CALL** block: Press the F AUTO soft key

**MISCELLANEOUS FUNCTION M?**

**3** **ENT** Enter a miscellaneous function (here, M3), and terminate the dialog with ENT

The part program now contains the following line:

**X+10 R0 FMAX M3**



**Actual position capture**

You can also generate a straight-line block by using the ACTUAL-POSITION-CAPTURE key:

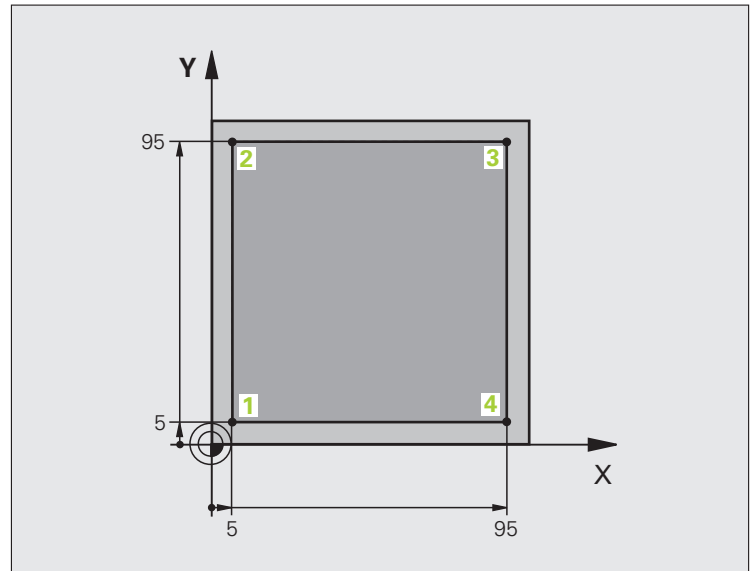
- ▶ In the Manual Operation mode, move the tool to the position you want to capture
- ▶ Switch the screen display to Programming and Editing
- ▶ Select the program block after which you want to insert the positioning block



- ▶ Press the ACTUAL-POSITION-CAPTURE key and select the axis soft key: The TNC generates a positioning block with the actual position coordinate



## Example: Linear movement



0 BEGIN PGM LINEAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4000	Call the tool in the spindle axis and with the spindle speed S
4 Z+250 R0 FMAX	Retract the tool in the spindle axis at rapid traverse FMAX
5 X-10 R0 FMAX	Pre-position the tool
6 Y-10 R0 FMAX	Pre-position the tool
7 Z+2 R0 FMAX	Pre-position the tool
8 Z-5 R0 F1000 M13	Move to working depth at feed rate F = 1000 mm/min
9 X+5 R- F500	Contour approach
10 Y+95 R+	Move to point 2
11 X+95 R+	Move to point 3
12 Y+5 R+	Move to point 4
13 X-10 R0	Move to last contour point 1 and retract
14 Z+250 R0 FMAX M30	Retract the tool, end program
15 END PGM LINEAR MM	



# 7

**Programming:  
Subprograms and  
program section repeats**



## 7.1 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.

### Labels

The beginnings of subprograms and program section repeats are marked in a part program by labels **LBL**.

A LABEL is identified by a number between 1 and 999 or by a name you define. Each LABEL number or LABEL name can be set only once in the program with LABEL SET. The number of label names you can enter is only limited by the internal memory.



Do not use a label number or label name more than once!

Label 0 (**LBL 0**) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

## 7.2 Subprograms

### Operating sequence

- 1 The TNC executes the part program up to the block in which a subprogram is called with **CALL LBL**.
- 2 The subprogram is then executed from beginning to end. The subprogram end is marked **LBL 0**.
- 3 The TNC then resumes the part program from the block after the subprogram call **CALL LBL**

### Programming notes

- A main program can contain up to 254 subprograms
- You can call subprograms in any sequence and as often as desired
- A subprogram cannot call itself
- Write subprograms at the end of the main program (behind the block with M2 or M30)
- If subprograms are located before the block with M2 or M30, they will be executed at least once even if they are not called

### Programming a subprogram



- ▶ 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
- ▶ To mark the end, press the LBL SET key and enter the label number "0"

### Calling a subprogram



- ▶ To call a subprogram, press the LBL CALL key
- ▶ **Label number:** Enter the label 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
- ▶ **Repeat REP:** Ignore the dialog question with the NO ENT key. Repeat REP is used only for program section repeats



**CALL LBL 0** is not permitted (Label 0 is only used to mark the end of a subprogram).

## 7.3 Program section repeats

### Label LBL

The beginning of a program section repeat is marked by the label **LBL**. The end of a program section repeat is identified by **CALL LBL n REPn**.

### Operating sequence

- 1 The TNC executes the part program up to the end of the program section (**CALL LBL n REPn**)
- 2 Then the program section between the called LBL and the label call **CALL LBL n REPn** is repeated the number of times entered after **REP**
- 3 The TNC then resumes the part 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

### Programming a program section repeat

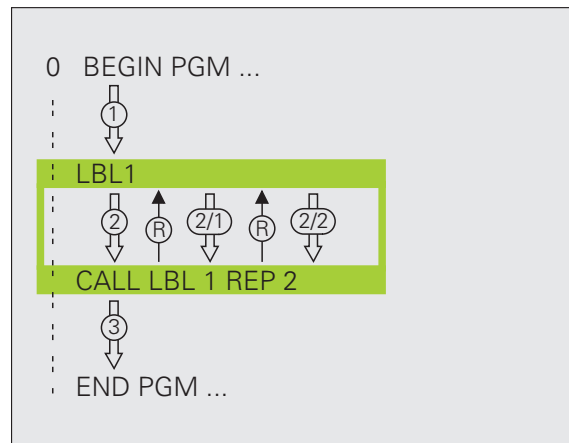


- ▶ To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the LBL NAME soft key to switch to text entry
- ▶ Enter the program section

### Calling a program section repeat



- ▶ Press the LBL CALL key
- ▶ **To call subprograms/section repeats:** Enter the label number of the subprogram to be called, then confirm with the ENT key. If you want to use a label name, press the " key to switch to text entry
- ▶ **Repeat REP:** Enter the number of repeats, then confirm with the ENT key



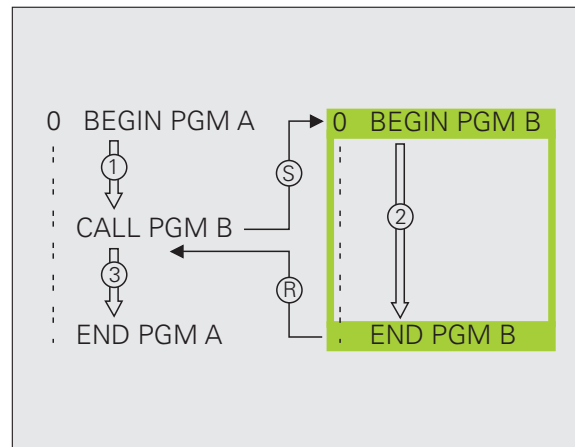
## 7.4 Separate program as subprogram

### Operating sequence

- 1 The TNC executes the part program up to the block in which another program is called with **CALL PGM**
- 2 Then the other program is run from beginning to end
- 3 The TNC then resumes the first (calling) part program with the block after the program call

### Programming notes

- No labels are needed to call any program as a subprogram
- The called program must not contain the miscellaneous functions M2 or M30. If you have defined subprograms with labels in the called program, you can then use M2 or M30 with the **FN 9: IF +0 EQU +0 GOTO LBL 99** jump function to force a jump over this program section.
- The called program must not contain a **CALL PGM** call into the calling program, otherwise an infinite loop will result.



## Calling any program as a subprogram

PGM  
CALL

PROGRAM

SELECT  
PROGRAM

- ▶ To select the functions for program call, press the PGM CALL key
- ▶ Press the PROGRAM soft key for the TNC to start the dialog for defining the program to be called. Use the screen keyboard to enter the path name (GOTO key), or
- ▶ press the SELECT PROGRAM soft key for the TNC to display a selection window in which you can select the program to be called. Confirm with the END key



If the program you want to call is located in the same directory as the program you are calling it from, then you only need to enter the program name.

If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path, e.g. **TNC:\ZW35\SCHRUPP\PGM1.H**

If you want to call a DIN/ISO program, enter the file type .I after the program name.

You can also call a program with Cycle **12 PGM CALL**.

As a rule, Q parameters are effective globally with a **PGM CALL**. So please note that changes to Q parameters in the called program can also influence the calling program.

## 7.5 Nesting

### Types of nesting

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Subprograms repeated
- Program section repeats within a subprogram

### Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: 19
- Maximum nesting depth for main program calls: 19, where a **CYCL CALL** acts like a main program call
- You can nest program section repeats as often as desired



Subprogram within a subprogram

Example NC blocks

0 BEGIN PGM SUBPGMS MM	
...	
17 CALL LBL "SP1"	Call the subprogram marked with LBL SP1
...	
35 Z+100 R0 FMAX M2	Last program block of the main program (with M2)
36 LBL "SP1"	Beginning of subprogram SP1
...	
39 CALL LBL 2	Call the subprogram marked with LBL 2
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM SUBPGMS MM	

Program execution

- 1 Main program SUBPGMS is executed up to block 17
- 2 Subprogram SP1 is called, and executed up to block 39
- 3 Subprogram 2 is called, and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called
- 4 Subprogram 1 is executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program SUBPGMS
- 5 Main program SUBPGMS is executed from block 18 up to block 35. Return jump to block 1 and end of program





## Repeating program section repeats

### Example NC blocks

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2	The program section between LBL 2 and this block
...	(block 20) is repeated twice
35 CALL LBL 1 REP 1	The program section between LBL 1 and this block
...	(block 15) is repeated once
50 END PGM REPS MM	

### Program execution

- 1 Main program REPS is executed up to block 27
- 2 Program section between block 20 and block 27 is repeated twice
- 3 Main program REPS is executed from block 28 to block 35
- 4 Program section between block 15 and block 35 is repeated once (including the program section repeat between 20 and block 27)
- 5 Main program REPS is executed from block 36 to block 50 (end of program)



Repeating a subprogram

Example NC blocks

0 BEGIN PGM SUBPGREP MM	
...	
10 LBL 1	Beginning of program section repeat 1
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2	The program section between LBL 1 and this block
...	(block 10) is repeated twice
19 Z+100 R0 FMAX M2	Last block of the main program with M2
20 LBL 2	Beginning of subprogram
...	
28 LBL 0	End of subprogram
29 END PGM SUBPGREP MM	

Program execution

- 1 Main program SUBPGREP is executed up to block 11
- 2 Subprogram 2 is called and executed
- 3 Program section between block 10 and block 12 is repeated twice:  
Subprogram 2 is repeated twice
- 4 Main program SUBPGREP is executed from block 13 to block 19;  
end of program

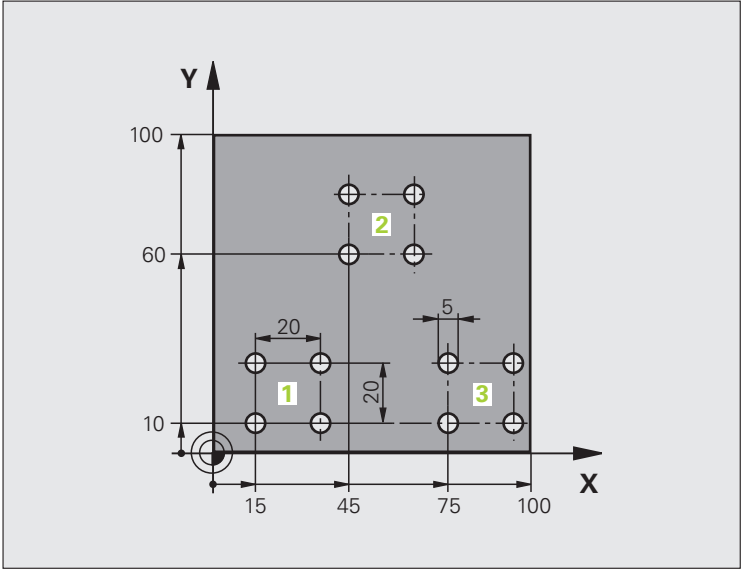


# 7.6 Programming examples

## Example: Groups of holes

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1



0 BEGIN PGM SP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Define the DRILLING cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	



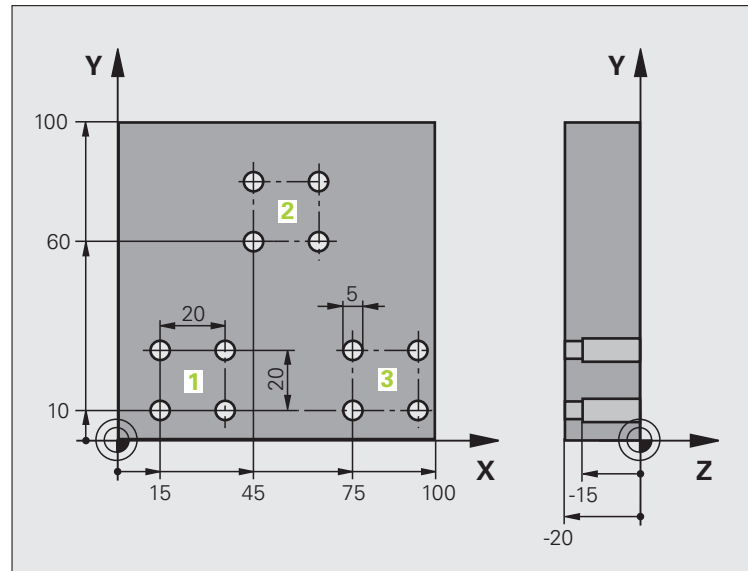
## 7.6 Programming examples

6	X+15 R0 FMAX M3	Move to starting point X for group 1
7	Y+10 R0 FMAX	Move to starting point Y for group 1
8	CALL LBL 1	Call the subprogram for the group
9	X+45 R0 FMAX	Move to starting point X for group 2
10	Y+60 R0 FMAX	Move to starting point Y for group 2
11	CALL LBL 1	Call the subprogram for the group
12	X+75 R0 FMAX	Move to starting point X for group 3
13	Y+10 R0 FMAX	Move to starting point Y for group 3
14	CALL LBL 1	Call the subprogram for the group
15	Z+250 R0 FMAX M30	End of main program
16	LBL 1	Beginning of subprogram 1: Group of holes
17	CYCL CALL	Hole 1
18	IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
19	IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
20	IX-20 R0 FMAX M99	Move to 4th hole, call cycle
21	LBL 0	End of subprogram 1
22	END PGM SP1 MM	

## Example: Group of holes with several tools

Program sequence

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2



0 BEGIN PGM SP2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Call tool: center drill
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Define the CENTERING cycle
Q200=2 ;SET-UP CLEARANCE	
Q202=-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=0.25 ;DWELL TIME AT DEPTH	
6 CALL LBL 1	Call subprogram 1 for the entire hole pattern

## 7.6 Programming examples

7	Z+250 R0 FMAX M6	Tool change
8	T00L CALL 2 Z S4000	Call tool: drill
9	FN 0: Q201 = -25	New depth for drilling
10	FN 0: Q202 = +5	New plunging depth for drilling
11	CALL LBL 1	Call subprogram 1 for the entire hole pattern
12	Z+250 R0 FMAX M6	Tool change
13	T00L CALL 3 Z S500	Call tool: reamer
14	CYCL DEF 201 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	
15	CALL LBL 1	Call subprogram 1 for the entire hole pattern
16	Z+250 R0 FMAX M2	End of main program
17	LBL 1	Beginning of subprogram 1: Entire hole pattern
18	X+15 R0 FMAX M3	Move to starting point X for group 1
19	Y+10 R0 FMAX	Move to starting point Y for group 1
20	CALL LBL 2	Call subprogram 2 for the group
21	X+45 R0 FMAX	Move to starting point X for group 2
22	Y+60 R0 FMAX	Move to starting point Y for group 2
23	CALL LBL 2	Call subprogram 2 for the group
24	X+75 R0 FMAX	Move to starting point X for group 3
25	Y+10 R0 FMAX	Move to starting point Y for group 3
26	CALL LBL 2	Call subprogram 2 for the group
27	LBL 0	End of subprogram 1
28	LBL 2	Beginning of subprogram 2: Group of holes
29	CYCL CALL	1st hole with active fixed cycle
30	IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
31	IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
32	IX-20 R0 FMAX M99	Move to 4th hole, call cycle
33	LBL 0	End of subprogram 2
34	END PGM SP2 MM	



# 8

**Programming:  
Q parameters**



## 8.1 Principle and overview

You can program entire families of parts in a single part program. You do this by entering variables called Q parameters instead of fixed numerical values.

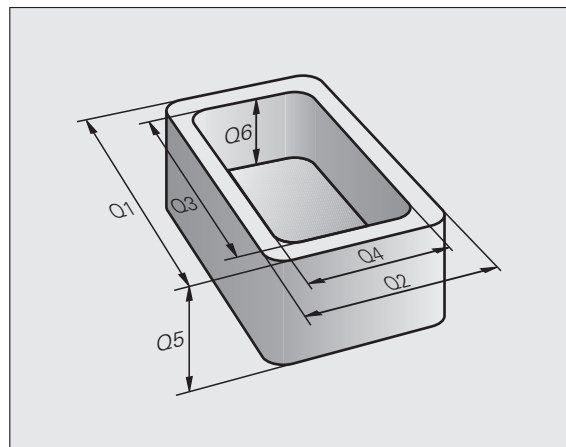
Q parameters can represent information such as:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

Q parameters also enable you to program contours that are defined with mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions. In conjunction with FK programming you can also combine contours that do not have NC-compatible dimensions with Q parameters.

Q parameters are designated by letters and a number between 0 and 1999. Parameters that take effect in different manners are available. Please refer to the following table:

Meaning	Range
Freely applicable parameters, as long as no overlapping with SL cycles can occur. They are globally effective for all programs stored in the TNC memory.	<b>Q0 to Q99</b>
Parameters for special TNC functions	<b>Q100 to Q199</b>
Parameters that are primarily used for cycles, globally effective for all programs stored in the TNC memory	<b>Q200 to Q1199</b>
Parameters that are primarily used for OEM cycles, and are globally effective for all programs stored in the TNC memory. This may require coordination with the machine manufacturer or supplier	<b>Q1200 to Q1399</b>
Parameters that are primarily used for <b>call-active</b> OEM cycles, globally effective for all programs that are stored in the TNC memory	<b>Q1400 to Q1499</b>
Parameters that are primarily used for <b>Def-active</b> OEM cycles, globally effective for all programs that are stored in the TNC memory	<b>Q1500 to Q1599</b>
Freely applicable parameters, globally effective for all programs stored in the TNC memory	<b>Q1600 to Q1999</b>





Meaning	Range
Freely usable <b>QL</b> parameters, only effective locally (within a program)	<b>QL0</b> to <b>QL499</b>
Freely usable <b>QR</b> parameters that are nonvolatile, i.e. they remain in effect even after a power interruption	<b>QR0</b> to <b>QR499</b>

**QS** parameters (the **S** stands for string) are also available on the TNC and enable you to process texts. In principle, the same ranges are available for **QS** parameters as for **Q** parameters (see table above).



Note that for the **QS** parameters the **QS100** to **QS199** range is reserved for internal texts.

Local parameters **QL** are only effective within the respective program, and are not applied as part of program calls or macros.

### Programming notes

You can mix **Q** parameters and fixed numerical values within a program.

**Q** parameters can be assigned numerical values between -999 999 999 and +999 999 999. The input range is limited to 15 digits, of which up to 9 may be before the decimal point. Internally the TNC calculates numbers up to a value of 10<sup>10</sup>.

You can assign a maximum of 254 characters to **QS** parameters.



Some **Q** and **QS** parameters are always assigned the same data by the TNC. For example, **Q108** is always assigned the current tool radius (see "Preassigned **Q** parameters", page 242).

The TNC saves numerical values internally in a binary number format (standard IEEE 754). Due to this standardized format some decimal numbers do not have an exact binary representation (round-off error). Keep this in mind especially when you use calculated **Q**-parameter contents for jump commands or positioning movements.



# Calling Q-parameter functions

When you are writing a part program, press the "Q" key (in the numeric keypad for numerical input and axis selection, below the +/- key). The TNC then displays the following soft keys:

Function group	Soft key	Page
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.	Page 184
Trigonometric functions	TRIGO- NOMETRY	Page 186
Function for calculating circles	CIRCLE CALCU- LATION	Page 188
If/then conditions, jumps	JUMP	Page 189
Other functions	DIVERSE FUNCTION	Page 192
Entering formulas directly	FORMULA	Page 226
Function for machining complex contours	CONTOUR FORMULA	See User's Manual for Cycles



The TNC shows the soft keys Q, QL and QR when you are defining or assigning a Q parameter. First press one of these soft keys to select the desired type of parameter, and then enter the parameter number.

If you have a USB keyboard connected, you can press the Q key to open the dialog for entering a formula.



# 8.2 Part families—Q parameters in place of numerical values

## Application

The Q parameter function **FN 0: ASSIGN** assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

### Example NC blocks

15 FN 0: Q10=25	Assign
...	Q10 is assigned the value 25
25 X +Q10	Means 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

$$R = Q1$$

Cylinder height

$$H = Q2$$

Cylinder Z1

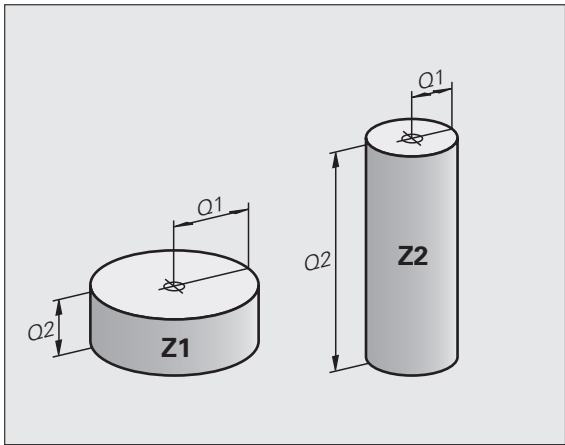
$$Q1 = +30$$

$$Q2 = +10$$

Cylinder Z2

$$Q1 = +10$$

$$Q2 = +50$$



# 8.3 Describing contours through mathematical operations

## Application

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- ▶ Select a Q-parameter function: Press the Q key (in the numerical keypad at right). The Q-parameter functions are displayed in a soft-key row
- ▶ Select the mathematical functions: Press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

## Overview

Function	Soft key
<b>FN 0: ASSIGN</b> Example: <b>FN 0: Q5 = +60</b> Assigns a numerical value	<div>FN0 X = Y</div>
<b>FN 1: ADDITION</b> Example: <b>FN 1: Q1 = -Q2 + -5</b> Calculates and assigns the sum of two values	<div>FN1 X + Y</div>
<b>FN 2: SUBTRACTION</b> Example <b>FN 2: Q1 = +10 - +5</b> Calculates and assigns the difference of two values	<div>FN2 X - Y</div>
<b>FN 3: MULTIPLICATION</b> Example: <b>FN 3: Q2 = +3 * +3</b> Calculates and assigns the product of two values	<div>FN3 X * Y</div>
<b>FN 4: DIVISION</b> Example: <b>FN 4: Q4 = +8 DIV +Q2</b> Calculates and assigns the quotient of two values <b>Not permitted:</b> Division by 0!	<div>FN4 X / Y</div>
<b>FN 5: SQUARE ROOT</b> Example: <b>FN 5: Q20 = SQRT 4</b> Calculates and assigns the square root of a number <b>Not permitted:</b> Calculating the square root of a negative value!	<div>FN5 SQRT</div>

To the right of the "=" character you can enter the following:

- 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:

Q

Call the Q parameter functions by pressing the Q key

BASIC  
ARITHM.

Select the mathematical functions: Press the BASIC ARITHMETIC soft key

FN0  
X = Y

Select the Q parameter function ASSIGN: Press the FN0 X = Y soft key

PARAMETER NO. FOR RESULT?

5

ENT

Enter the number of the Q parameter, e.g. 5

1ST VALUE OR PARAMETER?

10

ENT

Assign the value 10 to Q5

Q

Select the Q parameter functions: Press the Q key

BASIC  
ARITHM.

Select the mathematical functions: Press the BASIC ARITHMETIC soft key

FN3  
X \* Y

Select the Q parameter function MULTIPLICATION: Press the FN3 X \* Y soft key

PARAMETER NO. FOR RESULT?

12

ENT

Enter the number of the Q parameter, e.g. 12

1ST VALUE OR PARAMETER?

Q5

ENT

Enter Q5 for the first value

2ND VALUE OR PARAMETER?

7

ENT

Enter 7 for the second value

Example: Program blocks in the TNC

16 FN 0: Q5 = +10

17 FN 3: Q12 = +Q5 \* +7



## 8.4 Trigonometric functions

### Definitions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. In this case:

**Sine:**  $\sin \alpha = a / c$

**Cosine:**  $\cos \alpha = b / c$

**Tangent:**  $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

■ c is the side opposite the right angle

■ a is the side opposite the angle  $\alpha$

■ b is the third side.

The TNC can find the angle from the tangent:

$$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$$

#### Example:

$$a = 25 \text{ mm}$$

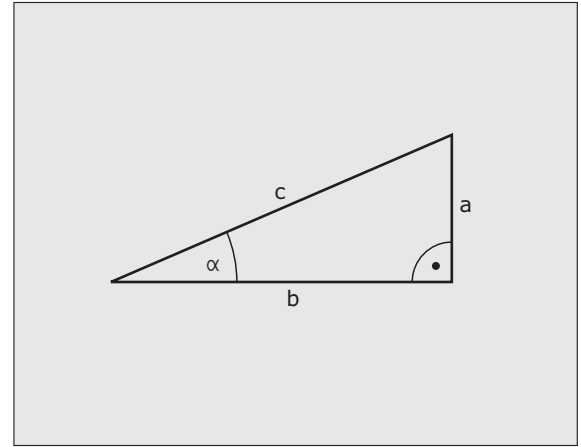
$$b = 50 \text{ mm}$$

$$\alpha = \arctan (a / b) = \arctan 0.5 = 26.57^\circ$$

Furthermore:

$$a^2 + b^2 = c^2 \text{ (where } a^2 = a \times a \text{)}$$

$$c = \sqrt{(a^2 + b^2)}$$



# Programming trigonometric functions

Press the ANGLE FUNCTION soft key to call the trigonometric functions. The TNC then displays the following soft keys:

Programming: Compare "Example: Programming fundamental operations."

Function	Soft key
<b>FN 6: SINE</b> Example: <b>FN 6: Q20 = SIN-Q5</b> Calculates and assigns the sine of an angle in degrees (°)	FNB SIN(X)
<b>FN 7: COSINE</b> Example: <b>FN 7: Q21 = COS-Q5</b> Calculates and assigns the cosine of an angle in degrees (°)	D7 COS(X)
<b>FN 8: ROOT SUM OF SQUARES</b> Example: <b>FN 8: Q10 = +5 LEN +4</b> Calculates and assigns length from two values	FNB X LEN Y
<b>FN 13: ANGLE</b> Example: <b>FN 13: Q20 = +25 ANG-Q1</b> Calculates the angle from the arc tangent of two sides or from the sine and cosine of the angle (0° < angle < 360°) and assigns it to a parameter	FN13 X ANG Y



## 8.5 Circle calculations

### Application

The TNC 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 if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Function	Soft key
FN 23: Determine the CIRCLE DATA from three points Example: <b>FN 23: Q20 = CDATA Q30</b>	<div>FN23 3 POINTS OF CIRCLE</div>

The coordinate pairs of three points on a circle must be saved in Q30 and the following five parameters—in this case, up to Q35.

The TNC then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.

Function	Soft key
FN 24: Determine the CIRCLE DATA from four points Example: <b>FN 24: Q20 = CDATA Q30</b>	<div>FN24 4 POINTS OF CIRCLE</div>

The coordinate pairs of four points on a circle must be saved in Q30 and the following seven parameters—in this case, up to Q37.

The TNC then saves the circle center in the reference axis (X if spindle axis is Z) in parameter Q20, the circle center in the minor axis (Y if spindle axis is Z) in parameter Q21, and the circle radius in parameter Q22.



Note that **FN 23** and **FN 24** automatically overwrite the resulting parameter and the two following parameters.





## 8.6 If-then decisions with Q parameters

### Application

The TNC can make logical if-then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see "Labeling subprograms and program section repeats", page 166). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter a **PGM CALL** program call after the block with the target label.

### Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

**FN 9: IF+10 EQU+10 GOTO LBL1**



Programming if-then decisions

Press the JUMP soft key to call the if-then conditions. The TNC then displays the following soft keys:

Function	Soft key
<b>FN 9:</b> IF EQUAL, JUMP Example <b>FN 9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"</b> If the two values or parameters are equal, jump to the given label	<div>FN9 IF X EQ V GOTO</div>
<b>FN 10:</b> IF UNEQUAL, JUMP Example: <b>FN 10: IF +10 NE -Q5 GOTO LBL 10</b> If the two values or parameters are unequal, jump to the given label.	<div>FN10 IF X NE V GOTO</div>
<b>FN 11:</b> IF GREATER THAN, JUMP Example: <b>FN 11: IF+Q1 GT+10 GOTO LBL 5</b> If the first value or parameter is greater than the second, jump to the given label.	<div>FN11 IF X GT V GOTO</div>
<b>FN 12:</b> IF LESS THAN, JUMP Example: <b>FN 12: IF+Q5 LT+0 GOTO LBL "ANYNAME"</b> If the first value or parameter is less than the second, jump to the given label	<div>FN12 IF X LT V GOTO</div>

Abbreviations used:


IF	:	If
EQU	:	Equal
NE	:	Not equal
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to



## 8.7 Checking and changing Q parameters

## Procedure

You can check Q parameters in all operating modes (when writing, testing and running programs) and also edit them.

- If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.
-  Call Q-parameter functions: Press the Q INFO soft key or the Q key.
- The TNC lists all parameters and their current values. Use the arrow keys or the GOTO key to select the desired parameter.
- If you would like to change the value, press the EDIT CURRENT FIELD soft key, enter the new value, and confirm with the ENT key.
- To leave the value unchanged, press the PRESENT VALUE soft key or end the dialog with the END key.



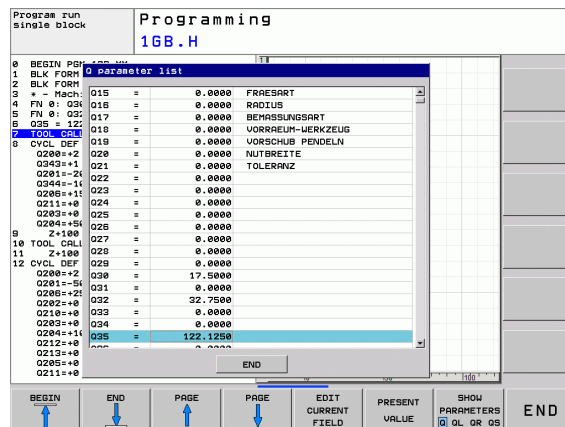
The parameters used by the TNC internally or in cycles are provided with comments.

If you want to check or edit local, global or string parameters, press the SHOW PARAMETERS Q QL QR QS soft key. The TNC then displays the specific parameter type. The functions previously described also apply.

PROGRAM  
+  
STATUSSTATUS OF  
Q PARAM.

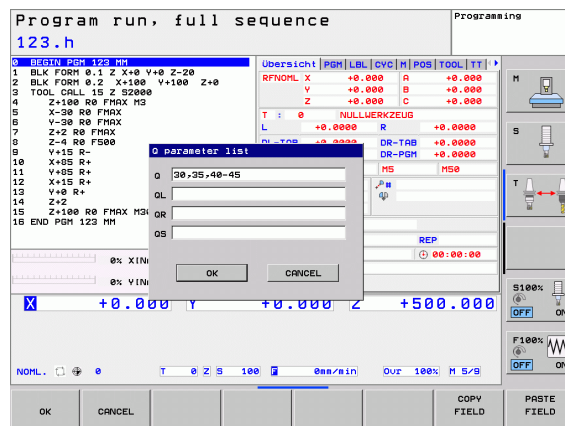
### Q PARAMETER LIST

- ▶ Call the soft-key row for screen layout
- ▶ Select the screen layout with additional status display:  
In the right half of the screen, the TNC shows the **Overview** status form
- ▶ Press the STATUS OF Q PARAM. soft key
- ▶ Press the Q PARAMETER LIST soft key
- ▶ The TNC opens a pop-up window in which you can enter the desired range for display of the Q parameters or string parameters. Multiple Q parameters are entered separated by commas (e.g. Q 1,2,3,4). To define display ranges, enter a hyphen (e.g. Q 10-14).



You can have the Q parameters be shown in the additional status display in the Manual, El. Handwheel, Single Block, Full Sequence and Test Run operating modes.








- If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.



## 8.8 Additional functions

### Overview

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key	Page
<b>FN 14:ERROR</b> Output of error messages		Page 193
<b>FN 16:F-PRINT</b> Formatted output of texts or Q-parameter values		Page 198
<b>FN 18:SYS-DATUM READ</b> Read system data		Page 202
<b>FN 19:PLC</b> Send values to the PLC		Page 211
<b>FN 20:WAIT FOR</b> Synchronize NC and PLC		Page 211
<b>FN 29:PLC</b> Transfer up to eight values to the PLC		Page 213
<b>FN 37:EXPORT</b> Export local Q parameters or QS parameters into a calling program		Page 214



## FN 14: ERROR: Display error messages

With the function **FN 14: ERROR** you can call messages under program control. The messages are predefined by the machine tool builder or by HEIDENHAIN. Whenever the TNC comes to a block with **FN 14** in the Program Run or Test Run mode, it interrupts the program run and displays a message. The program must then be restarted. The error numbers are listed in the table below.

Range of error numbers	Standard dialog text
0 ... 999	Machine-dependent dialog
1000 ... 1199	Internal error messages (see table at right)

### Example NC block

The TNC is to display the text stored under error number 254:

```
180 FN 14: ERROR = 254
```

### 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 < 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted

Error number	Text
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
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



Error number	Text
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D 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 allowed
1094	Tool name not allowed
1095	Software option not active
1096	Kinematics cannot be restored
1097	Function not permitted
1098	Contradictory workpc. blank dim.
1099	Measuring position not allowed





Error number	Text
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



FN 16: F-PRINT: Formatted output of text and Q-parameter values



With **FN 16** you can also output to the screen any messages from the NC program. Such messages are displayed by the TNC in a pop-up window.

The function **FN 16: F-PRINT** transfers Q parameter values and texts in a selectable format. If you send the values, the TNC saves the data in the file that you defined in the **FN 16** block.

To output the formatted texts and Q-parameter values, create a text file with the TNC's text editor. In this file you then define the output format and Q parameters you want to output.

Example of a text file to define the output format:

"MEASURING LOG OF IMPELLER CENTER OF GRAVITY";

"DATE: %2d-%2d-%4d",DAY,MONTH,YEAR4;

"TIME: %2d:%2d:%2d",HOUR,MIN,SEC;

"NO. OF MEASURED VALUES: = 1";

"X1 = %9.3LF", Q31;

"Y1 = %9.3LF", Q32;

"Z1 = %9.3LF", Q33;

When you create a text file, use the following formatting functions:

Special character	Function
" ....."	Define output format for texts and variables between the quotation marks
%9.3LF	Define format for Q parameter: 9 total characters (incl. decimal point), of which 3 are after the decimal, Long, Floating (decimal number)
%S	Format for text variable
%d	Format for integer
,	Separation character between output format and parameter
;	End of block character
\n	Line break



The following functions allow you to include the following additional information in the protocol log file:

Keyword	Function
CALL_PATH	Indicates the path for the NC program where you will find the FN16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN16. Example: M_CLOSE;
M_APPEND	Upon renewed output, appends the log to the existing log. Example: M_APPEND;
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_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_PORTUGUE	Outputs text only for Portuguese conversational language
L_HUNGARIA	Outputs text only for Hungarian conversational language
L_SLOVENIAN	Outputs text only for Slovenian conversational language
L_ALL	Outputs text independently of the conversational language



Keyword	Function
HOUR	Number of hours from the real-time clock
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

In the part program, program **FN 16: F-PRINT** to activate the output:

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/ TNC:\PROT1.TXT
```

The TNC then creates the file PROT1.TXT:

MEASURING LOG OF IMPELLER CENTER OF GRAVITY  
DATE: 27:11:2001  
TIME: 8:56:34  
NO. OF MEASURED VALUES : = 1  
X1 = 149.360  
Y1 = 25.509  
Z1 = 37.000



The **FN 16** function overwrites already existing log files with the same name. Use **M\_APPEND** if you want to append new log information to an existing log.

If you use **FN 16** more than once in the program, the TNC saves all texts in the file that you defined in the **FN 16** function. The file is not output until the TNC reads the **END PGM** block, or you press the NC stop button, or you close the file with **M\_CLOSE**.

In the **FN 16** block, program the format file and the log file with their respective extensions.

If you enter only the file name for the path of the log file, the TNC saves the log file in the directory in which the NC program with the **FN 16** function is located.

You can define a standard path for outputting protocol files via the user parameters **fn16DefaultPath** and **fn16DefaultPathSim** (Program Test).



### Displaying messages on the TNC screen

You can also use the function **FN 16** to display any messages from the NC program in a pop-up window on the TNC screen. This makes it easy to display explanatory texts, including long texts, at any point in the program in a way that the user has to react to them. You can also display Q-parameter contents if the protocol description file contains such instructions.

For the message to appear on the TNC screen, you need only enter **SCREEN:** as the name of the protocol file.

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/SCREEN:
```

If the message has more lines than fit in the pop-up window, you can use the arrow keys to page in the window.

To close the pop-up window, press the CE key. To have the program close the window, program the following NC block:

```
96 FN 16: F-PRINT TNC:\MASKE\MASKE1.A/SCLR:
```



All the previously described conventions apply to the protocol description file.

The **FN 16** function overwrites already existing log files with the same name. Use **M\_APPEND** if you want to append new log information to an existing log.

### Exporting messages

You can also use the **FN 16** function in the NC program in order to externally save the files generated with **FN 16**. Two possibilities are available for this:

Enter the complete target path in the **FN 16** function:

```
96 FN 16: F-PRINT TNC:\MSK\MSK1.A / PC325:\LOG\PR01.TXT
```



All the previously described conventions apply to the protocol description file.

The **FN 16** function overwrites already existing log files with the same name. Use **M\_APPEND** if you want to append new log information to an existing log.



## FN 18: SYS-DATUM READ

With the function **FN 18: SYS-DATUM READ** you can read system data and store it in Q parameters. You select the system data through a group name (ID number), and additionally through a number and an index.

Group name, ID no.	Number	Index	Meaning
Program information, 10	3	-	Number of the active fixed cycle
	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.
System jump addresses, 13	1	-	Label jumped to during M2/M30 instead of ending the current program. Value = 0: M2/M30 has the normal effect
	2	-	Label jumped to if FN14: ERROR after the NC CANCEL reaction instead of aborting the program with an error. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
	3	-	Label jumped to in the event of an internal server error (SQL, PLC, CFG) instead of aborting the program with an error message. Value = 0: Server error has the normal effect.
Machine status, 20	1	-	Active tool number
	2	-	Prepared tool number
	3	-	Active tool axis 0=X, 1=Y, 2=Z, 6=U, 7=V, 8=W
	4	-	Programmed spindle speed
	5	-	Active spindle status: -1=undefined, 0=M3 active, 1=M4 active, 2=M5 after M3, 3=M5 after M4
	7	-	Gear range
	8	-	Coolant status: 0=off, 1=on
	9	-	Active feed rate
	10	-	Index of prepared tool
	11	-	Index of active tool
Channel data, 25	1	-	Channel number
Cycle parameter, 30	1	-	Set-up clearance of active fixed cycle
	2	-	Drilling depth / milling depth of active fixed cycle
	3	-	Plunging depth of active fixed cycle

Group name, ID no.	Number	Index	Meaning
	4	-	Feed rate for pecking in active fixed cycle
	5	-	1st side length for rectangular pocket cycle
	6	-	2nd side length for rectangular pocket cycle
	7	-	1st side length for slot cycle
	8	-	2nd side length for slot cycle
	9	-	Radius for circular pocket cycle
	10	-	Feed rate for milling in active fixed cycle
	11	-	Direction of rotation for active fixed cycle
	12	-	Dwell time for active fixed cycle
	13	-	Thread pitch for Cycles 17, 18
	14	-	Finishing allowance for active fixed cycle
	15	-	Direction angle for rough out in active fixed cycle
	21	-	Probing angle
	22	-	Probing path
	23	-	Probing feed rate
Modal condition, 35	1	-	Dimensioning: 0 = Absolute (G90) 1 = Incremental (G91)
Data for SQL tables, 40	1	-	Result code for the last SQL command
Data from the tool table, 50	1	Tool no.	Tool length
	2	Tool no.	Tool radius
	3	Tool no.	Tool radius R2
	4	Tool no.	Tool length oversize DL
	5	Tool no.	Tool radius oversize DR
	6	Tool no.	Tool radius oversize DR2
	7	Tool no.	Tool locked (0 or 1)
	8	Tool no.	Number of the replacement tool
	9	Tool no.	Maximum tool age TIME1
	10	Tool no.	Maximum tool age TIME2
	11	Tool no.	Current tool age CUR. TIME



Group name, ID no.	Number	Index	Meaning
	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 teeth CUT
	16	Tool no.	TT: Wear tolerance for length LTOL
	17	Tool no.	TT: Wear tolerance for radius RTOL
	18	Tool no.	TT: Rotational direction DIRECT (0=positive/-1=negative)
	19	Tool no.	TT: Offset in plane R-OFFS
	20	Tool no.	TT: Offset in length L-OFFS
	21	Tool no.	TT: Break tolerance for length LBREAK
	22	Tool no.	TT: Break tolerance for radius RBREAK
	23	Tool no.	PLC value
	24	Tool no.	Probe-center offset in reference axis CAL-OF1
	25	Tool no.	Probe-center offset in minor axis CAL-OF2
	26	Tool no.	Spindle angle for calibration CAL-ANG
	27	Tool no.	Tool type for pocket table
	28	Tool no.	Maximum rpm NMAX
Pocket table data, 51	1	Pocket number	Tool number
	2	Pocket number	Special tool: 0=No, 1=Yes
	3	Pocket number	Fixed pocket: 0=No, 1=Yes
	4	Pocket number	Locked pocket: 0=No, 1=Yes
	5	Pocket number	PLC status
Pocket number of a tool in the tool-pocket table, 52	1	Tool no.	Pocket number
	2	Tool no.	Tool magazine number
Values programmed immediately after TOOL CALL, 60	1	-	Tool number T
	2	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W





Group name, ID no.	Number	Index	Meaning
	3	-	Spindle speed S
	4	-	Tool length oversize 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
Values programmed immediately after TOOL DEF, 61	1	-	Tool number T
	2	-	Length
	3	-	Radius
	4	-	Index
	5	-	Tool data programmed in TOOL DEF 1 = Yes, 0 = No
Active 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 and oversize from TOOL CALL	Active length
	3	1 = Without oversize 2 = With oversize 3 = With oversize and oversize from TOOL CALL	Rounding radius R2
Active transformations, 210	1	-	Basic rotation in MANUAL OPERATION mode
	2	-	Programmed rotation with Cycle 10
	3	-	Active mirrored axis
			0: Mirroring not active
			+1: X axis mirrored
			+2: Y axis mirrored
			+4: Z axis mirrored



Group name, ID no.	Number	Index	Meaning
			+64: U axis mirrored
			+128: V axis mirrored
			+256: W axis mirrored
			Combinations = Sum of individual axes
	4	1	Active scaling factor in X axis
	4	2	Active scaling factor in Y axis
	4	3	Active scaling factor in Z axis
	4	7	Active scaling factor in U axis
	4	8	Active scaling factor in V axis
	4	9	Active scaling factor in W axis
	5	1	3-D ROT A axis
	5	2	3-D ROT B axis
	5	3	3-D ROT C axis
	6	-	Tilted working plane active / inactive (-1/0) in a Program Run operating mode
	7	-	Tilted working plane active / inactive (-1/0) in a Manual operating mode
Active datum shift, 220	2	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Traverse range, 230	2	1 to 9	Negative software limit switch in axes 1 to 9
	3	1 to 9	Positive software limit switch in axes 1 to 9
	5	-	Software limit switch on or off: 0 = On, 1 = Off

Group name, ID no.	Number	Index	Meaning
Nominal position in the REF system, 240	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Current position in the active coordinate system, 270	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
TS triggering touch probe, 350	50	1	Touch probe type
		2	Line in the touch-probe table
	51	-	Effective length
	52	1	Effective ball radius
		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



Group name, ID no.	Number	Index	Meaning
	56	1	Maximum measuring range
		2	Safety clearance
	57	1	Spindle orientation possible: 0=No, 1=Yes
		2	Spindle-orientation angle
TT tool touch probe	70	1	Touch probe type
		2	Line in the touch-probe table
	71	1	Center point in reference axis (REF system)
		2	Center point in minor axis (REF system)
		3	Center point in tool axis (REF system)
	72	-	Probe contact radius
	75	1	Rapid traverse
		2	Measuring feed rate for stationary spindle
		3	Measuring feed rate for rotating spindle
	76	1	Maximum measuring range
		2	Safety clearance for linear measurement
		3	Safety clearance for radial measurement
	77	-	Spindle speed
	78	-	Probing direction
Reference point from touch probe cycle, 360	1	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length compensation but with probe radius compensation (workpiece coordinate system)
	2	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or probe radius compensation (machine coordinate system)
	3	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Result of measurement of the touch probe cycles 0 and 1 without probe radius or probe length compensation
	4	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length or stylus probe compensation (workpiece coordinate system)
	10	-	Oriented spindle stop



Group name, ID no.	Number	Index	Meaning
Value from the active datum table in the active coordinate system, 500	Line	Column	Read values
Basic transformation, 507	Line	1 to 6 (X, Y, Z, SPA, SPB, SPC)	Read the basic transformation of a preset
Axis offset, 508	Line	1 to 9 (X_OFFS, Y_OFFS, Z_OFFS, A_OFFS, B_OFFS, C_OFFS, U_OFFS, V_OFFS, W_OFFS)	Read the axis offset of a preset
Active preset, 530	1	-	Read the number of the active preset
Read data of the current tool, 950	1	-	Tool length L
	2	-	Tool radius R
	3	-	Tool radius R2
	4	-	Tool length oversize 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
	11	-	Current tool age CUR. TIME
	12	-	PLC status
	13	-	Maximum tooth length LCUTS
	14	-	Maximum plunge angle ANGLE
	15	-	TT: Number of 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
	20	-	TT: Offset in length L-OFFS



Group name, ID no.	Number	Index	Meaning
	21	-	TT: Break tolerance for length LBREAK
	22	-	TT: Break tolerance for radius RBREAK
	23	-	PLC value
	24	-	Tool type TYPE 0 = Milling cutter, 21 = Touch probe
	27	-	Corresponding line in the touch-probe table
	32	-	Point angle
	34	-	Lift off
Touch probe cycles, 990	1	-	Approach behavior: 0 = Standard behavior 1 = Effective radius, set-up clearance is zero
	2	-	0 = Probe monitoring off 1 = Probe monitoring on
	4	-	0 = Stylus not deflected 1 = Stylus deflected



Group name, ID no.	Number	Index	Meaning
Execution status, 992	10	-	Block scan active 1 = Yes, 0 = No
	11	-	Search phase
	14	-	Number of the last FN14 error
	16	-	Real execution active 1 = Execution , 2 = Simulation

**Example: Assign the value of the active scaling factor for the Z axis to Q25**

```
55 FN 18: SYSREAD Q25 = ID210 NR4 IDX3
```

## FN 19: PLC: Transfer values to the PLC

The function **FN 19: PLC** transfers up to two numerical values or Q parameters to the PLC.

Increments and units: 0.1  $\mu\text{m}$  or 0.0001°

**Example: Transfer the numerical value 10 (which means 1  $\mu\text{m}$  or 0.001°) to the PLC**

```
56 FN 19: PLC=+10/+Q3
```

## FN 20: WAIT FOR: NC and PLC synchronization



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

With the **FN 20: WAIT FOR** function you can synchronize the NC and PLC during a program run. The NC stops machining until the condition that you have programmed in the FN 20 block is fulfilled. The TNC can check the following PLC operands:

PLC operand	Abbreviation	Address range
Marker	<b>M</b>	0 to 4999
Input	<b>I</b>	0 to 31, 128 to 152 64 to 126 (first PL 401 B) 192 to 254 (second PL 401 B)
Output	<b>O</b>	0 to 30 32 to 62 (first PL 401 B) 64 to 94 (second PL 401 B)
Counter	<b>C</b>	48 to 79
Timer	<b>T</b>	0 to 95



PLC operand	Abbreviation	Address range
Byte	<b>B</b>	0 to 4095
Word	<b>W</b>	0 to 2047
Double word	<b>D</b>	2048 to 4095

The TNC 128 uses an extended interface for communication between the PLC and NC. This is a new, symbolic Application Programmer Interface (**API**). The familiar previous PLC-NC interface is also available and can be used if desired. The machine tool builder decides whether the new or old TNC API is used. Enter the name of the symbolic operand as string to wait for the defined condition of the symbolic operand.

The following conditions are permitted in the FN 20 block:

Condition	Abbreviation
Equal to	<b>==</b>
Less than	<b>&lt;</b>
Greater than	<b>&gt;</b>
Less than or equal	<b>&lt;=</b>
Greater than or equal	<b>&gt;=</b>

In addition, the **FN20: WAIT FOR SYNC** function is available. **WAIT FOR SYNC** is used whenever you read, for example, system data via **FN18** that require synchronization with real time. The TNC stops the look-ahead calculation and executes the subsequent NC block only when the NC program has actually reached that block.

**Example: Stop program run until the PLC sets marker 4095 to 1**

```
32 FN 20: WAIT FOR M4095==1
```

**Example: Stop program run until the PLC sets the symbolic operand to 1**

```
32 FN 20: APISPIN[0].NN_SPICONTROLINPOS==1
```

**Example: Pause internal look-ahead calculation, read current position in the X axis**

```
32 FN 20: WAIT FOR SYNC
33 FN 18: SYSREAD Q1 = ID270 NR1 IDX1
```





## FN 29: PLC: Transfer values to the PLC

The function FN 29: PLC transfers up to eight numerical values or Q parameters to the PLC.

Increments and units: 0.1  $\mu\text{m}$  or 0.0001°

**Example: Transfer the numerical value 10 (which means 1  $\mu\text{m}$  or 0.001°) to the PLC**

```
56 FN 29: PLC=+10/+Q3/+Q8/+7/+1/+Q5/+Q2/+15
```



## FN 37: EXPORT

You need the FN37: EXPORT function if you want to create your own cycles and integrate them in the TNC. The Q parameters 0 to 99 are effective only locally. This means that the Q parameters are effective only in the program in which they were defined. With the FN37: EXPORT function you can export locally effective Q parameters into another (calling) program.

**Example: The local Q parameter Q25 is exported**

```
56 FN37: EXPORT Q25
```

**Example: The local Q parameters Q25 to Q30 are exported**

```
56 FN37: EXPORT Q25 - Q30
```



The TNC exports the value that the parameter has at the time of the EXPORT command.

The parameter is exported only to the presently calling program.



## 8.9 Accessing tables with SQL commands

### Introduction

Accessing of tables is programmed on the TNC with SQL commands in **transactions**. A transaction consists of multiple SQL commands that guarantee an orderly execution of the table entries.



Tables are configured by the machine manufacturer. Names and designations required as parameters for SQL commands are also specified.

The following **terms** are used:

- **Table:** A table consists of x columns and y rows. It is saved as a file in the File Manager of the TNC, and is addressed with the path and file name (=table name). Synonyms can also be used for addressing, as an alternative to the path and file name.
- **Columns:** The number and names of the columns are specified when configuring the table. In some SQL commands the column name is used for addressing.
- **Rows:** The number of rows is variable. You can insert new rows. There are no row numbers or other designators. However, you can select rows based on the contents of a column. Rows can only be deleted in the table editor, not by an NC program.
- **Cell:** The part of a column in a row.
- **Table entry:** Content of a cell.
- **Result set:** During a transaction, the selected columns and rows are managed in the result set. You can view the result set as a sort of "intermediate memory," which temporarily assumes the set of selected columns and rows.
- **Synonym:** This term defines a name used for a table instead of its path and file name. Synonyms are specified by the machine manufacturer in the configuration data.



## A transaction

In principle, a transaction consists of the following actions:

- Address the table (file), select rows and transfer them to the result set.
- Read rows from the result set, change rows or insert new rows.
- Conclude transaction: If changes/insertions were made, the rows from the result set are placed in the table (file).

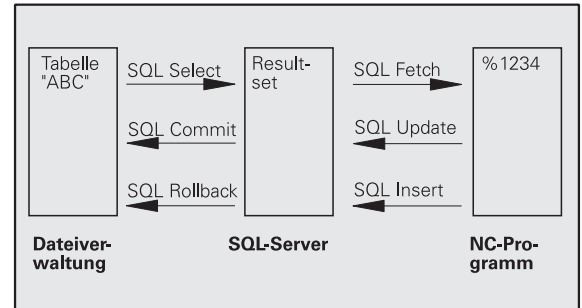
Other actions are also necessary so that table entries can be edited in an NC program and to ensure that other changes are not made to copies of the same table rows at the same time. This results in the following **transaction sequence**:

- 1 A Q parameter is specified for each column to be edited. The Q parameter is assigned to a column—it is "bound" (**SQL BIND...**).
- 2 Address the table (file), select rows and transfer them to the result set. In addition, you define which columns are transferred to the result set (**SQL SELECT...**).  
  
You can "lock" the selected rows. Other processes can then read these rows, but cannot change the table entries. You should always lock the selected rows when you are going to make changes (**SQL SELECT ... FOR UPDATE**).
- 3 Read rows from the result set, change rows or insert new rows:
  - Transfer one row of the result set into the Q parameters of your NC program (**SQL FETCH...**).
  - Prepare changes in the Q parameters and transfer one row from the result set (**SQL UPDATE...**).
  - Prepare new table row in the Q parameters and transfer into the result set as a new row (**SQL INSERT...**).
- 4 Conclude transaction:
  - If changes/insertions were made, the data from the result set is placed in the table (file). The data is now saved in the file. Any locks are canceled, and the result set is released (**SQL COMMIT...**).
  - If table entries were **not** changed or inserted (only read access), any locks are canceled and the result set is released (**SQL ROLLBACK... WITHOUT INDEX**).

Multiple transactions can be edited at the same time.



You must conclude a transaction, even if it consists solely of read accesses. Only this guarantees that changes/insertions are not lost, that locks are canceled, and that result sets are released.



## Result set

The selected rows are numbered in ascending order within the result set, starting from 0. This numbering is referred to as the **index**. The index is used for read- and write-accesses, enabling a row of the result set to be specifically addressed.

It can often be advantageous to sort the rows in the result set. Do this by specifying the table column containing the sorting criteria. Also select ascending or descending order (**SQL SELECT ... ORDER BY ...**).

The selected rows that were transferred to the result set are addressed with the **HANDLE**. All following SQL commands use the handle to refer to this "set of selected columns and rows."

When concluding a transaction, the handle is released (**SQL COMMIT...** or **SQL ROLLBACK...**). It is then no longer valid.

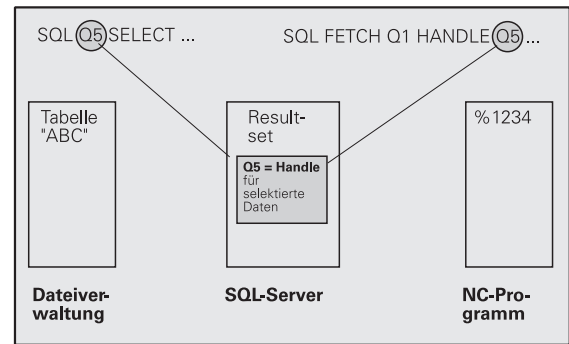
You can edit more than one result set at the same time. The SQL server assigns a new handle for each "Select" command.

### "Binding" Q parameters to columns

The NC program does not have direct access to the table entries in the result set. The data must be transferred in Q parameters. In the other direction, the data is first prepared in the Q parameters and then transferred to the result set.

Specify with **SQL BIND ...** which table columns are mapped to which Q parameters. The Q parameters are "bound" (assigned) to the columns. Columns that are not bound to Q parameters are not included in the read/write-processes.

If a new table row is generated with **SQL INSERT...**, the columns not bound to Q parameters are filled with default values.



Programming SQL commands



This function can only be programmed if you have entered the code number 555343.

Program SQL commands in the Programming mode:



- ▶ Call the SQL functions by pressing the SQL soft key
- ▶ Select an SQL command via soft key (see overview) or press the **SQL EXECUTE** soft key and program the SQL command

Overview of the soft keys

Function	Soft key
<b>SQL EXECUTE</b> Program a "Select" command.	
<b>SQL BIND</b> "Bind" a Q parameter to a table column.	
<b>SQL FETCH</b> Read table rows from the result set and save them in Q parameters.	
<b>SQL UPDATE</b> Save data from the Q parameters in an existing table row in the result set.	
<b>SQL INSERT</b> Save data from the Q parameters in a new table row in the result set.	
<b>SQL COMMIT</b> Transfer table rows from the result set into the table and conclude the transaction.	
<b>SQL ROLLBACK</b> <ul style="list-style-type: none"><li>■ If <b>INDEX</b> is not programmed: Discard any changes/insertions and conclude the transaction.</li><li>■ If <b>INDEX</b> is programmed: The indexed row remains in the result set. All other rows are deleted from the result set. The transaction is <b>not</b> concluded.</li></ul>	



# SQL BIND

**SQL BIND** binds a Q parameter to a table column. The SQL commands "Fetch," "Update" and "Insert" evaluate this binding (assignment) during data transfer between the result set and the NC program.

An **SQL BIND** command without a table or column name cancels the binding. Binding remains effective at most until the end of the NC program or subprogram.



- You can program any number of bindings. Read and write processes only take into account the columns that were entered in the "Select" command.
- **SQL BIND...** must be programmed **before** "Fetch," "Update" or "Insert" commands are programmed. You can program a "Select" command without a preceding "Bind" command.
- If in the "Select" command you include columns for which no binding is programmed, an error occurs during read/write processes (program interrupt).

SQL  
BIND

- ▶ **Parameter no. for result:** Q parameter that is bound (assigned) to the table column
- ▶ **Database: Column name:** Enter the table name and column name separated by a . (period)  
**Table name:** Synonym or path and file name of this table. The synonym is entered directly, whereas the path and file name are entered in single quotation marks  
**Column designation:** Designation of the table column as given in the configuration data

## Example: Bind a Q parameter to a table column

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
```

## Example: Cancel binding

```
91 SQL BIND Q881
92 SQL BIND Q882
93 SQL BIND Q883
94 SQL BIND Q884
```



## SQL SELECT

**SQL SELECT** selects table rows and transfers them to the result set.

The SQL server places the data in the result set row-by-row. The rows are numbered in ascending order, starting from 0. This row number, called the **INDEX**, is used in the SQL commands "Fetch" and "Update."

Enter the selection criteria in the **SQL SELECT...WHERE...** function. This lets you restrict the number of rows to be transferred. If you do not use this option, all rows in the table are loaded.

Enter the sorting criteria in the **SQL SELECT...ORDER BY...** function. Enter the column designation and the keyword for ascending/descending order. If you do not use this option, the rows are placed in random order.

Lock out the selected rows for other applications with the **SQL SELECT...FOR UPDATE** function. Other applications can continue to read these rows, but cannot change them. We strongly recommend using this option if you are making changes to the table entries.

**Empty result set:** If no rows match the selection criteria, the SQL server returns a valid handle but no table entries.



- **Parameter no. for result:** Q parameter for the handle. The SQL server returns the handle for the group of columns and rows selected with the current select command.  
In case of an error (selection could not be carried out), the SQL server returns the code 1.  
Code 0 identifies an invalid handle.
- **Database: SQL command text:** with the following elements:
  - **SELECT** (keyword):  
Name of the SQL command. Names of the table columns to be transferred. Separate column names with a , (comma) (see examples). Q parameters must be bound to all columns entered here.
  - **FROM** table name:  
Synonym or path and file name of this table. The synonym is entered directly, whereas the path name and table name are entered in single quotation marks (see examples of the SQL command, names of the table columns to be transferred—separate several columns by a comma). Q parameters must be bound to all columns entered here.
  - Optional:  
**WHERE** selection criteria:  
A selection criterion consists of a column name, condition (see table) and comparator. Link selection criteria with logical AND or OR. Program the comparator directly or with a Q parameter. A Q parameter is introduced with a colon and placed in single quotation marks (see example).
  - Optional:  
**ORDER BY** column name **ASC** to sort in ascending order—or  
**ORDER BY** column name **DESC** to sort in descending order.  
If neither ASC nor DESC are programmed, then ascending order is used as the default setting. The TNC places the selected rows in the indicated column.
  - Optional:  
**FOR UPDATE** (keyword):  
The selected rows are locked against write-accesses from other processes.

**Example: Select all table rows**

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"  
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"  
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"  
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"  
.  
.  
.  
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,  
MEAS_Z FROM TAB_EXAMPLE"
```

**Example: Selection of table rows with the WHERE function**

```
..  
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,  
MEAS_Z FROM TAB_EXAMPLE WHERE MEAS_NO<20"
```

**Example: Selection of table rows with the WHERE function and Q parameters**

```
..  
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,  
MEAS_Z FROM TAB_EXAMPLE WHERE  
MEAS_NO==:'Q11'"
```

**Example: Table name defined with path and file name**

```
..  
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,  
MEAS_Z FROM 'V:\TABLE\TAB_EXAMPLE' WHERE  
MEAS_NO<20"
```



Condition	Programming
Equal to	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR



SQL FETCH

**SQL FETCH** reads the row addressed with **INDEX** from the result set, and places the table entries in the bound (assigned) Q parameters. The result set is addressed with the **HANDLE**.

**SQL FETCH** takes into account all columns entered in the "Select" command.

SQL  
FETCH

- ▶ **Parameter no. for result:** Q parameter in which the SQL server reports the result:  
0: No error occurred.  
1: Error occurred (incorrect handle or index too large)
- ▶ **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)
- ▶ **Database: Index for SQL result:** Row number within the result set. The table entries of this row are read and are transferred into the bound Q parameters. If you do not enter an index, the first row is read (n=0). Either enter the row number directly or program the Q parameter containing the index.

**Example: Row number is transferred in a Q parameter**

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"  
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"  
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"  
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"  
. . .  
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,  
    MEAS_Z FROM TAB_EXAMPLE"  
. . .  
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

**Example: Row number is programmed directly**

```
. . .  
30 SQL FETCH Q1 HANDLE Q5 INDEX5
```



## SQL UPDATE

**SQL UPDATE** transfers the data prepared in the Q parameters into the row of the result set addressed with **INDEX**. The existing row in the result set is completely overwritten.

**SQL UPDATE** takes into account all columns entered in the "Select" command.

SQL  
UPDATE

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:  
0: No error occurred.  
1: Error occurred (incorrect handle, index too large, value outside of value range or incorrect data format)
- **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)
- **Database: Index for SQL result:** Row number within the result set. The table entries prepared in the Q parameters are written to this row. If you do not enter an index, the first row is written to (n=0). Either enter the row number directly or program the Q parameter containing the index.

## SQL INSERT

**SQL INSERT** generates a new row in the result set and transfers the data prepared in the Q parameters into the new row.

**SQL INSERT** takes into account all columns entered in the "Select" command. Table columns not entered in the "Select" command are filled with default values.

SQL  
INSERT

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:  
0: No error occurred.  
1: Error occurred (incorrect handle, value outside of value range or incorrect data format)
- **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)

### Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
```

### Example: Row number is programmed directly

```
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

### Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
40 SQL INSERT Q1 HANDLE Q5
```



# SQL COMMIT

**SQL COMMIT** transfers all rows in the result set back to the table. A lock set with **SELECT...FOR UPDATE** is canceled.

The handle given in the **SQL SELECT** command loses its validity.

SQL  
COMMIT

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:  
0: No error occurred.  
1: Error occurred (incorrect handle or equal entries in columns requiring unique entries)
- **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)

Example:

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL COMMIT Q1 HANDLE Q5
```

# SQL ROLLBACK

How **SQL ROLLBACK** is executed depends on whether **INDEX** is programmed:

- If **INDEX** is not programmed: The result set is **not** written back to the table (any changes/insertions are discarded). The transaction is closed and the handle given in the **SQL SELECT** command loses its validity. Typical application: Ending a transaction solely containing read-accesses.
- If **INDEX** is programmed: The indexed row remains. All other rows are deleted from the result set. The transaction is **not** concluded. A lock set with **SELECT...FOR UPDATE** remains for the indexed row. For all other rows it is reset.

SQL  
ROLLBACK

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:  
0: No error occurred.  
1: Error occurred (incorrect handle)
- **Database: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**)
- **Database: Index for SQL result:** Row that is to remain in the result set. Either enter the row number directly or program the Q parameter containing the index

Example:

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL ROLLBACK Q1 HANDLE Q5
```



## 8.10 Entering formulas directly

### Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the FORMULA soft key to call the mathematical functions. The TNC displays the following soft keys in several soft-key rows:

Mathematical function	Soft key
<b>Addition</b> Example: Q10 = Q1 + Q5	+
<b>Subtraction</b> Example: Q25 = Q7 – Q108	-
<b>Multiplication</b> Example: Q12 = 5 * Q5	*
<b>Division</b> Example: Q25 = Q1 / Q2	/
<b>Opening parenthesis</b> Example: Q12 = Q1 * (Q2 + Q3)	(
<b>Closing parenthesis</b> Example: Q12 = Q1 * (Q2 + Q3)	)
<b>Square of a value</b> Example: Q15 = SQ 5	SQ
<b>Square root</b> Example: Q22 = SQRT 25	SQRT
<b>Sine of an angle</b> Example: Q44 = SIN 45	SIN
<b>Cosine of an angle</b> Example: Q45 = COS 45	COS
<b>Tangent of an angle</b> Example: Q46 = TAN 45	TAN
<b>Arc sine</b> Inverse of the sine. Determines the angle from the ratio of the side opposite the angle and the hypotenuse Example: Q10 = ASIN 0.75	ASIN
<b>Arc cosine</b> Inverse of the cosine. Determines the angle from the ratio of the side adjacent to the angle and the hypotenuse Example: Q11 = ACOS Q40	ACOS



Mathematical function	Soft key
<b>Arc tangent</b> Inverse of the tangent. Determines the angle from the ratio of the opposite side to the adjacent side Example: Q12 = ATAN Q50	ATAN
<b>Powers of values</b> Example: Q15 = 3^3	^
<b>Constant "pi" (3.14159)</b> Example: Q15 = PI	PI
<b>Natural logarithm (LN) of a number</b> Base 2.7183 Example: Q15 = LN Q11	LN
<b>Logarithm of a number, base 10</b> Example: Q33 = LOG Q22	LOG
<b>Exponential function, 2.7183 to the power of n</b> Example: Q1 = EXP Q12	EXP
<b>Negate (multiplication by -1)</b> Example: Q2 = NEG Q1	NEG
<b>Truncate decimal places</b> Form an integer Example: Q3 = INT Q42	INT
<b>Absolute value of a number</b> Example: Q4 = ABS Q22	ABS
<b>Truncate places before the decimal point</b> Form a fraction Example: Q5 = FRAC Q23	FRAC
<b>Check algebraic sign of a number</b> Example: Q12 = SGN Q50 If result for Q12 = 1, then Q50 >= 0 If result for Q12 = -1, then Q50 < 0	SGN
<b>Calculate modulo value (division remainder)</b> Example: Q12 = 400 % 360 Result: Q12 = 40	%



## Rules for formulas

Mathematical formulas are programmed according to the following rules:

### Higher-level operations are performed first

12 Q1 = 5 \* 3 + 2 \* 10 = 35

1st calculation:  $5 * 3 = 15$

2nd calculation:  $2 * 10 = 20$

3rd calculation:  $15 + 20 = 35$

or

13 Q2 = SQ 10 - 3^3 = 73

1st calculation: 10 squared = 100

2nd calculation: 3 to the power of 3 = 27

3rd calculation:  $100 - 27 = 73$

### Distributive law

Law for calculating with parentheses


$$a * (b + c) = a * b + a * c$$




# Programming example



Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.



  To select the formula entering function, press the Q key and the FORMULA soft key, or use the shortcut:


 Press the Q key on the ASCII keyboard


**PARAMETER NO. FOR RESULT?**


 **25** Enter the parameter number



  Shift the soft-key row and select the arc tangent function

  Shift the soft-key row and open the parentheses

 **12** Enter Q parameter number 12

 Select division

 **13** Enter Q parameter number 13

  Close parentheses and conclude formula entry

## Example NC block

**37 Q25 = ATAN (Q12/Q13)**



## 8.11 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 **FN 16:F-PRINT** function to create variable logs.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) up to a length of 256 characters to a string parameter. You can also check and process the assigned or imported values by using the functions described below. As in Q-parameter programming, you can use a total of 2000 QS parameters (see also "Principle and overview" on page 180).

The **STRING FORMULA** and **FORMULA Q-parameter** functions contain various functions for processing the string parameters.

STRING FORMULA functions	Soft key	Page
Assign string parameters	<div>STRING</div>	Page 231
Chain-link string parameters		Page 231
Convert a numerical value to a string parameter	<div>TOCHAR</div>	Page 233
Copy a substring from a string parameter	<div>SUBSTR</div>	Page 234
FORMULA string functions	Soft key	Page
Convert a string parameter to a numerical value	<div>TONUMB</div>	Page 235
Check a string parameter	<div>INSTR</div>	Page 236
Find the length of a string parameter	<div>STRLEN</div>	Page 237
Compare alphabetic priority	<div>STRCOMP</div>	Page 238



When you use a **STRING FORMULA**, the result of the arithmetic operation is always a string. When you use the **FORMULA** function, the result of the arithmetic operation is always a numeric value.



## Assigning string parameters

You have to assign a string variable before you use it. Use the **DECLARE STRING** command to do so.

SPEC  
FCT

- Show the soft-key row with special functions

PROGRAM  
FUNCTIONS

- Select the menu for defining various plain-language functions

STRING  
FUNCTIONS

- Select string functions

DECLARE  
STRING

- Select the **DECLARE STRING** function

**Example NC block:**

```
37 DECLARE STRING QS10 = "WORKPIECE"
```



## Chain-linking string parameters

With the concatenation operator (string parameter || string parameter) you can make a chain of two or more string parameters.



- ▶ Show the soft-key row with special functions



- ▶ Select the menu for defining various plain-language functions



- ▶ Select string functions



- ▶ Select the STRING FORMULA function
- ▶ Enter the number of the string parameter in which the TNC 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 TNC displays the concatenation symbol ||
- ▶ Confirm your entry with 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, QS13 and QS14**

```
37 QS10 = QS12 || QS13 || QS14
```

Parameter contents:

- QS12: Workpiece
- QS13: Status:
- QS14: Scrap
- QS10: Workpiece Status: Scrap

## Converting a numerical value to a string parameter

With the **TOCHAR** function, the TNC converts a numerical value to a string parameter. This enables you to chain numerical values with string variables.

SPEC  
FCT

- Show the soft-key row with special functions

PROGRAM  
FUNCTIONS

- Select the menu for defining various plain-language functions

STRING  
FUNCTIONS

- Select string functions

STRING  
FORMULA

- Select the STRING FORMULA function

TOCHAR

- Select the function for converting a numerical value to a string parameter
- Enter the number or the desired Q parameter to be converted, and confirm with the ENT key
- If desired, enter the number of decimal places that the TNC 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**

```
37 QS11 = TOCHAR ( DAT+Q50 DECIMALS3 )
```



## 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

- ▶ Select the menu for defining various plain-language functions

STRING  
FUNCTIONS

- ▶ Select string functions

STRING  
FORMULA

- ▶ Select the STRING FORMULA function
- ▶ Enter the number of the string parameter in which the TNC is to save the copied string. Confirm with the ENT key

SUBSTR

- ▶ Select the function for cutting out 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



Remember that the first character of a text sequence starts internally with the zeroth place.

**Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG2)**

```
37 QS13 = SUBSTR ( SRC_QS10 BEG2 LEN4 )
```

## 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 must contain only one numerical value. Otherwise the TNC will output an error message.



FORMULA

- ▶ Select Q-parameter functions
- ▶ Select the FORMULA function
- ▶ Enter the number of the parameter in which the TNC is to save the numerical value. Confirm with the ENT key



TONUMB

- ▶ Shift the soft-key row
- ▶ Select the function for converting a string parameter to a numerical value
- ▶ Enter the number of the Q parameter to be converted, 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**

```
37 Q82 = TONUMB ( SRC_QS11 )
```



## Checking a string parameter

The **INSTR** function checks whether a string parameter is contained in another string parameter.



- ▶ Select Q-parameter functions
- ▶ Select the FORMULA function
- ▶ Enter the number of the Q parameter in which the TNC is to save the place at which the search text begins. Confirm with the ENT key
- ▶ Shift the soft-key row
- ▶ 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, and confirm with the ENT key
- ▶ Enter the number of the place starting from which the TNC is to 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



Remember that the first character of a text sequence starts internally with the zeroth place.

If the TNC 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 is found in more than one place, the TNC 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.**

```
37 Q50 = INSTR ( SRC_QS10 SEA_QS13 BEG2 )
```



## Finding the length of a string parameter

The **STRLEN** function returns the length of the text saved in a selectable string parameter.



- ▶ Select Q-parameter functions



- ▶ Select the FORMULA function
- ▶ Enter the number of the Q parameter in which the TNC is to save the ascertained string length. Confirm with the ENT key



- ▶ Shift the soft-key row
- ▶ Select the function for finding the text length of a string parameter
- ▶ Enter the number of the QS parameter whose length the TNC is to ascertain, and confirm with the ENT key
- ▶ Close the parenthetical expression with the ENT key and confirm your entry with the END key

### Example: Find the length of QS15

```
37 Q52 = STRLEN ( SRC_QS15 )
```



## Comparing alphabetic priority

The **STRCOMP** function compares string parameters for alphabetic priority.



- ▶ Select Q-parameter functions
- ▶ Select the FORMULA function
- ▶ Enter the number of the Q parameter in which the TNC is to save the result of comparison. Confirm with the ENT key
- ▶ Shift the soft-key row
- ▶ Select the function for comparing string parameters
- ▶ Enter the number of the first QS parameter to be compared, and confirm with the ENT key
- ▶ Enter the number of the second QS parameter to be compared, and confirm with the ENT key
- ▶ Close the parenthetical expression with the ENT key and confirm your entry with the END key



The TNC returns the following results:

- **0**: The compared QS parameters are identical
- **-1**: The first QS parameter **precedes** the second QS parameter alphabetically
- **+1**: The first QS parameter **follows** the second QS parameter alphabetically

**Example: QS12 and QS14 are compared for alphabetic priority**

```
37 Q52 = STRCOMP ( SRC_QS12 SEA_QS14 )
```

## Reading machine parameters

Use the **CFGREAD** function to read out TNC machine parameters as numerical values or as strings.

In order to read out a machine parameter, you must use the TNC's configuration editor to determine the parameter name, parameter object, and, if they have been assigned, the group name and index:

Type	Meaning	Example	Icon
Key	Group name of the machine parameter (if assigned)	CH_NC	
Entity	Parameter object (the name starts with "Cfg...")	CfgGeoCycle	
Attribute	Name of the machine parameter	displaySpindleErr	
Index	List index of a machine parameter (if assigned)	[0]	



If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts. To display the actual system names of the parameters, press the key for the screen layout and then the SHOW SYSTEM NAME soft key. Follow the same procedure to return to the standard display.

Each time you want to interrogate a machine parameter with the **CFGREAD** function, you must first define a QS parameter with attribute, entity and key.

The following parameters are read in the **CFGREAD** function's dialog:

- **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 string of a machine parameter

In order to store the content of a machine parameter as a string in a QS parameter:

SPEC  
FCT

PROGRAM  
FUNCTIONS

STRING  
FUNCTIONS

STRING  
FORMULA

- ▶ Show the soft-key row with special functions
- ▶ Select the menu for defining various plain-language functions
- ▶ Select string functions
- ▶ Select the STRING FORMULA function
- ▶ Enter the number of the string parameter in which the TNC is to save the machine parameter. Confirm with the ENT key
- ▶ Select the CFGREAD function
- ▶ Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the ENT key
- ▶ Enter the number for the index, or skip the dialog with NO ENT, whichever applies
- ▶ Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Read as a string the axis designation of the fourth axis

Parameter settings in the configuration editor

DisplaySettings  
CfgDisplayData  
axisDisplayOrder  
[0] to [5]

14 DECLARE STRING QS11 = ""	Assign string parameter for key
15 DECLARE STRING QS12 = "CfgDisplayData"	Assign string parameter for entity
16 DECLARE STRING QS13 = "axisDisplayOrder"	Assign string parameter for parameter name
17 QS1 = CFGREAD( KEY_QS11 TAG_QS12 ATR_QS13 IDX3 )	Read out machine parameter



### Reading a numerical value of a machine parameter

In order to store the value of a machine parameter as a numerical value in a Q parameter:



FORMULA

- ▶ Select Q-parameter functions
- ▶ Select the FORMULA function
- ▶ Enter the number of the Q parameter in which the TNC is to save the machine parameter. Confirm with the ENT key
- ▶ Select the CFGREAD function
- ▶ Enter the numbers of the string parameters for the key, entity and attribute, then confirm with the ENT key
- ▶ Enter the number for the index, or skip the dialog with NO ENT, whichever applies
- ▶ Close the parenthetical expression with the ENT key and confirm your entry with the END key

Example: Read overlap factor as Q parameter

#### Parameter settings in the configuration editor

```
ChannelSettings
CH_NC
  CfgGeoCycle
    pocketOverlap
```

14 DECLARE STRING QS11 = "CH_NC"	Assign string parameter for key
15 DECLARE STRING QS12 = "CfgGeoCycle"	Assign string parameter for entity
16 DECLARE STRING QS13 = "pocketOverlap"	Assign string parameter for parameter name
17 Q50 = CFGREAD( KEY_QS11 TAG_QS12 ATR_QS13 )	Read out machine parameter



## 8.12 Preassigned Q parameters

The Q parameters Q100 to Q199 are assigned values by the TNC. The following types of information are assigned to Q parameters:

- Values from the PLC
- Tool and spindle data
- Data on operating status
- Results of measurements from touch probe cycles etc.

The TNC saves the values for the preassigned Q parameters Q108, Q114 and Q115 to Q117 in the unit of measure used by the active program.



Do not use preassigned Q parameters (or QS parameters) between **Q100** and **Q199** (**QS100** and **QS199**) as calculation parameters in NC programs. Otherwise you might receive undesired results.

### Values from the PLC: Q100 to Q107

The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

### Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or **TOOL DEF** block)
- Delta value DR from the tool table
- Delta value DR from the **TOOL CALL** block



The TNC remembers the current tool radius even if the power is interrupted.

## Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = -1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

## Spindle status: Q110

The value of the parameter Q110 depends on the M function last programmed for the spindle.

M function	Parameter value
No spindle status defined	Q110 = -1
M3: Spindle ON, clockwise	Q110 = 0
M4: Spindle ON, counterclockwise	Q110 = 1
M5 after M3	Q110 = 2
M5 after M4	Q110 = 3

## Coolant on/off: Q111

M function	Parameter value
M8: Coolant ON	Q111 = 1
M9: Coolant OFF	Q111 = 0

## Overlap factor: Q112

The overlap factor for pocket milling (pocketOverlap) is assigned to Q112.



## Unit of measurement for dimensions in the program: Q113

During nesting with PGM CALL, the value of the parameter Q113 depends on the dimensional data of the program from which the other programs are called.

Dimensional data of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

## Tool length: Q114

The current value for the tool length is assigned to Q114.



The TNC remembers the current tool length even if the power is interrupted.

## Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates refer to the datum point that is active in the Manual Operation mode.

The length of the stylus and the radius of the ball tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
4th axis Machine-dependent	Q118
5th axis Machine-dependent	Q119



## Deviation between actual value and nominal value during automatic tool measurement with the TT 130

Deviation of actual from nominal value	Parameter value
Tool length	Q115
Tool radius	Q116



# 9

**Programming:  
Miscellaneous functions**



## 9.1 Entering miscellaneous functions M and STOP

### Fundamentals

With the TNC'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



The machine tool builder may add some M functions that are not described in this User's Manual. Refer to your machine manual.

You can enter up to two M functions at the end of a positioning block or in a separate block. The TNC displays the following dialog question:

#### Miscellaneous function M ?

You usually enter only the number of the M function in the programming dialog. Some M functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the Manual Operation and EI. Handwheel modes of operation, the M functions are entered with the M soft key.



Please note that some M functions become effective at the start of a positioning block, and others at the end, regardless of their position in the NC block.

M functions come into effect in the block in which they are called.

Some M functions are effective only in the block in which they are programmed. Unless the M function is only effective blockwise, either you must cancel it in a subsequent block with a separate M function, or it is automatically canceled by the TNC at the end of the program.

### Entering an M function in a STOP block

If you program a STOP block, the program run or test run is interrupted at the block, for example for tool inspection. You can also enter an M function in a STOP block:



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

Example NC blocks

```
87 STOP M6
```

# 9.2 Miscellaneous functions for program run control, spindle and coolant

## Overview



The machine tool builder can influence the behavior of the miscellaneous functions described below. Refer to your machine manual.

M	Effect	Effective at block...	Start	End
<b>M0</b>	Stop program run Spindle STOP			■
<b>M1</b>	Optional program STOP Spindle STOP if necessary Coolant OFF if necessary (not effective during Test Run, function determined by the machine tool builder)			■
<b>M2</b>	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (depends on the <b>clearMode</b> machine parameter)			■
<b>M3</b>	Spindle ON clockwise		■	
<b>M4</b>	Spindle ON counterclockwise		■	
<b>M5</b>	Spindle STOP			■
<b>M6</b>	Tool change Spindle STOP Program STOP			■
<b>M8</b>	Coolant ON		■	
<b>M9</b>	Coolant OFF			■
<b>M13</b>	Spindle ON clockwise Coolant ON		■	
<b>M14</b>	Spindle ON counterclockwise Coolant ON		■	
<b>M30</b>	Same as M2			■



## 9.3 Miscellaneous functions for coordinate data

### Programming machine-referenced coordinates: M91/M92

#### Scale reference point

On the scale, a reference mark indicates the position of the scale reference point.

#### Machine datum

The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)
- Moving to machine-referenced positions (such as tool change positions)
- Setting the workpiece datum

The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

#### Standard behavior

The TNC references coordinates to the workpiece datum (see "Datum setting without a 3-D touch probe", page 284).

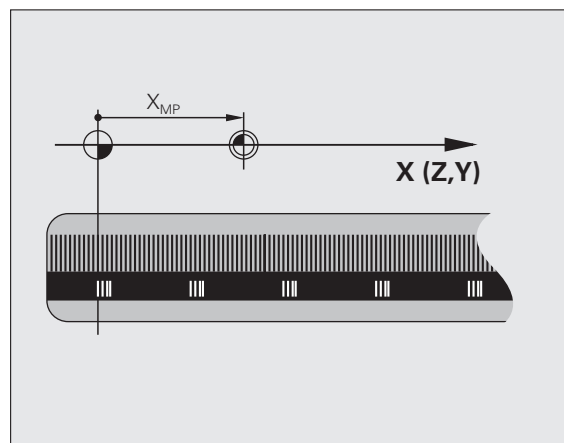
#### Behavior with M91—Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.



If you program incremental coordinates in an M91 block, enter them with respect to the last programmed M91 position. If no M91 position is programmed in the active NC block, then enter the coordinates with respect to the current tool position.

The coordinate values on the TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF (see "Status displays", page 63).



## Behavior with M92—Additional machine datum



In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.



Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

### Effect

M91 and M92 are effective only in the blocks in which they are programmed.

M91 and M92 take effect at the start of block.

### Workpiece datum

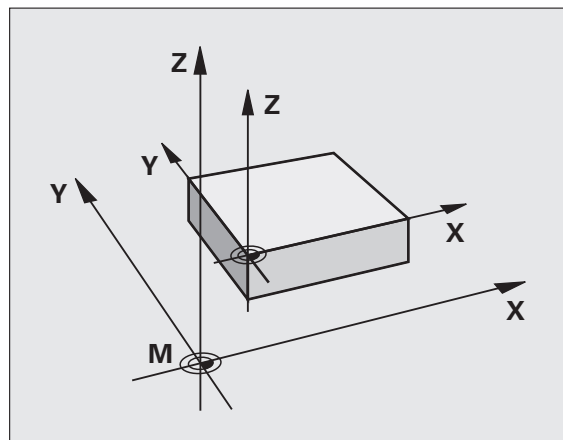
If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes.

If datum setting is inhibited for all axes, the TNC no longer displays the SET DATUM soft key in the Manual Operation mode.

The figure shows coordinate systems with the machine datum 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 set datum (see "Showing the workpiece blank in the working space", page 325).



## Reducing display of a rotary axis to a value less than 360°: M94

### Standard behavior

The TNC moves the tool from the current angular value to the programmed angular value.

Example:

Current angular value:	538°
Programmed angular value:	180°
Actual distance of traverse:	-358°

### Behavior with M94

At the start of block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. As an alternative you can enter a rotary axis after M94. The TNC then reduces the display only of this axis.

### Example NC blocks

To reduce display of all active rotary axes:

```
M94
```

To reduce display of the C axis only:

```
M94 C
```

To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

```
C+180 FMAX M94
```

### Effect

M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.



## 9.4 Miscellaneous functions for contouring behavior

### Feed rate factor for plunging movements: M103

#### Standard behavior

The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

#### Behavior with M103

The TNC 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%:

$$FZMAX = FPROG \times F\%$$

#### Programming M103

If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor F.

#### Effect

M103 becomes effective at the start of block.  
To cancel M103, program M103 once again without a factor.

#### Example NC blocks

The feed rate for plunging is to be 20 % of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
17 X+20 Y+20 RL F500 M103 F20	500
18 Y+50	500
19 IZ-2.5	100
20 IY+5 IZ-5	141
21 IX+50	500
22 Z+5	500



## Feed rate in millimeters per spindle revolution: M136

### Standard behavior

The TNC moves the tool at the programmed feed rate F in mm/min.

### Behavior with M136



In inch-programs, M136 is not permitted in combination with the new alternate feed rate FU.

The spindle is not permitted to be controlled when M136 is active.

With M136, the TNC does not move the tool in mm/min, but rather at the programmed feed rate F in millimeters per spindle revolution. If you change the spindle speed by using the spindle override, the TNC changes the feed rate accordingly.

### Effect

M136 becomes effective at the start of block.

You can cancel M136 by programming M137.

## Retraction from the contour in the tool-axis direction: M140

### Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

### Behavior with M140

With M140 MB (move back) you can enter a path in the direction of the tool axis for departure from the contour.

### Input

If you enter M140 in a positioning block, the TNC continues the dialog and asks for the desired path of tool departure from the contour. Enter the requested path that the tool should follow when departing the contour, or press the MB MAX soft key to move to the limit of the traverse range.

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 TNC moves the tool along the entered path at rapid traverse.

### Effect

M140 is effective only in the block in which it is programmed.

M140 becomes effective at the start of block.

### Example NC blocks

Block 250: Retract the tool 50 mm from the contour.

Block 251: Move the tool to the limit of the traverse range.

```
250 L X+0 Y+38.5 F125 M140 MB 50 F750
```

```
251 L X+0 Y+38.5 F125 M140 MB MAX
```



M140 is also effective if the tilted-working-plane function is active. On machines with tilting heads, the TNC then moves the tool in the tilted coordinate system.

With **M140 MB MAX** you can only retract in the positive direction.

Always define a TOOL CALL with a tool axis before entering **M140**, otherwise the direction of traverse is not defined.



## Suppressing touch probe monitoring: M141

### Standard behavior

When the stylus is deflected, the TNC outputs an error message as soon as you attempt to move a machine axis.

### Behavior with M141

The TNC 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 connection with measuring cycle 3 in order to retract the stylus by means of a positioning block after it has been deflected.



#### **Danger of collision!**

If you use M141, make sure that you retract the touch probe in the correct direction.

M141 functions only for movements with straight-line blocks.

### Effect

M141 is effective only in the block in which it is programmed.

M141 becomes effective at the start of block.

# 10

**Programming: Special  
functions**



## 10.1 Overview of special functions

The TNC provides the following powerful special functions for a large number of applications:

Function	Description
Working with text files	Page 271
Working with freely definable tables	Page 261

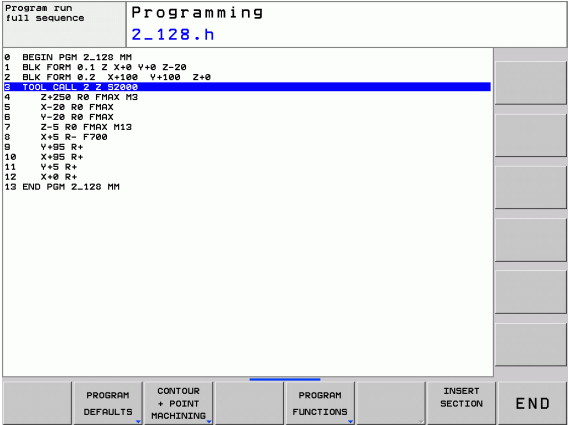
Press the SPEC FCT and the corresponding soft keys to access further special functions of the TNC. The following tables will give you an overview of which functions are available.

### Main menu for SPEC FCT special functions



► Press the special functions key

Function	Soft key	Description
Define program defaults	PROGRAM DEFAULTS	Page 259
Functions for contour and point machining	CONTOUR + POINT MACHINING	Page 259
Define various plain-language functions	PROGRAM FUNCTIONS	Page 260
Define structure items	INSERT SECTION	Page 122



# Program defaults menu

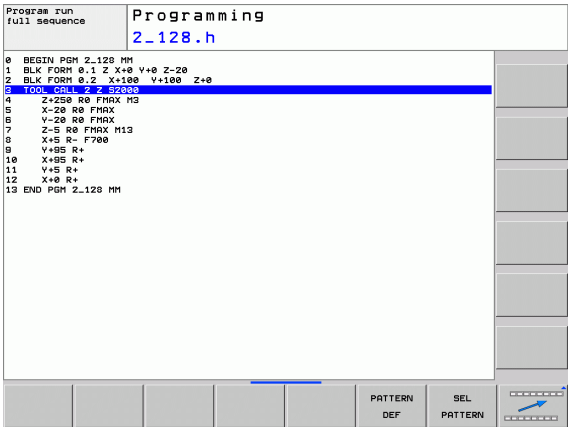
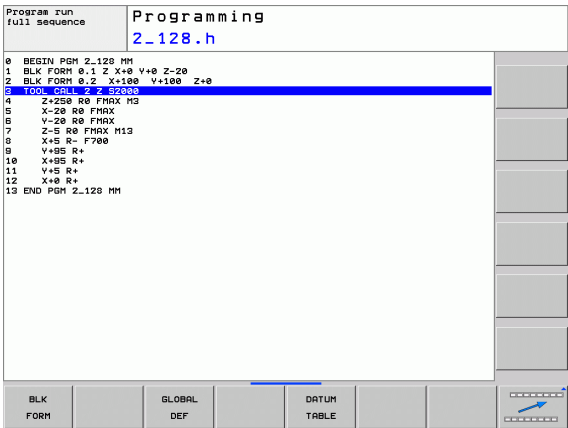
**PROGRAM  
DEFAULTS** ▶ Select the program defaults menu

Function	Soft key	Description
Define the workpiece blank	BLK FORM	Page 80
Select datum table	DATUM TABLE	See User's Manual for Cycles
Define global cycle parameters	GLOBAL DEF	See User's Manual for Cycles

## Functions for contour and point machining menu

**CONTOUR  
+ POINT  
MACHINING** ▶ Select the menu for functions for contour and point machining

Function	Soft key	Description
Define regular machining pattern	PATTERN DEF	See User's Manual for Cycles
Select the point file with machining positions	SEL PATTERN	See User's Manual for Cycles

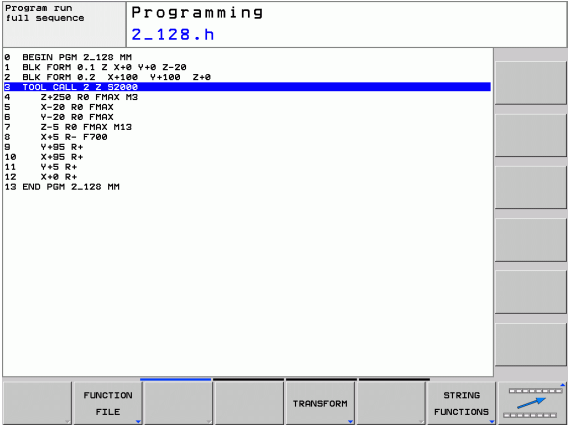


Menu for defining various plain-language functions

PROGRAM  
FUNCTIONS

► Select the menu for defining various plain-language functions

Function	Soft key	Description
Define file functions	<div>FUNCTION FILE</div>	Page 268
Define coordinate transformations	<div>TRANSFORM</div>	Page 269
Define string functions	<div>STRING FUNCTIONS</div>	Page 230
Add comments	<div>INSERT COMMENT</div>	Page 121



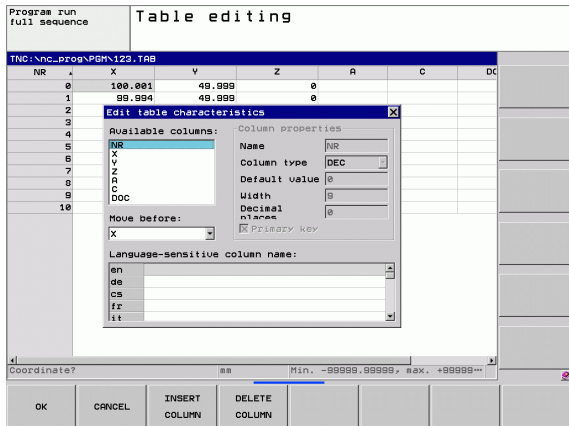




## Editing the table format

- Press the EDIT FORMAT soft key (2nd soft-key level): The TNC opens the editor form, in which the table structure is shown. The meanings of the structure commands (header entries) are shown in the following table.

Structure command	Meaning
<b>Available columns:</b>	List of all columns contained in the table
<b>Move before:</b>	The entry highlighted in <b>Available columns</b> is moved in front of this column
<b>Name</b>	Column name: Is displayed in the header
<b>Column type</b>	<b>TEXT:</b> Text input <b>SIGN:</b> Algebraic sign + or – <b>BIN:</b> Binary number <b>DEC:</b> Decimal, positive, whole number (cardinal number) <b>HEX:</b> Hexadecimal number <b>INT:</b> Whole number <b>LENGTH:</b> Length (is converted in inch programs) <b>FEED:</b> Feed rate (mm/min or 0.1 ipm) <b>IFEED:</b> Feed rate (mm/min or ipm) <b>FLOAT:</b> Floating decimal point <b>BOOL:</b> Logical value <b>INDEX:</b> Index <b>TSTAMP:</b> Permanently defined format for date and time
<b>Default value</b>	Default value for the fields in this column
<b>Width</b>	Width of the column (number of characters)
<b>Primary key</b>	First table column
<b>Language-sensitive column name</b>	Language-sensitive dialogs



You can use a connected mouse or the TNC keyboard to navigate in the form. Navigation using the TNC keyboard:



- Press the navigation keys to go to the input fields. Use the arrow keys to navigate within an input field. To open pop-down menus, press the GOTO key.



In a table that already has lines, you cannot 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.

#### Exiting the structure editor

- Press the OK soft key The TNC closes the editor form and applies the changes. All changes are discarded by pressing the CANCEL soft key.



## Switching between table and form view

All tables with the file extension **.TAB** can be opened in either list view or form view.



- Press the key for setting the screen layout. Select the respective soft key for list view or form view (form view: with or without dialog texts)

In the form view the TNC lists the line numbers with the contents of the first column in the left half of the screen.

In the right half you can change the data.

- Press the ENT key or the arrow key to move to the next input field
- To select another line, press the green navigation key (folder symbol). This moves the cursor to the left window, and you can select the desired line with the arrow keys. Press the green navigation key to switch back to the input window.

Program run full sequence		Table editing	
TNC:\nc\prog\pdm\123.TAB		NR: 0	
NR	X	NR	
0	100.001	Koordinate	100.001
1	99.994	Koordinate	49.999
2	99.999	Koordinate	0
3	100.002	Koordinate	
4	99.990	Koordinate	
5		Koordinate	
6		Bemerkung	PAT 1
7			
8			
9			
10			

Co MIN. -99999.99999, MAX.
Koordinate (mm) 1/1

TABLE  
FILTER
HIDE/  
SORT/  
COLUMNS
EDIT  
FORMAT
MORE  
FUNCTIONS
EDIT  
CURRENT  
FIELD
SORT

## FN26: TABOPEN: Opening a freely definable table

With **FN 26: TABOPEN** you can define a table to be written to with FN27, or to be read from with **FN 28**.



Only one table can be open in an NC program. A new block with TABOPEN automatically closes the last opened table.

The table to be opened must have the file name extension .TAB.

**Example: Open the table TAB1.TAB, which is saved in the directory TNC:DIR1.**

```
56 FN 26: TABOPEN TNC:\DIR1\TAB1.TAB
```



## FN 27: TABWRITE: Writing to a freely definable table

After you have opened a table with **FN 26: TABOPEN**, you can use the function **FN 27: TABWRITE** to write to it.

You can define and write to several column names in a TABWRITE block. The column names must be written between quotation marks and separated by a comma. You define the values that the TNC is to write to the respective column with Q parameters.



Note that by default the **FN 27: TABWRITE** function writes values to the currently open table also in the Test Run mode. The **FN18 ID992 NR16** function enables you to query in which operating mode the program is to be run. If the **FN27** function is to be run only in the Program Run operating modes, you can skip the respective program section by using a jump command (see "If-then decisions with Q parameters" on page 189).

You can write only to numerical table fields.

If you wish to write to more than one column in a block, you must save the values under successive Q parameter numbers.

### Example:

You wish to write to the columns "Radius," "Depth" and "D" in line 5 of the presently opened table. The values to be written to the table must be saved in the Q parameters Q5, Q6 and Q7.

```
53 FN 0: Q5 = 3.75
```

```
54 FN 0: Q6 = -5
```

```
55 FN 0: Q7 = 7.5
```

```
56 FN 27: TABWRITE 5/"RADIUS,DEPTH,D" = Q5
```



## FN28: TABREAD: Reading a freely definable table

After you have opened a table with **FN26: TABOPEN**, you can use function **FN28: TABREAD** to read from it.

You can define and read several column names in a TABREAD block. The column names must be written between quotation marks and separated by a comma. In the **FN 28** block you can define the Q parameter number in which the TNC is to write the value that is first read.



You can read only numerical table fields.

If you wish to read from more than one column in a block, the TNC will save the values under successive Q parameter numbers.

### Example:

You wish to read the values of the columns "Radius," "Depth" and "D" from line 6 of the presently opened table. Save the first value in Q parameter Q10 (second value in Q11, third value in Q12).

```
56 FN 28: TABREAD Q10 = 6/"RADIUS,DEPTH,D"
```



## 10.3 File functions

### Application

The **FILE FUNCTION** features are used to copy, move and delete files from within the part program.



You must not use **FILE** functions on programs or files, to which you have previously made reference with functions such as **CALL PGM** or **CYCL DEF 12 PGM CALL**.

### Defining file functions



- ▶ Press the special functions key
- ▶ Select the program functions
- ▶ Select the file functions: The TNC displays the available functions

Function	Meaning	Soft key
FILE COPY	Copy a file: Enter the name and path of the file to be copied, as well as the target path.	FILE COPY
FILE MOVE	Move a file: Enter the name and path of the file to be moved, as well as the target path.	FILE MOVE
FILE DELETE	Delete a file: Enter the path and name of the file to be deleted.	FILE DELETE










# 10.4 Defining coordinate transformations

## Overview

As an alternative to the coordinate transformation Cycle 7 **DATUM SHIFT**, you can use the **TRANS DATUM** plain-language function. Just as in Cycle 7, you can use **TRANS DATUM** to directly program shift values or activate a line from a selectable datum table. In addition, there is also the **TRANS DATUM RESET** function, which you can easily use to reset a datum shift.

## TRANS DATUM AXIS

You can define a datum shift by entering values in the respective axes with the **TRANS DATUM AXIS** function. You can define up to nine coordinates in one block, and incremental entries are possible. Proceed as follows for the definition:

- 
  - Show the soft-key row with special functions
- 
  - Select the menu for defining various plain-language functions
- 
  - Select transformations
- 
  - Select datum shifting with **TRANS DATUM**
- 
  - Select the value input soft key
  - Enter the datum shift in the affected axes, confirming with the ENT key each time



Values entered as absolute values refer to the workpiece datum, which is specified either by datum setting or with a preset from the preset table.

Incremental values always refer to the datum which was last valid (this may be a datum which has already been shifted).

### Example: NC block

```
13 TRANS DATUM AXIS X+10 Y+25 Z+42
```



## TRANS DATUM TABLE

You can define a datum shift by selecting a datum number from a datum table with the **TRANS DATUM TABLE** function. Proceed as follows for the definition:



- ▶ Show the soft-key row with special functions



- ▶ Select the menu for defining various plain-language functions



- ▶ Select transformations



- ▶ Select datum shifting with **TRANS DATUM**



- ▶ Press the arrow key to move to **TRANS AXIS**



- ▶ Select datum shifting with **TRANS DATUM TABLE**
- ▶ If desired, enter the name of the datum table from which you want to activate the datum number, and confirm with the ENT key. If you do not want to define a datum table, confirm with the NO ENT key
- ▶ Enter the line number to be activated by the TNC, and confirm with the ENT key



If you did not define a datum table in the **TRANS DATUM TABLE** block, then the TNC uses the datum table already selected in the NC program with **SEL TABLE**, or the datum table with status M selected in one of the Program Run modes.

### Example: NC block

```
13 TRANS DATUM TABLE TABLINE25
```

## TRANS DATUM RESET

Use the **TRANS DATUM RESET** function to cancel a datum shift. How you previously defined the datum is irrelevant. Proceed as follows for the definition:



- ▶ Show the soft-key row with special functions



- ▶ Select the menu for defining various plain-language functions



- ▶ Select transformations



- ▶ Select datum shifting with **TRANS DATUM**



- ▶ Press the arrow key to move to **TRANS AXIS**



- ▶ Select the **TRANS DATUM RESET** datum shift

### Example: NC block

```
13 TRANS DATUM RESET
```



# 10.5 Creating text files

## Application

You can use the TNC’s text editor to write and edit texts. Typical applications:







- Recording test results
- Documenting working procedures
- Creating formula collections

Text files are type .A files (ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

## Opening and exiting text files

- ▶ Select the Programming and Editing mode of operation
- ▶ Call the file manager: Press the PGM MGT key
- ▶ Display type .A files: Press the SELECT TYPE and then the SHOW .A soft keys
- ▶ Select a file and open it with the SELECT soft key or ENT key, or create 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 a part program

Cursor movements	Soft key
Move cursor one word to the right	
Move cursor one word to the left	
Go to next screen page	
Go to previous screen page	
Go to beginning of file	
Go to 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.

The line in which the cursor is presently located is depicted in a different color. 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 placed in the buffer memory
- Move the cursor to the location where you wish to insert the text, and press the RESTORE LINE/WORD soft key

Function	Soft key
Delete and temporarily store a line	DELETE LINE
Delete and temporarily store a word	DELETE WORD
Delete and temporarily store a character	DELETE CHAR
Insert a line or word from temporary storage	INSERT LINE / WORD



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
  - ▶ 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:

Function	Soft key
Delete the selected block and store temporarily	CUT OUT BLOCK
Store the selected block temporarily without erasing (copy)	INSERT BLOCK

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
  - ▶ Press the INSERT BLOCK soft key for the text block to be 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
  - ▶ Press the APPEND TO FILE soft key. The TNC displays the dialog prompt **Destination file =**
  - ▶ Enter the path and name of the destination file. The TNC appends the selected text to the specified file. If no target file with the specified name is found, the TNC 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
  - ▶ Press the READ FILE soft key. The TNC displays the dialog prompt **File name =**
  - ▶ 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. Two functions are available:

### 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
- ▶ Exit the search function: Press the END soft key

### Finding any text

- ▶ Select the search function: Press the FIND soft key. The TNC displays the dialog prompt **Find text:**
- ▶ Enter the text that you wish to find
- ▶ Find the text: Press the EXECUTE soft key
- ▶ Exit the search function: Press the END soft key



# 11

**Manual operation and  
setup**



## 11.1 Switch-on, switch-off

### Switch-on



Switch-on and crossing over the reference points can vary depending on the machine tool. Refer to your machine manual.

Switch on the power supply for TNC and machine. The TNC then displays the following dialog:

#### SYSTEM STARTUP

TNC is started

#### POWER INTERRUPTED



TNC message that the power was interrupted—clear the message

#### COMPILE A PLC PROGRAM

The PLC program of the TNC is automatically compiled

#### RELAY EXT. DC VOLTAGE MISSING



Switch on external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit

#### MANUAL OPERATION TRAVERSE REFERENCE POINTS



Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or



Cross the reference points in any sequence: Press and hold the machine axis direction button for each axis until the reference point has been traversed



If your machine is equipped with absolute encoders, you can leave out crossing the reference marks. In such a case, the TNC is ready for operation immediately after the machine control voltage is switched on.



The TNC is now ready for operation in the Manual Operation mode.



The reference points need only be crossed if the machine axes are to be moved. If you intend only to edit or test programs, you can select the Programming and Editing or Test Run modes of operation immediately after switching on the control voltage.

You can cross the reference points later by pressing the PASS OVER REFERENCE soft key in the Manual Operation mode.

## Switch-off

To prevent data from being lost at switch-off, you need to shut down the operating system of the TNC as follows:

- ▶ Select the Manual Operation mode



- ▶ Select the function for shutting down, confirm again with the YES soft key
- ▶ When the TNC displays the message **NOW IT IS SAFE TO TURN POWER OFF** in a pop-up window, you may cut off the power supply to the TNC



Inappropriate switch-off of the TNC can lead to data loss!

Remember that pressing the END key after the control has been shut down restarts the control. Switch-off during a restart can also result in data loss!



## 11.2 Moving the machine axes

### Note



Traversing with the machine axis direction buttons can vary depending on the machine tool. The machine manual provides further information.

### Moving the axis using the machine axis direction buttons



Select the Manual Operation mode



Press the machine axis direction button and hold it as long as you wish the axis to move, or



Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button



To stop the axis, press the machine STOP button

You can move several axes at a time with these two methods. You can change the feed rate at which the axes are traversed with the F soft key, see "Spindle speed S, feed rate F and miscellaneous functions M", page 281.

## Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.



Select the Manual Operation or El. Handwheel mode



Shift the soft-key row



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

**JOG INCREMENT =**



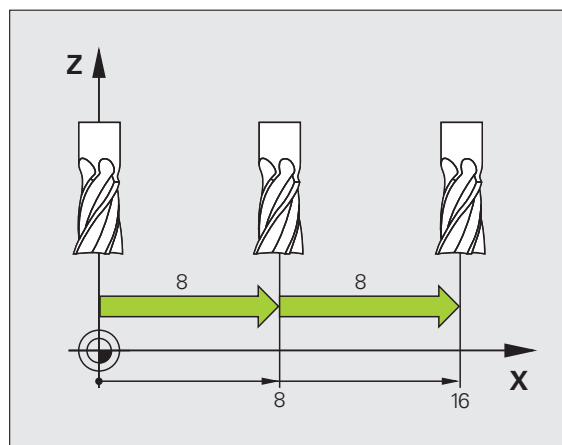
Enter the jog increment in mm, and confirm with the ENT key



Press the machine axis direction button as often as desired



The maximum permissible value for infeed is 10 mm.



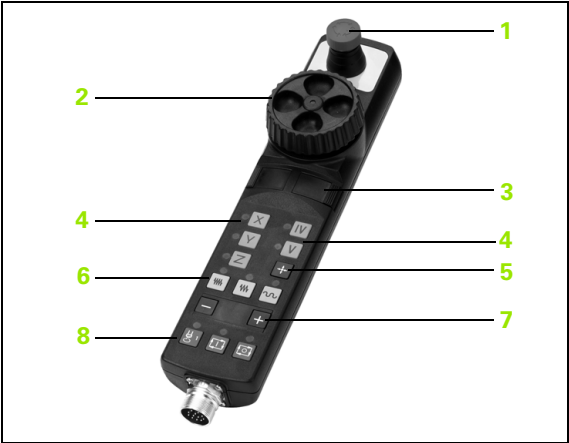
Traversing with the HR 410 electronic handwheel

The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip.

You can only move the machine axes when a permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:

- 1 EMERGENCY STOP button
- 2 Handwheel
- 3 Permissive buttons
- 4 Axis address keys
- 5 Actual-position-capture key
- 6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
- 7 Direction in which the TNC moves the selected axis
- 8 Machine function (set by the machine tool builder)



The red indicator lights show the axis and feed rate you have selected.

Procedure



Select the El. Handwheel operating mode



Press and hold a permissive button



Select the axis



Select the feed rate



Move the active axis in the positive direction, or



Move the active axis in the negative direction

## 11.3 Spindle speed S, feed rate F and miscellaneous functions M

### Function

In the Manual Operation and El. Handwheel operating modes, you can enter the spindle speed S, feed rate F and the miscellaneous functions M with soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous functions."



The machine tool builder determines which miscellaneous functions M are available on your control and what effects they have.

### Entering values

#### Spindle speed S, miscellaneous function M

S

Enter the spindle speed: Press the S soft key

SPINDLE SPEED S =

1000



Enter the desired spindle speed and confirm your entry with the machine START button

The spindle speed S with the entered rpm is started with a miscellaneous function M. Proceed in the same way to enter a miscellaneous function M.

#### Feed rate F

After entering a feed rate F, you must confirm your entry with the ENT key instead of the machine START button.

The following is valid for feed rate F:

- If you enter F=0, then the lowest feed rate from the machine parameter **manualFeed** is effective.
- If the feed rate entered exceeds the value defined in the machine parameter **maxFeed**, then the parameter value is effective.
- F is not lost during a power interruption



## Changing the spindle speed and feed rate

With the override knobs you can vary the spindle speed S and feed rate F from 0 % to 150 % of the set value.



The override knob for spindle speed is only functional on machines with infinitely variable spindle drive.



## Activating feed-rate limitation



The feed-rate limit depends on the machine. The machine manual provides further information.

When the F LIMITED soft key is set to ON, the TNC limits the maximum permissible axis speed to the safely limited speed specified by the machine manufacturer.



► Select the **Manual Operation** mode



► Scroll to the last soft-key row



► Switch on/off feed rate limit



## 11.4 Datum setting without a 3-D touch probe

### Note



Datum setting with a 3-D touch probe: (see "Datum setting with a 3-D touch probe" on page 303).

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

### Preparation

- ▶ Clamp and align the workpiece
- ▶ Insert the zero tool with known radius into the spindle
- ▶ Ensure that the TNC is showing the actual position values





## Workpiece presetting with axis keys



### Protective measure

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness  $d$  on it. Then enter a tool axis datum value that is larger than the desired datum by the value  $d$ .



Select the **Manual** Operation mode



Move the tool slowly until it touches (scratches) the workpiece surface



Select the axis

### DATUM SETTING Z=



ENT

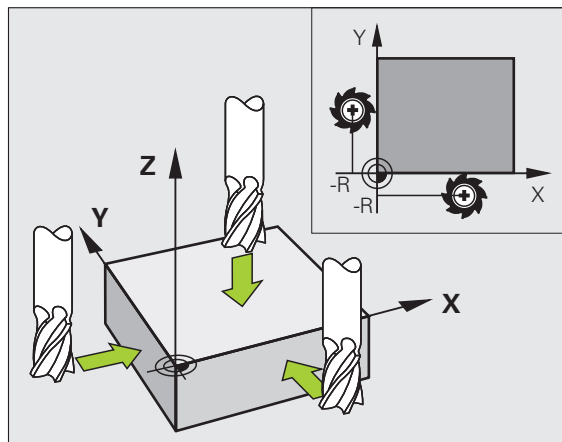
Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness  $d$  of the shim. In the tool axis, offset the tool radius

Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length  $L$  of the tool or enter the sum  $Z=L+d$



The TNC automatically saves the datum set with the axis keys in line 0 of the preset table.



## Datum management with the preset table



You should definitely use the preset table if until now you have been working with older TNC controls with REF-based datum tables.

The preset table can contain any number of lines (datums). To optimize the file size and the processing speed, you should use only as many lines as you need for datum management.

For safety reasons, new lines can be inserted only at the end of the preset table.

### Saving the datums in the preset table

The preset table has the name **PRESET.PR**, and is saved in the directory **TNC:\table**. **PRESET.PR** is editable in the **Manual Operation** and **E1. Handwheel** modes only if the **CHANGE PRESET** soft key was pressed.

It is permitted to copy the preset table into another directory (for data backup). Lines that were written by your machine tool builder are also always write-protected in the copied tables. You therefore cannot edit them.

Never change the number of lines in the copied tables! That could cause problems when you want to reactivate the table.

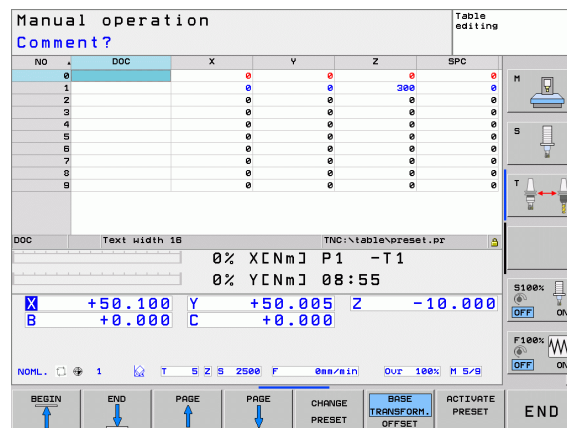
To activate the preset table copied to another directory you have to copy it back to the directory **TNC:\table\**.

There are two methods for saving datums and/or basic rotations in the preset table:

- Via touch probe cycles in the **Manual Operation** and **E1. Handwheel** modes (see "Using the 3-D touch probe" on page 291)
- Manual entry (see description below)



The line 0 in the preset table is write protected. In line 0, the TNC always saves the datum that you most recently set manually via the axis keys or via soft key. If the datum set manually is active, the TNC displays the text **PR MAN(0)** in the status display.



## Manually saving the datums in the preset table

In order to set datums in the preset table, proceed as follows:



Select the **Manual Operation** mode



Move the tool slowly until it touches (scratches) the workpiece surface, or position the measuring dial correspondingly



Display the preset table: The TNC opens the preset table and sets the cursor to the active table row



Select functions for entering the presets: The TNC displays the available possibilities for entry in the soft-key row. See the table below for a description of the entry possibilities



Select the line in the preset table that you want to change (the line number is the preset number)








If needed, select the column (axis) in the preset table that you want to change















Use the soft keys to select one of the available entry possibilities (see the following table)



Function	Soft key
Directly transfer the actual position of the tool (the measuring dial) as the new datum: This function only saves the datum in the axis which is currently highlighted	
Assign any value to the actual position of the tool (the measuring dial): This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window	
Incrementally shift a datum already stored in the table: This function only saves the datum in the axis which is currently highlighted. Enter the desired corrective value with the correct sign in the pop-up window. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm	
Directly enter the new datum without calculation of the kinematics (axis-specific). Only use this function if your machine has a rotary table, and you want to set the datum to the center of the rotary table by entering 0. This function only saves the datum in the axis which is currently highlighted. Enter the desired value in the pop-up window. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm	
Write the currently active datum to a selectable line in the table: This function saves the datum in all axes, and then activates the appropriate row in the table automatically. If inch display is active: Enter the value in inches, and the TNC will internally convert the entered values to mm	

## Editing the preset table

Editing function in table mode	Soft key
Select beginning of table	
Select end of table	
Select previous page in table	
Select next page in table	
Select the functions for preset entry	
Activate the datum of the selected line of the preset table	
Add the entered number of lines to the end of the table (2nd soft-key row)	
Copy the highlighted field (2nd soft-key row)	
Insert the copied field (2nd soft-key row)	
Reset the selected line: The TNC enters - in all columns (2nd soft-key row)	
Insert a single line at the end of the table (2nd soft-key row)	
Delete a single line at the end of the table (2nd soft-key row)	

## Activating a datum from the preset table in the Manual Operation mode



When activating a datum from the preset table, the TNC resets the active datum shift, mirroring, rotation and scaling factor.



Select the **Manual Operation** mode



Display the preset table



Select the datum number you want to activate, or



**4**



With the GOTO key, select the datum number that you want to activate. Confirm with the ENT key



Activate the preset



Confirm activation of the datum. The TNC sets the display and—if defined—the basic rotation



Exit the preset table

## Activating a datum from the preset table in an NC program

To activate datums from the preset table during program run, use Cycle 247. In Cycle 247 you define the number of the datum that you want to activate (see User's Manual, Cycles, Cycle 247 SET DATUM).

# 11.5 Using the 3-D touch probe

## Overview

The following touch probe cycles are available in the Manual Operation mode:








HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used.



The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. The machine manual provides further information.





The touch probe cycles are available only with the **Touch probe function** software option (option number 17). If you are using a HEIDENHAIN touch probe, this option is available automatically.

Function	Soft key	Page
Calibrate the touch probe		Page 298
Set the datum in any axis		Page 303
Set a circle center as datum		Page 304
Set a center line as datum		Page 304
Touch probe system data management		Page 467



## Functions in touch probe cycles

Soft keys that are used to select the probing direction or a probing routine are displayed in the manual touch probe cycles. The soft keys displayed vary depending on the respective cycle:

Soft key	Function
	Select the probing direction
	Capture the actual position
	Probe hole (inside circle) automatically
	Probe stud (outside circle) automatically

### Automatic probing routine for holes and studs



If you use a function for probing a circle automatically, the TNC automatically positions the touch probe to the respective touch points. Ensure that the positions can be approached without collision.

If you use a probing routine for probing a hole or a stud automatically, the TNC opens a form with the required input fields.

Input fields in the **Measure a stud** and **Measure hole** forms:

Input field	Function
<b>Stud diameter?</b> or <b>Hole diameter?</b>	Diameter of probe contact (optional for holes)
<b>Safety clearance?</b>	Distance to the probe contact in the plane
<b>Incr. clearance height?</b>	Positioning of touch probe in spindle axis direction (starting from the current position)
<b>Starting angle?</b>	Angle for the first probing operation (0° = Positive direction of principal axis, i.e. in X+ for spindle axis Z). All other probe angles result from the number of touch points.
<b>Number of touch points?</b>	Number of probing operations (3 to 8)
<b>Angular length?</b>	Probing a full circle (360°) or a circle segment (angular length < 360°)



Position the touch probe approximately in the center of the hole (inside circle) or near the first touch point on the stud (outside circle), and select the soft key for the first probing direction. Once you press the machine START button to start the touch probe cycle, the TNC automatically performs all prepositioning movements and probing operations.

The TNC positions the touch probe to the individual touch points, taking the safety clearance into account. If a clearance height has been defined, the TNC positions the touch probe to clearance height in the spindle axis beforehand.

The TNC approaches the position at the feed rate **FMAX** defined in the touch probe table. The defined probing feed rate **F** is used for the actual probing operation.



Before starting the automatic probing routine, you need to preposition the touch probe near the first touch point. Offset the touch probe by approximately the safety clearance (value from touch probe table + value from input form) opposite to the probing direction.

For an inside circle with a large diameter, the TNC can also preposition the touch probe on a circular arc at the positioning feed rate **FMAX**. This requires that you enter a safety clearance for prepositioning and the hole diameter in the input form. Position the touch probe inside the hole at a position that is offset by approximately the safety clearance from the wall. For prepositioning, keep in mind the starting angle for the first probing operation (with an angle of 0°, the TNC probes in the positive direction of the principal axis).

## Selecting touch probe cycles

- Select the Manual Operation or El. Handwheel mode of operation



- Select the touch probe functions by pressing the TOUCH PROBE soft key. The TNC displays additional soft keys (see overview table).



- Select the touch probe cycle by pressing the appropriate soft key, for example PROBING POS, for the TNC to display the associated menu



When you select a manual probing function, the TNC opens a form displaying all data required. The content of the forms varies depending on the respective function.

You can also enter values in some of the fields. Use the arrow keys to move to the desired input field. You can position the cursor only in fields that can be edited. Fields that cannot be edited appear dimmed.



## Recording measured values from the touch-probe cycles



The TNC must be specially prepared by the machine tool builder for use of this function. The machine manual provides further information.

After executing any selected touch probe cycle, the TNC displays the soft key WRITE LOG TO FILE. If you press this soft key, the TNC will record the current values determined in the active touch probe cycle.

If you store the measuring results, the TNC creates the text file TCHPRMAN.TXT. Unless you define a specific path in the machine parameter **fn16DefaultPath**, the TNC will store the TCHPRMAN.TXT file in the main directory **TNC:\**.



When you press the WRITE LOG TO FILE soft key, the TCHPRMAN.TXT file must not be active in the **Programming** mode of operation. The TNC will otherwise display an error message.

The TNC stores the measured data in the TCHPRMAN.TXT file only. If you execute several touch probe cycles in succession and want to store the resulting measured data, you must make a backup of the contents stored in TCHPRMAN.TXT between the individual cycles by copying or renaming the file.

Format and content of the TCHPRMAN.TXT file are preset by the machine tool builder.

## Writing the measured values from touch probe cycles to datum tables



Use this function if you want to save measured values in the workpiece coordinate system. If you want to save measured values in the fixed machine coordinate system (REF coordinates), press the ENTER IN PRESET TABLE soft key (see "Writing the measured values from touch probe cycles to the preset table" on page 296).

With the ENTER IN DATUM TABLE soft key, the TNC can write the values measured during a touch probe cycle in a datum table:

- ▶ Select any probe function
- ▶ Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run)
- ▶ Enter the datum number in the **Number in table=** input box
- ▶ Press the ENTER IN DATUM TABLE soft key. The TNC saves the datum in the indicated datum table under the entered number



## Writing the measured values from touch probe cycles to the preset table



Use this function if you want to save measured values in the machine-based coordinate system (REF coordinates). If you want to save measured values in the workpiece coordinate system, press the ENTER IN DATUM TABLE soft key (see "Writing the measured values from touch probe cycles to datum tables" on page 295).

With the ENTER IN PRESET TABLE soft key, the TNC can write the values measured during a probe cycle in the preset table. The measured values are then stored referenced to the machine-based coordinate system (REF coordinates). The preset table has the name PRESET.PR, and is saved in the directory TNC:\table\.

- ▶ Select any probe function
- ▶ Enter the desired coordinates of the datum in the appropriate input boxes (depends on the touch probe cycle being run)
- ▶ Enter the preset number in the **Number in table:** input box
- ▶ Press the ENTER IN PRESET TABLE soft key. The TNC saves the datum in the preset table under the entered number

## 11.6 Calibrating a 3-D touch probe

### Introduction

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe, otherwise the TNC cannot provide precise measuring results.



Always calibrate a touch probe in the following cases:

- Commissioning
- Stylus breakage
- Stylus exchange
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up
- Change of active tool axis

When you press the OK soft key after calibration, the calibration values are applied to the active touch probe. The updated tool data become effective immediately, and a new tool call is not necessary.

During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

The TNC provides calibration cycles for calibrating the length and the radius:




- Press the TOUCH PROBE soft key



- Display the calibration cycles: Press CALIBRATE TS

- Select the calibration cycle

Calibration cycles of the TNC:

Soft key	Function	Page
	Calibrate the length	Page 298
	Measure the radius and the center offset using a calibration ring	Page 299
	Measure the radius and the center offset using a stud or a calibration pin	Page 299



## Calibrating the effective length



HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used.

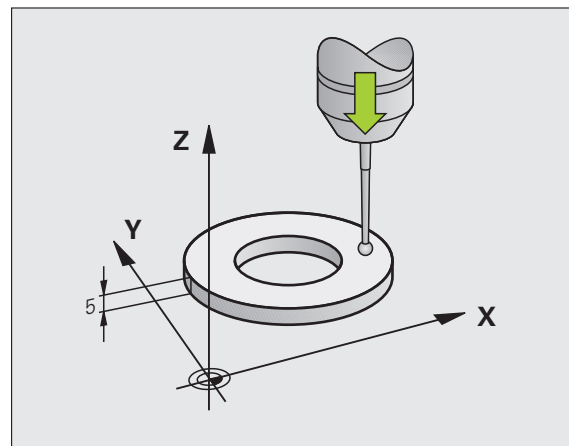


The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

- Set the datum in the spindle axis such that for the machine tool table  $Z=0$ .



- Select the calibration function for the touch probe length: Press the CAL. L soft key. The TNC opens a menu window with input fields
- Datum for length: Enter the height of the ring gauge
- New cal. spindle angle: Spindle angle that is used for the calibration. The TNC uses CAL\_ANG from the touch probe table as a default value. If you change the value, the TNC saves the value to the touch probe table during calibration.
- Move the touch probe to a position just above the ring gauge
- To change the traverse direction (if necessary), press a soft key or an arrow key
- To probe the upper surface of the ring gauge, press the machine START button
- Check the results (change the values if required)
- Press the OK soft key for the values to take effect
- Press the END soft key to terminate the calibrating function



## Calibrating the effective radius and compensating center offset



HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used.



The center offset can be determined only with a suitable touch probe.

If you want to calibrate using the outside of an object, you need to preposition the touch probe above the center of the calibration sphere or calibration pin. Ensure that the touch points can be approached without collision.

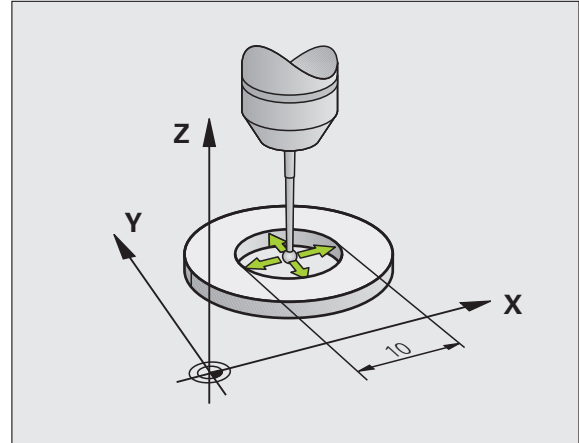
After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The calibration function can determine the offset between touch-probe axis and spindle axis by probing from opposite orientations (rotation by 180°) and can compute the compensation.

The calibration routine varies depending on how your touch probe can be oriented:

- No orientation possible or orientation possible in only one direction: The TNC executes eight probing operations and determines only the effective ball tip radius (column R in tool.t).
- Orientation possible in two directions (e.g. HEIDENHAIN touch probes with cable): The TNC executes eight probing operations, rotates the touch probe by 180° and then executes four more probing operations. The center offset (CAL\_OF in tchprobe.tp) is determined in addition to the radius by probing from opposite orientations.
- Any orientation possible (e.g. HEIDENHAIN infrared touch probes): For probing routine, see "orientation possible in two directions."

When calibrating the ball tip radius, the TNC executes an automatic probing routine. During the first probing cycle, the TNC determines the center of the calibration ring or stud (coarse measurement) and positions the touch probe in the center. Then the ball tip radius is determined during the actual calibration process (fine measurement). If the touch probe allows probing from opposite orientations, the center offset is determined during another cycle.

The characteristic of whether and how your touch probe can be oriented is already defined in HEIDENHAIN touch probes. Other touch probes are configured by the machine tool builder.



Proceed as follows for manual calibration using a calibration ring:

- ▶ In the Manual Operation mode, position the ball tip inside the bore of the ring gauge



- ▶ Select the calibration function: Press the CAL. R soft key
- ▶ Enter the diameter of the ring gauge
- ▶ Enter the safety clearance
- ▶ New cal. spindle angle: Spindle angle that is used for the calibration. The TNC uses CAL\_ANG from the touch probe table as a default value. If you change the value, the TNC saves the value to the touch probe table during calibration
- ▶ Start the probing procedure: Press the machine START button. The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the TNC calculates the center offset
- ▶ Check the results (change the values if required)
- ▶ Press the OK soft key for the values to take effect
- ▶ Press the END soft key to terminate the calibrating function



In order to be able to determine the ball-tip center offset, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information.



Proceed as follows for manual calibration with a stud or calibration pin:

- In the Manual Operation mode, position the ball tip above the center of the calibration pin



- Select the calibration function: Press the CAL. R soft key
- Enter the diameter of the stud
- Enter the safety clearance
- New cal. spindle angle: Spindle angle that is used for the calibration. The TNC uses CAL\_ANG from the touch probe table as a default value. If you change the value, the TNC saves the value to the touch probe table during calibration
- Start the probing procedure: Press the machine START button. The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius. If probing from opposite orientations is possible, the TNC calculates the center offset
- Check the results (change the values if required)
- Press the END soft key to terminate the calibrating function



In order to be able to determine the ball-tip center offset, the TNC needs to be specially prepared by the machine manufacturer. The machine manual provides further information.

## Displaying calibration values

The TNC saves the effective length and effective radius of the touch probe in the tool table. The TNC saves the ball-tip center offset of the touch probe in the touch-probe table in the **CAL\_OF1** (principal axis) and **CAL\_OF2** (secondary axis) columns. You can display the values on the screen by pressing the TOUCH PROBE TABLE soft key.



Make sure that you have activated the correct tool number before using the touch probe, regardless of whether you wish to run the touch probe cycle in automatic mode or manual mode.



For more information about the touch probe table, refer to the User's Manual for Cycle Programming.

Table editing

Test run

TNC:\table\ichprobe.id							
NO	TYPE	CAL_OF1	CAL_OF2	CAL_RNG	P	FMAX	DIST
1	TS120	0	0	0	500	+2000	10
2	TS120	0	0	0	500	+2000	10

Selection of the touch probe?

BEGIN

END

PAGE

PAGE

EDIT

OFF

ON

FIND

END

H

S

T

S100%

OFF

ON

F100%

OFF




ON



# 11.7 Datum setting with a 3-D touch probe

## Overview

The following soft-key functions are available for setting the datum on an aligned workpiece:

Soft key	Function	Page
	Datum setting in any axis	Page 303
	Setting a circle center as datum	Page 304
	Center line as datum	Page 307

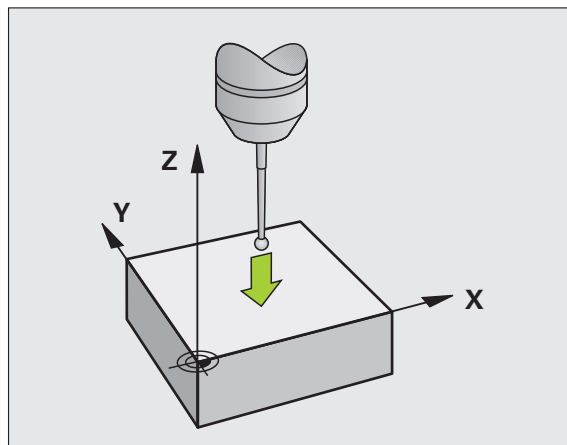


HEIDENHAIN grants a warranty for the function of the touch probe cycles only if HEIDENHAIN touch probes are used.

## Datum setting in any axis



- ▶ Select the probing function: Press the PROBING POS soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Use the soft keys to select the probe axis and direction in which you want to set the datum, such as Z in direction Z–
- ▶ Start the probing procedure: Press the machine START button
- ▶ Datum: Enter the nominal coordinate and confirm your entry with the SET DATUM soft key, see "Writing the measured values from touch probe cycles to datum tables", page 295
- ▶ Terminate the probing function: Press the END soft key



## Circle center as datum

With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.

### Inside circle:

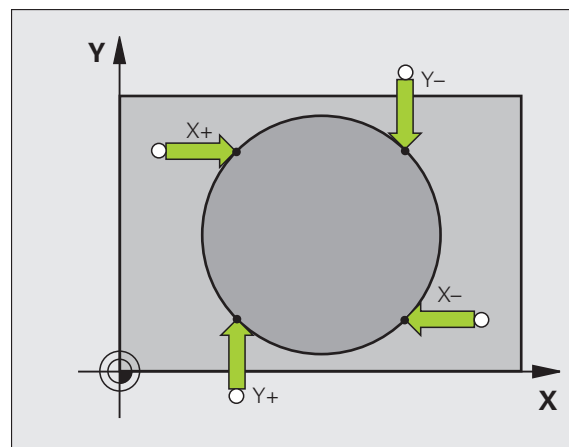
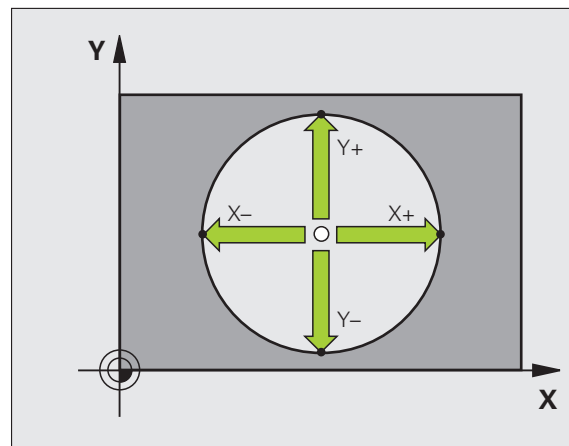
The TNC probes the inside wall of a circle in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- ▶ Position the touch probe approximately in the center of the circle
  - ▶ Select the probing function: Press the PROBING CC soft key
  - ▶ Select the probing direction or press the soft key for the automatic probing routine
  - ▶ Start the probing procedure: Press the machine START button. The touch probe probes the inside wall of the circle in the selected direction. If you are not using the automatic probing routine, you need to repeat this procedure. After the third probing operation, you can have the TNC calculate the center (four touch points are recommended).
  - ▶ Terminate the probing procedure and switch to the evaluation menu: Press the EVALUATE soft key
  - ▶ **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles to datum tables", page 295, or see "Writing the measured values from touch probe cycles to the preset table", page 296)
  - ▶ Terminate the probing function: Press the END soft key



The TNC needs only three touch points to calculate outside or inside circles, e.g. for circle segments. More precise results are obtained if you measure circles using four touch points, however. You should always preposition the touch probe in the center, or as close to the center as possible.



## Outside circle:

- ▶ Position the touch probe at a position near the first touch point outside of the circle
- ▶ Select the probe direction by soft key
- ▶ Start the probing procedure: Press the machine START button. If you are not using the automatic probing routine, you need to repeat this procedure. After the third probing operation, you can have the TNC calculate the center (four touch points are recommended).
- ▶ Terminate the probing procedure and switch to the evaluation menu: Press the EVALUATE soft key
- ▶ **Datum:** Enter the coordinates of the datum and confirm your entry with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles to datum tables", page 295, or see "Writing the measured values from touch probe cycles to the preset table", page 296)
- ▶ Terminate the probing function: Press the END soft key

After the probing procedure is completed, the TNC displays the current coordinates of the circle center and the circle radius PR.



## Setting the datum using multiple holes/cylindrical studs

A second soft-key row provides a soft key for using multiple holes or cylindrical studs to set the datum. You can set the intersection of two or more elements as datum.

Select the probing function for the intersection of holes/cylindrical studs:



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



- ▶ Hole is to be probed automatically: Define by soft key



- ▶ Circular stud is to be probed automatically: Define by soft key

Preposition the touch probe approximately in the center of the hole or near the first touch point of the circular stud. After you have pressed the NC Start key, the TNC automatically probes the points on the circle.

Move the touch probe to the next hole, repeat the probing operation and have the TNC repeat the probing procedure until all the holes have been probed to set the datum.

Setting the datum in the intersection of multiple holes:

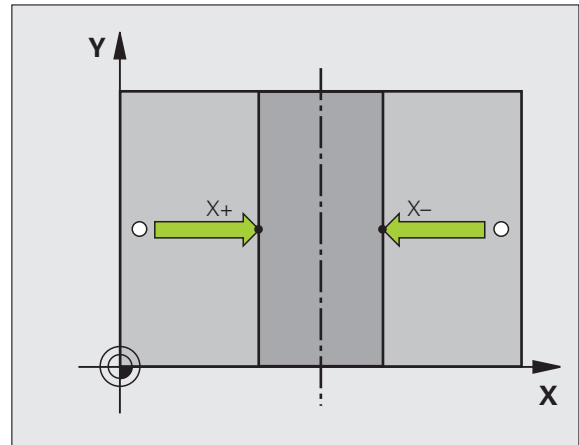
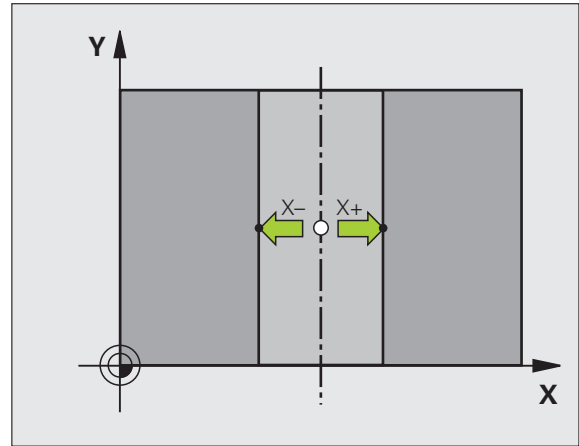


- ▶ Preposition the touch probe approximately in the center of the hole
- ▶ Hole is to be probed automatically: Define by soft key
- ▶ Start the probing procedure: Press the machine START button. The touch probe probes the circle automatically.
- ▶ Repeat the probing procedure for the remaining elements
- ▶ Terminate the probing procedure and switch to the evaluation menu: Press the EVALUATE soft key
- ▶ **Datum:** In the menu window, enter both coordinates of the circle center, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles to datum tables", page 295, or see "Writing the measured values from touch probe cycles to the preset table", page 296)
- ▶ Terminate the probing function: Press the END soft key

## Center line as datum



- ▶ Select the probe function: Press the PROBING soft key
- ▶ Position the touch probe at a position near the first touch point
- ▶ Select the probing direction by soft key
- ▶ Start the probing procedure: Press the NC Start button
- ▶ Position the touch probe at a position near the second touch point
- ▶ Start the probing procedure: Press the NC Start button
- ▶ **Datum:** Enter the coordinate of the datum in the menu window, confirm with the SET DATUM soft key, or write the value to a table (see "Writing the measured values from touch probe cycles to datum tables", page 295, or see "Writing the measured values from touch probe cycles to the preset table", page 296).
- ▶ Terminate the probing function: Press the END key



## Measuring workpieces with a 3-D touch probe

You can also use the touch probe in the Manual Operation and El. Handwheel operating modes to make simple measurements on the workpiece. With a 3-D touch probe you can determine:

- Position coordinates, and from them,
- Dimensions and angles on the workpiece

### Finding the coordinates of a position on an aligned workpiece



- ▶ Select the probing function: Press the PROBING POS soft key
- ▶ Move the touch probe to a position near the touch point
- ▶ Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection
- ▶ Start the probing procedure: Press the machine START button

The TNC shows the coordinates of the touch point as reference point.



## Measuring workpiece dimensions

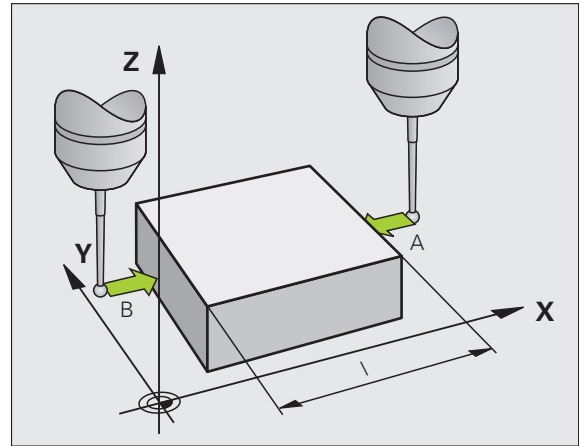


- ▶ Select the probing function: Press the PROBING POS soft key
- ▶ Position the touch probe at a position near the first touch point A
- ▶ Select the probing direction by soft key
- ▶ Start the probing procedure: Press the machine START button
- ▶ If you need the current datum later, write down the value that appears in the Datum display
- ▶ Datum: Enter "0"
- ▶ Cancel the dialog: Press the END key
- ▶ Select the probing function again: Press the PROBING POS soft key
- ▶ Position the touch probe at a position near the second touch point B
- ▶ Select the probe direction with the soft keys: Same axis but from the opposite direction
- ▶ Start the probing procedure: Press the machine START button

The value displayed as datum is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

- ▶ Select the probing function: Press the PROBING POS soft key
- ▶ Probe the first touch point again
- ▶ Set the datum to the value that you wrote down previously
- ▶ Cancel the dialog: Press the END key



## Using touch probe functions with mechanical probes or dial gauges

If you do not have an electronic 3-D touch probe on your machine, you can also use all the previously described manual touch probe functions (exception: calibration function) with mechanical probes or by simply touching the workpiece with the tool.

In place of the electronic signal generated automatically by a 3-D touch probe during probing, you can manually initiate the trigger signal for capturing the **probing position** by pressing a key. Proceed as follows:



- ▶ Select any touch probe function by soft key
- ▶ Move the mechanical probe to the first position to be captured by the TNC
- ▶ Confirm the position: Press the actual-position-capture soft key for the TNC to save the current position
- ▶ Move the mechanical probe to the next position to be captured by the TNC
- ▶ Confirm the position: Press the actual-position-capture soft key for the TNC to save the current position
- ▶ If required, move to additional positions and capture as described previously
- ▶ **Datum:** In the menu window, enter the coordinates of the new datum, confirm with the SET DATUM soft key, or write the values to a table (see "Writing the measured values from touch probe cycles to datum tables", page 295, or see "Writing the measured values from touch probe cycles to the preset table", page 296)
- ▶ Terminate the probing function: Press the END key

# 12

**Positioning with manual  
data input**



## 12.1 Programming and executing simple machining operations

The Positioning with Manual Data Input mode of operation is particularly convenient for simple machining operations or to pre-position the tool. You can write the a short program in HEIDENHAIN conversational programming and execute it immediately. You can also call TNC cycles. The program is stored in the file \$MDI. In the Positioning with MDI mode of operation, the additional status displays can also be activated.

### Positioning with manual data input (MDI)



#### Limitation

The following functions are not available in the MDI mode:

- FK free contour programming
- Program section repeats
- Subprogramming
- Path compensation
- The programming graphics
- Program call **PGM CALL**
- The program-run graphics



Select the Positioning with MDI mode of operation.  
Program the file \$MDI as you wish



Start the program run: Press the machine START key

#### Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

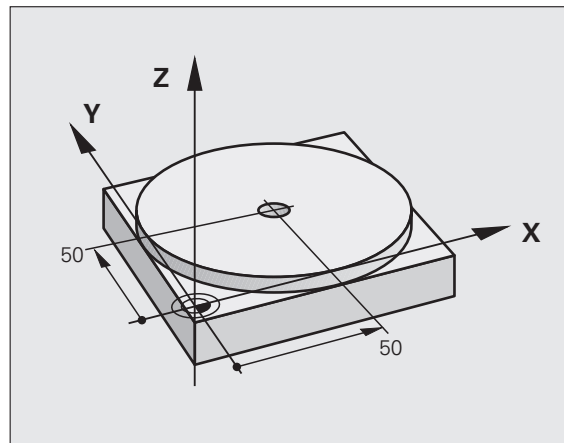
First you preposition the tool with straight-line blocks to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle **200 DRILLING**.

```
0 BEGIN PGM $MDI MM
```

```
1 TOOL CALL 1 Z S2000
```

```
2 Z+200 R0 FMAX
```

```
3 X+50 R0 FMAX
```



Call the tool: tool axis Z,  
spindle speed 2000 rpm

Retract the tool (F MAX = rapid traverse)

Move the tool at F MAX to a position above the hole


<b>3 Y+50 R0 FMAX M3</b>	Move the tool at F MAX to a position above the hole, spindle on
<b>4 CYCL DEF 200 DRILLING</b>	Define the DRILLING cycle
<b>Q200=5 ;SET-UP CLEARANCE</b>	Set-up clearance of the tool above the hole
<b>Q201=-15 ;DEPTH</b>	Hole depth (algebraic sign=working direction)
<b>Q206=250 ;FEED RATE FOR PLNGNG</b>	Feed rate for drilling
<b>Q202=5 ;PLUNGING DEPTH</b>	Depth of each infeed before retraction
<b>Q210=0 ;DWELL TIME AT TOP</b>	Dwell time after every retraction in seconds
<b>Q203=-10 ;SURFACE COORDINATE</b>	Coordinate of the workpiece surface
<b>Q204=20 ;2ND SET-UP CLEARANCE</b>	Set-up clearance of the tool above the hole
<b>Q211=0.2 ;DWELL TIME AT DEPTH</b>	Dwell time in seconds at the hole bottom
<b>5 CYCL CALL</b>	Call the DRILLING cycle
<b>6 Z+200 R0 FMAX M2</b>	Retract the tool
<b>7 END PGM \$MDI MM</b>	End of program

DRILLING cycle: See "DRILLING (Cycle 200)" on page 389




## Protecting and erasing programs in \$MDI


The \$MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:




Select the Programming and Editing mode of operation



Press the PGM MGT key (program management) to call the file manager



Move the highlight to the \$MDI file




Select the file copying function: Press the COPY soft key


DESTINATION FILE =

HOLE

Enter the name under which you want to save the current contents of the \$MDI file



Copy the file



Close the file manager: Press the END soft key

For more information: see "Copying a single file", page 101.



# 13

**Test run and  
program run**



## 13.1 Graphics

### Application

In the program run modes of operation as well as in the Test Run mode, the TNC graphically simulates the machining of the workpiece. Using soft keys, select whether you desire:

- Plan view
- Projection in three planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter. For this purpose, enter  $R2 = R$  in the tool table.

The TNC will not show a graphic if

- the current program has no valid workpiece blank definition
- no program is selected



The TNC graphic does not show a radius oversize **DR** that has been programmed in the **TOOL CALL** block.

A graphic simulation is only possible under certain conditions for program sections or programs in which rotary axis movements are defined. The graphic may not be displayed correctly by the TNC.



Setting the speed of the test run



The most recently set speed remains active, even if the power is interrupted, until you change it.

After you have started a program, the TNC displays the following soft keys with which you can set the simulation speed:

Functions	Soft key
Perform the test run at the same speed at which the program will be run (programmed feed rates are taken into account)	
Increase the test speed incrementally	
Decrease the test speed incrementally	
Test run at the maximum possible speed (default setting)	

You can also set the simulation speed before you start a program:






- ▶ Shift the soft-key row
- ▶ Select the function for setting the simulation speed
- ▶ Select the desired function by soft key, e.g. incrementally increasing the test speed



Overview of display modes

The TNC displays the following soft keys in the Program Run and Test Run modes of operation:

View	Soft key
Plan view	
Projection in three planes	
3-D view	

Limitations during program run



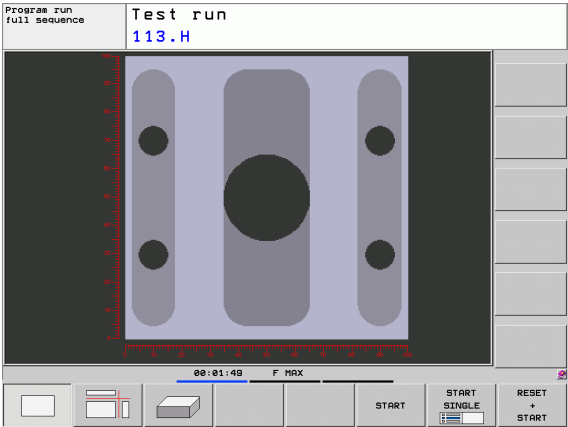
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text **ERROR** in the graphics window. The machining process is continued, however.

Plan view

This is the fastest of the graphic display modes.



- ▶ Press the soft key for plan view
- ▶ Regarding depth display, remember: The deeper the surface, the darker the shade









# Projection in 3 planes

Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes.

Details can be isolated in this display mode for magnification (see "Magnifying details", page 322).

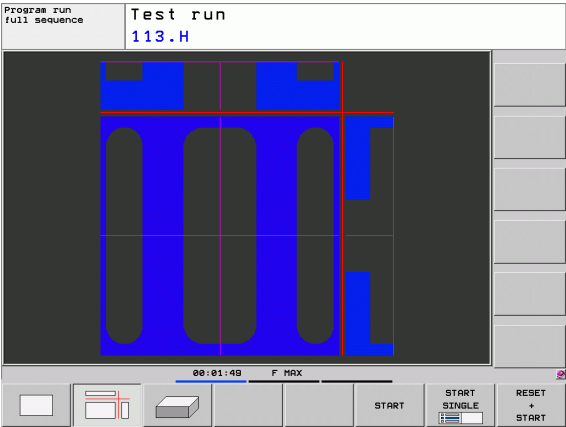
In addition, you can shift the sectional planes with the corresponding soft keys:

- ▶ Select the soft key for projection in three planes
- ▶ Shift the soft-key row until the soft key for the functions for shifting the sectional plane appears
- ▶ Select the functions for shifting the sectional plane. The TNC offers the following soft keys:

Function	Soft keys
Shift the vertical sectional plane to the right or left	 
Shift the vertical sectional plane forward or backward	 
Shift the horizontal sectional plane upwards or downwards	 

The positions of the sectional planes are visible during shifting.

The default setting of the sectional plane is selected such that it lies in the working plane in the workpiece center and in the tool axis on the top surface.



## 3-D view

The workpiece is displayed in three dimensions.

You can rotate the 3-D display about the vertical and horizontal axes via soft keys. If there is a mouse attached to your TNC, you can also perform this function by holding down the right mouse button and dragging the mouse.

The shape of the workpiece can be depicted by a frame overlay at the beginning of the graphic simulation.

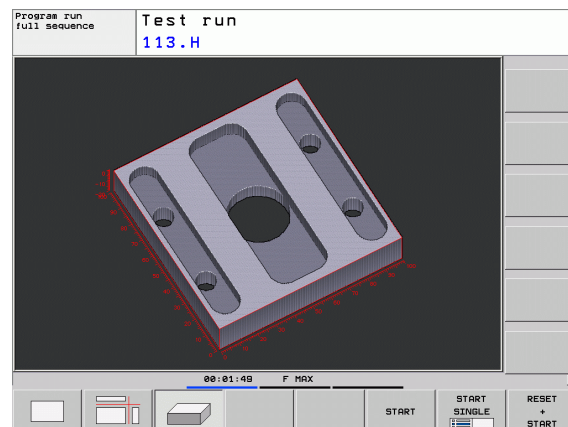
In the Test Run mode of operation you can isolate details for magnification, see "Magnifying details", page 322.



► Press the soft key for 3-D view



The speed of the 3-D graphics depends on the tooth length (**LCUTS** column in the tool table). If **LCUTS** is defined as 0 (basic setting), the simulation calculates an infinitely long tooth length, which leads to a long processing time.










Rotating and magnifying/reducing the 3-D view



► Shift the soft-key row until the soft key for the rotating and magnification/reduction appears



► Select functions for rotating and magnifying/reducing:

Function	Soft keys
Rotate in 5° steps about the vertical axis	 
Tilt in 5° steps about the horizontal axis	 
Magnify the graphic stepwise	
Reduce the graphic stepwise	
Reset image to programmed size	

If there is a mouse attached to your TNC, you can also perform the functions described above with the mouse:

- In order to rotate the graphic shown in three dimensions: Hold the right mouse button down and move the mouse. After you release the right mouse button, the TNC orients the workpiece to the defined orientation
- In order to shift the graphic shown: Hold the center mouse button or the wheel button down and move the mouse. The TNC shifts the workpiece in the corresponding direction. After you release the center mouse button, the TNC shifts the workpiece to the defined position
- In order to zoom in on a certain area with the mouse: Draw a rectangular zoom area while holding the left mouse button down. After you release the left mouse button, the TNC zooms in on the defined area of the workpiece
- In order to quickly zoom in and out with the mouse: Rotate the wheel button forward or backward



Magnifying details

You can magnify details in all display modes in the Test Run mode and a Program Run mode.

The graphic simulation or the program run, respectively, must first have been stopped. A detail magnification is always effective in all display modes.

Changing the detail magnification

The soft keys are listed in the table

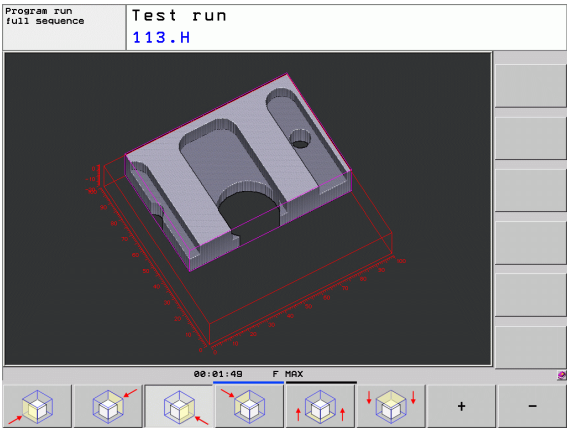
- ▶ Interrupt the graphic simulation, if necessary
- ▶ Shift the soft-key row in the Test Run mode, or in a Program Run mode, respectively, until the soft key for detail magnification appears



- ▶ Shift the soft-key row until the soft-key for the detail magnification functions appears



- ▶ Select the functions for detail magnification
- ▶ Press the corresponding soft key to select the workpiece surface (see table below)
- ▶ To reduce or magnify the blank form, press and hold the MINUS or PLUS soft key, respectively
- ▶ Restart the test run or program run by pressing the START soft key (RESET + START returns the workpiece to its original state)



Function	Soft keys	
Select the left/right workpiece surface		
Select the front/back workpiece surface		
Select the top/bottom workpiece surface		
Shift the sectional plane to reduce or magnify the workpiece blank		
Select the isolated detail		





After a new workpiece detail magnification is selected, the control "forgets" previously simulated machining operations. The TNC then displays machined areas as unmachined areas.

If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. To clear the error message, reduce or enlarge the workpiece blank.



## Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.



Function	Soft key
Restore workpiece to the detail magnification in which it was last shown	
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM	



With the WINDOW BLK FORM soft key, you return the displayed workpiece blank to its originally programmed dimensions, even after isolating a detail without TRANSFER DETAIL.

## Displaying the tool

You can display the tool during simulation in the plan view and in the projection in three planes. The TNC depicts the tool in the diameter defined in the tool table.

Function	Soft key
Do not display the tool during simulation	
Display the tool during simulation	



# Measuring the machining time

## Program Run modes of operation

The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.

### Test run

The timer displays the time that the TNC calculates for the duration of tool movements that are executed at feed rate. Dwell times are included in the calculation by the TNC. The time calculated by the TNC can only conditionally be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

### Activating the stopwatch function



- ▶ Shift the soft-key row until the soft-key for the stopwatch functions appears



- ▶ Select the stopwatch functions

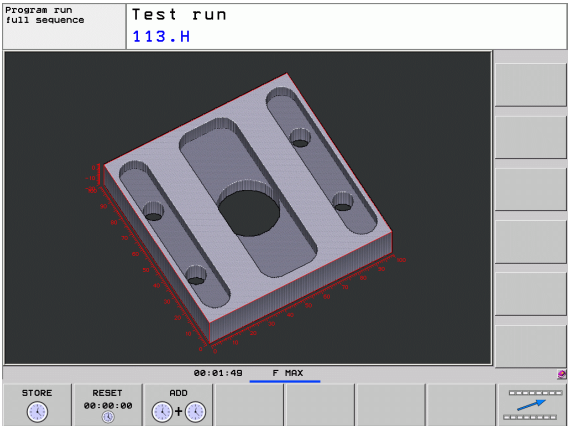


- ▶ Select the desired function by soft key, e.g. saving the displayed time

Stopwatch functions	Soft key
Store displayed time	
Display the sum of stored time and displayed time	
Clear displayed time	



During the test run, the TNC resets the machining time as soon as a new **BLK FORM** is evaluated.





# 13.2 Showing the workpiece blank in the working space


## Application

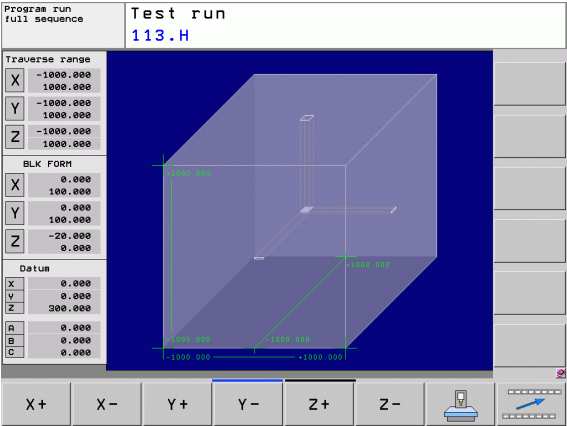
This MOD function enables you to graphically check the position of the workpiece blank or reference point in the machine's working space and to activate work space monitoring in the Test Run mode of operation. This function is activated with the **BLANK IN WORKSPACE** soft key. You can activate or deactivate the function with the **SW limit monitoring** soft key (2nd soft-key row).

Another transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The TNC takes the dimensions from the workpiece blank definition of the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the traverse-range cuboid.

For a test run it normally does not matter where the workpiece blank is located within the working space. However, if you activate working-space monitoring, you must graphically shift the workpiece blank so that it lies within the working space. Use the soft keys shown in the table.

You can also activate the current datum for the Test Run operating mode (see the last line of the following table).





Function	Soft keys	
Shift workpiece blank in positive/negative X direction	X +	X -
Shift workpiece blank in positive/negative Y direction	Y +	Y -
Shift workpiece blank in positive/negative Z direction	Z +	Z -
Show workpiece blank referenced to the set datum		
Switch monitoring function on or off	SW limit monitoring	



# 13.3 Functions for program display

## Overview

In the Program Run modes of operation as well as in the Test Run mode, the TNC provides the following soft keys for displaying a part program in pages:

Functions	Soft key
Go back in the program by one screen	
Go forward in the program by one screen	
Go to the beginning of the program	
Go to the end of the program	



## 13.4 Test Run

### Application

In the Test Run mode of operation you can simulate programs and program sections to reduce programming errors during program run. The TNC checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

- Blockwise test run
- Interruption of test at any block
- Optional block skip
- Functions for graphic simulation
- Measuring the machining time
- Additional status display





## Caution: Danger of collision!

The TNC cannot graphically simulate all traverse motions actually performed by the machine. These include

- Traverse motions during tool change, if the machine manufacturer defined them in a tool-change macro or via the PLC
- Positioning movements that the machine manufacturer defined in an M-function macro
- Positioning movements that the machine manufacturer performs via the PLC

HEIDENHAIN therefore recommends proceeding with caution for every new program, even when the program test did not output any error message, and no visible damage to the workpiece occurred.

After a tool call, the TNC always starts a program test at the following position:

- In the machining plane at the position  $X=0$ ,  $Y=0$
- In the tool axis, 1 mm above the **MAX** point defined in the **BLK FORM**

If you call the same tool, the TNC resumes program simulation from the position last programmed before the tool call.

In order to ensure unambiguous behavior during program run, after a tool change you should always move to a position from which the TNC can position the tool for machining without causing a collision.



Your machine tool builder can also define a tool-change macro for the Test Run operating mode. This macro will simulate the exact behavior of the machine. Please refer to your machine tool manual.

Executing a test run

If the central tool file is active, a tool table must be active (status S) to conduct a test run. Select a tool table via the file manager (PGM MGT) in the Test Run mode of operation

With the BLANK IN WORK SPACE function, you activate working space monitoring for the test run, see "Showing the workpiece blank in the working space", page 325.



- ▶ Select the Test Run operating mode
- ▶ Call the file manager with the PGM MGT key and select the file you wish to test, or
- ▶ Go to the program beginning: Select line 0 with the GOTO key and confirm your entry with the ENT key

The TNC then displays the following soft keys:

Functions	Soft key
Reset the blank form and test the entire program	<div>RESET + START</div>
Test the entire program	<div>START</div>
Test each program block individually	<div>START SINGLE</div>
Halt test run (soft key only appears once you have started the test run)	<div>STOP</div>

You can interrupt the test run and continue it again at any point—even within a fixed cycle. In order to continue the test, the following actions must not be performed:

- Selecting another block with the arrow keys or the GOTO key
- Making changes to the program
- Switching the operating mode
- Selecting a new program



## 13.5 Program run

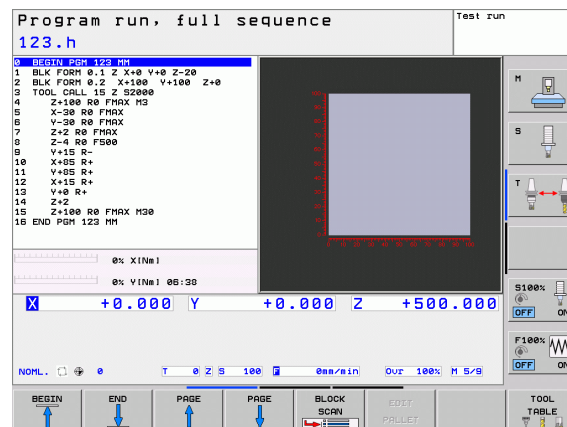
### Application

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the Program Run, Single Block mode of operation you must start each block separately by pressing the machine START button.

The following TNC functions are available in the program run modes of operation:

- Interrupting the program run
- Starting the program run from a certain block
- Optional block skip
- Editing the tool table TOOL.T
- Checking and changing Q parameters
- Superimposing handwheel positioning
- Functions for graphic simulation
- Additional status display



## Running a part program

### Preparation

- 1 Clamp the workpiece to the machine table
- 2 Set the datum
- 3 Select the required tables (status M).
- 4 Select the part program (status M)



You can adjust the feed rate and spindle speed with the override knobs.



It is possible to reduce the feed rate when starting the NC program using the FMAX soft key. The reduction applies to all rapid traverse and feed rate movements. The value you enter is no longer in effect after the machine has been turned off and on again. In order to re-establish the respectively defined maximum feed rate after switch-on, you need to re-enter the corresponding value.

The behavior of this function varies depending on the respective machine. Refer to your machine manual.

### Program Run, Full Sequence

- Start the part program with the machine START button

### Program Run, Single Block

- Start each block of the part program individually with the machine START button



Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Pressing the machine STOP button
- Switching to Program Run, Single Block

If the TNC registers an error during program run, it automatically interrupts the machining process.


Programmed interruptions

You can program interruptions directly in the part program. The TNC interrupts the program run at a block containing one of the following entries:

- **STOP** (with and without a miscellaneous function)
- Miscellaneous functions **M0**, **M2** or **M30**
- Miscellaneous function **M6** (defined by the machine tool builder)

Interruption through the machine STOP button

- ▶ Press the machine STOP button: The block that the TNC is currently executing is not completed. The NC stop signal in the status display blinks (see table)
- ▶ If you do not wish to continue the machining process, you can reset the TNC with the INTERNAL STOP soft key. The NC stop signal in the status display goes out. In this case, the program must be restarted from the program beginning

Symbol	Meaning
	Program run is stopped

Interrupting the machining process by switching to the Program Run, Single Block mode of operation

You can interrupt a program that is being run in the Program Run, Full Sequence mode of operation by switching to the Program Run, Single Block mode. The TNC interrupts the machining process at the end of the current block.





## Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the Manual Operation mode.

### Application example:

#### Retracting the spindle after tool breakage

- ▶ Interrupt machining
- ▶ Enable the external direction keys: Press the MANUAL TRAVERSE soft key
- ▶ Move the axes with the machine axis-direction keys



On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis-direction keys. Refer to your machine manual.



## Resuming program run after an interruption



If you cancel a program with INTERNAL STOP, you have to start the program with the RESTORE POS. AT N function or with GOTO "0".

If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.

If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT N function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations (e.g. datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined



Note that the stored data remain active until they are reset (e.g. if you select a new program).

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION soft key).

### Resuming program run with the START button

You can resume program run by pressing the machine START button if the program was interrupted in one of the following ways:

- The machine STOP button was pressed
- Programmed interruption

### Resuming program run after an error

If the error message is not blinking:

- ▶ Remove the cause of the error
- ▶ Clear the error message from the screen: Press the CE key
- ▶ Restart the program, or resume program run where it was interrupted

### If the error message is blinking:

- ▶ Press and hold the END key for two seconds. This induces a TNC system restart
- ▶ Remove the cause of the error
- ▶ Start again

If you cannot correct the error, write down the error message and contact your service agency.

## Mid-program startup (block scan)



The RESTORE POS AT N feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.

With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.



Mid-program startup must not begin in a subprogram.

All necessary programs and tables must be selected in a program run mode of operation (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.

After a block scan, return the tool to the calculated position with RESTORE POSITION.

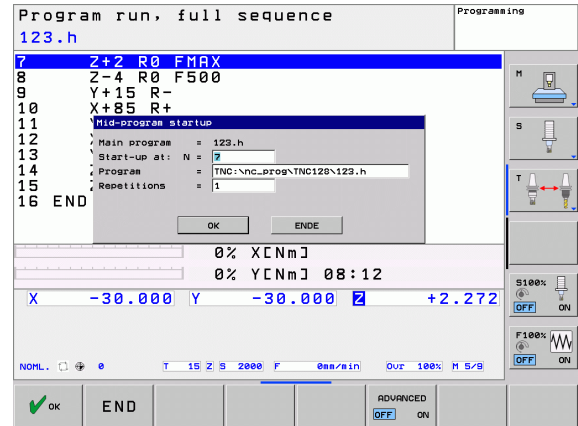
Tool length compensation does not take effect until after the tool call and a following positioning block. This applies, too, if you have only changed the tool length.



The TNC skips all touch probe cycles in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

You may not use mid-program startup if the following occurs after a tool change in the machining program:

- The stretch filter is active
- The program is started in a threading cycle (Cycles 17, 18, 19, 206, 207 and 209) or the subsequent program block
- Touch-probe cycles 0, 1 and 3 are used before program start



- Go to the first block of the current program to start a block scan:  
Enter GOTO "0"



- Select mid-program startup: Press the MID-PROGRAM STARTUP soft key
- **Start-up at N:** Enter the block number N at which the block scan should end
- **Program:** Enter the name of the program containing block N
- **Repetitions:** If block N is located in a program section repeat or in a subprogram that is to be run repeatedly, enter the number of repetitions to be calculated in the block scan
- Start mid-program startup: Press the machine START button
- Contour approach (see following section)

## Entering a program with the GOTO key



If you use the GOTO block number key for going into a program, neither the TNC nor the PLC will execute any functions that ensure a safe start.

If you use the GOTO block number key for going into a subprogram,

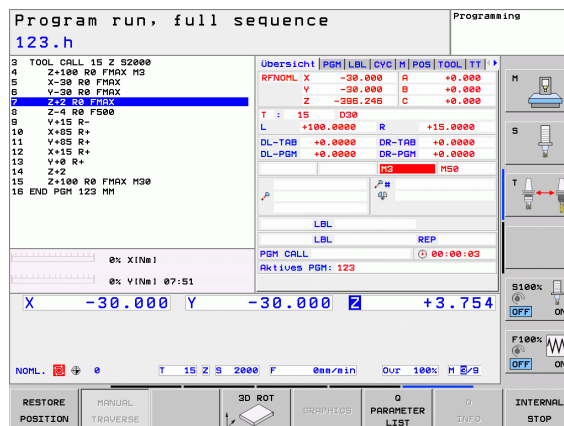
- the TNC will skip the end of the subprogram (**LBL 0**)
- the TNC will reset function M126 (Shorter-path traverse of rotary axes)

In such cases you must always use the mid-program startup function.

## 13.5 Program run

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function
- Return to the contour after a block scan with RESTORE POS AT N, for example after an interruption with INTERNAL STOP
- Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption

- ▶ Press the **RESTORE POSITION** soft key
- ▶ Restore machine status, if required
- ▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine **START** button, or
- ▶ To move the axes in any sequence, press the soft keys **RESTORE X**, **RESTORE Z**, etc., and activate each axis with the machine **START** button
- ▶ To resume machining, press the machine **START** button



## 13.6 Optional block skip

### Application

In a test run or program run, the control can skip over blocks that begin with a slash "/":



- ▶ To run or test the program without the blocks preceded by a slash, set the soft key to ON



- ▶ To run or test the program with the blocks preceded by a slash, set the soft key to OFF



This function does not work for **TOOL DEF** blocks.

After a power interruption the TNC returns to the most recently selected setting.

### Inserting the "/" character

- ▶ In the **Programming** mode you select the block in which the character is to be inserted



- ▶ Select the INSERT soft key

### Erasing the "/" character

- ▶ In the **Programming** mode you select the block in which the character is to be deleted



- ▶ Select the REMOVE soft key

## 13.7 Optional program-run interruption

### Application

The TNC optionally interrupts program run at blocks containing M1. If you use M1 in the Program Run mode, the TNC does not switch off the spindle or coolant.



- ▶ Do not interrupt program run or test run at blocks containing M1: Set soft key to OFF



- ▶ Interrupt program run or test run at blocks containing M1: Set soft key to ON





# 14

**MOD functions**



## 14.1 Selecting MOD functions

The MOD functions provide additional input possibilities and displays. In addition you can enter code numbers to enable access to protected areas. The available MOD functions depend on the selected operating mode.

### Selecting the MOD functions

Call the operating mode in which you wish to change the MOD functions. Open the pop-up window with the MOD functions:

- MOD
 ▶ Select the MOD functions: Press the MOD key. The TNC opens a pop-up window displaying the available MOD -functions.

### Changing the settings

There are three possibilities for changing a setting, depending on the function selected:

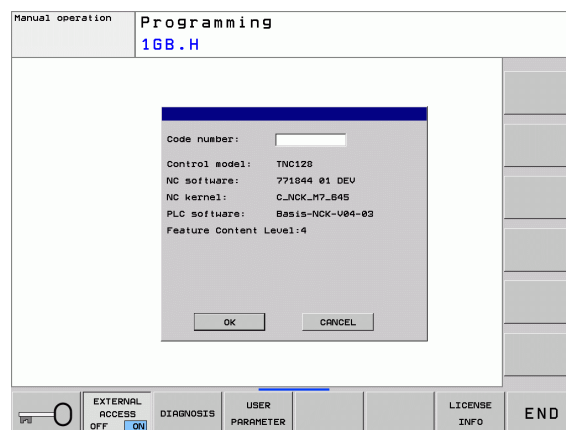
- Enter a numerical value directly, e.g. when determining the traverse range limit
- Change a setting by pressing the ENT key, e.g. when setting program input
- Change a setting via a selection window



If more than one possibility is available for a particular setting, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the setting with the ENT key. If you don't want to change the setting, close the window again with END.

### Exiting the MOD functions

- ▶ Exit MOD functions: Press the END soft key or END key



## Overview of MOD functions

The functions available depend on the momentarily selected operating mode:

Programming:

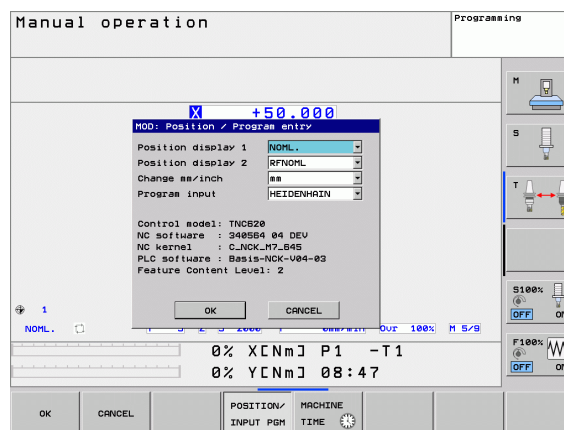
- Display software numbers
- Enter code number
- Machine-specific user parameters, if applicable
- Legal information

Test run:

- Display software numbers
- Showing the active tool table in the test run
- Showing the active datum table in the test run

In all other modes:

- Display software numbers
- Select position displays
- Define unit of measurement (mm/inches)
- Set the programming language for MDI
- Select the axes for actual position capture
- Display operating times



## 14.2 Software numbers

### Application

The following software numbers are displayed on the TNC screen after the "Software version" MOD function has been selected:

- **Control model**: Designation of the control (managed by HEIDENHAIN)
- **NC software**: Number of the NC software (managed by HEIDENHAIN)
- **NC software**: Number of the NC software (managed by HEIDENHAIN)
- **NCK**: Number of the NC software (managed by HEIDENHAIN)
- **PLC software**: Number or name of the PLC software (managed by your machine tool builder)

In the "FCL information" MOD function, the TNC shows the following information:

- **Feature Content Level (FCL)**: Development level of the software installed on the control (see "Feature content level (upgrade functions)" on page 8)



# 14.3 Entering code numbers

## Application

The TNC requires a code number for the following functions:

Function	Code number
Selecting user parameters	123
Configuring an Ethernet card	NET123
Enabling special functions for Q parameter programming	555343



## 14.4 Setting the data interfaces

### Serial interfaces on the TNC 128

The TNC 128 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is permanent and cannot be changed except for setting the baud rate (machine parameter **baudRateLsv2**). You can also specify another type of transmission (interface). The settings described below are therefore effective only for the respective newly defined interface.

### Application

To set up a data interface, select the file management (PGM MGT) and press the MOD key. Press the MOD key again and enter the code number 123. The TNC shows the user parameter **GfgSerialInterface**, in which you can enter the following settings:

### Setting the RS-232 interface

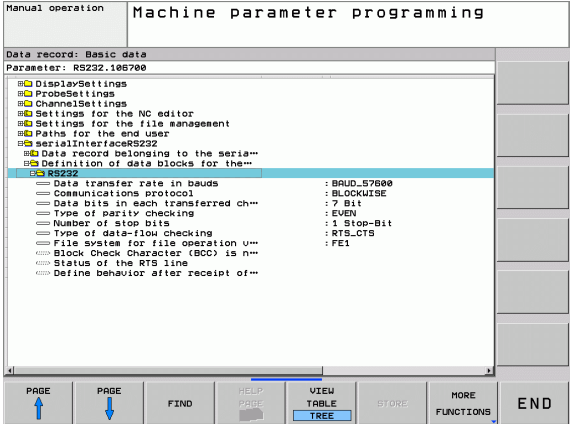
Open the RS232 folder. The TNC then displays the following settings:

### Setting the baud rate (baudRate)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

### Setting the protocol (protocol)

The data transfer protocol controls the data flow of a serial transmission (comparable to MP5030 of the iTNC 530).



Here, the BLOCKWISE setting designates a form of data transfer where data is transmitted in blocks. This is not to be confused with the blockwise reception and processing of data by older TNC contouring controls. Blockwise reception of an NC program and simultaneous machining of the program is not possible!

Communications protocol	Selection
Standard data transfer	STANDARD
Packet-based data transfer	BLOCKWISE
Transmission without protocol	RAW_DATA



## Setting the data bits (dataBits)

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

## Parity check (parity)

The parity bit helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

## Setting the stop bits (stopBits)

The start bit and one or two stop bits enable the receiver to synchronize to every transmitted character during serial data transmission.

## Setting the handshake (flowControl)

By handshaking, two devices control data transfer between them. A distinction is made between software handshaking and hardware handshaking.

- No data flow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS\_CTS): Transmission stop is active through RTS
- Software handshaking (XON\_XOFF): Transmission stop is active through DC3 (XOFF)



Settings for data transfer with the TNCserver  
PC software

Enter the following settings in the user parameters  
(serialInterfaceRS232 / definition of data blocks for the serial  
ports / RS232):

Parameter	Selection
Data transfer rate in baud	Has to match the setting in TNCserver
Data transmission protocol	BLOCKWISE
Data bits in each transferred character	7 bits
Type of parity checking	EVEN
Number of stop bits	1 stop bit
Specify type of handshake:	RTS_CTS
File system for file operations	FE1

Setting the operating mode of the external  
device (fileSystem)



The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the FE2 and FEX modes.

External device	Operating mode	Symbol
PC with HEIDENHAIN data transfer software TNCremoNT	LSV2	
HEIDENHAIN floppy disk units	FE1	
Non-HEIDENHAIN devices such as printers, scanners, punchers, PC without TNCremoNT	FEX	





## Software for data transfer

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremo data transfer software. With TNCremo, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.



You can download the current version of TNCremo free of charge from the HEIDENHAIN Filebase ([www.heidenhain.de](http://www.heidenhain.de), <Services and Documentation>, <Software>, <PC Software>, <TNCremoNT>).

System requirements for TNCremo:

- PC with 486 processor or higher
- Windows 95, Windows 98, Windows NT 4.0, Windows 2000, Windows XP or Windows Vista operating systems
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

### Installation under Windows

- ▶ Start the SETUP.EXE installation program with the file manager (Explorer)
- ▶ Follow the setup program instructions

### Starting TNCremoNT under Windows

- ▶ Click <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremo>

When you start TNCremo for the first time, TNCremo automatically tries to set up a connection with the TNC.



## Data transfer between the TNC and TNCremoNT



Before you transfer a program from the TNC to the PC, you must make absolutely sure that you have already saved the program currently selected on the TNC. The TNC saves changes automatically when you switch the mode of operation on the TNC, or when you select the file manager via the PGM MGT key.

Check whether the TNC is connected to the correct serial port on your PC or to the network.

Once you have started TNCremoNT, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using the menu items <File> and <Change directory>, you can change the active directory or select another directory on your PC.

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

- ▶ Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**
- ▶ To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**
- ▶ To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**

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

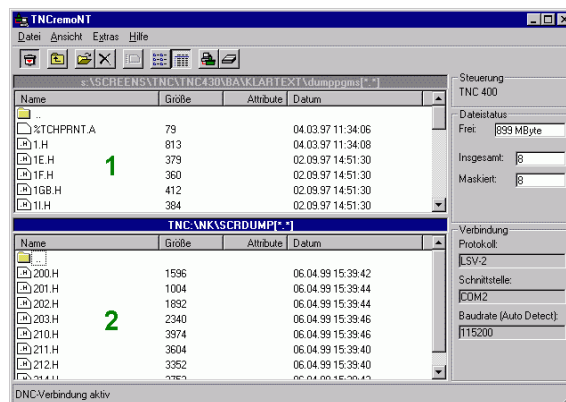
- ▶ Select <Extras>, <TNCserver>. TNCremoNT is now in server mode. It can receive data from the TNC and send data to the TNC
- ▶ You can now call the file management functions on the TNC by pressing the PGM MGT key (see "Data transfer to or from an external data medium" on page 114) and transfer the desired files

### Exiting TNCremoNT

Select the menu items <File>, <Exit>



Refer also to the TNCremoNT context-sensitive help texts where all of the functions are explained in more detail. The help texts must be called with the F1 key.



## 14.5 Ethernet interface

### Introduction

The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data via the Ethernet card with

- the **smb** protocol (**s**erver **m**essage **b**lock) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with support from the NFS (Network File System)

### Connection possibilities

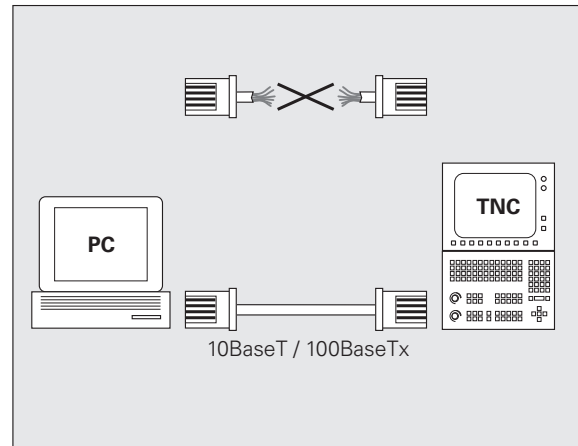
You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX or 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.



The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (100BaseTX or 10BaseT).

No great effort is required to connect the TNC directly to a PC that has an Ethernet card. Simply connect the TNC (port X26) and the PC with an Ethernet crossover cable (trade names: crossed patch cable or STP cable).



Connecting the control to the network



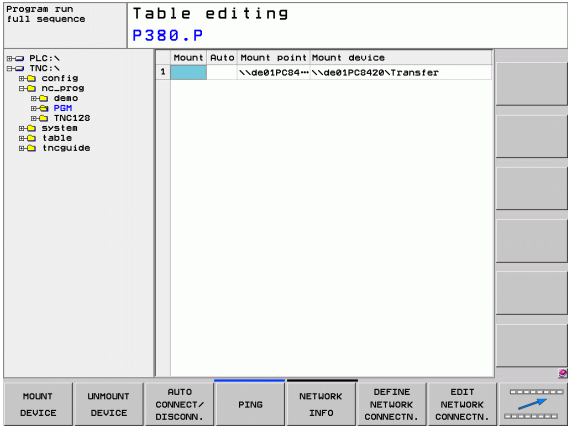
Make sure that the person configuring your TNC is a network specialist.

Please note that the TNC performs an automatic restart if you change the IP address of the TNC.

Overview of functions for configuring the network

► In the file manager (PGM MGT), press the **Network** soft key

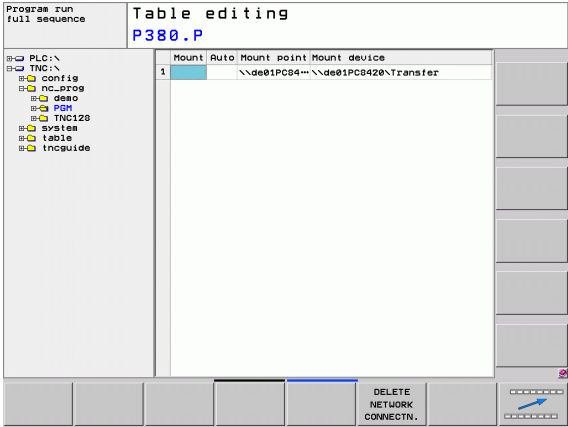
Function	Soft key
Establish the connection to the selected network drive. Successful connection is indicated by a check mark under Mount.	MOUNT DEVICE
Separate the connection to a network drive.	UNMOUNT DEVICE
Activate or deactivate the Automount function (= automatic connection of the network drive during control start-up). The status of the function is indicated by a check mark under Auto in the network drive table.	AUTO MOUNT
Use the ping function to check whether a connection to a particular remote station in the network is available. The address is entered as four decimal numbers separated by points (dotted-decimal notation).	PING
The TNC displays an overview window with information on the active network connections.	NETWORK INFO
Configure the access to network drives. (Selectable only after entry of the MOD code number NET123)	DEFINE NETWORK CONNECTN.
Open the dialog window for editing the data of an existing network connection. (Selectable only after entry of the MOD code number NET123)	EDIT NETWORK CONNECTN.
Configure the network address of the control. (Selectable only after entry of the MOD code number NET123)	CONFIGURE NETWORK
Delete an existing network connection. (Selectable only after entry of the MOD code number NET123)	DELETE NETWORK CONNECTN.



### Configuring the control's network address

- ▶ Connect the TNC (port X26) with a network or a PC
- ▶ In the file manager (PGM MGT), select the **Network** soft key
- ▶ Press the MOD key. Then enter the code number **NET123**
- ▶ Press the **CONFIGURE NETWORK** soft key to enter the network setting for a specific device (see figure at center right)
- ▶ The dialog window for the network configuration opens

Setting	Meaning
HOSTNAME	Name under which the control logs onto the network. If you use a host-name server, you must enter the "Fully Qualified Host Name" here. If you do not enter a name here, the control uses the so-called null authentication.
DHCP	DHCP = Dynamic Host Configuration Protocol In the drop-down menu, select <b>YES</b> , then the control automatically obtains its network address (IP address), the subnet mask, the default router and any required broadcast address from a DHCP server in the network. The DHCP server identifies the control by its host name. Your company network must be specially prepared for this function. Contact your network administrator.
IP ADDRESS	Network address of the control: In each of the four adjacent input fields you can enter three digits of the IP address. To move to the next field, press the ENT key. Your network specialist can give you a network address for the control.
SUBNET MASK	Serves to differentiate between the network ID and the host ID of the network: Your network specialist assigns the subnet mask of the control.



Setting	Meaning
BROADCAST	The broadcast address of the control is needed only if it is different from the standard setting. The standard setting is formed from the net and host ID, in which all bits are set to 1.
ROUTER	Network address of default router: This entry is required only if your network consists of several subnetworks interconnected by routers.

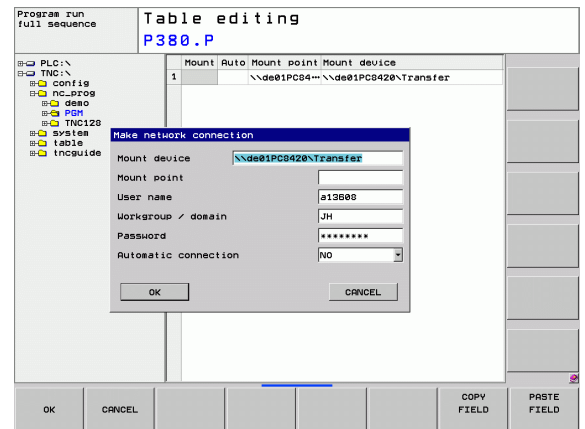


The entered network configuration does not become effective until the control is rebooted. After the network configuration is concluded with the OK button or soft key, the control asks for confirmation and reboots.

## Configuring network access to other devices (mount)

- ▶ Connect the TNC (port X26) with a network or a PC
- ▶ In the file manager (PGM MGT), select the **Network** soft key
- ▶ Press the MOD key. Then enter the code number **NET123**
- ▶ Press the **DEFINE NETWORK CONNECTN.** soft key
- ▶ The dialog window for the network configuration opens

Setting	Meaning
Mount device	<ul style="list-style-type: none"> <li>■ Connection over NFS: Directory name to be mounted. This is formed from the network address of the device, a colon, a slash and the name of the directory. Entry of the network address as four decimal numbers separated by points (dotted-decimal notation), e.g. 160.1.180.4/PC. When entering the path name, pay attention to capitalization.</li> <li>■ To connect individual Windows computers via SMB: Enter the network name and the share name of the computer, e.g. \\PC1791NT\PC</li> </ul>
Mount point	Device name: The device name entered here is displayed on the control in the program management for the mounted network, e.g. WORLD: (The name must end with a colon!)
File system	File system type: <ul style="list-style-type: none"> <li>■ NFS: Network File System</li> <li>■ SMB: Windows network</li> </ul>



Setting	Meaning
NFS option	<p><b>rsize</b>: Packet size in bytes for data reception</p> <p><b>wsiz</b>e: Packet size for data transmission in bytes</p> <p><b>time0</b>=: Time in tenths of a second, after which the control repeats an unanswered Remote Procedure Call.</p> <p><b>soft</b>: If <b>YES</b> is entered, the Remote Procedure Call is repeated until the NFS server answers. If <b>NO</b> is entered, it is not repeated.</p>
SMB option	<p>Options that concern the SMB file system type: Options are given without space characters, separated only by commas. Pay attention to capitalization.</p> <p>Options:</p> <p><b>ip</b>: IP address of the Windows PC to which the control is to be connected</p> <p><b>username</b>: User name with which the control should log in</p> <p><b>workgroup</b>: Workgroup under which the control should log in</p> <p><b>password</b>: Password with which the control is to log on (up to 80 characters)</p> <p>Further SMB options: Input of further options for the Windows network</p>
Automatic connection	<p>Automount (YES or NO): Here you specify whether the network will be automatically mounted when the control starts up. Devices that are not automatically mounted can be mounted anytime in the program management.</p>



You do not need to indicate the protocol with the TNC 128. It uses the communications protocol according to RFC 894.



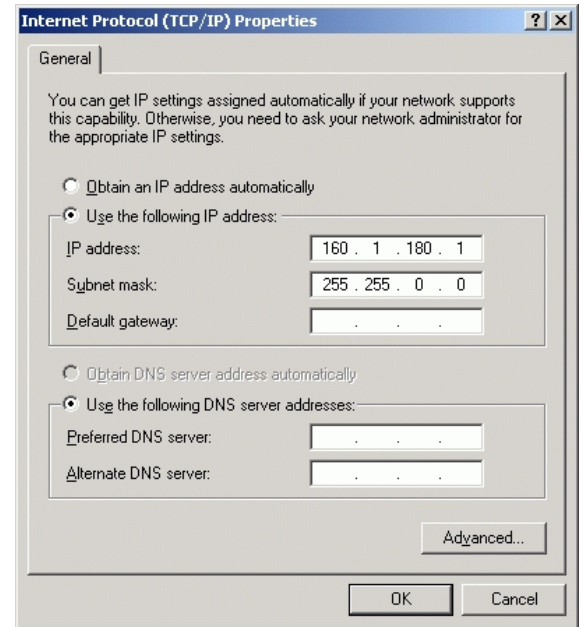
## Settings on a PC with Windows 2000

**Prerequisite:**

The network card must already be installed on the PC and ready for operation.

If the PC that you want to connect the TNC to is already integrated in your company network, then keep the PC's network address and adapt the TNC's network address accordingly.

- ▶ To open Network Connections, click <Start>, <Control Panel>, <Network and Dial-up Connections>, and then Network Connections
- ▶ Right-click the <LAN connection> symbol, and then <Properties> in the menu that appears
- ▶ Double-click <Internet Protocol (TCP/IP)> to change the IP settings (see figure at top right)
- ▶ If it is not yet active, select the <Use the following IP address> option
- ▶ In the <IP address> input field, enter the same IP address that you entered for the PC network settings on the iTNC, e.g. 160.1.180.1
- ▶ Enter 255.255.0.0 in the <Subnet mask> input field
- ▶ Confirm the settings with <OK>
- ▶ Save the network configuration with <OK>. You may have to restart Windows now





# 14.6 Position display types

## Application

In the Manual Operation mode and in the Program Run modes of operation, you can select the type of coordinates to be displayed.

The figure at right shows the different tool positions:

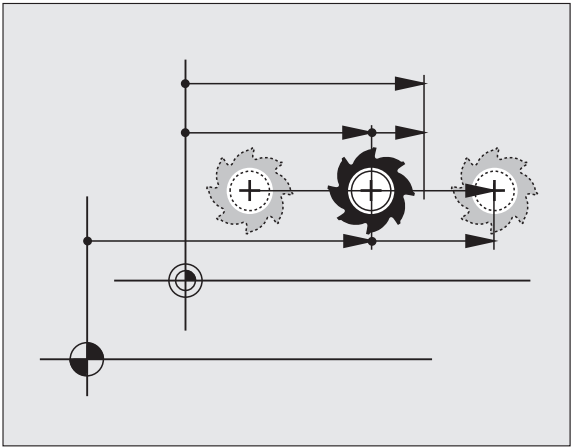
- Starting position
- Target position of the tool
- Workpiece datum
- Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	NOML.
Actual position; current tool position	ACTL.
Reference position; the actual position relative to the machine datum	REF ACTL
Reference position; the nominal position relative to the machine datum	REF NOML
Servo lag; difference between nominal and actual positions (following error)	LAG
Distance remaining to the programmed position; difference between actual and target positions	DIST

With the MOD function **Position display 1**, you can select the position display in the status display.

With the MOD function **Position display 2**, you can select the position display in the status display.



## 14.7 Unit of measurement

### Application

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm), set the Change MM/INCH function to mm. The value is displayed to 3 decimal places.
- To select the inch system (e.g. X = 0.6216 inches), set the Change MM/INCH function to inches. The value is displayed to 4 decimal places.

If you would like to activate the inch display, the TNC shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.



# 14.8 Displaying operating times

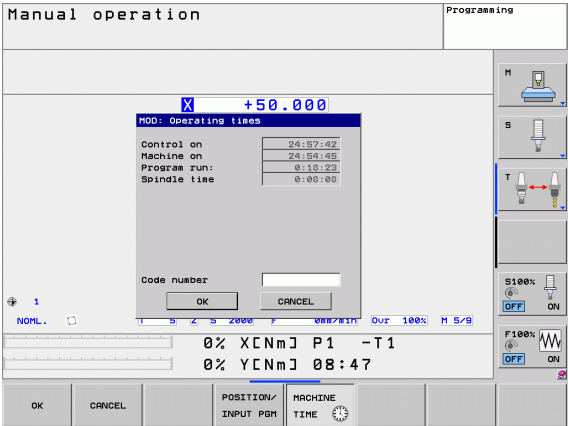
## Application

The MACHINE TIME soft key enables you to see various types of operating times:

Operating time	Meaning
Control on	Operating time of the control since being put into service
Machine on	Operating time of the machine tool since being put into service
Program run	Duration of controlled operation since being put into service



The machine tool builder can provide further operating time displays. The machine manual provides further information.





# 15

**Fundamentals of cycles**



## 15.1 Introduction

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and several special functions are also available as cycles.

Most cycles use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, **Q200** is always assigned the set-up clearance, **Q202** the plunging depth, etc.



### Danger of collision!

Cycles sometimes execute extensive operations. For safety reasons, you should run a graphical program test before machining.



If you use indirect parameter assignments in cycles with numbers greater than 200 (e.g. **Q210 = Q1**), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. **Q210**) directly in such cases.

If you define a feed-rate parameter for fixed cycles greater than 200, then instead of entering a numerical value you can use soft keys to assign the feed rate defined in the **TOOL CALL** block (FAUTO soft key). You can also use the feed-rate alternatives **FMAX** (rapid traverse), **FZ** (feed per tooth) and **FU** (feed per rev), depending on the respective cycle and the function of the feed-rate parameter.

Note that, after a cycle definition, a change of the **FAUTO** feed rate has no effect, because internally the TNC assigns the feed rate from the **TOOL CALL** block when processing the cycle definition.

If you want to delete a block that is part of a cycle, the TNC asks you whether you want to delete the whole cycle.

# 15.2 Available cycle groups

## Overview of fixed cycles



► The soft-key row shows the available groups of cycles

Cycle group	Soft key	Page
Cycles for pecking, reaming, boring, counterboring, and tapping	DRILLING/ THREAD	Page 386
Cycles for milling rectangular pockets and studs	POCKETS/ STUDS/ SLOTS	Page 426
Coordinate transformation cycles for shifting, mirroring, enlarging and reducing contours	COORD. TRANSF.	Page 442
Cycles for creating point patterns	PATTERN	Page 376
Special cycles such as dwell time, program call and oriented spindle stop	SPECIAL CYCLES	Page 458
Cycles for automatic tool measurement (enabled by the machine tool builder)		Page 469



► If required, switch to machine-specific fixed cycles.  
These fixed cycles can be integrated by your machine tool builder.



## 15.3 Working with fixed cycles

### Machine-specific cycles

In addition to the HEIDENHAIN cycles, many machine tool builders offer their own cycles in the TNC. These cycles are available in a separate cycle-number range:

- Cycles 300 to 399  
Machine-specific cycles that are to be defined through the CYCLE DEF key
- Cycles 500 to 599  
Machine-specific touch probe cycles that are to be defined through the TOUCH PROBE key



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

Sometimes machine-specific cycles use transfer parameters that HEIDENHAIN already uses in standard cycles. The TNC executes DEF-active cycles as soon as they are defined (see also "Calling cycles" on page 366). It executes CALL-active cycles only after they have been called (see also "Calling cycles" on page 366). When DEF-active cycles and CALL-active cycles are used simultaneously, it is important to prevent overwriting of transfer parameters already in use. Use the following procedure:

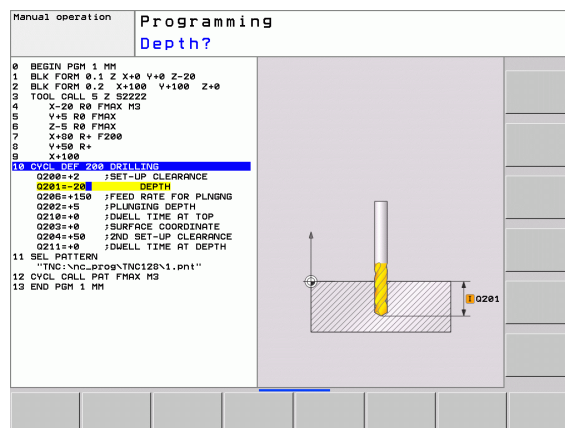
- As a rule, always program DEF-active cycles before CALL-active cycles
- Only program a DEF-active cycle between the definition and call of a CALL-active cycle if there is no common use of specific transfer parameters



## Defining a cycle using soft keys

CYCL  
DEFDRILLING/  
THREAD262  


- ▶ The soft-key row shows the available groups of cycles
- ▶ Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles
- ▶ Select the desired cycle, e.g. DRILLING. The TNC initiates the programming dialog and requests all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is requested in the dialog prompt is highlighted
- ▶ Enter all parameters requested by the TNC and conclude each entry with the ENT key
- ▶ The TNC ends the dialog when all required data has been entered



## Defining a cycle using the GOTO function

CYCL  
DEFGOTO  


- ▶ The soft-key row shows the available groups of cycles
- ▶ The TNC shows an overview of cycles in a pop-up window
- ▶ Choose the desired cycle with the arrow keys, or
- ▶ Enter the cycle number and confirm it with the ENT key. The TNC then initiates the cycle dialog as described above

### Example NC blocks

7 CYCL DEF 200 DRILLING

Q200=2 ;SET-UP CLEARANCE

Q201=3 ;DEPTH

Q206=150 ;FEED RATE FOR PLNGNG

Q202=5 ;PLUNGING DEPTH

Q210=0 ;DWELL TIME AT TOP

Q203=+0 ;SURFACE COORDINATE

Q204=50 ;2ND SET-UP CLEARANCE

Q211=0.25 ;DWELL TIME AT DEPTH



## Calling cycles



### Prerequisites

The following data must always be programmed before a cycle call:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF)

For some cycles, additional prerequisites must be observed. They are detailed in the descriptions for each cycle.

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:

- Coordinate transformation cycles
- Cycle 9 DWELL TIME
- All touch probe cycles

You can call all other cycles with the functions described as follows.

### Calling a cycle with CYCL CALL

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



- ▶ To program the cycle call, press the CYCL CALL key
- ▶ Press the CYCL CALL M soft key to enter a cycle call
- ▶ If necessary, enter the miscellaneous function M (for example **M3** to switch the spindle on), or end the dialog by pressing the END key

### Calling a cycle with CYCL CALL PAT

The **CYCL CALL PAT** function calls the most recently defined fixed cycle at all positions that you defined in a PATTERN DEF pattern definition (see "PATTERN DEF pattern definition" on page 368) or in a point table (see "Point tables" on page 381).

### Calling a cycle with M99/89

The **M99** function, which is active only in the block in which it is programmed, calls the last defined fixed cycle once. You can program **M99** at the end of a positioning block. The TNC moves to this position and then calls the last defined fixed cycle.

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

To cancel the effect of **M89**, program:

- **M99** in the positioning block in which you move to the last starting point, or
- Use **CYCL DEF** to define a new fixed cycle



# 15.4 PATTERN DEF pattern definition

## Application

You use the **PATTERN DEF** function to easily define regular machining patterns, which you can call with the **CYCL CALL PAT** function. As in the cycle definitions, support graphics that illustrate the respective input parameter are also available for pattern definitions.



**PATTERN DEF** is to be used only in connection with the tool axis Z.

The following machining patterns are available:

Machining pattern	Soft key	Page
POINT Definition of up to any 9 machining positions		Page 370
ROW Definition of a single row, straight or rotated		Page 371
PATTERN Definition of a single pattern, straight, rotated or distorted		Page 372
FRAME Definition of a single frame, straight, rotated or distorted		Page 373
CIRCLE Definition of a full circle		Page 374
PITCH CIRCLE Definition of a pitch circle		Page 375



## Entering PATTERN DEF



► Select the Programming and Editing operating mode



► Press the special functions key



► Select the functions for contour and point machining



► Open a **PATTERN DEF** block



► Select the desired machining pattern, e.g. a single row

► Enter the required definitions, and confirm each entry with the ENT key

## Using PATTERN DEF

As soon as you have entered a pattern definition, you can call it with the **CYCL CALL PAT** function (see "Calling a cycle with CYCL CALL PAT" on page 367). The TNC then performs the most recently defined machining cycle on the machining pattern you defined.



A machining pattern remains active until you define a new one, or select a point table with the **SEL PATTERN** function.

You can use the mid-program startup function to select any point at which you want to start or continue machining (see User's Manual, Test Run and Program Run sections).



Defining individual machining positions



You can enter up to 9 machining positions. Confirm each entry with the ENT key.

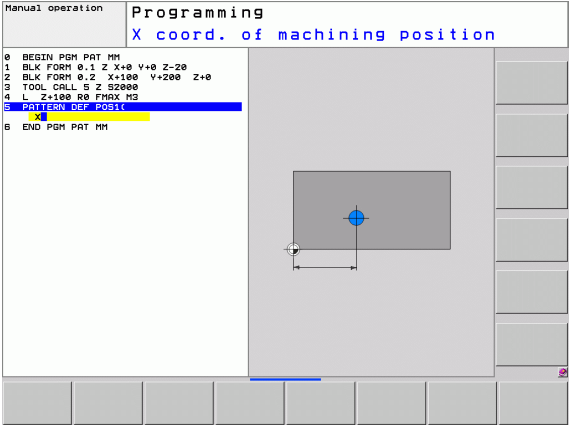
If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- ▶ **X coord. of machining position** (absolute): Enter X coordinate
- ▶ **Y coord. of machining position** (absolute): Enter Y coordinate
- ▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

```
10 Z+100 R0 FMAX
11 PATTERN DEF
POS1 (X+25 Y+33.5 Z+0)
POS2 (X+50 Y+75 Z+0)
```



# Defining a single row



If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



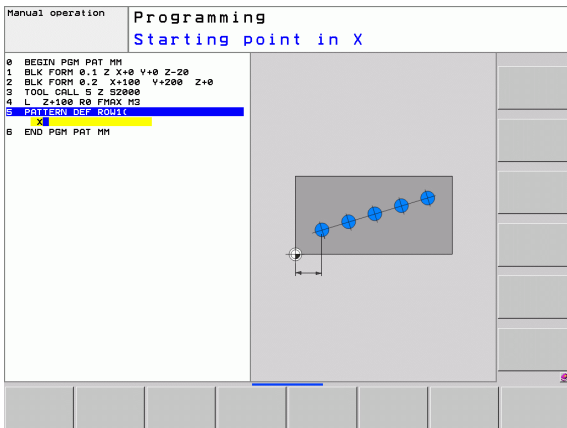
- **Starting point in X** (absolute): Coordinate of the starting point of the row in the X axis
- **Starting point in Y** (absolute): Coordinate of the starting point of the row in the Y axis
- **Spacing of machining positions (incremental)**: Distance between the machining positions. You can enter a positive or negative value
- **Number of repetitions**: Total number of machining operations
- **Rot. position of entire pattern (absolute)**: Angle of rotation around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

## Example: NC blocks

10 Z+100 R0 FMAX

11 PATTERN DEF

ROW1 (X+25 Y+33.5 D+8 NUM5 ROT+0 Z+0)



Defining a single pattern



If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

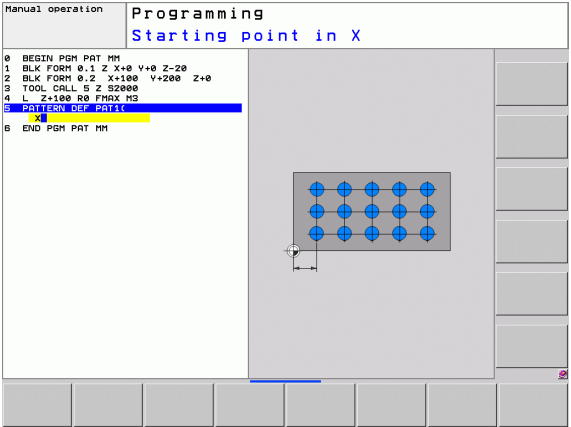
The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotated position** of the entire pattern.



- ▶ **Starting point in X** (absolute): Coordinate of the starting point of the pattern in the X axis
- ▶ **Starting point in Y** (absolute): Coordinate of the starting point of the pattern in the Y axis
- ▶ **Spacing of machining positions X (incremental)**: Distance between the machining positions in the X direction. You can enter a positive or negative value
- ▶ **Spacing of machining positions Y (incremental)**: Distance between the machining positions in the Y direction. You can enter a positive or negative value
- ▶ **Number of columns**: Total number of columns in the pattern
- ▶ **Number of lines**: Total number of rows in the pattern
- ▶ **Rot. position of entire pattern (absolute)**: Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- ▶ **Rotary pos. ref. ax.:** Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value
- ▶ **Rotary pos. minor ax.:** Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value
- ▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

```
10 Z+100 R0 FMAX
11 PATTERN DEF
PAT1 (X+25 Y+33.5 DX+8 DY+10 NUMX5
NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0)
```





## Defining individual frames



If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.

The **Rotary pos. ref. ax.** and **Rotary pos. minor ax.** parameters are added to a previously performed **rotated position** of the entire pattern.



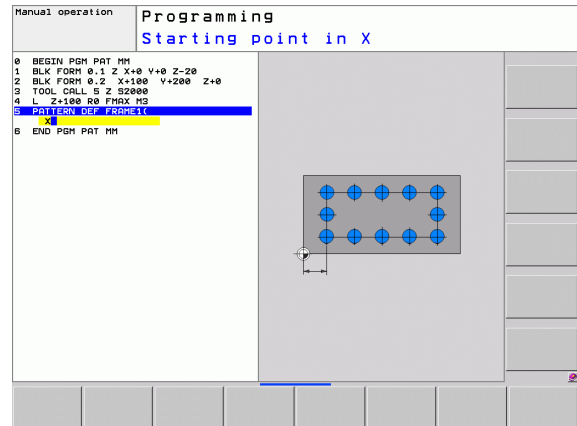
- ▶ **Starting point in X (absolute):** Coordinate of the starting point of the frame in the X axis
- ▶ **Starting point in Y (absolute):** Coordinate of the starting point of the frame in the Y axis
- ▶ **Spacing of machining positions X (incremental):** Distance between the machining positions in the X direction. You can enter a positive or negative value
- ▶ **Spacing of machining positions Y (incremental):** Distance between the machining positions in the Y direction. You can enter a positive or negative value
- ▶ **Number of columns:** Total number of columns in the pattern
- ▶ **Number of lines:** Total number of rows in the pattern
- ▶ **Rot. position of entire pattern (absolute):** Angle of rotation by which the entire pattern is rotated around the entered starting point. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- ▶ **Rotary pos. ref. ax.:** Angle of rotation around which only the reference axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value
- ▶ **Rotary pos. minor ax.:** Angle of rotation around which only the minor axis of the machining plane is distorted with respect to the entered starting point. You can enter a positive or negative value
- ▶ **Coordinate of workpiece surface (absolute):** Enter Z coordinate at which machining is to begin

### Example: NC blocks

```
10 Z+100 R0 FMAX
```

```
11 PATTERN DEF
```

```
FRAME1 (X+25 Y+33.5 DX+8 DY+10 NUMX5  
NUMY4 ROT+0 ROTX+0 ROTY+0 Z+0)
```



Defining a full circle



If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- ▶ **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- ▶ **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- ▶ **Number of repetitions**: Total number of machining positions on the circle
- ▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

Example: NC blocks

```
10 Z+100 R0 FMAX
11 PATTERN DEF
CIRC1 (X+25 Y+33 D80 START+45 NUM8 Z+0)
```

Manual operation

Programming

Bolt-hole circle center X

0 BEGIN PGM PAT MM

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

2 BLK FORM 0.2 X+100 Y+200 Z+0

3 TOOL CALL 5 Z S2000

4 L Z+100 R0 FMAX

5 PATTERN DEF CIRC1(

6 END PGM PAT MM



# Defining a pitch circle

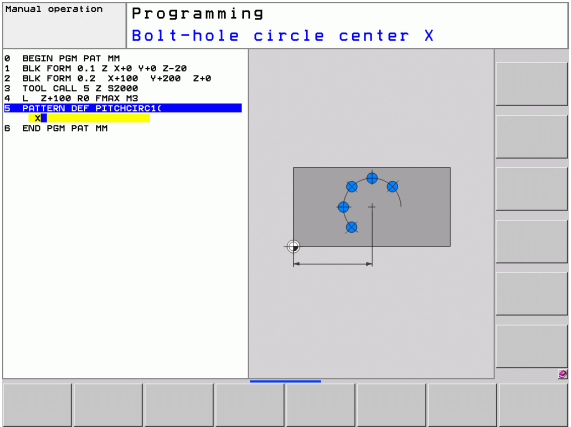
If you have defined a **workpiece surface in Z** not equal to 0, then this value is effective in addition to the workpiece surface **Q203** that you defined in the machining cycle.



- ▶ **Bolt-hole circle center X** (absolute): Coordinate of the circle center in the X axis
- ▶ **Bolt-hole circle center Y** (absolute): Coordinate of the circle center in the Y axis
- ▶ **Bolt-hole circle diameter**: Diameter of the bolt-hole circle
- ▶ **Starting angle**: Polar angle of the first machining position. Reference axis: Reference axis of the active machining plane (e.g. X for tool axis Z). You can enter a positive or negative value
- ▶ **Stepping angle/end angle**: Incremental polar angle between two machining positions. You can enter a positive or negative value. As an alternative you can enter the end angle (switch via soft key)
- ▶ **Number of repetitions**: Total number of machining positions on the circle
- ▶ **Coordinate of workpiece surface** (absolute): Enter Z coordinate at which machining is to begin

## Example: NC blocks

```
10 Z+100 R0 FMAX
11 PATTERN DEF
PITCHCIRC1 (X+25 Y+33 D80 START+45 STEP30
NUM8 Z+0)
```



## 15.5 POLAR PATTERN (Cycle 220)

### Cycle run

- 1 The TNC moves the tool at rapid traverse from its current position to the starting point for the first machining operation.

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
  - Approach the starting point in the machining plane
  - Move to the set-up clearance above the workpiece surface (spindle axis)
- 2 From this position the TNC executes the last defined fixed cycle
  - 3 The tool then approaches on a straight line or circular arc the starting point for the next machining operation. The tool stops at the set-up clearance (or the 2nd set-up clearance)
  - 4 This process (1 to 3) is repeated until all machining operations have been executed

### Please note while programming:



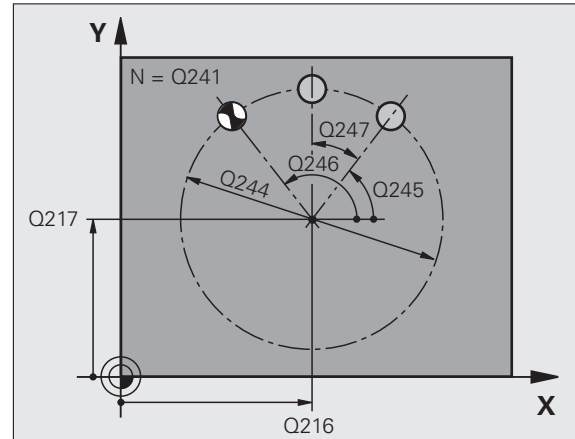
Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

If you combine Cycle 220 with one of the fixed cycles, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 220 will be effective for the selected fixed cycle.

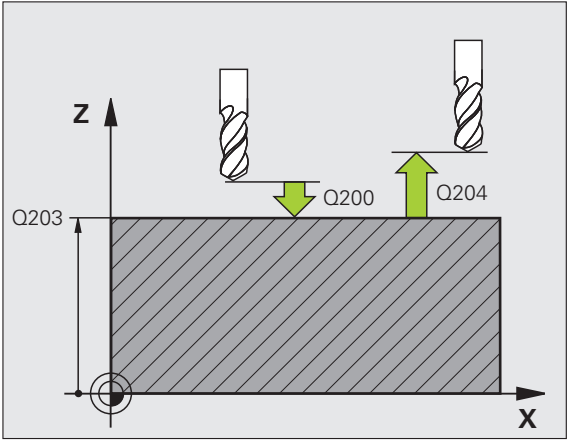
## Cycle parameters



- ▶ **Center in 1st axis** Q216 (absolute): Center of the pitch circle in the reference axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Center in 2nd axis** Q217 (absolute): Center of the pitch circle in the minor axis of the working plane. Input range -99999.9999 to 99999.9999
- ▶ **Pitch circle diameter** Q244: Diameter of the pitch circle. Input range 0 to 99999.9999
- ▶ **Starting angle** Q245 (absolute): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle. Input range -360.000 to 360.000
- ▶ **Stopping angle** Q246 (absolute): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to full circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise. Input range -360.000 to 360.000
- ▶ **Stepping angle** Q247 (incremental): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the TNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the angle step determines the working direction (negative = clockwise). Input range -360.000 to 360.000
- ▶ **Number of repetitions** Q241: Number of machining operations on a pitch circle. Input range 1 to 99999



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Move to clearance height** Q301: Definition of how the tool is to move between machining processes:  
**0**: Move to the set-up clearance between operations  
**1**: Move to the 2nd set-up clearance between machining operations



Example: NC blocks

53	CYCLE DEF 220 POLAR PATTERN
Q216	=+50 ;CENTER IN 1ST AXIS
Q217	=+50 ;CENTER IN 2ND AXIS
Q244	=80 ;PITCH CIRCLE DIA.
Q245	=+0 ;STARTING ANGLE
Q246	=+360 ;STOPPING ANGLE
Q247	=+0 ;STEPPING ANGLE
Q241	=8 ;NUMBER OF OPERATIONS
Q200	=2 ;SET-UP CLEARANCE
Q203	=+30 ;SURFACE COORDINATE
Q204	=50 ;2ND SET-UP CLEARANCE
Q301	=1 ;MOVE TO CLEARANCE



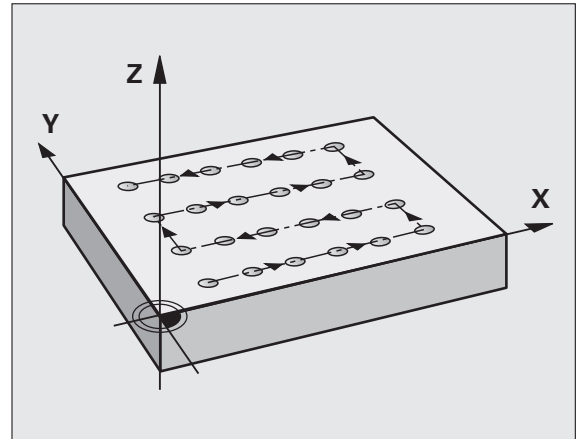
## 15.6 CARTESIAN PATTERN (Cycle 221)

### Cycle run

- 1 The TNC automatically moves the tool from its current position to the point of the first machining operation

Sequence:

- Move to the 2nd set-up clearance (spindle axis)
  - Approach the starting point in the machining plane
  - Move to the set-up clearance above the workpiece surface (spindle axis)
- 2 From this position the TNC executes the last defined fixed cycle
  - 3 The tool then approaches the starting point of the next machining operation in the positive reference axis direction at the set-up clearance (or the 2nd set-up clearance)
  - 4 This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line
  - 5 The tool subsequently moves to the last point on the second line where it executes the machining operation
  - 6 From this position the tool approaches the starting point of the next machining operation in the negative reference axis direction
  - 7 This process (6) is repeated until all machining operations in the second line have been executed
  - 8 The tool then moves to the starting point of the next line
  - 9 All subsequent lines are machined in a reciprocating movement



### Please note while programming:



Cycle 221 is DEF active, which means that Cycle 221 automatically calls the last defined fixed cycle.

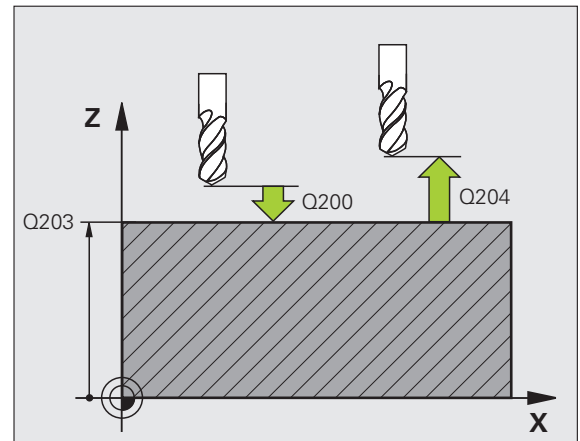
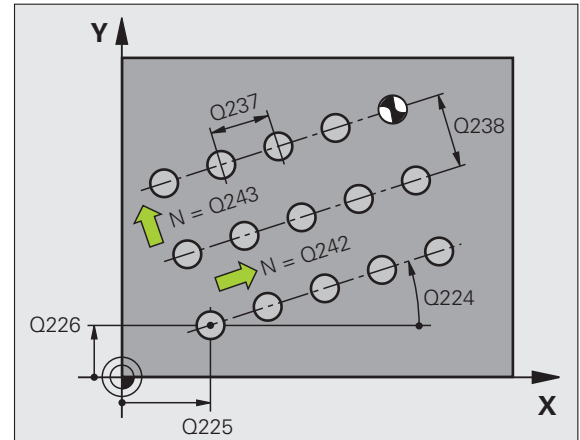
If you combine Cycle 221 with one of the fixed cycles, the set-up clearance, workpiece surface, 2nd set-up clearance and angle of rotation that you defined in Cycle 221 will be effective for the selected fixed cycle.

The slot position 0 is not allowed if you use Cycle 254 Circular Slot in combination with Cycle 221.

## Cycle parameters



- ▶ **Starting point 1st axis** Q225 (absolute): Coordinate of the starting point in the reference axis of the machining plane
- ▶ **Starting point 2nd axis** Q226 (absolute): Coordinate of the starting point in the minor axis of the machining plane
- ▶ **Spacing in 1st axis** Q237 (incremental): Spacing between each point on a line
- ▶ **Spacing in 2nd axis** Q238 (incremental): Spacing between each line
- ▶ **Number of columns** Q242: Number of machining operations on a line
- ▶ **Number of lines** Q243: Number of lines
- ▶ **Angle of rotation** Q224 (absolute): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point
- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur
- ▶ **Move to clearance height** Q301: Definition of how the tool is to move between machining processes:  
**0:** Move to the set-up clearance between operations  
**1:** Move to the 2nd set-up clearance between machining operations



## Example: NC blocks

```
54 CYCL DEF 221 CARTESIAN PATTERN
```

```
Q225=+15 ;STARTING POINT 1ST AXIS
```

```
Q226=+15 ;STARTING POINT 2ND AXIS
```

```
Q237=+10 ;SPACING IN 1ST AXIS
```

```
Q238=+8 ;SPACING IN 2ND AXIS
```

```
Q242=6 ;NUMBER OF COLUMNS
```

```
Q243=4 ;NUMBER OF LINES
```

```
Q224=+15 ;ANGLE OF ROTATION
```

```
Q200=2 ;SET-UP CLEARANCE
```

```
Q203=+30 ;SURFACE COORDINATE
```

```
Q204=50 ;2ND SET-UP CLEARANCE
```

```
Q301=1 ;MOVE TO CLEARANCE
```



# 15.7 Point tables

## Application

You should create a point table (\*.pnt) whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

## Creating a point table

Select the **Programming and Editing** mode of operation.

PGM  
MGT

Press the PGM MGT key to call the file manager

FILE NAME?

ENT

Enter the name and file type of the point table and confirm your entry with the ENT key


MM

Select the unit of measure: Press the MM or INCH soft key. The TNC switches to the program blocks window and displays an empty point table

INSERT  
LINE

With the soft key INSERT LINE, insert new lines and enter the coordinates of the desired machining position

Repeat the process until all desired coordinates have been entered



The name of the point table must begin with a letter.

Use the soft keys X OFF/ON, Y OFF/ON, Z OFF/ON (second soft-key row) to specify which coordinates you want to enter in the point table.



## Hiding single points from 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.



In the table, select the point to be hidden



Select the FADE column



Activate hiding, or



Deactivate hiding

## Selecting a point table in the program

In the Programming and Editing mode of operation, select the program for which you want to activate the point table:



Press the PGM CALL key to call the function for selecting the point table



Press the POINT TABLE soft key

Enter the name of the point table and confirm your entry with the END key. If the point table is not stored in the same directory as the NC program, you must enter the complete path

### Example NC block

```
7 SEL PATTERN "TNC:\DIRKT5\NUST35.PNT"
```

## Calling a cycle in connection with point tables



With **CYCL CALL PAT** the TNC runs the point table that you last defined (even if you defined the point table in a program that was nested with **CALL PGM**).

If you want the TNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with **CYCLE CALL PAT**:



- ▶ To program the cycle call, press the CYCL CALL key
- ▶ Press the CYCL CALL PAT soft key to call a point table
- ▶ Enter the feed rate at which the TNC is to move from point to point (if you make no entry the TNC will move at the last programmed feed rate; **FMAX** is not valid)
- ▶ If required, enter a miscellaneous function M, then confirm with the END key

The TNC retracts the tool to the safety clearance between the starting points. Depending on which is greater, the TNC uses either the spindle axis coordinate from the cycle call or the value from cycle parameter Q204 as the clearance height.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103.

### Effect of the point tables with Cycle 12

The TNC interprets the points as an additional datum shift.

### Effect of the point tables with Cycles 200 to 207

The TNC interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.

### Effect of the point tables with Cycles 251 to 256

The TNC interprets the points of the working plane as coordinates of the cycle starting point. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0.





# 16











**Drilling, boring and  
thread cycles**



# 16.1 Fundamentals

## Overview

The TNC offers 10 cycles for all types of drilling, boring and threading operations:

Cycle	Soft key	Page
240 CENTERING With automatic pre-positioning, 2nd set-up clearance, optional entry of the centering diameter or centering depth		Page 387
200 DRILLING With automatic pre-positioning, 2nd set-up clearance		Page 389
201 REAMING With automatic pre-positioning, 2nd set-up clearance		Page 391
202 BORING With automatic pre-positioning, 2nd set-up clearance		Page 393
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing		Page 397
204 BACK BORING With automatic pre-positioning, 2nd set-up clearance		Page 401
205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance		Page 405
241 SINGLE-LIP DEEP-HOLE DRILLING With automatic pre-positioning to deepened starting point, shaft speed and coolant definition		Page 409
206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		Page 416
207 RIGID TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance		Page 418



## 16.2 CENTERING (Cycle 240)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
- 2 The tool is centered at the programmed feed rate **F** to the entered centering diameter or centering depth.
- 3 If defined, the tool remains at the centering depth.
- 4 Finally, the tool moves to set-up clearance or—if programmed—to the 2nd set-up clearance at rapid traverse **FMAX**.

### Please note while programming:



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter **Q344** (diameter) or **Q201** (depth) determines the working direction. If you program the diameter or depth = 0, the cycle will not be executed.



#### **Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

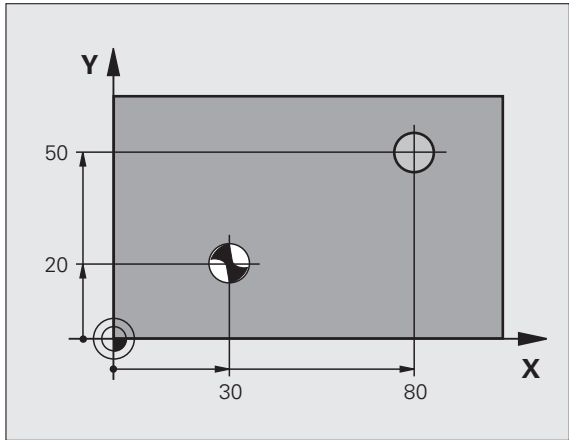
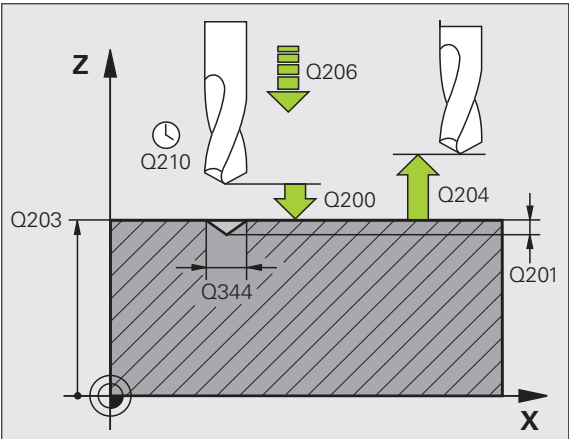
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive diameter or depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!



Cycle parameters



- ▶ **Set-up clearance Q200** (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- ▶ **Select depth/diameter (0/1) Q343**: Select whether centering is based on the entered diameter or depth. If the TNC is to center based on the entered diameter, the point angle of the tool must be defined in the **T-ANGLE** column of the tool table TOOL.T.  
**0**: Centering based on the entered depth  
**1**: Centering based on the entered diameter
- ▶ **Depth Q201** (incremental): Distance between workpiece surface and centering bottom (tip of centering taper). Only effective if Q343=0 is defined. Input range -99999.9999 to 99999.9999
- ▶ **Diameter (algebraic sign) Q344**: Centering diameter. Only effective if Q343=1 is defined. Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging Q206**: Traversing speed of the tool during centering in mm/min. Input range: 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Dwell time at depth Q211**: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Coordinate of workpiece surface Q203** (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance Q204** (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



Example: NC blocks

11	CYCL DEF 240 CENTERING
	Q200=2 ;SET-UP CLEARANCE
	Q343=1 ;SELECT DEPTH/DIA.
	Q201=+0 ;DEPTH
	Q344=-9 ;DIAMETER
	Q206=250 ;FEED RATE FOR PLNGNG
	Q211=0.1 ;DWELL TIME AT DEPTH
	Q203=+20 ;SURFACE COORDINATE
	Q204=100 ;2ND SET-UP CLEARANCE
12	X+30 R0 FMAX
13	Y+20 R0 FMAX M3 M99
14	X+80 R0 FMAX
15	Y+50 R0 FMAX M99





## 16.3 DRILLING (Cycle 200)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
- 2 The tool drills to the first plunging depth at the programmed feed rate **F**.
- 3 The TNC returns the tool at **FMAX** to the set-up clearance, dwells there (if a dwell time was entered), and then moves at **FMAX** to the set-up clearance above the first plunging depth.
- 4 The tool then advances with another infeed at the programmed feed rate **F**.
- 5 The TNC repeats this process (2 to 4) until the programmed depth is reached.
- 6 The tool is retracted from the hole bottom to the set-up clearance or—if programmed—to the 2nd set-up clearance at **FMAX**.

### Please note while programming:



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.



#### **Danger of collision!**

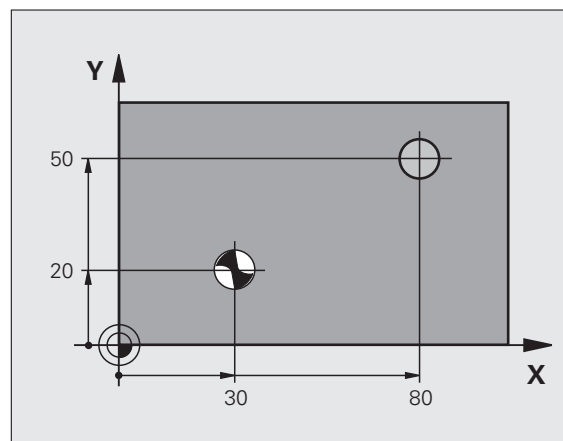
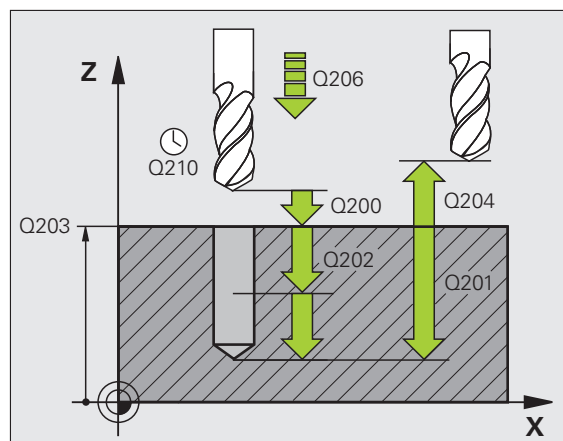
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

# Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Enter a positive value. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- ▶ **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000



## Example: NC blocks

```

11 CYCL DEF 200 DRILLING
    Q200=2 ;SET-UP CLEARANCE
    Q201=-15 ;DEPTH
    Q206=250 ;FEED RATE FOR PLNGNG
    Q202=5 ;PLUNGING DEPTH
    Q210=0 ;DWELL TIME AT TOP
    Q203=+20 ;SURFACE COORDINATE
    Q204=100 ;2ND SET-UP CLEARANCE
    Q211=0.1 ;DWELL TIME AT DEPTH
12 X+30 FMAX
13 Y+20 FMAX M3 M99
14 X+80 FMAX
15 Y+50 FMAX M99
    
```

## 16.4 REAMING (Cycle 201)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 The tool reams to the entered depth at the programmed feed rate **F**.
- 3 If programmed, the tool remains at the hole bottom for the entered dwell time.
- 4 The tool then retracts to the set-up clearance at the feed rate **F**, and from there—if programmed—to the 2nd set-up clearance at **FMAX**.

### Please note while programming:



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.



#### **Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

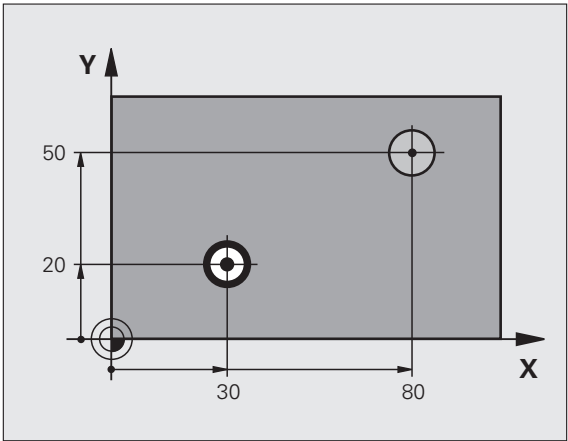
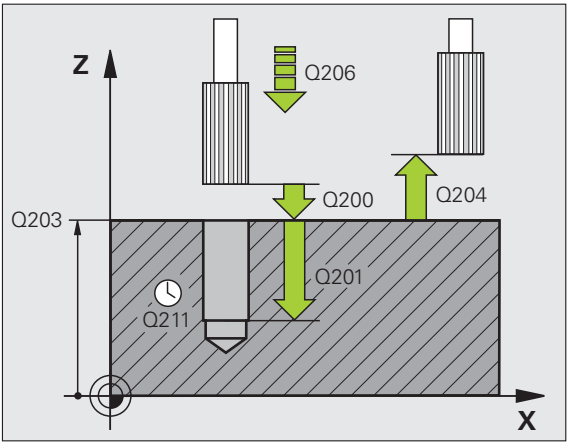
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!



Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool during reaming in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the reaming feed rate. Input range 0 to 99999.999
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range 0 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999



Example: NC blocks

11	CYCL DEF 201 REAMING
	Q200=2 ;SET-UP CLEARANCE
	Q201=-15 ;DEPTH
	Q206=100 ;FEED RATE FOR PLNGNG
	Q211=0.5 ;DWELL TIME AT DEPTH
	Q208=250 ;RETRACTION FEED RATE
	Q203=+20 ;SURFACE COORDINATE
	Q204=100 ;2ND SET-UP CLEARANCE
12	X+30 FMAX
13	Y+20 FMAX M3 M99
14	X+80 FMAX
15	Y+50 FMAX M99



## 16.5 BORING (Cycle 202)

### Cycle run

- 1** The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
- 2** The tool bores to the programmed depth at the feed rate for plunging.
- 3** If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
- 4** The TNC then orients the spindle to the position that is defined in parameter Q336.
- 5** If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
- 6** The TNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at **FMAX**. If Q214=0, the tool point remains on the wall of the hole.



**Please note while programming:**

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

**Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

Select a disengaging direction in which the tool moves away from the edge of the hole.

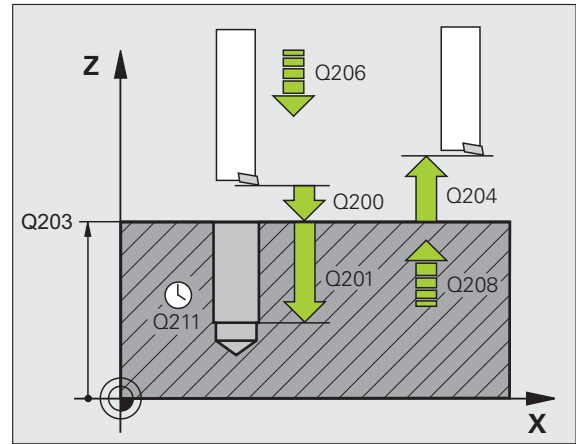
Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

During retraction the TNC automatically takes an active rotation of the coordinate system into account.

## Cycle parameters

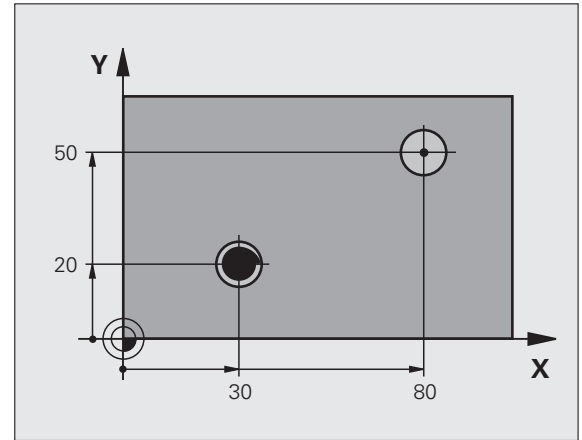


- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool during boring at mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter  $Q208 = 0$ , the tool retracts at feed rate for plunging. Input range 0 to 99999.999, alternatively **FMAX, FAUTO**
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.999



## 16.5 BORING (Cycle 202)

- **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation)
- 0** Do not retract tool
  - 1** Retract tool in the negative ref. axis direction
  - 2** Retract tool in the negative minor axis direction
  - 3** Retract tool in the positive ref. axis direction
  - 4** Retract tool in the positive minor axis direction
- **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before retracting it. Input range -360.000 to 360.000



**Example:**

```

10 Z+100 R0 FMAX
11 CYCL DEF 202 BORING
    Q200=2 ;SET-UP CLEARANCE
    Q201=-15 ;DEPTH
    Q206=100 ;FEED RATE FOR PLNGNG
    Q211=0.5 ;DWELL TIME AT DEPTH
    Q208=250 ;RETRACTION FEED RATE
    Q203=+20 ;SURFACE COORDINATE
    Q204=100 ;2ND SET-UP CLEARANCE
    Q214=1 ;DISENGAGING DIRECTN
    Q336=0 ;ANGLE OF SPINDLE
12 X+30 FMAX
12 Y+20 FMAX M3 M99
14 X+80 FMAX
14 Y+50 FMAX M99

```



## 16.6 UNIVERSAL DRILLING (Cycle 203)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 The tool drills to the first plunging depth at the programmed feed rate **F**.
- 3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at the retraction feed rate to the set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at **FMAX** to the set-up clearance above the first PLUNGING DEPTH.
- 4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
- 5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
- 6 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.



**Please note while programming:**

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

**Danger of collision!**

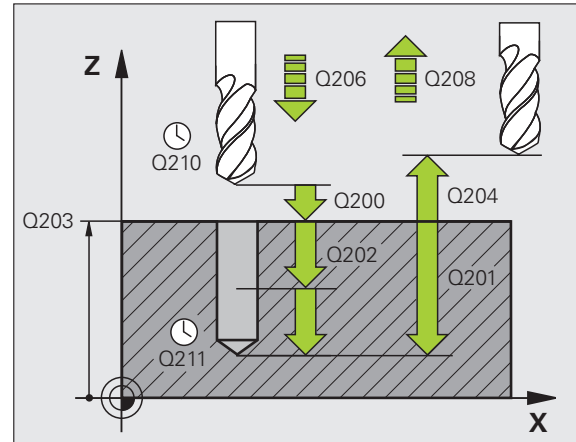
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

## Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth and no chip breaking is defined
- ▶ **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip removal. Input range 0 to 3600.0000
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202 after each infeed. Input range 0 to 99999.9999



- ▶ **No. of breaks before retracting** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip removal. For chip breaking, the TNC retracts the tool each time by the value in Q256. Input range 0 to 99999
- ▶ **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999
- ▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q206. Input range 0 to 99999.999, alternatively **FMAX, FAUTO**
- ▶ **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. Input range 0.1000 to 99999.9999

Example: NC blocks

11	CYCL DEF 203	UNIVERSAL DRILLING
Q200=2	;SET-UP CLEARANCE	
Q201=-20	;DEPTH	
Q206=150	;FEED RATE FOR PLNGNG	
Q202=5	;PLUNGING DEPTH	
Q210=0	;DWELL TIME AT TOP	
Q203=+20	;SURFACE COORDINATE	
Q204=50	;2ND SET-UP CLEARANCE	
Q212=0.2	;DECREMENT	
Q213=3	;NR OF BREAKS	
Q205=3	;MIN. PLUNGING DEPTH	
Q211=0.25	;DWELL TIME AT DEPTH	
Q208=500	;RETRACTION FEED RATE	
Q256=0.2	;DIST FOR CHIP BRKNG	

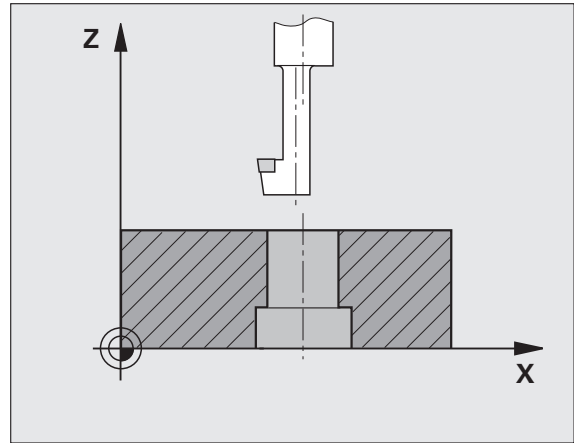


## 16.7 BACK BORING (Cycle 204)

### Cycle run

This cycle allows holes to be bored from the underside of the workpiece.

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the set-up clearance above the workpiece surface.
- 2 The TNC then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.
- 3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached the set-up clearance on the underside of the workpiece.
- 4 The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.
- 5 If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. Another oriented spindle stop is carried out and the tool is once again displaced by the off-center distance.
- 6 The TNC moves the tool at the pre-positioning feed rate to the set-up clearance and then—if entered—to the 2nd set-up clearance at **FMAX**.



**Please note while programming:**

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.

Special boring bars for upward cutting are required for this cycle.



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

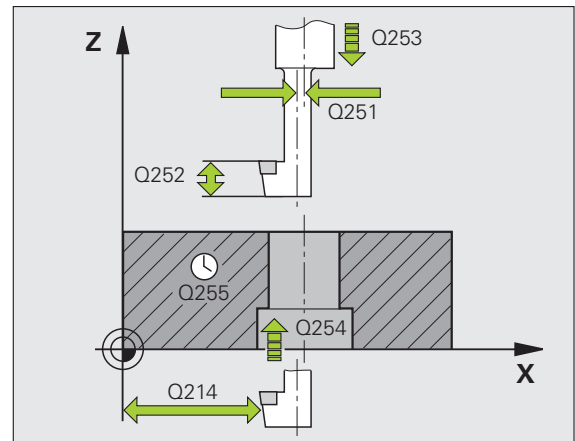
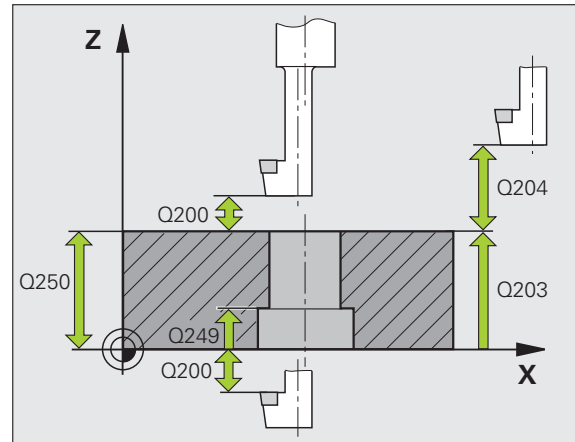
**Danger of collision!**

Check the position of the tool tip when you program a spindle orientation to the angle that you enter in **Q336** (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

## Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Depth of counterbore** Q249 (incremental): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction. Input range -99999.9999 to 99999.9999
- ▶ **Material thickness** Q250 (incremental): Thickness of the workpiece. Input range 0.0001 to 99999.9999
- ▶ **Off-center distance** Q251 (incremental): Off-center distance for the boring bar; value from tool data sheet. Input range 0.0001 to 99999.9999
- ▶ **Tool edge height** Q252 (incremental): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet. Input range 0.0001 to 99999.9999
- ▶ **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the workpiece. Input range 0 to 99999.999; alternatively **FMAX**, **FAUTO**
- ▶ **Feed rate for back boring** Q254: Traversing speed of the tool during back boring in mm/min. Input range 0 to 99999.999; alternatively **FAUTO**, **FU**
- ▶ **Dwell time** Q255: Dwell time in seconds at the top of the bore hole. Input range 0 to 3600.000



- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation). Input of 0 is not permitted
  - 1 Retract tool in the negative ref. axis direction
  - 2 Retract tool in the negative minor axis direction
  - 3 Retract tool in the positive ref. axis direction
  - 4 Retract tool in the positive minor axis direction
- ▶ **Angle for spindle orientation** Q336 (absolute): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole. Input range -360.0000 to 360.0000

Example: NC blocks

11 CYCL DEF 204 BACK BORING	
Q200=2	;SET-UP CLEARANCE
Q249=+5	;DEPTH OF COUNTERBORE
Q250=20	;MATERIAL THICKNESS
Q251=3.5	;OFF-CENTER DISTANCE
Q252=15	;TOOL EDGE HEIGHT
Q253=750	;F PRE-POSITIONING
Q254=200	;F COUNTERSINKING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE





## 16.8 UNIVERSAL PECKING (Cycle 205)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 If you enter a deepened starting point, the TNC moves at the defined positioning feed rate to the set-up clearance above the deepened starting point.
- 3 The tool drills to the first plunging depth at the programmed feed rate **F**.
- 4 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to the set-up clearance, and then at **FMAX** to the entered starting position above the first plunging depth.
- 5 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
- 6 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
- 7 The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.



**Please note while programming:**

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

If you enter different advance stop distances for **Q258** and **Q259**, the TNC will change the advance stop distances between the first and last plunging depths at the same rate.

If you use **Q379** to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. Retraction movements are not changed by the TNC, therefore they are calculated with respect to the coordinate of the workpiece surface.

**Danger of collision!**

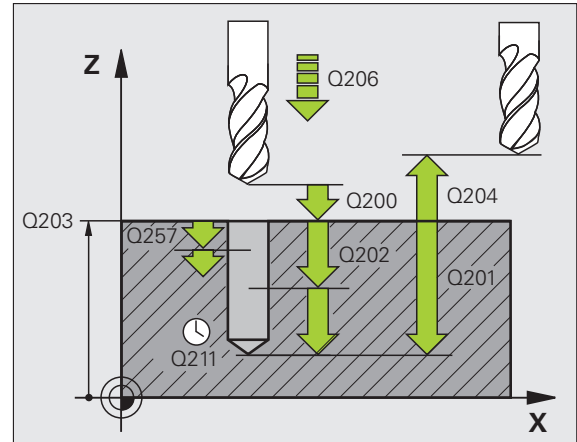
Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

## Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole (tip of drill taper). Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Plunging depth** Q202 (incremental): Infeed per cut. Input range 0 to 99999.9999. The depth does not have to be a multiple of the plunging depth. The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Decrement** Q212 (incremental): Value by which the TNC decreases the plunging depth Q202. Input range 0 to 99999.9999
- ▶ **Minimum plunging depth** Q205 (incremental): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205. Input range 0 to 99999.9999
- ▶ **Upper advanced stop distance** Q258 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth. Input range 0 to 99999.9999
- ▶ **Lower advanced stop distance** Q259 (incremental): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth. Input range 0 to 99999.9999



- ▶ **Infeed depth for chip breaking** Q257 (incremental): Depth at which the TNC carries out chip breaking. No chip breaking if 0 is entered. Input range 0 to 99999.9999
- ▶ **Retraction rate for chip breaking** Q256 (incremental): Value by which the TNC retracts the tool during chip breaking. The TNC retracts the tool at a feed rate of 3000 mm/min. Input range 0.1000 to 99999.9999.
- ▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position of drilling if a shorter tool has already pilot drilled to a certain depth. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance to the deepened starting point. Input range 0 to 99999.9999
- ▶ **Feed rate for pre-positioning** Q253: Traversing velocity of the tool during positioning from the set-up clearance to a deepened starting point in mm/min. Effective only if Q379 is entered not equal to 0. Input range 0 to 99999.999, alternatively **FMAX**, **FAUTO**

Example: NC blocks

11	CYCL DEF 205	UNIVERSAL PECKING
Q200=2	;SET-UP CLEARANCE	
Q201=-80	;DEPTH	
Q206=150	;FEED RATE FOR PLNGNG	
Q202=15	;PLUNGING DEPTH	
Q203=+100	;SURFACE COORDINATE	
Q204=50	;2ND SET-UP CLEARANCE	
Q212=0.5	;DECREMENT	
Q205=3	;MIN. PLUNGING DEPTH	
Q258=0.5	;UPPER ADV. STOP DIST.	
Q259=1	;LOWER ADV. STOP DIST.	
Q257=5	;DEPTH FOR CHIP BRKNG	
Q256=0.2	;DIST FOR CHIP BRKNG	
Q211=0.25	;DWELL TIME AT DEPTH	
Q379=7.5	;STARTING POINT	
Q253=750	;F PRE-POSITIONING	



## 16.9 SINGLE-LIP DEEP-HOLE DRILLING (Cycle 241)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 Then the TNC moves the tool at the defined positioning feed rate to the set-up clearance above the deepened starting point and activates the drilling speed (**M3**) and the coolant. The TNC executes the approach motion with the direction of rotation defined in the cycle, with clockwise, counterclockwise or stationary spindle.
- 3 The tool drills to the entered drilling depth at the programmed feed rate **F**.
- 4 If programmed, the tool remains at the hole bottom for chip breaking. Then the TNC switches off the coolant and resets the drilling speed to the value defined for retraction.
- 5 After the dwell time at the hole bottom, the tool is retracted to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.

### Please note while programming:



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.



#### **Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

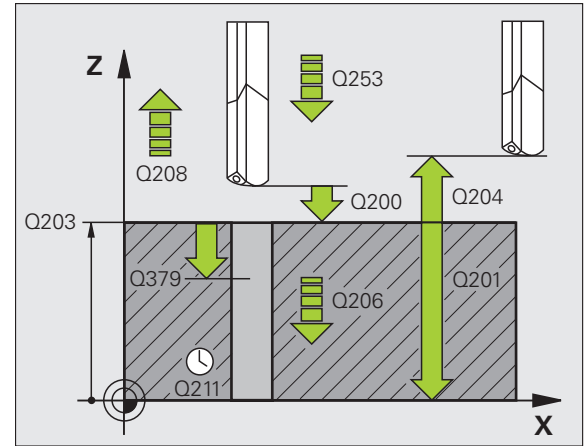
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!



## Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of hole. Input range -99999.9999 to 99999.9999
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU**
- ▶ **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom. Input range 0 to 3600.0000
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position for actual drilling operation. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance to the deepened starting point. Input range 0 to 99999.9999
- ▶ **Feed rate for pre-positioning** Q253: Traversing velocity of the tool during positioning from the set-up clearance to the deepened starting point in mm/min. Effective only if Q379 is entered not equal to 0. Input range 0 to 99999.999, alternatively **FMAX, FAUTO**
- ▶ **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q206. Input range 0 to 99999.999, alternatively **FMAX, FAUTO**



- ▶ **Rotat. dir. of entry/exit (3/4/5)** Q426: Desired direction of spindle rotation when tool moves into and retracts from the hole. Input range:  
**3:** Spindle rotation with M3  
**4:** Spindle rotation with M4  
**5:** Movement with stationary spindle
- ▶ **Spindle speed of entry/exit** Q427: Desired spindle speed when tool moves into and retracts from the hole. Input range 0 to 99999
- ▶ **Drilling speed** Q428: Desired speed for drilling. Input range 0 to 99999
- ▶ **M function for coolant on?** Q429: M function for switching on the coolant. The TNC switches the coolant on if the tool is in the hole at the deepened starting point. Input range 0 to 999
- ▶ **M function for coolant off?** Q430: M function for switching off the coolant. The TNC switches the coolant off if the tool is at the hole depth. Input range 0 to 999

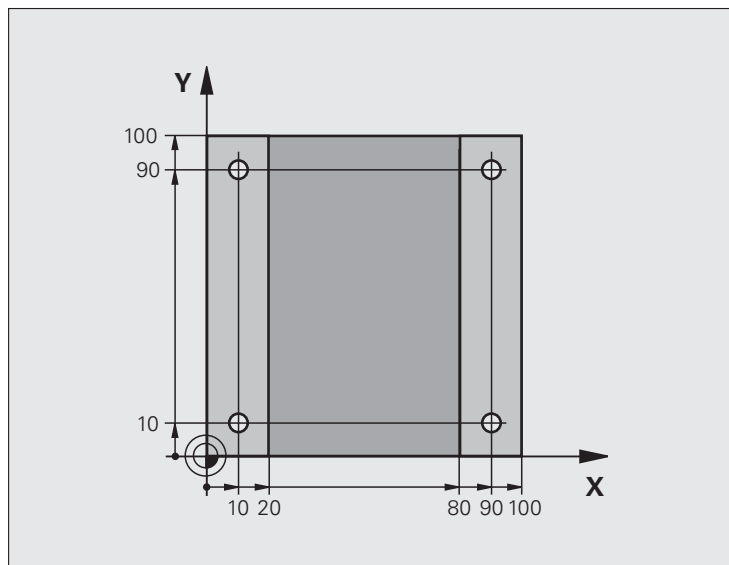
**Example: NC blocks**

11 CYCL DEF 241 SINGLE-LIP D.H.DRLNG
Q200=2 ;SET-UP CLEARANCE
Q201=-80 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q211=0.25 ;DWELL TIME AT DEPTH
Q203=+100 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q379=7.5 ;STARTING POINT
Q253=750 ;F PRE-POSITIONING
Q208=1000 ;RETRACTION FEED RATE
Q426=3 ;DIR. OF SPINDLE ROT.
Q427=25 ;ROT. SPEED INFED/OUT
Q428=500 ;DRILLING SPEED
Q429=8 ;COOLANT ON
Q430=9 ;COOLANT OFF



# 16.10 Programming examples

## Example: Drilling cycles



0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S4500	Tool call (tool radius 3)
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Cycle definition
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;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	



6	X+10 R0 FMAX M3	Approach hole 1, spindle ON
8	Y+10 R0 FMAX M99	Approach hole 1, call cycle
9	X+90 R0 FMAX M99	Approach hole 2, call cycle
9	Y+90 R0 FMAX M99	Approach hole 3, call cycle
9	X+10 R0 FMAX M99	Approach hole 4, call cycle
11	Z+250 R0 FMAX M2	Retract the tool, end program
12	END PGM C200 MM	



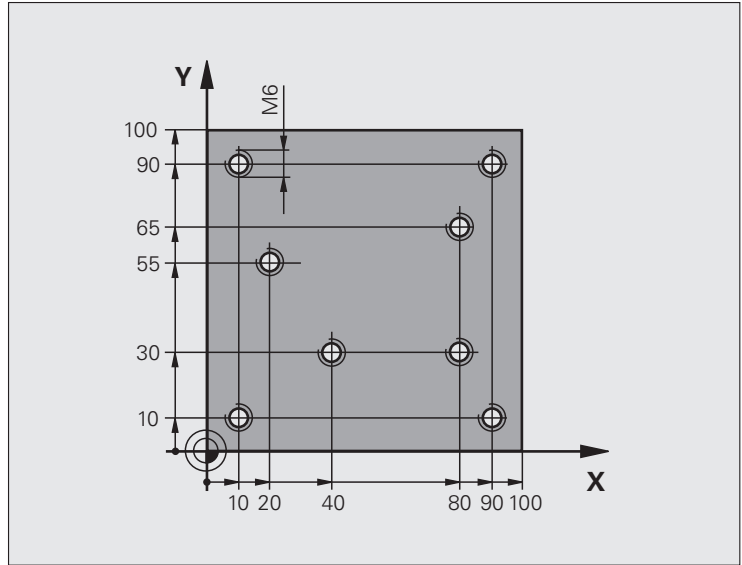
## Example: Using drilling cycles in connection with PATTERN DEF

The drill hole coordinates are stored in the pattern definition **PATTERN DEF POS** and are called by the TNC with **CYCL CALL PAT**:

The tool radii are selected so that all work steps can be seen in the test graphics.

### Program sequence

- Centering (tool radius 4)
- Drilling (tool radius 2.4)
- Tapping (tool radius 3)



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Call the centering tool (tool radius 4)
4 Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
	The TNC positions to the clearance height after every cycle
5 PATTERN DEF	Define all drilling positions in the point pattern
POS1( X+10 Y+10 Z+0 )	
POS2( X+40 Y+30 Z+0 )	
POS3( X+20 Y+55 Z+0 )	
POS4( X+10 Y+90 Z+0 )	
POS5( X+90 Y+90 Z+0 )	
POS6( X+80 Y+65 Z+0 )	
POS7( X+80 Y+30 Z+0 )	
POS8( X+90 Y+10 Z+0 )	

<b>6 CYCL DEF 240 CENTERING</b>	Cycle definition: CENTERING
<b>Q200=2 ;SET-UP CLEARANCE</b>	
<b>Q343=0 ;SELECT DEPTH/DIA.</b>	
<b>Q201=-2 ;DEPTH</b>	
<b>Q344=-10 ;DIAMETER</b>	
<b>Q206=150 ;FEED RATE FOR PLNGN</b>	
<b>Q211=0 ;DWELL TIME AT DEPTH</b>	
<b>Q203=+0 ;SURFACE COORDINATE</b>	
<b>Q204=50 ;2ND SET-UP CLEARANCE</b>	
<b>7 CYCL CALL PAT F5000 M13</b>	Call the cycle in connection with the hole pattern
<b>8 Z+100 R0 FMAX</b>	Retract the tool, change the tool
<b>9 TOOL CALL 2 Z S5000</b>	Call the drilling tool (radius 2.4)
<b>10 Z+10 R0 F5000</b>	Move tool to clearance height (enter a value for F)
<b>11 CYCL DEF 200 DRILLING</b>	Cycle definition: drilling
<b>Q200=2 ;SET-UP CLEARANCE</b>	
<b>Q201=-25 ;DEPTH</b>	
<b>Q206=150 ;FEED RATE FOR PECKING</b>	
<b>Q202=5 ;PLUNGING DEPTH</b>	
<b>Q210=0 ;DWELL TIME AT TOP</b>	
<b>Q203=+0 ;SURFACE COORDINATE</b>	
<b>Q204=50 ;2ND SET-UP CLEARANCE</b>	
<b>Q211=0.2 ;DWELL TIME AT DEPTH</b>	
<b>12 CYCL CALL PAT F5000 M13</b>	Call the cycle in connection with the hole pattern
<b>13 Z+100 R0 FMAX</b>	Retract the tool
<b>14 TOOL CALL 3 Z S200</b>	Call the tapping tool (radius 3)
<b>15 Z+50 R0 FMAX</b>	Move tool to clearance height
<b>16 CYCL DEF 206 TAPPING NEW</b>	Cycle definition for tapping
<b>Q200=2 ;SET-UP CLEARANCE</b>	
<b>Q201=-25 ;DEPTH OF THREAD</b>	
<b>Q206=150 ;FEED RATE FOR PECKING</b>	
<b>Q211=0 ;DWELL TIME AT DEPTH</b>	
<b>Q203=+0 ;SURFACE COORDINATE</b>	
<b>Q204=50 ;2ND SET-UP CLEARANCE</b>	
<b>17 CYCL CALL PAT F5000 M13</b>	Call the cycle in connection with the hole pattern
<b>18 Z+100 R0 FMAX M2</b>	Retract the tool, end program
<b>19 END PGM 1 MM</b>	



## 16.11 TAPPING NEW with floating tap holder (Cycle 206)

### Cycle run

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 The tool taps to the total hole depth in one movement.
- 3 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
- 4 At the set-up clearance, the direction of spindle rotation reverses once again.

### Please note while programming:



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

For tapping right-hand threads activate the spindle with **M3**, for left-hand threads use **M4**.



#### Danger of collision!

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

## Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch. Input range 0 to 99999.9999
- ▶ **Total hole depth** Q201 (thread length, incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999
- ▶ **Feed rate F** Q206: Traversing speed of the tool during tapping. Input range 0 to 99999.999; alternatively **FAUTO**
- ▶ **Dwell time at bottom** Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction. Input range 0 to 3600.0000
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

The feed rate is calculated as follows:  $F = S \times p$

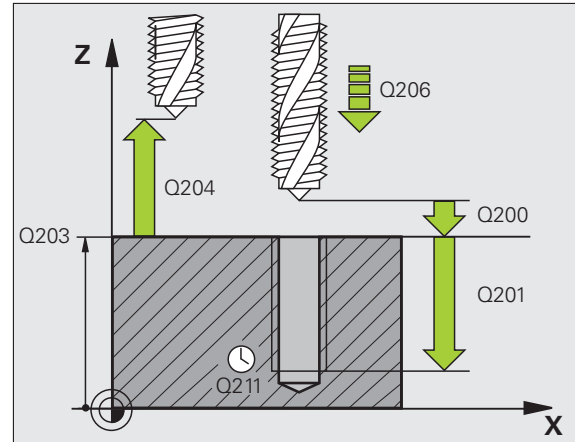
F: Feed rate (mm/min)

S: Spindle speed (rpm)

p: Thread pitch (mm)

### Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.



### Example: NC blocks

```

25 CYCL DEF 206 TAPPING NEW
  Q200=2    ;SET-UP CLEARANCE
  Q201=-20  ;DEPTH
  Q206=150  ;FEED RATE FOR PLNGNG
  Q211=0.25 ;DWELL TIME AT DEPTH
  Q203=+25  ;SURFACE COORDINATE
  Q204=50   ;2ND SET-UP CLEARANCE
  
```



## 16.12 RIGID TAPPING without a floating tap holder NEW (Cycle 207)

### Cycle run

The TNC cuts the thread without a floating tap holder in one or more passes.

- 1 The TNC positions the tool in the spindle axis at rapid traverse **FMAX** to the entered set-up clearance above the workpiece surface.
- 2 The tool taps to the total hole depth in one movement.
- 3 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. If programmed, the tool moves to the 2nd set-up clearance at **FMAX**.
- 4 The TNC brings spindle rotation to a stop at the set-up clearance.

**Please note while programming:**

Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

This cycle is effective only for machines with servo-controlled spindle.



Program a positioning block for the starting point (hole center) in the working plane with radius compensation **R0**.

The algebraic sign for the total hole depth parameter determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the feed-rate override is used during tapping, the TNC automatically adjusts the feed rate.

The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with **M3** (or **M4**).

**Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!



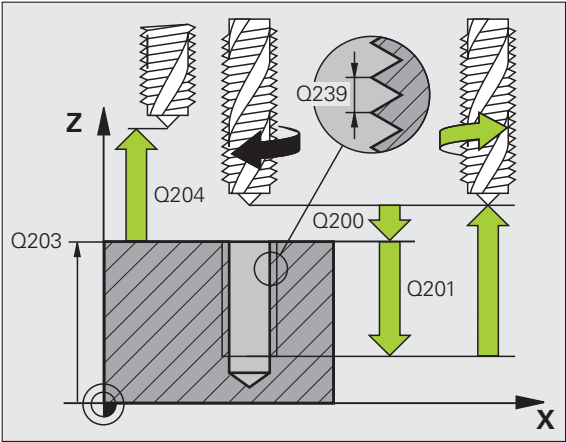
### Cycle parameters



- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip (at starting position) and workpiece surface. Input range 0 to 99999.9999
- ▶ **Total hole depth** Q201 (incremental): Distance between workpiece surface and end of thread. Input range -99999.9999 to 99999.9999
- ▶ **Pitch** Q239  
Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:  
+= right-hand thread  
-= left-hand thread  
Input range -99.9999 to 99.9999
- ▶ **Coordinate of workpiece surface** Q203 (absolute): Coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999

#### Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the MANUAL OPERATION soft key. If you press MANUAL OPERATION, you can retract the tool under program control. Simply press the positive axis direction button of the active spindle axis.



#### Example: NC blocks

26	CYCL DEF 207 RIGID TAPPING NEW
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q239=+1	;PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE





# 16.13 Programming example

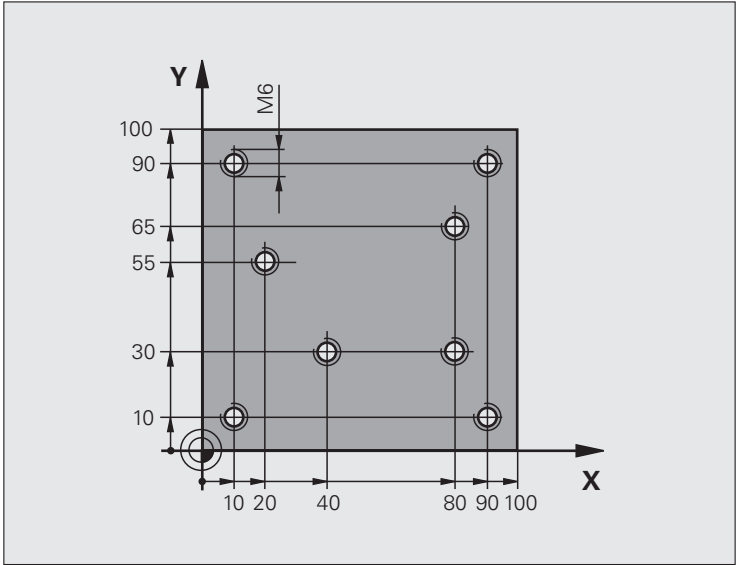
## Example: Thread milling

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the TNC with **CYCL CALL PAT**.

The tool radii are selected so that all work steps can be seen in the test graphics.

### Program sequence

- Centering
- Drilling
- Tapping



0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Y+0	
3 TOOL CALL 1 Z S5000	Call tool: centering drill
4 Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
	The TNC positions to the clearance height after every cycle
5 SEL PATTERN "TAB1"	Definition of point table
6 CYCL DEF 200 DRILLING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q201=-2 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGN	
Q202=2 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	



10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
	Feed rate between points: 5000 mm/min
11 Z+100 R0 FMAX M6	Retract the tool, change the tool
12 TOOL CALL 2 Z S5000	Call tool: drill
13 Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PECKING	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWELL TIME AT DEPTH	
15 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
16 Z+100 R0 FMAX M6	Retract the tool, change the tool
17 TOOL CALL 3 Z S200	Call tool: tap
18 Z+50 R0 FMAX	Move tool to clearance height
19 CYCL DEF 206 TAPPING NEW	Cycle definition for tapping
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH OF THREAD	
Q206=150 ;FEED RATE FOR PECKING	
Q211=0 ;DWELL TIME AT DEPTH	
Q203=+0 ;SURFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2ND SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
20 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
21 Z+100 R0 FMAX M2	Retract the tool, end program
22 END PGM 1 MM	

## Point table TAB1.PNT

TAB1. PNTMM
NRXYZ
0+10+10+0
1+40+30+0
2+90+10+0
3+80+30+0
4+80+65+0
5+90+90+0
6+10+90+0
7+20+55+0
[END]





# 17



**Fixed cycles: Pocket  
milling / stud milling /  
slot milling**



# 17.1 Fundamentals

## Overview

The TNC offers 2 cycles for machining pockets and studs:

Cycle	Soft key	Page
251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging		Page 427
256 RECTANGULAR STUD Roughing/finishing cycle with stepover, if multiple passes are required		Page 432



## 17.2 RECTANGULAR POCKET (Cycle 251)

### Cycle run

Use Cycle 251 RECTANGULAR POCKET to completely machine rectangular pockets. Depending on the cycle parameters, the following machining alternatives are available:

- Complete machining: Roughing, floor finishing, side finishing
- Only roughing
- Only floor finishing and side finishing
- Only floor finishing
- Only side finishing

### Roughing

- 1 The tool plunges into the workpiece at the pocket center and advances to the first plunging depth.
- 2 The TNC roughs out the pocket from the inside out, taking the overlap factor (parameter Q370) and the finishing allowances (parameters Q368 and Q369) into account.
- 3 At the end of the roughing operation, the TNC moves the tool away from the pocket wall, then moves by the set-up clearance above the current infeed depth and returns from there at rapid traverse to the pocket center.
- 4 This process is repeated until the programmed pocket depth is reached.

### Finishing

- 5 Inasmuch as finishing allowances are defined, the TNC then finishes the pocket walls, in multiple infeeds if so specified.
- 6 Then the TNC finishes the floor of the pocket from the inside out.



**Please note while programming:**

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (pocket position).

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end of the cycle, the TNC returns the tool to the starting position.

At the end of a roughing operation, the TNC positions the tool back to the pocket center at rapid traverse. The tool is above the current pecking depth by the set-up clearance. Enter the set-up clearance so that the tool cannot jam because of chips.

**Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

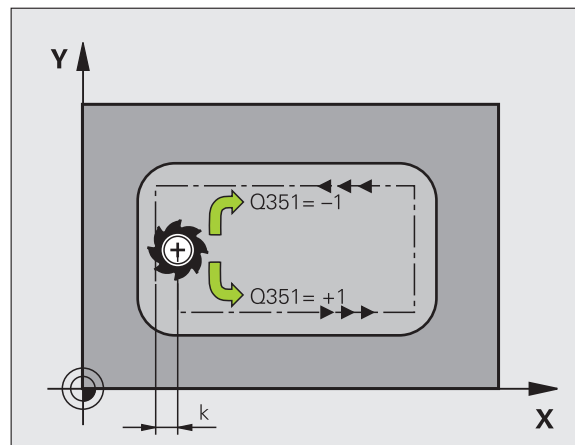
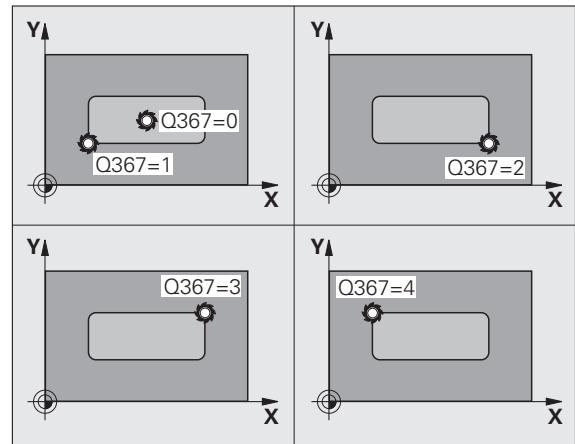
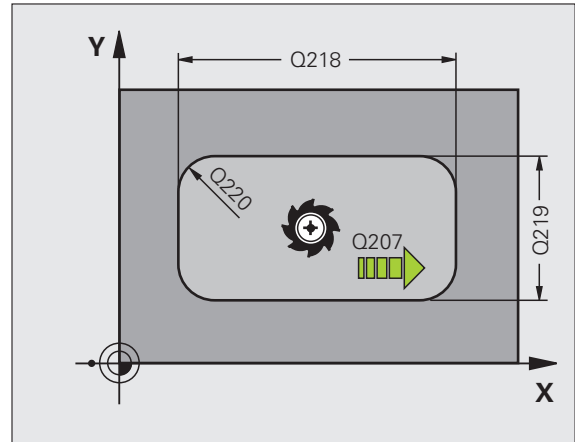
If you call the cycle with machining operation 2 (only finishing), then the TNC positions the tool in the center of the pocket at rapid traverse to the first plunging depth.



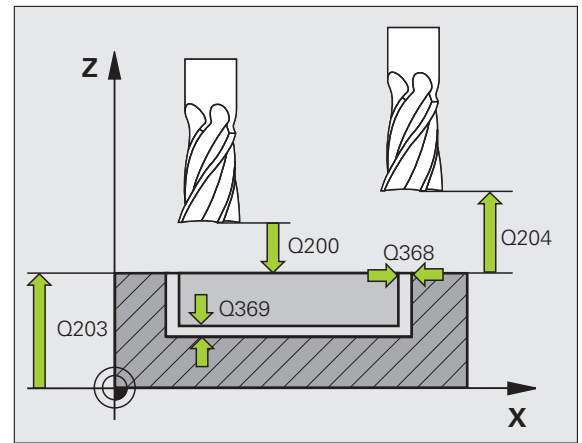
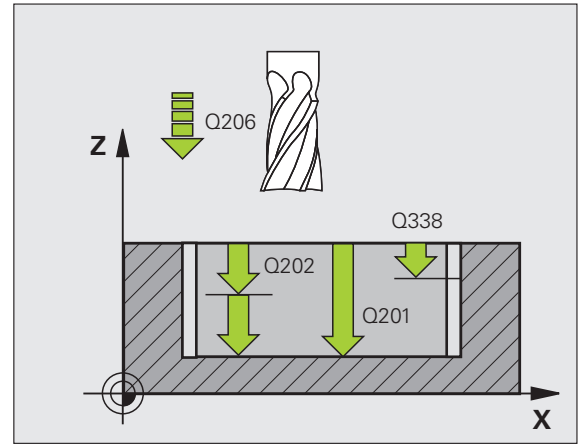
## Cycle parameters



- **Machining operation (0/1/2)** Q215: Define the machining operation:
  - 0:** Roughing and finishing
  - 1:** Only roughing
  - 2:** Only finishing
 Side finishing and floor finishing are only executed if the respective finishing allowance (Q368, Q369) has been defined
- **1st side length** Q218 (incremental): Pocket length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- **2nd side length** Q219 (incremental): Pocket length, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- **Depth** Q201 (incremental): Distance between workpiece surface and bottom of pocket. Input range -99999.9999 to 99999.9999
- **Pocket position** Q367: Position of the pocket in reference to the position of the tool when the cycle is called:
  - 0:** Tool position = Center of pocket
  - 1:** Tool position = Lower left corner
  - 2:** Tool position = Lower right corner
  - 3:** Tool position = Upper right corner
  - 4:** Tool position = Upper left corner
- **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU, FZ**
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU, FZ**
- **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.9999; alternatively **FAUTO, FU, FZ**
- **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane. Input range 0 to 99999.9999
- **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999



- ▶ **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999
- ▶ **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- ▶ **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- ▶ **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- ▶ **Climb or up-cut** Q351: Type of milling operation with M3:  
 +1 = Climb milling  
 -1 = Up-cut milling
- ▶ **Path overlap factor** Q370:  $Q370 \times \text{tool radius} = \text{stepover factor } k$ . Input range 0.1 to 1.9999.



Example: NC blocks

8	CYCL DEF 251	RECTANGULAR POCKET
	Q215=0	;MACHINING OPERATION
	Q218=80	;1ST SIDE LENGTH
	Q219=60	;2ND SIDE LENGTH
	Q201=-20	;DEPTH
	Q367=0	;POCKET POSITION
	Q202=5	;PLUNGING DEPTH
	Q207=500	;FEED RATE FOR MILLING
	Q206=150	;FEED RATE FOR PLNGNG
	Q385=500	;FEED RATE FOR FINISHING
	Q368=0.2	;ALLOWANCE FOR SIDE
	Q369=0.1	;ALLOWANCE FOR FLOOR
	Q338=5	;INFEED FOR FINISHING
	Q200=2	;SET-UP CLEARANCE
	Q203=+0	;SURFACE COORDINATE
	Q204=50	;2ND SET-UP CLEARANCE
	Q351=+1	;CLIMB OR UP-CUT
	Q370=1	;TOOL PATH OVERLAP
9	X+50 R0 FMAX	
10	Y+50 R0 FMAX M3 M99	

17.2 RECTANGULAR POCKET (Cycle 251)

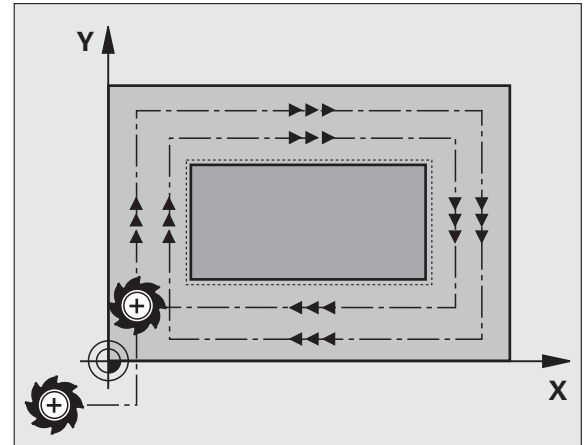


## 17.3 RECTANGULAR STUD (Cycle 256)

### Cycle run

Use Cycle 256 to machine a rectangular stud. If a dimension of the workpiece blank is greater than the maximum possible stepover, then the TNC performs multiple stepovers until the finished dimension has been machined.

- 1 The tool moves from the cycle starting position (stud center) in the negative X direction to the starting position for stud machining. The starting position is to the left of the unmachined stud and is offset by the set-up clearance + tool radius.
- 2 If the tool is at the 2nd set-up clearance, it moves at rapid traverse **FMAX** to the set-up clearance, and from there it advances to the first plunging depth at the feed rate for plunging.
- 3 The tool then moves to the stud contour and machines one revolution.
- 4 If the finished dimension cannot be machined with one revolution, the TNC performs a stepover with the current factor, and machines another revolution. The TNC takes the dimensions of the workpiece blank, the finished dimension, and the permitted stepover into account. This process is repeated until the defined finished dimension has been reached.
- 5 The tool then departs the contour and returns to the starting point of stud machining.
- 6 The TNC then plunges the tool to the next plunging depth, and machines the stud at this depth.
- 7 This process is repeated until the programmed stud depth is reached.



**Please note while programming:**

Pre-position the tool in the machining plane to the starting position with radius compensation **R0**. Note parameter Q367 (stud position).

The TNC automatically pre-positions the tool in the tool axis. Note parameter Q204 (2nd set-up clearance).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.

At the end, the TNC positions the tool back to the set-up clearance, or to the 2nd set-up clearance if one was programmed.

**Danger of collision!**

Use the machine parameter **displayDepthErr** to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis to set-up clearance **below** the workpiece surface!

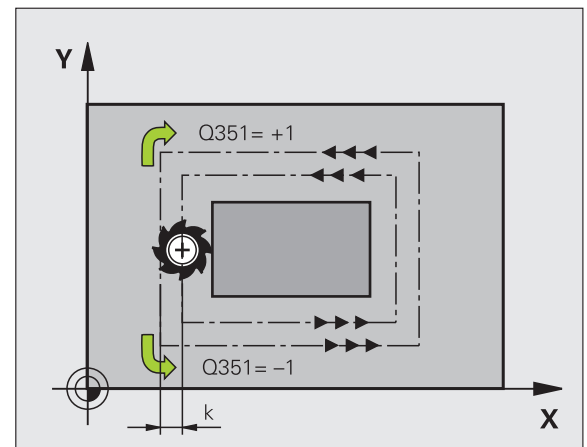
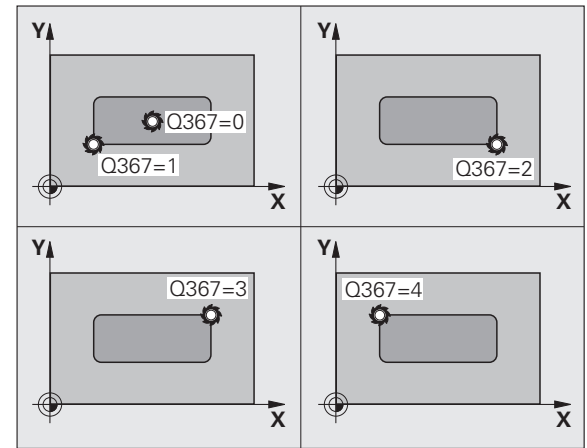
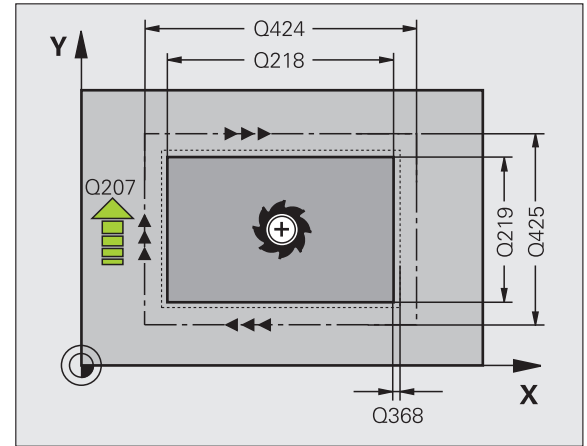
Leave enough room next to the stud for the approach motion. Minimum: tool diameter + 2 mm



## Cycle parameters



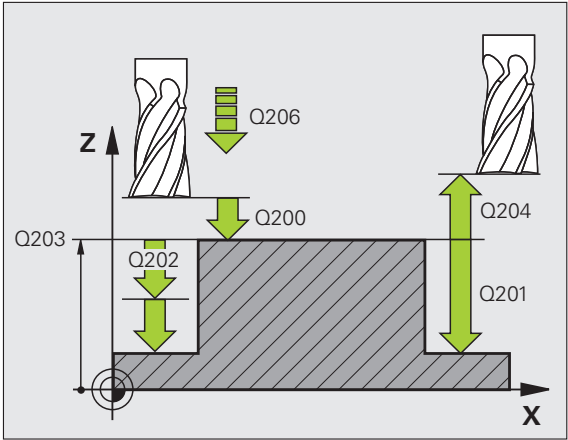
- ▶ **Machining operation (0/1/2)** Q215: Define the machining operation:  
**0:** Roughing and finishing  
**1:** Only roughing  
**2:** Only finishing  
Side finishing and floor finishing are only executed if the finishing allowance (Q368) has been defined
- ▶ **1st side length** Q218: Stud length, parallel to the reference axis of the working plane. Input range 0 to 99999.9999
- ▶ **Workpiece blank side length 1** Q424: Length of the stud blank, parallel to the reference axis of the working plane. Enter **Workpiece blank side length 1** greater than **1st side length**. The TNC performs multiple stepovers if the difference between blank dimension 1 and finished dimension 1 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- ▶ **2nd side length** Q219: Stud length, parallel to the minor axis of the working plane. Enter **Workpiece blank side length 2** greater than **2nd side length**. The TNC performs multiple stepovers if the difference between blank dimension 2 and finished dimension 2 is greater than the permitted stepover (tool radius multiplied by path overlap **Q370**). The TNC always calculates a constant stepover. Input range 0 to 99999.9999
- ▶ **Depth** Q201 (incremental): Distance between workpiece surface and bottom of stud. Input range -99999.9999 to 99999.9999
- ▶ **Stud position** Q367: Position of the stud in reference to the position of the tool when the cycle is called:  
**0:** Tool position = Center of stud  
**1:** Tool position = Lower left corner  
**2:** Tool position = Lower right corner  
**3:** Tool position = Upper right corner  
**4:** Tool position = Upper left corner
- ▶ **Plunging depth** Q202 (incremental): Infeed per cut. Enter a value greater than 0. Input range 0 to 99999.9999
- ▶ **Feed rate for milling** Q207: Traversing speed of the tool during milling in mm/min. Input range 0 to 99999.999; alternatively **FAUTO, FU, FZ**
- ▶ **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Input range 0 to 99999.999; alternatively **FMAX, FAUTO, FU, FZ**



- ▶ **Feed rate for finishing** Q385: Traversing speed of the tool during side and floor finishing in mm/min. Input range 0 to 99999.9999; alternatively **FAUTO, FU, FZ**
- ▶ **Workpiece blank side length 2** Q425: Length of the stud blank, parallel to the minor axis of the working plane. Input range 0 to 99999.9999
- ▶ **Finishing allowance for side** Q368 (incremental): Finishing allowance in the working plane, is left over after machining. Input range 0 to 99999.9999
- ▶ **Finishing allowance for floor** Q369 (incremental value): Finishing allowance in the tool axis. Input range 0 to 99999.9999
- ▶ **Infeed for finishing** Q338 (incremental): Infeed per cut. Q338=0: Finishing in one infeed. Input range 0 to 99999.9999



- **Set-up clearance** Q200 (incremental): Distance between tool tip and workpiece surface. Input range 0 to 99999.9999
- **Workpiece surface coordinate** Q203 (absolute): Absolute coordinate of the workpiece surface. Input range -99999.9999 to 99999.9999
- **2nd set-up clearance** Q204 (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur. Input range 0 to 99999.9999
- **Climb or up-cut** Q351: Type of milling operation with M3:  
 +1 = Climb milling  
 -1 = Up-cut milling
- **Path overlap factor** Q370:  $Q370 \times \text{tool radius} = \text{Stepover factor } k$ . Input range 0.1 to 1.9999.



Example: NC blocks

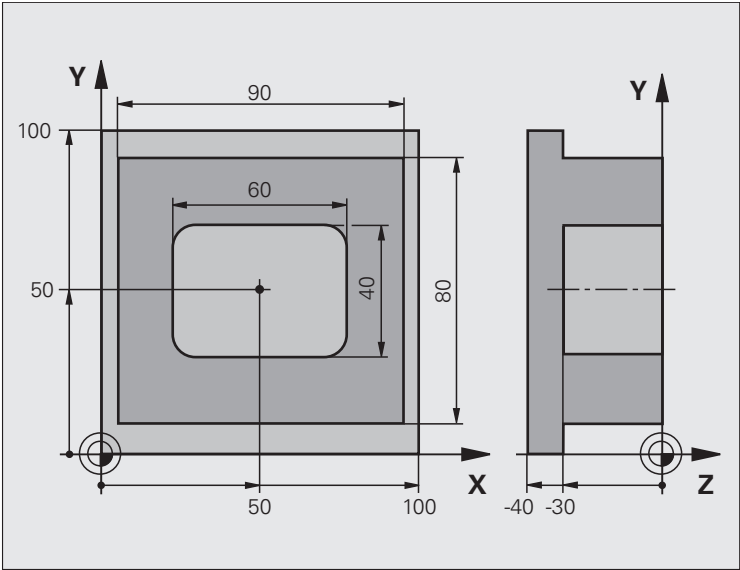
8 CYCL DEF 256 RECTANGULAR STUD	
Q215=0	;MACHINING OPERATION
Q218=60	;1ST SIDE LENGTH
Q424=74	;WORKPC. BLANK SIDE 1
Q219=40	;2ND SIDE LENGTH
Q425=60	;WORKPC. BLANK SIDE 2
Q201=-20	;DEPTH
Q367=0	;STUD POSITION
Q202=5	;PLUNGING DEPTH
Q207=500	;FEED RATE FOR MILLING
Q206=150	;FEED RATE FOR PLNGNG
Q385=500	;FEED RATE FOR FINISHING
Q368=0.2	;ALLOWANCE FOR SIDE
Q369=0.1	;ALLOWANCE FOR FLOOR
Q338=5	;INFEEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q351=+1	;CLIMB OR UP-CUT
Q370=1	;TOOL PATH OVERLAP
9	X+50 R0 FMAX
9	Y+50 R0 FMAX M3 M99





# 17.4 Programming examples

## Example: Milling pockets, studs and slots



0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Definition of workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
4 Z+250 R0 FMAX	Retract the tool



5	CYCL DEF 256 RECTANGULAR STUD	Define cycle for machining the contour outside
	Q218=90 ;1ST SIDE LENGTH	
	Q424=100 ;WORKPC. BLANK SIDE 1	
	Q219=80 ;2ND SIDE LENGTH	
	Q425=100 ;WORKPC. BLANK SIDE 2	
	Q201=-30 ;DEPTH	
	Q367=0 ;STUD POSITION	
	Q202=5 ;PLUNGING DEPTH	
	Q207=250 ;FEED RATE FOR MILLING	
	Q206=250 ;FEED RATE FOR PLNGNG	
	Q385=750 ;FEED RATE FOR FINISHING	
	Q368=0 ;ALLOWANCE FOR SIDE	
	Q369=0.1 ;ALLOWANCE FOR FLOOR	
	Q338=5 ;INFEED FOR FINISHING	
	Q200=2 ;SET-UP CLEARANCE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=20 ;2ND SET-UP CLEARANCE	
	Q351=+1 ;CLIMB OR UP-CUT	
	Q370=1 ;TOOL PATH OVERLAP	
6	X+50 R0	Call cycle for machining the contour outside
7	Y+50 R0 M3 M99	Call cycle for machining the contour outside

8 CYCL DEF 251 RECTANGULAR POCKET	Define RECTANGULAR POCKET cycle
Q215=0 ;MACHINING OPERATION	
Q218=60 ;1ST SIDE LENGTH	
Q219=40 ;2ND SIDE LENGTH	
Q201=-30 ;DEPTH	
Q367=+0 ;POCKET POSITION	
Q202=5 ;PLUNGING DEPTH	
Q207=500 ;FEED RATE FOR MILLING	
Q206=150 ;FEED RATE FOR PLNGNG	
Q385=750 ;FEED RATE FOR FINISHING	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q338=5 ;INFEEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q351=+1 ;CLIMB OR UP-CUT	
Q370=1 ;TOOL PATH OVERLAP	
9 X+50 R0 FMAX	Call CIRCULAR POCKET cycle
10 Y+50 R0 FMAX M99	Call CIRCULAR POCKET cycle
11 Z+250 R0 FMAX M30	Tool change
12 END PGM C210 MM	





# 18






**Cycles: Coordinate  
transformations**



# 18.1 Fundamentals

## Overview

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The TNC provides the following coordinate transformation cycles:

Cycle	Soft key	Page
7 DATUM SHIFT For shifting contours directly within the program or from datum tables		Page 443
247 DATUM SETTING Datum setting during program run		Page 449
8 MIRROR IMAGE Mirroring contours		Page 450
11 SCALING For increasing or reducing the size of contours		Page 452
26 AXIS-SPECIFIC SCALING FACTOR For increasing or reducing the size of contours with scaling factors for each axis		Page 453

## Effect of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called separately. It remains in effect until it is changed or canceled.

### To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M2, M30, or an END PGM block (depending on machine parameter **clearMode**).
- Select a new program



## 18.2 DATUM SHIFT (Cycle 7, DIN/ISO: G54)

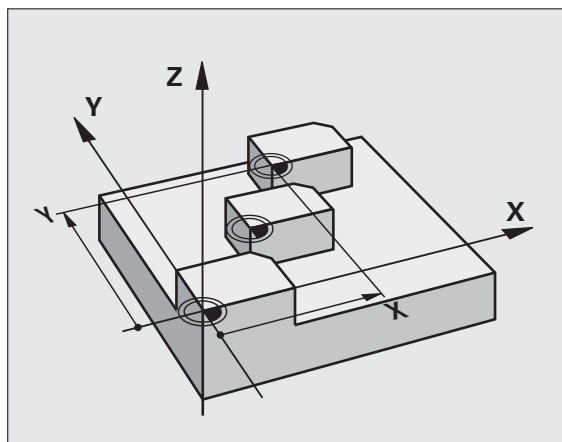
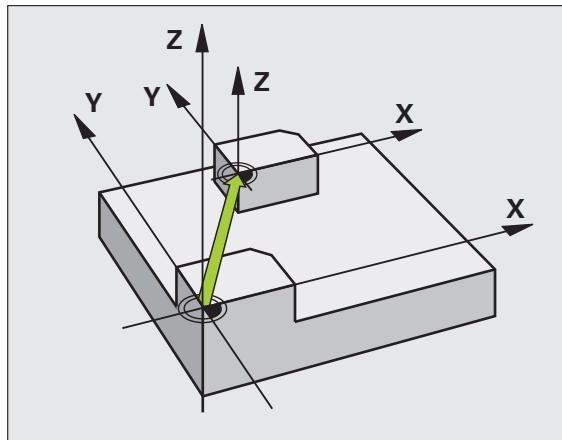
### Effect

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.

When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.

### Resetting

- Program a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.
- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.



### Cycle parameters



- **Datum shift:** Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid—this can be a datum which has already been shifted. Input range: Up to six NC axes, each from -99999.9999 to 99999.9999

#### Example: NC blocks

```
13 CYCL DEF 7.0 DATUM SHIFT
```

```
14 CYCL DEF 7.1 X+60
```

```
16 CYCL DEF 7.3 Z-5
```

```
15 CYCL DEF 7.2 Y+40
```



## 18.3 DATUM SHIFT with datum tables (Cycle 7)

### Effect

Datum tables are used for:

- Frequently recurring machining sequences at various locations on the workpiece
- Frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.

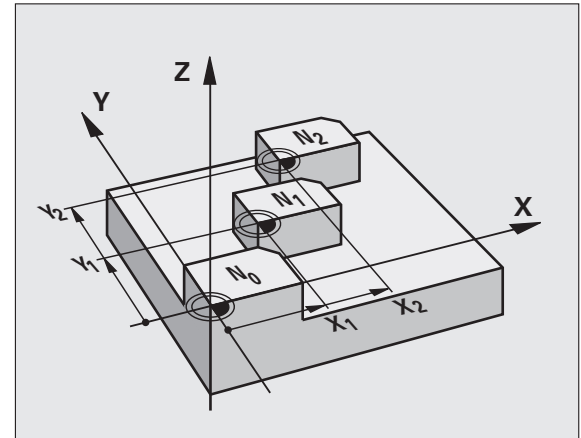
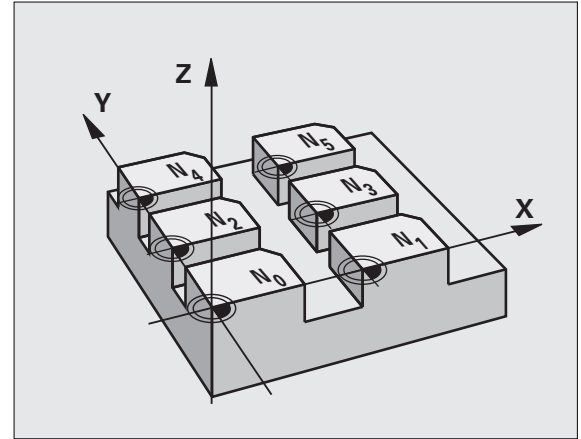
### Resetting

- Call a datum shift to the coordinates  $X=0$ ;  $Y=0$  etc. from the datum table.
- Execute a datum shift to the coordinates  $X=0$ ,  $Y=0$  etc. directly with a cycle definition

### Status displays

In the additional status display, the following data from the datum table are shown:

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number





**Please note while programming:****Danger of collision!**

Datums from a datum table are **always and exclusively** referenced to the current datum (preset).



If you are using datum shifts with datum tables, then use the **SEL TABLE** function to activate the desired datum table from the NC program.

If you work without **SEL TABLE**, then you must activate the desired datum table before the test run or the program run. (This applies also to the programming graphics).

- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table receives the status S.
- Use the file management in a program run mode to select the desired table for program run: The table receives the status M.

The coordinate values from datum tables are only effective with absolute coordinate values.

New lines can only be inserted at the end of the table.

If you create datum tables, the file name has to start with a letter.



Cycle parameters



- ▶ **Datum shift:** Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number entered in the Q parameter. Input range: 0 to 9999

Example: NC blocks

```
77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1 #5
```

Selecting a datum table in the part program

With the **SEL TABLE** function you select the table from which the TNC takes the datums:



- ▶ Select the functions for program call: Press the PGM CALL key
- ▶ Press the DATUM TABLE soft key
- ▶ Select the complete path name of the datum table or the file with the SELECT soft key and confirm your entry with the END key



Program a **SEL TABLE** block before Cycle 7 Datum Shift.  
A datum table selected with **SEL TABLE** remains active until you select another datum table with **SEL TABLE** or through PGM MGT.



Editing the datum table in the Programming and Editing mode of operation



After you have changed a value in a datum table, you must save the change with the ENT key. Otherwise the change might not be included during program run.

Select the datum table in the **Programming and Editing** mode of operation



- ▶ Call the file manager: Press the PGM MGT key
- ▶ Display the datum tables: Press the SELECT TYPE and SHOW .D soft keys
- ▶ Select the desired table or enter a new file name
- ▶ Edit the file. The soft-key row comprises the following functions for editing:

Function	Soft key
Go to beginning of table	
Select end of table	
Go to previous page	
Go to next page	
Insert line (only possible at end of table)	
Delete line	
Find	
Go to beginning of line	
Go to end of line	
Copy the current value	
Insert the copied value	
Add the entered number of lines (datums) to the end of the table	



### Configuring the datum table

If you do not wish to define a datum for an active axis, press the DEL key. Then the TNC clears the numerical value from the corresponding input field.

Manual operation

Table editing

TNC:\nc\_prog\pgm\zeroshift.d

D	X	Y	Z	R	B
0	100.334	50.002	0	0.0	0.0
1	200.524	50.007	0	0.0	0.0
2	300.001	49.999	0	0.0	0.0
3	400.994	50.001	0	0.0	0.0
4	0.0	0.0	0.0	0.0	0.0
5	0.0	0.0	0.0	0.0	0.0
6	0.0	0.0	0.0	0.0	0.0
7	0.0	0.0	0.0	0.0	0.0
8	0.0	0.0	0.0	0.0	0.0
9	0.0	0.0	0.0	0.0	0.0
10	0.0	0.0	0.0	0.0	0.0
11	0.0	0.0	0.0	0.0	0.0
12	0.0	0.0	0.0	0.0	0.0
13	0.0	0.0	0.0	0.0	0.0
14	0.0	0.0	0.0	0.0	0.0
15	0.0	0.0	0.0	0.0	0.0
16	0.0	0.0	0.0	0.0	0.0
17	0.0	0.0	0.0	0.0	0.0
18	0.0	0.0	0.0	0.0	0.0
19	0.0	0.0	0.0	0.0	0.0
20	0.0	0.0	0.0	0.0	0.0
21	0.0	0.0	0.0	0.0	0.0
22	0.0	0.0	0.0	0.0	0.0

BEGIN

END

PAGE

PAGE

FIND

END

Min. -99999.99999, Max. +99999.99999

### Exiting a datum table

Select a different type of file in file management and choose the desired file.



After you have changed a value in a datum table, you must save the change with the ENT key. Otherwise the change may not be included during program run.

### Status displays

In the additional status display, the TNC shows the values of the active datum shift.



# 18.4 DATUM SETTING (Cycle 247)

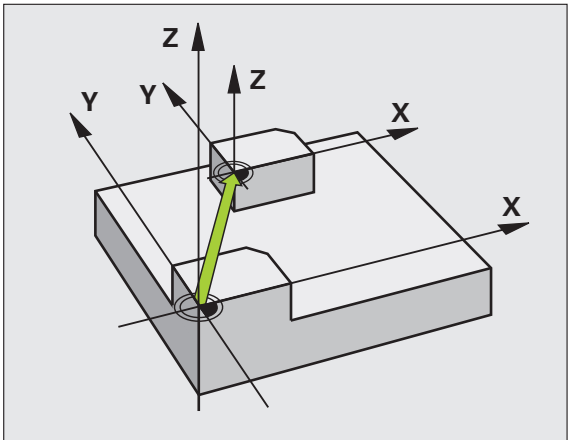
## Effect

With the DATUM SETTING cycle you can activate as the new datum a preset defined in a preset table.

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

## Status display

In the status display the TNC shows the active preset number behind the datum symbol.



## Please note before programming:



When activating a datum from the preset table, the TNC resets the datum shift, mirroring, rotation, scaling factor and axis-specific scaling factor.

If you activate preset number 0 (line 0), then you activate the datum that you last set in a manual operating mode.

Cycle 247 is not functional in Test Run mode.

## Cycle parameters



► **Number for datum?**: Enter the number of the datum to be activated from the preset table. Input range: 0 to 65535

### Example: NC blocks

```
13 CYCL DEF 247 DATUM SETTING
Q339=4 ;DATUM NUMBER
```

## Status displays

In the additional status display (POS. DISP. STATUS) the TNC shows the active preset number behind the **datum** dialog.



## 18.5 MIRROR IMAGE (Cycle 8)

### Effect

The TNC can machine the mirror image of a contour in the working plane.

The mirroring cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

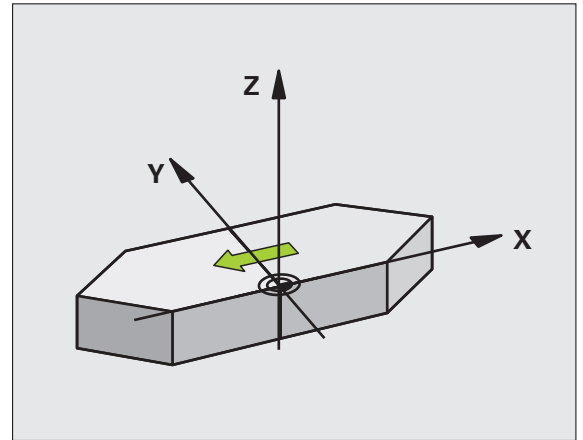
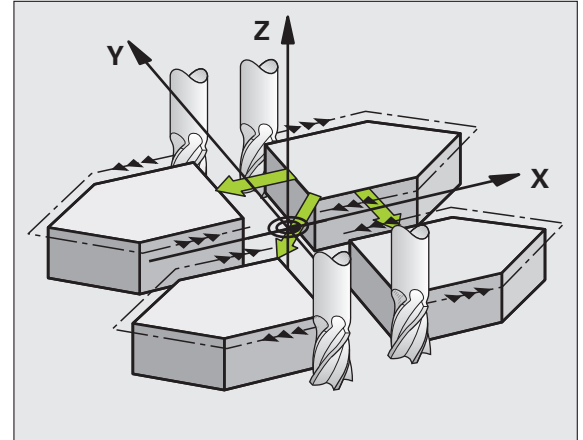
- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

The result of the mirroring depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also "jumps" to another location.

### Resetting

Program the MIRROR IMAGE cycle once again with NO ENT.



### Please note while programming:



If you mirror only one axis, the machining direction is reversed for the milling cycles (Cycles 2xx). Exception: Cycle 208, in which the direction defined in the cycle applies.

## Cycle parameters



- **Mirrored axis?**: Enter the axis to be mirrored. You can mirror all axes except for the spindle axis—including rotary axes—with the exception of the spindle axis and its associated auxiliary axis. You can enter up to three axes. Input range: Up to three NC axes **X, Y, Z, U, V, W, A, B, C**

### Example: NC blocks

```
79 CYCL DEF 8.0 MIRROR IMAGE
```

```
80 CYCL DEF 8.1 X Y Z
```



## 18.6 SCALING FACTOR (Cycle 11)

### Effect

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

SCALING becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

- All three coordinate axes at the same time
- Dimensions in cycles

### Prerequisite

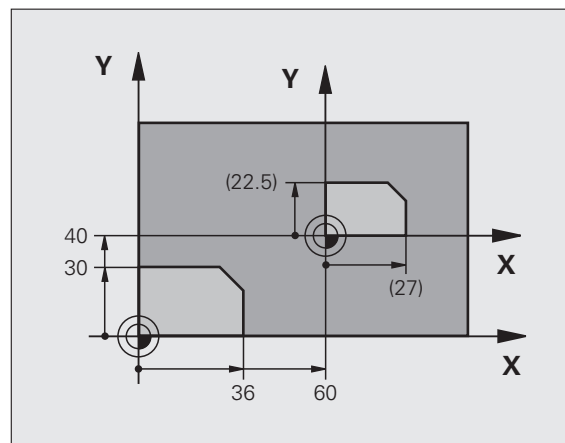
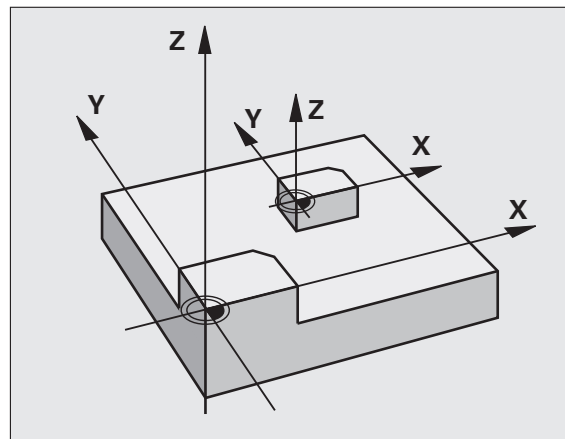
It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction: SCL less than 1 (down to 0.000 001)

### Resetting

Program the SCALING cycle once again with a scaling factor of 1.



### Cycle parameters



- **Scaling factor?:** Enter the scaling factor SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Effect" above). Input range: 0.000000 to 99.999999

### Example: NC blocks

```
11 CALL LBL 1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL 1
```



## 18.7 AXIS-SPECIFIC SCALING (Cycle 26)

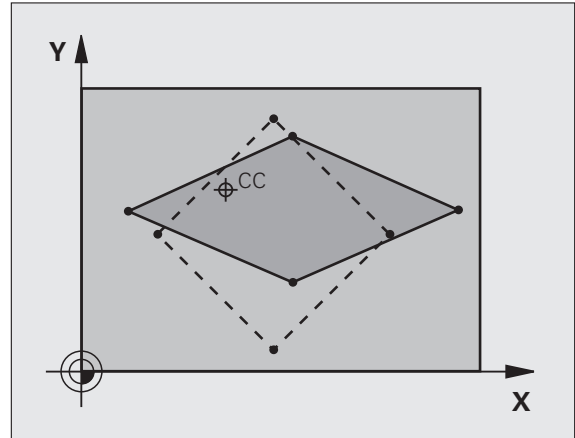
### Effect

With Cycle 26 you can account for shrinkage and oversize factors for each axis.

SCALING becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

### Resetting

Program the SCALING cycle once again with a scaling factor of 1 for the same axis.



### Please note while programming:



Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

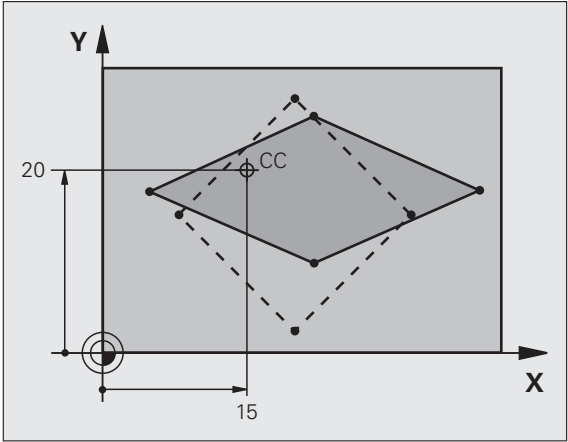
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING) with reference to the active datum.

Cycle parameters



- **Axis and scaling factor:** Select the coordinate axis/axes by soft key and enter the factor(s) involved in enlarging or reducing. Input range: 0.000000 to 99.999999
- **Center coordinates:** Enter the center of the axis-specific enlargement or reduction. Input range: -99999.9999 to 99999.9999



Example: NC blocks

```
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1
```

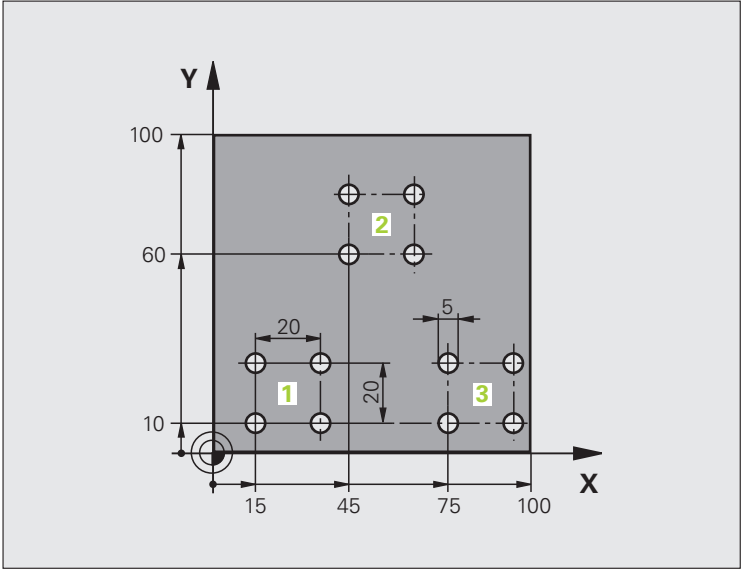


# 18.8 Programming examples

## Example: Groups of holes

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1



0 BEGIN PGM SP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL CALL 1 Z S5000	Tool call
4 Z+250 R0 FMAX	Retract the tool
5 CYCL DEF 200 DRILLING	Define the DRILLING cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
6 CYCL DEF 7.0 DATUM SHIFT	Datum shift for group 1
7 CYCL DEF 7.1 X+15	
8 CYCL DEF 7.2 Y+10	
9 CALL LBL 1	Call the subprogram for the group



## 18.8 Programming examples

10 CYCL DEF 7.0 DATUM SHIFT	Datum shift for group 2
11 CYCL DEF 7.1 X+75	
12 CYCL DEF 7.2 Y+10	
13 CALL LBL 1	Call the subprogram for the group
14 CYCL DEF 7.0 DATUM SHIFT	Datum shift for group 3
15 CYCL DEF 7.1 X+45	
16 CYCL DEF 7.2 Y+60	
17 CALL LBL 1	Call the subprogram for the group
18 Z+250 R0 FMAX M30	End of main program
19 LBL 1	Beginning of subprogram 1: Group of holes
20 X+0 R0 FMAX	Move to 1st hole
21 Y+0 R0 FMAX M99 M3	Move to 1st hole, call cycle
22 X+20 R0 FMAX M99	Move to 2nd hole, call cycle
23 Y+20 R0 FMAX M99	Move to 3rd hole, call cycle
24 X-20 R0 FMAX M99	Move to 4th hole, call cycle
25 LBL 0	End of subprogram 1
26 END PGM SP1 MM	



# 19




**Cycles: Special functions**



# 19.1 Fundamentals

## Overview

The TNC provides four cycles for the following special purposes:

Cycle	Soft key	Page
9 DWELL TIME		Page 459
12 PROGRAM CALL		Page 460
13 SPINDLE ORIENTATION		Page 462

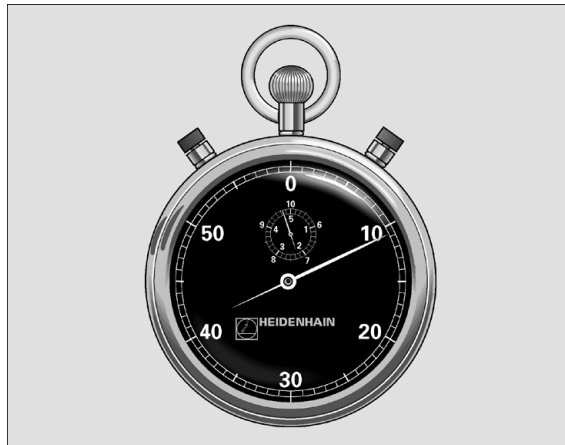


## 19.2 DWELL TIME (Cycle 9)

### Function

This causes the execution of the next block within a running program to be delayed by the programmed DWELL TIME. A dwell time can be used for such purposes as chip breaking.

The cycle becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



Example: NC blocks

```
89 CYCL DEF 9.0 DWELL TIME
```

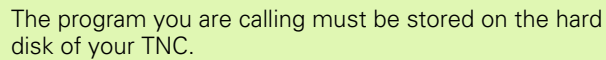
```
90 CYCL DEF 9.1 DWELL 1.5
```

### Cycle parameters



- **Dwell time in seconds:** Enter the dwell time in seconds. Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs and then called like fixed cycles.



If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path, for example **TNC:\KLAR35\FK1\50.H**.

If you want to define a DIN/ISO program to be a cycle, enter the file type .l behind the program name.

As a rule, Q parameters are globally effective when called with Cycle 12. So please note that changes to Q parameters in the called program can also influence the calling program.



Cycle parameters



- **Program name:** Enter the name of the program you want to call and, if necessary, the directory it is located in or
- Activate the file select dialog with the SELECT soft key and select the program to be called

Call the program with

- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

**Example: Designate program 50 as a cycle and call it with M99**

```
55 CYCL DEF 12.0 PGM CALL
56 CYCL DEF 12.1 PGM TNC:\KLAR35\FK1\50.H
57 X+20 FMAX
58 Y+50 FMAX M99
```



## 19.4 ORIENTED SPINDLE STOP (Cycle 13)

### Cycle function



Machine and TNC must be specially prepared by the machine tool builder for use of this cycle.

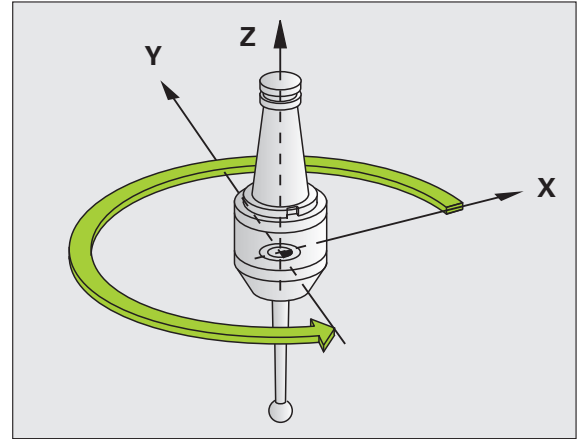
The TNC can control the machine tool spindle and rotate it to a given angular position.

Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set by the machine manufacturer (see your machine manual).



### Example: NC blocks

```
93 CYCL DEF 13.0 ORIENTATION
```

```
94 CYCL DEF 13.1 ANGLE 180
```

### Please note while programming:



Cycle 13 is used internally for Cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

### Cycle parameters



- **Angle of orientation:** Enter the angle referenced to the reference axis of the working plane. Input range: 0.0000° to 360.0000°

# 20

**Touch probe cycles**



## 20.1 General information about touch probe cycles



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



The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe. The machine manual provides further information.

The touch probe cycles are available only with the **Touch probe function** software option (option number 17). If you are using a HEIDENHAIN touch probe, this option is available automatically.

### Method of function

Whenever the TNC runs a touch probe cycle, the 3-D touch probe approaches the workpiece in one linear axis. The machine tool builder determines the probing feed rate in a machine parameter (see "Before You Start Working with Touch Probe Cycles" later in this chapter).

When the probe stylus contacts the workpiece,

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

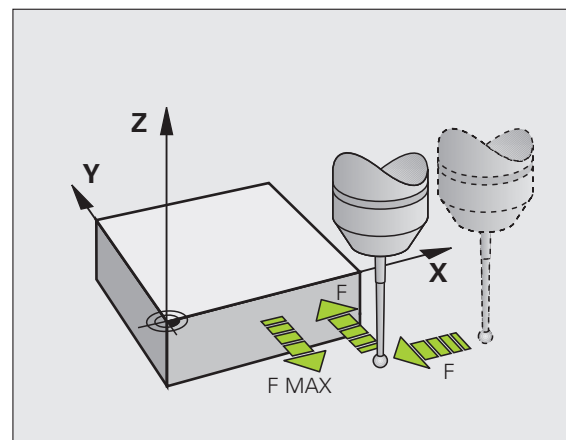
If the stylus is not deflected within a defined distance, the TNC displays an error message (distance: **DIST** from touch probe table).

### Touch probe cycles in the Manual Operation and El. Handwheel modes

In the Manual Operation and El. Handwheel modes, the TNC provides touch probe cycles that allow you to:

- Calibrate the touch probe
- Set datums

The manual touch probe cycles are described in the "Manual operation and setup" chapter (see "Using the 3-D touch probe" on page 291).



## 20.2 Before you start working with touch probe cycles

To make it possible to cover the widest possible range of applications, machine parameters enable you to determine the behavior common to all touch probe cycles.

### Maximum traverse to touch point: **DIST** in touch probe table

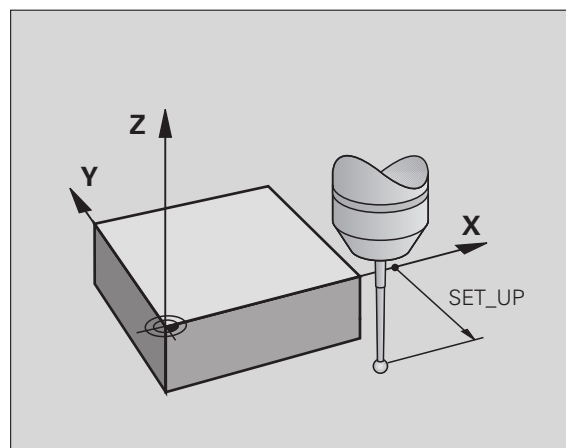
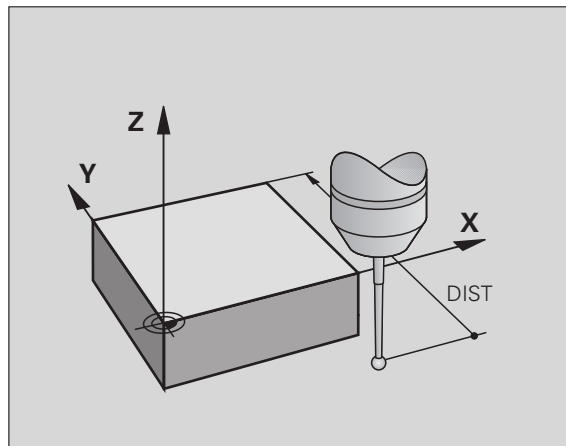
If the stylus is not deflected within the path defined in **DIST**, the TNC outputs an error message.

### Set-up clearance to touch point: **SET\_UP** in touch probe table

In **SET\_UP** you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to **SET\_UP**.

### Orient the infrared touch probe to the programmed probe direction: **TRACK** in touch probe table

To increase measuring accuracy, you can use **TRACK = ON** to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction.



## Touch trigger probe, probing feed rate: **F** in touch probe table

In **F** you define the feed rate at which the TNC is to probe the workpiece.

## Touch trigger probe, rapid traverse for positioning: **FMAX**

In **FMAX** you define the feed rate at which the TNC pre-positions the touch probe, or positions it between measuring points.

## Touch trigger probe, rapid traverse for positioning: **F\_PREPOS** in touch probe table

In **F\_PREPOS** you define whether the TNC is to position the touch probe at the feed rate defined in **FMAX** or at rapid traverse.

- Input value = **FMAX\_PROBE**: Position at feed rate from **FMAX**
- Input value = **FMAX\_MACHINE**: Pre-position at rapid traverse

## Executing touch probe cycles

All touch probe cycles are DEF active. This means that the TNC runs the cycle automatically as soon as the TNC executes the cycle definition in the program run.

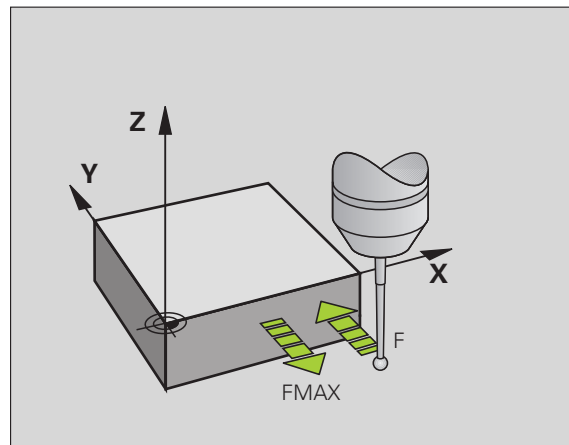


### Danger of collision!

When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

Touch probe cycles with a number greater than 400 position the touch probe according to a positioning logic:

- If the current coordinate of the south pole of the stylus is less than the coordinate of the clearance height (defined in the cycle), the TNC retracts the touch probe in the probe axis to the clearance height and then positions it in the working plane to the first starting position.
- If the current coordinate of the south pole of the stylus is greater than the coordinate of the clearance height, the TNC first positions the probe in the working plane to the first starting position and then moves it immediately to the measuring height in the touch probe axis.



# 20.3 Touch probe table

## General information

Various data is stored in the touch probe table that defines the probe behavior during the probing process. If you run several touch probes on your machine tool, you can save separate data for each touch probe.

## Editing touch probe tables

To edit the touch probe table, proceed as follows:





- 
- ▶ Select the Manual Operation mode
  - ▶ Select the touch probe functions by pressing the TOUCH PROBE soft key. The TNC displays additional soft keys: see table above
  - ▶ Select the touch probe table: Press the TCH PROBE TABLE soft key
  - ▶ Set the EDIT soft key to ON
  - ▶ Using the arrow keys, select the desired setting
  - ▶ Perform desired changes
  - ▶ Exit the touch probe table: Press the END soft key

Table editing

Test run

TNC \> table \ tchprobe.tbl

NO.	TYPE	CAL_OF1	CAL_OF2	CAL_RNG	F	FMAX	DIST
1	TS120	0	0	0	500	+2000	10
2	TS120	0	0	0	500	+2000	10

M

S

T

5100%

OFF

ON

F100%

OFF

ON

Selection of the touch probe?

BEGIN

END

PAGE

PAGE

EDIT

OFF

ON

FIND

END



## Touch probe data

Abbr.	Inputs	Dialog
<b>NO</b>	Number of the touch probe: Enter this number in the tool table (column: <b>TP_NO</b> ) under the appropriate tool number	–
<b>TYPE</b>	Selection of the touch probe used	<b>Selection of touch probe?</b>
<b>CAL_OF1</b>	Offset of the touch probe axis to the spindle axis for the reference axis	<b>TS center misalignmt. ref. axis?</b> [mm]
<b>CAL_OF2</b>	Offset of the touch probe axis to the spindle axis for the minor axis	<b>TS center misalignmt. aux. axis?</b> [mm]
<b>CAL_ANG</b>	The TNC orients the touch probe to the orientation angle before calibration or probing (if orientation is possible)	<b>Spindle angle for calibration?</b>
<b>F</b>	Feed rate at which the TNC is to probe the workpiece	<b>Probing feed rate?</b> [mm/min]
<b>FMAX</b>	Feed rate at which the touch probe pre-positions, or is positioned between the measuring points	<b>Rapid traverse in probing cycle?</b> [mm/min]
<b>DIST</b>	If the stylus is not deflected within the defined path, the TNC outputs an error message	<b>Maximum measuring path?</b> [mm]
<b>SET_UP</b>	In <b>SET_UP</b> you define how far from the defined (or calculated) touch point the TNC is to pre-position the touch probe. The smaller the value you enter, the more exactly you must define the touch point position. In many touch probe cycles you can also define a set-up clearance that is added to the <b>SET_UP</b> machine parameter.	<b>Set-up clearance?</b> [mm]
<b>F_PREPOS</b>	Defining speed with pre-positioning: <ul style="list-style-type: none"> <li>■ Pre-positioning with speed from <b>FMAX</b>: <b>FMAX_PROBE</b></li> <li>■ Pre-positioning with machine rapid traverse: <b>FMAX_MACHINE</b></li> </ul>	<b>Pre-positioning at rap. traverse?</b> ENT/NO ENT
<b>TRACK</b>	To increase measuring accuracy, you can use <b>TRACK = ON</b> to have an infrared touch probe oriented in the programmed probe direction before every probe process. In this way the stylus is always deflected in the same direction: <ul style="list-style-type: none"> <li>■ <b>ON</b>: Perform spindle tracking</li> <li>■ <b>OFF</b>: Do not perform spindle tracking</li> </ul>	<b>Orient touch probe cycles?</b> Yes=ENT, No=NOENT





## 20.4 Fundamentals of tool measurement

### Overview



When running touch probe cycles, Cycle 8 MIRROR IMAGE, Cycle 11 SCALING and Cycle 26 AXIS-SPECIFIC SCALING must not be active.

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



The TNC and the machine tool must be set up by the machine tool builder for use of the TT touch probe.

Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.






The touch probe cycles are available only with the **Touch probe function** software option (option number 17). If you are using a HEIDENHAIN touch probe, this option is available automatically.

In conjunction with the TNC's tool measurement cycles, the tool touch probe enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for at the end of the touch probe cycle. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill
- Tool measurement while the tool is rotating
- Measurement of individual teeth



You can program the cycles for tool measurement in the Programming and Editing mode of operation via the TOUCH PROBE key. The following cycles are available:

Cycle	New format	Page
Calibrating the TT, Cycle 480		Page 474
Measuring the tool length, Cycle 481		Page 476
Measuring the tool radius, Cycle 482		Page 478
Measuring the tool length and radius, Cycle 483		Page 480
Measuring the tool length and radius, Cycle 483		Page 475



The measuring cycles can be used only when the central tool file TOOL.T is active.

Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with **TOOL CALL**.



Setting the machine parameters



Before you start work with the TT cycles, inspect all machine parameters defined in **ProbeSettings** > **CfgToolMeasurement** and **CfgTTRoundStylus**.  
The TNC uses the feed rate for probing defined in **probingFeed** when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows:

$n = \text{maxPeriphSpeedMeas} / (r \bullet 0.0063)$  where

- n Shaft speed [rpm]
- maxPeriphSpeedMeas Maximum permissible surface cutting speed in m/min
- r Active tool radius in mm

The feed rate for probing is calculated from:

$v = \text{meas. tolerance} \bullet n$  where

- v Feed rate for probing in mm/min
- Measuring tolerance Measuring tolerance [mm], depending on maxPeriphSpeedMeas
- n Shaft speed in rpm

probingFeedCalc determines the calculation of the probing feed rate:

probingFeedCalc = ConstantTolerance:

The measuring tolerance remains constant regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (maxPeriphSpeedMeas) and the permissible tolerance (measureTolerance1), the sooner you will encounter this effect.

probingFeedCalc = VariableTolerance:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

Tool radius	Measuring tolerance
Up to 30 mm	measureTolerance1
30 to 60 mm	2 • measureTolerance1
60 to 90 mm	3 • measureTolerance1
90 to 120 mm	4 • measureTolerance1



probingFeedCalc = ConstantFeed:

The feed rate for probing remains constant; the error of measurement, however, rises linearly with the increase in tool radius:

Measuring tolerance =  $r \bullet \text{measureTolerance1} / 5 \text{ mm}$ , where

- r Active tool radius in mm
- measureTolerance1 Maximum permissible error of measurement

Entries in the tool table TOOL.T

Abbr.	Inputs	Dialog
CUT	Number of teeth (20 teeth maximum)	Number of teeth?
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: length?
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Wear tolerance: radius?
DIRECT.	Cutting direction of the tool for measuring the tool during rotation	Cutting direction (M3 = -)?
R_OFFS	Tool length measurement: Tool offset between stylus center and tool center. Default setting: No value entered (offset = tool radius)	Tool offset: radius?
L_OFFS	Tool radius measurement: tool offset in addition to offsetToolAxis between upper surface of stylus and lower surface of tool. Default: 0	Tool offset: length?
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: length?
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: radius?



Input examples for common tool types

Tool type	CUT	TT:R_OFFS	TT:L_OFFS
Drill	– (no function)	0 (no offset required because tool tip is to be measured)	
End mill with diameter < 19 mm	4 (4 teeth)	0 (no offset required because tool diameter is smaller than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from <b>offsetToolAxis</b> is used.)
End mill with diameter > 19 mm	4 (4 teeth)	R (offset required because tool diameter is larger than the contact plate diameter of the TT)	0 (no additional offset required during radius measurement. Offset from <b>offsetToolAxis</b> is used)
Radius cutter	4 (4 teeth)	0 (no offset required because the south pole of the ball is to be measured)	5 (always define the tool radius as the offset so that the diameter is not measured in the radius)



## 20.5 Calibrating the TT (Cycle 480)

### Cycle run

The TT is calibrated with the measuring cycle TCHPROBE 480. The calibration process is automatic. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.

### Please note while programming:



The functioning of the calibration cycle is dependent on machine parameter **CfgToolMeasurement**. Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The position of the TT within the machine working space must be defined by setting the Machine Parameters **centerPos** > [0] to [2].

If you change the setting of any of the Machine Parameters **centerPos** > [0] to [2], you must recalibrate.

### Cycle parameters



- **Clearance height:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999

### Example: NC blocks

```
6 TOOL CALL 1 Z
```

```
7 TCH PROBE 480 CALIBRATE TT
```

```
Q260=+100 ;CLEARANCE HEIGHT
```

## 20.6 Calibrating the wireless TT 449 (Cycle 484)

### Fundamentals

With Cycle 484, you calibrate the wireless infrared TT 449 tool touch probe. The calibration process is not completely automated, because the TT's position on the table is not defined.

### Cycle run

- ▶ Insert the calibrating tool
- ▶ Define and start the calibration cycle
- ▶ Position the calibrating tool manually above the center of the touch probe and follow the instructions in the pop-up window. Ensure that the calibrating tool is located above the measuring surface of the probe contact

The calibration process is semi-automatic. The TNC also measures the center misalignment of the calibrating tool by rotating the spindle by 180° after the first half of the calibration cycle.

The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.



The calibration tool should have a diameter of more than 15 mm and protrude approx. 50 mm from the chuck. This configuration causes a deformation of 0.1 µm per 1 N of probing force.

### Please note while programming:



The functioning of the calibration cycle is dependent on MP6500. Refer to your machine manual.

Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.

The TT needs to be recalibrated if you change its position on the table.

### Cycle parameters

Cycle 484 has no cycle parameters.



## 20.7 Measuring the tool length (Cycle 481)

### Cycle run

To measure the tool length, program Cycle TCH PROBE 481. Via input parameters you can select the desired type of measurement for determining the length of a tool:

- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the tool while it is rotating.
- If the tool diameter is smaller than the diameter of the measuring surface of the TT, or if you are measuring the length of a drill or spherical cutter, you measure the tool while it is at standstill.
- If the tool diameter is larger than the diameter of the measuring surface of the TT, you measure the individual teeth of the tool while it is at standstill.

### Cycle for measuring a tool during rotation

The control determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe and then moving it toward the measuring surface of the TT until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (**TT: R\_OFFS**).

### Cycle for measuring a tool during standstill (e.g. for drills)

The control positions the tool to be measured over the center of the measuring surface. It then moves the non-rotating tool toward the measuring surface of the TT until it touches the surface. To activate this function, enter zero for the tool offset: Radius (**TT: R\_OFFS**) in the tool table.

### Cycle for measuring individual teeth

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in **offsetToolAxis**. You can enter an additional offset with tool offset: Length (**TT: L\_OFFS**) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation.

### Please note while programming:



Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

You can run an individual tooth measurement of tools with **up to 20 teeth**.



Cycle parameters



- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  - 0.0:** Tool is within the tolerance
  - 1.0:** Tool is worn (**LTOL** exceeded)
  - 2.0:** Tool is broken (**LBREAK** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **Clearance height:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Cutter measurement? 0=No / 1=Yes:** Choose whether the control is to measure the individual teeth (maximum of 20 teeth)

Example: NC blocks

6	TOOL CALL 12 Z
7	TCH PROBE 481 TOOL LENGTH
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH



## 20.8 Measuring the tool radius (Cycle 482)

### Cycle run

To measure the tool radius, program Cycle TCH PROBE 482. Via input parameters you can select the desired type of measurement for determining the radius of a tool:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in

**offsetToolAxis**. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the control measures the radius of each tooth with the aid of oriented spindle stops.

### Please note while programming:



Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgToolMeasurement**. Refer to your machine manual.

# Cycle parameters



- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  - 0.0:** Tool is within the tolerance
  - 1.0:** Tool is worn (**RTOL** exceeded)
  - 2.0:** Tool is broken (**RBREAK** exceeded). If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **Clearance height:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Cutter measurement? 0=No / 1=Yes:** Choose whether the control is also to measure the individual teeth (maximum of 20 teeth)

## Example: NC blocks in new format

6	TOOL CALL 12 Z
7	TCH PROBE 482 TOOL RADIUS
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH



## 20.9 Measuring the tool length and radius (Cycle 483)

### Cycle run

To measure the tool length and radius, program Cycle TCH PROBE 483. This cycle is particularly suitable for the first measurement of tools, as it saves time when compared with individual measurement of length and radius. Via input parameters you can select the desired type of measurement:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

The TNC measures the tool in a fixed programmed sequence. First it measures the tool radius, then the tool length. The sequence of measurement is the same as for Cycles 481 and 482.

### Please note while programming:



Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

Cylindrical tools with diamond surfaces can be measured with stationary spindle. To do so, define in the tool table the number of teeth **CUT** as 0 and adjust machine parameter **CfgToolMeasurement**. Refer to your machine manual.

# Cycle parameters



- **Measure tool=0 / Check tool=1:** Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R and the tool length L in the central tool file TOOL.T by the delta values DR = 0 and DL = 0. If you wish to inspect a tool, the TNC compares the measured data with the tool data stored in TOOL.T. The TNC calculates the deviations and enters them as positive or negative delta values DR and DL in TOOL.T. The deviations are also available in the Q parameters Q115 and Q116. If the delta values are greater than the permissible tool tolerances for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- **Parameter number for result?:** Parameter number in which the TNC stores the status of the measurement:
  - 0.0:** Tool is within the tolerance
  - 1.0:** Tool is worn (**LTOL** or/and **RTOL** exceeded)
  - 2.0:** Tool is broken (**LBREAK** or/and **RBREAK** exceeded).
 If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- **Clearance height:** Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from **safetyDistStylus**). Input range -99999.9999 to 99999.9999
- **Cutter measurement? 0=No / 1=Yes:** Choose whether the control is also to measure the individual teeth (maximum of 20 teeth)

## Example: NC blocks in new format

6	TOOL CALL 12 Z
7	TCH PROBE 483 MEASURE TOOL
Q340=1	;CHECK
Q260=+100	;CLEARANCE HEIGHT
Q341=1	;PROBING THE TEETH





# 21

**Tables and overviews**



## 21.1 Machine-specific user parameters

### Application

The parameter values are entered in the **configuration editor**.



To enable you to set machine-specific functions, your machine tool builder can define which machine parameters are available as user parameters. Furthermore, your machine tool builder can integrate additional machine parameters, which are not described in the following, into the TNC.

Refer to your machine manual.

The machine parameters are grouped as parameter objects in a tree structure in the configuration editor. Each parameter object has a name (e.g. **CfgDisplayLanguage**) that gives information about the parameters it contains. A parameter object, also called "entity", is marked with an "E" in the folder symbol in the tree structure. Some machine parameters have a key name to identify them unambiguously. The key name assigns the parameter to a group (e.g. X for X axis). The respective group folder bears the key name and is marked by a "K" in the folder symbol.










If you are in the configuration editor for the user parameters, you can change the display of the existing parameters. In the default setting, the parameters are displayed with short, explanatory texts. To display the actual system names of the parameters, press the key for the screen layout and then the SHOW SYSTEM NAME soft key. Follow the same procedure to return to the standard display.






### Calling the configuration editor

- ▶ Select the **Programming** mode of operation
- ▶ Press the **MOD** key
- ▶ Enter the code number **123**
- ▶ Press the **END** soft key to exit the configuration editor

The icon at the beginning of each line in the parameter tree shows additional information about this line. The icons have the following meanings:

-  Branch exists but is closed
-  Branch is open
-  Empty object, cannot be opened
-  Initialized machine parameter
-  Uninitialized (optional) machine parameter
-  Can be read but not edited
-  Cannot be read or edited

The type of the configuration object is identified by its folder symbol:

-  Key (group name)
-  List
-  Entity or parameter object



Displaying help texts

The **HELP** key enables you to call a help text for each parameter object or attribute.

If the help text does not fit on one page (1/2 is then displayed at the upper right, for example), press the **HELP PAGE** soft key to scroll to the second page.

To exit the help text, press the **HELP** key again.

Additional information, such as the unit of measure, the initial value, or a selection list, is also displayed. If the selected machine parameter matches a parameter in the TNC, the corresponding MP number is shown.

Parameter list

Parameter settings

- DisplaySettings
  - Settings for screen display
    - Sequence of the displayed axes
      - [0] to [5]
        - Depends on the available axes**
      - Type of position display in the position window
        - NOML.**
        - ACTL.**
        - REF ACTL**
        - REF NOML**
        - LAG**
        - DIST**
      - Type of position display in the status display
        - NOML.**
        - ACTL.**
        - REF ACTL**
        - REF NOML**
        - LAG**
        - DIST**
      - Definition of decimal separator for position display
        - .
      - Feed rate display in Manual Operation operating mode
        - at axis key: Display the feed rate only if axis-direction key is pressed**
        - always minimum: Always display the feed rate**
      - Display of spindle position in the position display
        - during closed loop: Display spindle position only if spindle is in position control loop**
        - during closed loop and M5: Display the spindle position only if spindle is in position control loop and M5 is active**
      - Display or hide the PRESET TABLE soft key.
        - True: Preset Table soft key is not displayed**
        - False: Display the Preset Table soft key**



Parameter settings	
DisplaySettings	
Display step for the individual axes	
List of all available axes	
Display step for position display in mm or degrees	
	0.1
	0.05
	0.01
	0.005
	0.001
	0.0005
	0.0001
	0.00005
	0.00001
Display step for position display in inches	
	0.005
	0.001
	0.0005
	0.0001
	0.00005
	0.00001
<hr/>	
DisplaySettings	
Definition of the unit of measure valid for the display	
	<b>metric: Use metric system</b>
	<b>inch: Use inch system</b>
<hr/>	
DisplaySettings	
Format of the NC programs and cycle display	
Program entry in HEIDENHAIN plain language or in DIN/ISO	
	<b>HEIDENHAIN: Program entry in plain language in MDI mode</b>
	<b>ISO: Program entry in MDI mode in DIN/ISO format</b>
Depiction of the cycles	
	<b>TNC_STD: Display the cycles with comments</b>
	<b>TNC_PARAM: Display the cycles without comments</b>
<hr/>	
DisplaySettings	
Behavior during control startup	
	<b>True: Display "Power Interrupted" message</b>
	<b>False: Do not display "Power Interrupted" message</b>
<hr/>	



Parameter settings

DisplaySettings

Settings of the NC and PLC conversational language  
NC conversational language

- ENGLISH
- GERMAN
- CZECH
- FRENCH
- ITALIAN
- SPANISH
- PORTUGUESE
- SWEDISH
- DANISH
- FINNISH
- DUTCH
- POLISH
- HUNGARIAN
- RUSSIAN
- CHINESE
- CHINESE\_TRAD
- SLOVENIAN
- ESTONIAN
- KOREAN
- LATVIAN
- NORWEGIAN
- ROMANIAN
- SLOVAK
- TURKISH
- LITHUANIAN

PLC conversational language  
**See NC conversational language**  
PLC error message language  
**See NC conversational language**  
Language for online help  
**See NC conversational language**

DisplaySettings

Behavior during control startup  
Acknowledge the "Power interrupted" message  
**TRUE: Start-up of the control is not continued until the message has been acknowledged**  
**FALSE: The "Power interrupted" message does not appear**  
Depiction of the cycles  
**TNC\_STD: Display the cycles with comments**  
**TNC\_PARAM: Display the cycles without comments**

DisplaySettings

Settings for program-run graphics  
Type of graphic display  
**High (requires much computing power):The orientation of linear and rotary axes is considered for the program-run graphics (3-D).**  
**Low: Only the orientation of linear axes is considered for the program-run graphics (2.5-D).**  
**Disabled: The program-run graphics are deactivated**



## Parameter settings

### ProbeSettings

#### Configuration of a round stylus

Coordinates of the stylus center

**[0]: X coordinate of the stylus center with respect to the machine datum**

**[1]: Y coordinate of the stylus center with respect to the machine datum**

**[2]: Z coordinate of the stylus center with respect to the machine datum**

Safety clearance above the stylus for pre-positioning

**0.001 to 99 999.9999 [mm]: Set-up clearance in tool-axis direction**

Safety zone around the stylus for prepositioning

**0.001 to 99 999.9999 [mm]: Set-up clearance in the plane perpendicular to the tool axis**

### CfgToolMeasurement

M function for spindle orientation

**-1: Spindle orientation directly by the NC**

**0: Function inactive**

**1 to 999: Number of the M function for spindle orientation**

Probing direction for tool radius measurement

**X\_Positive, Y\_Positive, X\_Negative, Y\_Negative (depending on the tool axis)**

Distance from lower edge of tool to upper edge of stylus

**0.001 to 99.9999 [mm]: Offset of stylus to tool**

Rapid traverse in probing cycle

**10 to 300 000 [mm/min]: Rapid traverse in probing cycle**

Probing feed rate for tool measurement

**1 to 3 000 [mm/min]: Rapid traverse during tool measurement**

Calculation of the probing feed rate

**ConstantTolerance: Calculation of the probing feed rate with constant tolerance**

**VariableTolerance: Calculation of the probing feed rate with variable tolerance**

**ConstantFeed: Constant probing feed rate**

Max. permissible surface cutting speed at the tooth edge

**1 to 129 [m/min]: Permissible surface cutting speed at the circumference of the milling tool**

Maximum permissible speed during tool measurement

**0 to 1 000 [1/min]: Maximum permissible speed**

Maximum permissible measuring error for tool measurement

**0.001 to 0.999 [mm]: First maximum permissible measurement error**

Maximum permissible measuring error for tool measurement

**0.001 to 0.999 [mm]: Second maximum permissible measurement error**

Probing routine

**MultiDirections: Probing from multiple directions**

**SingleDirection: Probing from one direction**



Parameter settings

ChannelSettings  
CH\_NC

Active kinematics

Kinematics to be activated

**List of machine kinematics**

Configuration of the fixed cycles

Overlap factor for pocket milling

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

Display the "Spindle?" error message if M3/M4 is not active

**On: Issue error message**

**Off: No error message**

Display the "Enter a negative depth" error message

**On: Issue error message**

**Off: No error message**

Behavior when moving to wall of slot in the cylinder surface

**LineNormal. Approach on a straight line**

**CircleTangential: Approach on a circular path**

M function for spindle orientation

**-1: Spindle orientation directly by the NC**

**0: Function inactive**

**1 to 999: Number of the M function for spindle orientation**

Specify the behavior of the NC program

Reset the machining time when program starts

**True: Machining time is reset**

**False: Machining time is not reset**



## Parameter settings

Settings for the NC editor

Generate backup files

**TRUE: Generate backup file after editing NC programs**

**FALSE: Do not generate backup file after editing NC programs**

Behavior of the cursor after deletion of lines

**TRUE: Cursor is placed on the preceding line after deletion (iTNC behavior)**

**FALSE: Cursor is placed on the following line after deletion**

Behavior of the cursor on the first or last line

**TRUE: Cursor jumps from end to beginning of program**

**FALSE: Cursor does not jump from end to beginning of program**

Line break with multiline blocks

**ALL: Always display all lines**

**ACT: Only display the lines of the active block completely**

**NO: Only display all lines when block is edited**

Activate help

**TRUE: Always display help graphics during input**

**FALSE: Only display help graphics when the CYCLE HELP soft key is set to ON. The CYCLE HELP OFF/ON soft key is displayed in Programming operating mode after pressing the "Screen layout" key**

Behavior of the soft-key row after a cycle entry

**TRUE: The cycle soft-key row remains active after a cycle definition**

**FALSE: The cycle soft-key row is hidden after a cycle definition**

Safety check when deleting blocks

**TRUE: Display confirmation question when deleting an NC block**

**FALSE: Do not display confirmation question when deleting an NC block**

Line number up to which a test of the NC program is to be run

**100 to 9999: Program length for which the geometry is to be checked**

DIN/ISO programming: Block number increment

**0 to 250: Increments for creation of DIN/ISO blocks in the program**

Line number to which identical syntax elements are searched for

**500 to 9999: Search for cursored elements with up / down arrow keys**

Paths for the end user

List of drives and/or directories

**Drives or directories entered here are shown in the TNC's file manager**

FN 16 output path for execution

**Path for FN 16 output when no path is defined in the program**

FN 16 output path for the Programming and Test Run op. modes

**Path for FN 16 output when no path is defined in the program**

Settings for the file management

Display of dependent files

**MANUAL: Dependent files are displayed**

**AUTOMATIC: Dependent files are not displayed**

Universal Time (Greenwich Mean Time)

Time difference to universal time [h]

**-12 to 13: Time difference in hours relative to Greenwich Mean Time**

Universal time (l)

Time difference to universal time [h]

**-12 to 13: Time difference in hours relative to Greenwich Mean Time**

serial Interface: See "Setting the data interfaces" on page 346.



## 21.2 Pin layouts and connecting cables for the data interfaces

### RS-232-C/V.24 interface for HEIDENHAIN devices



The interface complies with the requirements of EN 50 178 for **low voltage electrical separation**.

When using the 25-pin adapter block:

TNC		Conn. cable 365725-xx			Adapter block 310085-01		Conn. cable 274545-xx		
Male	Assignment	Female	Color	Female	Male	Female	Male	Color	Female
1	Do not assign	1		1	1	1	1	White/Brown	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

When using the 9-pin adapter block:

TNC		Conn. cable 355484-xx			Adapter block 363987-02		Conn. cable 366964-xx		
Male	Assignment	Female	Color	Male	Female	Male	Female	Color	Female
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	White/Green	8	8	8	8	White/Green	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.





### Non-HEIDENHAIN devices

The connector layout of a non-HEIDENHAIN device may substantially differ from that of a HEIDENHAIN device.

It depends on the unit and the type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363987-02		Conn. cable 366964-xx		
Female	Male	Female	Color	Female
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	White/Green	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

### Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	



## 21.3 Technical information

### Explanation of symbols

- Standard
- Axis option
- ◆ Software option 1
- Software option 2

User functions	
Short description	<ul style="list-style-type: none"><li>■ Basic version: 3 axes plus closed-loop spindle</li><li>□ 1st additional axis for 4 axes plus closed-loop spindle</li><li>□ 2nd additional axis for 5 axes plus closed-loop spindle</li></ul>
Program entry	HEIDENHAIN conversational programming
Position data	<ul style="list-style-type: none"><li>■ Nominal positions for lines and arcs in Cartesian coordinates</li><li>■ Incremental or absolute dimensions</li><li>■ Display and entry in mm or inches</li></ul>
Tool compensation	<ul style="list-style-type: none"><li>■ Tool radius in the working plane and tool length</li></ul>
Tool tables	Multiple tool tables with any number of tools
Parallel operation	Creating a program with graphical support while another program is being run
Program jumps	<ul style="list-style-type: none"><li>■ Subroutines</li><li>■ Program-section repeat</li><li>■ Any desired program as subroutine</li></ul>
Fixed cycles	<ul style="list-style-type: none"><li>■ Cycles for drilling, and conventional and rigid tapping</li><li>■ Roughing rectangular pockets and rectangular studs</li><li>■ Cycles for pecking, reaming, boring, and counterboring</li><li>■ Finishing rectangular pockets and rectangular studs</li><li>■ Cartesian and polar point patterns</li><li>■ OEM cycles (special cycles developed by the machine tool builder) can also be integrated</li></ul>
Coordinate transformation	<ul style="list-style-type: none"><li>■ Datum shift, mirroring</li><li>■ Scaling factor (axis-specific)</li></ul>
<b>Q parameters</b> Programming with variables	<ul style="list-style-type: none"><li>■ Mathematical functions =, +, -, *, /, sin <math>\alpha</math>, cos <math>\alpha</math>, root calculation</li><li>■ Logical comparisons (=, <math>\neq</math>, &lt;, &gt;)</li><li>■ Calculating with parentheses</li><li>■ tan <math>\alpha</math>, arc sine, arc cosine, arc tangent, <math>a^n</math>, <math>e^n</math>, ln, log, absolute value of a number, the constant <math>\pi</math>, negation, truncation of digits before or after the decimal point</li><li>■ Functions for calculation of circles</li><li>■ String parameters</li></ul>



User functions	
<b>Programming aids</b>	<ul style="list-style-type: none"> <li>■ Calculator</li> <li>■ Complete list of all current error messages</li> <li>■ Context-sensitive help function for error messages</li> <li>■ Graphic support for the programming of cycles</li> <li>■ Comment blocks in the NC program</li> </ul>
<b>Actual position capture</b>	<ul style="list-style-type: none"> <li>■ Actual positions can be transferred directly into the NC program</li> </ul>
<b>Test run graphics</b> Display modes	<ul style="list-style-type: none"> <li>■ Graphic simulation before program run, even while another program is being run</li> <li>■ Plan view / projection in 3 planes / 3-D view</li> <li>■ Magnification of details</li> </ul>
<b>Programming graphics</b>	<ul style="list-style-type: none"> <li>■ In the Programming mode, the contour of the NC blocks is drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running</li> </ul>
<b>Program Run graphics</b> Display modes	<ul style="list-style-type: none"> <li>■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view</li> </ul>
<b>Machining time</b>	<ul style="list-style-type: none"> <li>■ Calculating the machining time in the Test Run mode of operation</li> <li>■ Display of the current machining time in the Program Run modes</li> </ul>
<b>Returning to the contour</b>	<ul style="list-style-type: none"> <li>■ Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining</li> <li>■ Program interruption, contour departure and return</li> </ul>
<b>Datum tables</b>	<ul style="list-style-type: none"> <li>■ Multiple datum tables, for storing workpiece-related datums</li> </ul>
<b>Touch probe cycles</b>	<ul style="list-style-type: none"> <li>■ Touch probe calibration</li> <li>■ Datum setting, manual</li> <li>■ Cycles for automatic tool measurement</li> </ul>
Specifications	
<b>Components</b>	<ul style="list-style-type: none"> <li>■ Operating panel</li> <li>■ TFT color flat-panel display with soft keys</li> </ul>
<b>Program memory</b>	<ul style="list-style-type: none"> <li>■ Minimum 2 GB</li> </ul>
<b>Input resolution and display step</b>	<ul style="list-style-type: none"> <li>■ Up to 0.1 µm for linear axes</li> <li>■ Up to 0.0001° for rotary axes</li> </ul>
<b>Input range</b>	<ul style="list-style-type: none"> <li>■ Maximum 999 999 999 mm or 999 999 999°</li> </ul>
<b>Block processing time</b> 3-D straight line without radius compensation	<ul style="list-style-type: none"> <li>■ 6 ms</li> </ul>
<b>Axis feedback control</b>	<ul style="list-style-type: none"> <li>■ Position loop resolution: Signal period of the position encoder/1024</li> <li>■ Cycle time of position controller: 3 ms</li> <li>■ Cycle time of speed controller: 200 µs</li> </ul>
<b>Range of traverse</b>	<ul style="list-style-type: none"> <li>■ Maximum 100 m (3937 inches)</li> </ul>



Specifications	
Spindle speed	■ Maximum 100 000 rpm (analog speed command signal)
Error compensation	■ Linear and nonlinear axis error, backlash, thermal expansion ■ Static friction
Data interfaces	■ One each RS-232-C /V.24 max. 115 kilobaud ■ Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with the HEIDENHAIN software TNCremo ■ Ethernet interface 100BaseT Approx. 40 to 80 Mbps (depending on file type and network utilization) ■ 3 x USB 2.0
Ambient temperature	■ Operation: 0 °C to +45 °C ■ Storage: –30 °C to +70 °C
Accessories	
Electronic handwheels	■ One <b>HR 410</b> portable handwheel, or ■ One <b>HR 130</b> panel-mounted handwheel, or ■ Up to three <b>HR 150</b> panel-mounted handwheels via HRA 110 handwheel adapter
Touch probes	■ <b>TS 220</b> : 3-D touch trigger probe with cable connection, or ■ <b>KT 130</b> : Simple touch trigger probe with cable connection ■ <b>TT 140</b> : 3-D touch trigger probe for tool measurement
Touch probe function (option number 17)	
Touch probe cycles	■ Datum setting in manual mode ■ Automatic tool measurement
HEIDENHAIN DNC (option number 18)	
	■ Communication with external PC applications over COM component



**Software option for additional conversational languages** (option number 41)

<b>Additional conversational languages</b>	<input type="checkbox"/> Slovenian
	<input type="checkbox"/> Norwegian
	<input type="checkbox"/> Slovak
	<input type="checkbox"/> Latvian
	<input type="checkbox"/> Korean
	<input type="checkbox"/> Estonian
	<input type="checkbox"/> Turkish
	<input type="checkbox"/> Romanian
	<input type="checkbox"/> Lithuanian

**Python OEM Process** (option number 46)

<input type="checkbox"/> Python applications on the TNC
---



Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5, 4: places before the decimal point, places after the decimal point) [mm]
Tool numbers	0 to 32 767.9 (5, 1)
Tool names	16 characters, enclosed by quotation marks with <b>TOOL CALL</b> . Permitted special characters: #, \$, %, &, -
Delta values for tool compensation	–99.9999 to +99.9999 (2, 4) [mm]
Spindle speeds	0 to 99 999.999 (5, 3) [rpm]
Feed rates	0 to 99 999.999 (5, 3) [mm/min] or [mm/tooth] or [mm/rev]
Dwell time in Cycle 9	0 to 3600.000 (4, 3) [s]
Thread pitch in various cycles	–99.9999 to +99.9999 (2, 4) [mm]
Angle of spindle orientation	0 to 360.0000 (3, 4) [°]
Datum numbers in Cycle 7	0 to 2999 (4, 0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2, 6)
Miscellaneous functions M	0 to 999 (4, 0)
Q parameter numbers	0 to 1999 (4, 0)
Q parameter values	–99 999.9999 to +99 999.9999 (9, 6)
Labels (LBL) for program jumps	0 to 999 (5, 0)
Labels (LBL) for program jumps	Any text string in quotes (" ")
Number of program section repeats REP	1 to 65 534 (5, 0)
Error number with Q parameter function FN14	0 to 1199 (4, 0)



# Overview tables

## Fixed cycles

Cycle number	Cycle designation	DEF active	CALL active
7	Datum shift	■	
8	Mirroring	■	
9	Dwell time	■	
11	Scaling factor	■	
12	Program call	■	
13	Oriented spindle stop	■	
200	Drilling		■
201	Reaming		■
202	Boring		■
203	Universal drilling		■
204	Back boring		■
205	Universal pecking		■
206	Tapping with a floating tap holder, new		■
207	Rigid tapping, new		■
240	Centering		■
241	Single-lip deep-hole drilling		■
247	Datum setting	■	
251	Rectangular pocket (complete machining)		■
256	Rectangular stud (complete machining)		■



## Miscellaneous functions

M	Effect	Effective at block...	Start	End	Page
<b>M0</b>	Program run STOP/Spindle STOP/Coolant OFF			■	Page 249
<b>M1</b>	Optional program STOP/Spindle STOP/Coolant OFF			■	Page 339
<b>M2</b>	Program run STOP/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1			■	Page 249
<b>M3</b>	Spindle ON clockwise		■		Page 249
M4	Spindle ON counterclockwise		■		
M5	Spindle STOP			■	
<b>M6</b>	Tool change/Stop program run (depending on machine parameter)/Spindle STOP			■	Page 249
<b>M8</b>	Coolant ON		■		Page 249
M9	Coolant OFF			■	
<b>M13</b>	Spindle ON clockwise/Coolant ON		■		Page 249
M14	Spindle ON counterclockwise/Coolant ON		■		
<b>M30</b>	Same function as M2			■	Page 249
<b>M89</b>	Vacant miscellaneous function <b>or</b> Cycle call, modally effective (depending on machine parameter)		■	■	Page 367
<b>M91</b>	Within the positioning block: Coordinates are referenced to machine datum		■		Page 250
<b>M92</b>	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position		■		Page 250
<b>M94</b>	Reduce the rotary axis display to a value below 360°		■		Page 252
<b>M99</b>	Blockwise cycle call			■	Page 367
<b>M140</b>	Retraction from the contour in the tool-axis direction		■		Page 255
<b>M141</b>	Suppress touch probe monitoring		■		Page 256



**Symbole**

- 3-D touch probes ... 362, 464
  - Calibrating
    - Triggering ... 297
- 3-D view ... 320

**A**

- Accessories ... 73
- Actual position capture ... 85
- ASCII files ... 271
- Automatic tool measurement ... 146, 472
- Axis-specific scaling ... 453

**B**

- Back boring ... 401
- Backup, data ... 94
- Baud rate, setting the ... 346, 347
- Block
  - Deleting ... 87
  - Inserting, editing ... 87
- BMP files, opening ... 113
- Bolt hole circle ... 376
- Boring ... 393

**C**

- Calculating with parentheses ... 226
- Calculator ... 123
- Centering ... 387
- Circle calculations ... 188
- Code numbers ... 345
- Comments, adding ... 121
- Context-sensitive help ... 132
- Conversational programming ... 83
- Coordinate transformation ... 269, 442
- Copying program sections ... 89
- Cycle
  - Calling ... 366
  - Defining ... 365
- Cycles and point tables ... 383

**D**

- Data backup ... 120
- Data interface
  - Pin layout ... 492
  - Setting ... 346
- Data transfer rate ... 346, 347
- Data transfer software ... 349
- Datum management ... 286
- Datum setting ... 284
  - Without a 3-D touch probe ... 284
- Datum setting, manual
  - Center line as datum ... 307
  - Circle center as datum ... 304
  - In any axis ... 303
- Datum shift ... 269
  - Coordinate input ... 269
  - Resetting ... 270
  - Via datum table ... 270
  - With datum tables ... 444
  - Within the program ... 443
- Datum table
  - Confirming probed values ... 295
- Datum, setting the ... 79
- Deepened starting point for drilling ... 408, 410
- Dialog ... 83
- Directory ... 95, 100
  - Copying ... 103
  - Creating ... 100
  - Deleting ... 105
- Drilling ... 389, 397, 405
  - Deepened starting point ... 408, 410
- Drilling and boring cycles ... 386
- Dwell time ... 459

**E**

- Error messages ... 127
  - Help with ... 127
- Ethernet interface
  - Connecting and disconnecting network drives ... 116
  - Connection possibilities ... 351
  - Introduction ... 351
- Excel files, opening ... 110
- External data transfer
  - iTNC 530 ... 114

**F**

- FCL ... 344
- FCL function ... 8
- Feature content level ... 8
- Feed rate ... 281
  - Changing ... 282
  - Input possibilities ... 84
- Feed rate factor for plunging movements M103 ... 253
- Feed rate in millimeters per spindle revolution M136 ... 254
- File
  - Creating ... 100
- File functions ... 268
- File management ... 95
  - Calling ... 97
  - Copying a file ... 101
  - Copying a table ... 103
  - Deleting a file ... 104
- Directories ... 95
  - Copying ... 103
  - Creating ... 100
- External data transfer ... 114
- File
  - Creating ... 100
  - File name ... 93
  - File selection ... 98
  - File type ... 92
    - External file types ... 94
  - Overview of functions ... 96
  - Overwriting files ... 102
  - Protecting a file ... 108
  - Renaming a file ... 107
  - Tagging files ... 106
- File status ... 97
- FN14: ERROR: Display error messages ... 193
- FN15: PRINT: Formatted output of texts ... 198
- FN18: SYSREAD: Read system data ... 202
- FN19: PLC: Transfer values to the PLC ... 211



## F

FN20: WAIT FOR: NC and PLC synchronization ... 211  
 FN23: CIRCLE DATA: Calculating a circle from 3 points ... 188  
 FN24: CIRCLE DATA: Calculating a circle from 4 points ... 188  
 FN26: TABOPEN: Opening a freely definable table ... 265  
 FN27: TABWRITE: Writing to a freely definable table ... 266  
 FN28: TABREAD: Reading a freely definable table ... 267  
 Form view ... 264  
 Fundamentals ... 76

## G

GIF files, opening ... 113  
 Graphic simulation ... 323  
   Tool display ... 323  
 Graphics  
   Display modes ... 318  
   During programming ... 125  
     Detail enlargement ... 126  
     Magnification of details ... 322  
 Graphics files, opening ... 113

## H

Hard disk ... 92  
 Help files, downloading ... 137  
 Help system ... 132  
 Help with error messages ... 127  
 HTML files, displaying ... 110

## I

Indexed tools ... 148  
 Information on formats ... 498  
 INI files, opening ... 112  
 Internet files, displaying ... 110  
 iTNC 530 ... 56

## J

JPG files, opening ... 113

## L

Local Q parameters, defining ... 182

## M

M functions  
   See "Miscellaneous functions"  
 M91, M92 ... 250  
 Machine axes, moving the ... 278  
   In increments ... 279  
   With the electronic handwheel ... 280  
   With the machine axis direction buttons ... 278  
 Machine parameters for 3-D touch probes ... 465  
 Machine parameter, reading out ... 239  
 Machining patterns ... 368  
 Machining time, measuring the ... 324  
 Machining, interrupting ... 332  
 Mid-program startup ... 335  
   After power failure ... 335  
 Mirror image ... 450  
 Miscellaneous functions  
   Entering ... 248  
   For contouring behavior ... 253  
   For coordinate data ... 250  
   For program run control ... 249  
   For spindle and coolant ... 249  
 MOD function  
   Exiting ... 342  
   Overview ... 343  
   Select ... 342

## N

NC and PLC synchronization ... 211  
 NC error messages ... 127  
 Nesting ... 171  
 Network connection ... 116  
 Nonvolatile Q parameters, defining ... 182

## O

Operating modes ... 60  
 Operating panel ... 59  
 Operating times ... 359  
 Option number ... 344  
 Oriented spindle stop ... 462  
 Output of data to the server ... 201

## P

Parametric programming: See Q parameter programming  
 Part families ... 183  
 Path ... 95  
 Pattern definition ... 368  
 PDF viewer ... 109  
 Pecking ... 405, 409  
   Deepened starting point ... 408, 410  
 Pin layout for data interfaces ... 492  
 Plan view ... 318  
 PLC and NC synchronization ... 211  
 PNG files, opening ... 113  
 Point pattern  
   Cartesian ... 379  
   Polar ... 376  
 Point tables ... 381  
 Positioning  
   With manual data input (MDI) ... 312  
 Positioning logic ... 466  
 Preset table ... 286  
   Confirming probed values ... 296  
 Principal axes ... 77  
 Probed values, writing to datum tables ... 295  
 Probed values, writing to preset table ... 296  
 Probing feed rate ... 466  
 Program  
   Editing ... 86  
   Open new ... 81  
   Structure ... 80  
   Structuring ... 122  
 Program call  
   Any desired program as subroutine ... 169  
   Via cycle ... 460

**P**

Program defaults ... 259  
 Program management: see File management  
 Program name: See File management, File name  
 Program run  
   Executing ... 331  
   Interrupting ... 332  
   Mid-program startup ... 335  
   Optional block skip ... 338  
   Overview ... 330  
   Resuming after an interruption ... 334  
 Program sections, copying ... 89  
 Program-section repeat ... 168  
 Projection in three planes ... 319

**Q**

Q parameter programming ... 180, 230  
   Additional functions ... 192  
   Basic arithmetic (assign, add, subtract, multiply, divide, square root) ... 184  
   Circle calculations ... 188  
   If-then decisions ... 189  
   Programming notes ... 181, 232, 233, 234, 236, 238  
   Trigonometric functions ... 186  
 Q parameters  
   Checking ... 191  
   Formatted output ... 198  
   Local QL parameters ... 180  
   Nonvolatile QR parameters ... 180  
   Preassigned ... 242  
   Transferring values to the PLC ... 211, 213, 214

**R**

Radius compensation ... 156  
   Input ... 157  
 Rapid traverse ... 140  
 Reaming ... 391  
 Rectangular pocket  
   Roughing+finishing ... 427  
 Rectangular stud ... 432  
 Reference points, crossing over ... 276  
 Reference system ... 77  
 Replacing texts ... 91  
 Retraction from the contour ... 255  
 Returning to the contour ... 337  
 Rotary axis  
   Reducing display M94 ... 252

**S**

Scaling factor ... 452  
 Screen ... 57  
 Screen layout ... 58  
 Screen output of data ... 201  
 Search function ... 90  
 Secondary axes ... 77  
 Single-lip deep-hole drilling ... 409  
 Software number ... 344  
 SPEC FCT ... 258  
 Special functions ... 258  
 Specifications ... 494  
 Spindle speed, changing the ... 282  
 Spindle speed, entering ... 151  
 SQL commands ... 215  
 Status display ... 63  
   Additional ... 64  
   General ... 63  
 String parameters ... 230  
 Structuring programs ... 122  
 Subprogram ... 167  
 Switch-off ... 277  
 Switch-on ... 276

**T**

Table access ... 215  
 Tapping  
   With a floating tap holder ... 416  
   Without floating tap holder ... 418  
 Teach in ... 85, 163  
 Test run  
   Executing ... 329  
   Overview ... 326  
   Speed setting ... 317  
 Text file  
   Delete functions ... 272  
   Finding text sections ... 274  
   Opening and exiting ... 271  
 Text files, opening ... 112  
 Text variables ... 230  
 TNCguide ... 132  
 TNCremo ... 349  
 TNCremoNT ... 349  
 Tool change ... 152  
 Tool compensation  
   Length ... 155  
   Radius ... 156

**T**

Tool data  
   Calling ... 151  
   Delta values ... 143  
   Entering into tables ... 144  
   Entering them into the program ... 143  
   Indexing ... 148  
 Tool length ... 142  
 Tool measurement ... 146, 472  
   Calibrating the TT ... 474, 475  
   Machine parameters ... 471  
   Measuring tool length and radius ... 480  
   Tool length ... 476  
   Tool radius ... 478  
 Tool movements, programming ... 83  
 Tool name ... 142  
 Tool number ... 142  
 Tool radius ... 142  
 Tool table  
   Editing functions ... 148  
   Editing, exiting ... 147  
   Input possibilities ... 144  
 Tool usage file ... 153  
 Tool usage test ... 153  
 Touch probe cycles  
   Manual Operation mode ... 291  
   See User's Manual for Touch Probe Cycles  
 Touch probe data ... 468  
 Touch probe functions, use with mechanical probes or dial gauges ... 310  
 Touch probe monitoring ... 256  
 Touch probe table ... 467  
 TRANS DATUM ... 269  
 Trigonometric functions ... 186  
 Trigonometry ... 186  
 TXT files, opening ... 112

**U**

Unit of measure, selection ... 81  
 Universal drilling ... 397, 405  
 USB devices, connecting/removing ... 117  
 User parameters  
   Machine-specific ... 484



## V

Version numbers ... 345

## W

Window manager ... 71

Working space, monitoring ... 325, 329

Workpiece blank, defining ... 81

Workpiece measurement ... 308

Workpiece positions

    Absolute ... 78

    Incremental ... 78

## Z

ZIP archive ... 111

# HEIDENHAIN

---

## DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

**83301 Traunreut, Germany**

☎ +49 8669 31-0

FAX +49 8669 5061

E-mail: [info@heidenhain.de](mailto:info@heidenhain.de)

---

**Technical support** FAX +49 8669 32-1000

**Measuring systems** ☎ +49 8669 31-3104

E-mail: [service.ms-support@heidenhain.de](mailto:service.ms-support@heidenhain.de)

**TNC support** ☎ +49 8669 31-3101

E-mail: [service.nc-support@heidenhain.de](mailto:service.nc-support@heidenhain.de)

**NC programming** ☎ +49 8669 31-3103

E-mail: [service.nc-pgm@heidenhain.de](mailto:service.nc-pgm@heidenhain.de)

**PLC programming** ☎ +49 8669 31-3102

E-mail: [service.plc@heidenhain.de](mailto:service.plc@heidenhain.de)

**Lathe controls** ☎ +49 8669 31-3105

E-mail: [service.lathe-support@heidenhain.de](mailto:service.lathe-support@heidenhain.de)

---

[www.heidenhain.de](http://www.heidenhain.de)

---

## Touch probes from HEIDENHAIN

help you reduce non-productive time and  
improve the dimensional accuracy of the finished workpieces.

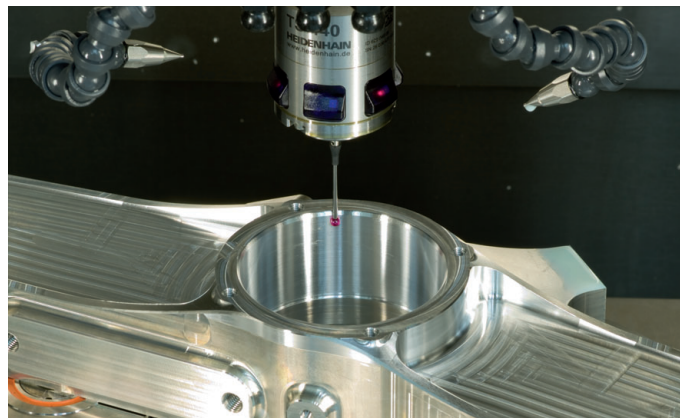
### Workpiece touch probes

**TS 220** Signal transmission by cable

**TS 440, TS 444** Infrared transmission

**TS 640, TS 740** Infrared transmission

- Workpiece alignment
- Setting datums
- Workpiece measurement



### Tool touch probes

**TT 140** Signal transmission by cable

**TT 449** Infrared transmission

**TL** Contact-free laser systems

- Tool measurement
- Wear monitoring
- Tool breakage detection

