0 0 HEIDENHAIN Programming and editing Program run, full sequence Ø BEGIN PGM 17000 MM 1 BLK FORM 0.1 Z X-20 Y-32 Z-53 2 BLK FORM 0.2 IX+40 IY+64 IZ+53 3 L Z+100 R0 FMAX 4 TOOL CALL 51 Z 51000 -----5 L Z+100 R0 FMAX 6 L X+0 Y+0 R0 F9999 7 L Z+1 R0 F9999 M3 8 CYCL DEF 5.0 CIRCULAR POCKET 9 CYCL DEF 5.1 SET UP1 -21 99% S-OVR 15:35 115% F-OVR LIMIT 1 +13.000 Y +0.000+C ų × **A +26.000 2 +100.000 0 S 67.825 I 4 . MAN ACTL T 53 Z 5 1241 M 5/9 UINDOU TRANSFER . + + BLK -DETAIL FORM 0 \odot



iTNC 530

NC Software 340 422-xx 340 423-xx 340 480-xx 340 481-xx

> User's Manual ISO Programming

> > English (en) 4/2003





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					
iTNC 530	340 422-xx					
iTNC 530E	340 423-xx					
iTNC 530, dual-processor version	340 480-xx					
iTNC 530E, dual-processor version	340 481-xx					

The suffix E indicates the export version of the TNC. The export version of the TNC has the following limitations:

Linear movement is possible in no more than 4 axes simultaneously.

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

Some TNC functions have to be implemented by the machine tool builder and are therefore not universally available on all machines. These functions include:

- Probing function for the 3-D touch probe
- Tool measurement with the TT 130
- Rigid tapping
- Returning to the contour after an interruption

5

In addition, the iTNC 530 also has two software option packets that can be enabled by you or your machine tool builder:

Software option 1

Cylinder surface interpolation (Cycles 27 and 28)

Feed rate in mm/min on rotary axes: M116

Tilting the machining plane (Cycle 19 and 3D-ROT soft key in the manual operating mode)

Circle in 3 axes (with tilted working plane)

Software option 2

Block processing time 0.5 ms instead of 3.6 ms

5 axis interpolation

Spline interpolation

3-D machining:

- M114: Automatic compensation of machine geometry when working with tilted axes
- M128: Maintaining the position of the tool tip when positioning with tilted axes (TCPM)
- M144: Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block
- Additional parameters finishing/roughing and tolerance for rotary axes in Cycle 32 (G62)
- LN blocks (3-D compensation)

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.



Touch Probe Cycles User's Manual:

All of the touch probe functions are described in a separate manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID number: 375 319-xx.

Location of use

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.

Functions new since the predecessor versions 340 420-xx and 340 421-xx

- Datum management via the preset table (see "Datum management with the preset table" on page 54)
- New milling cycle RECTANGULAR POCKET (see "RECTANGULAR POCKET (Cycle G251)" on page 283)
- New milling cycle CIRCULAR POCKET (see "CIRCULAR POCKET (Cycle G252)" on page 288)
- New milling cycle SLOT MILLING (see "SLOT MILLING (Cycle G253)" on page 291)
- New milling cycle CIRCULAR SLOT (see "CIRCULAR SLOT (Cycle G254)" on page 295)
- The CYCL CALL POS function provides a new possibility for calling fixed cycles (see "Calling a cycle with G79:G01 (CYCL CALL POS)" on page 225)
- Cycle 205 UNIVERSAL PECKING has been expanded: a deeper starting point for pecking can now be entered (see "UNIVERSAL PECKING (Cycle G205)" on page 244)
- Point pattern on circle cycle has been expanded: Traverse between machining positions is selectable on a straight line or pitch circle (see "CIRCULAR PATTERN (Cycle G220)" on page 325)
- Special features of the iTNC 530 with Windows 2000 (see "iTNC 530 with Windows 2000 (Option)" on page 537)
- Management of dependent files (see "Changing the setting for dependent files" on page 490)
- Testing network connections with the ping monitor (see "Test network connection" on page 488)
- Generating a file with version numbers (see "Code Numbers" on page 477)
- Cycle 210 SLOT WITH RECIPROCATING PLUNGING has been expanded by the parameter for the feed-rate for plunging during finishing (see "SLOT with reciprocating plunge-cut (Cycle G210)" on page 314)
- Cycle 211 CIRCULAR SLOT has been expanded by the parameter for the feed-rate for plunging during finishing (see "CIRCULAR SLOT with reciprocating plunge-cut (Cycle G211)" on page 317)

7

Functions changed since the predecessor versions 340420-xx and 340 421-xx

- The datum shift in datum table function was changed. REF-based datums are no longer available. Instead, the preset table was introduced (see "DATUM SHIFT with datum tables (Cycle G53)" on page 381)
- The function of Cycle 247 was changed. Cycle 247 now activates a preset from the preset table (see "DATUM SETTING (Cycle G247)" on page 384)
- Machine parameter 7475 is now meaningless (see "Compatibility machine parameters for datum tables" on page 518)

New/changed descriptions in this manual

- Meaning of software numbers after the MOD functions have been selected (see "Software Numbers and Option Numbers" on page 476).
- Calling machining cycles (see "Calling a cycle" on page 224)
- Programming example with new milling cycles (see "Example: Milling pockets, studs and slots" on page 320)
- New description of the TE 530 keyboard unit (see "Keyboard" on page 35)
- Overwriting tool data from an external PC (see "Using an external PC to overwrite individual tool data" on page 144)
- Connecting the iTNC directly with a Windows PC (see "Connecting the iTNC directly with a Windows PC" on page 483)

9

Contents

Introduction

Manual Operation and Setup

Positioning with Manual Data Input (MDI)

Programming: Fundamentals of File **Man**agement, Programming Aids

Programming: Tools

Programming: Programming Contours

Programming: Miscellaneous Functions

Programming: Cycles

Programming: Subprograms and Program Section Repeats

Programming: Q Parameters

Test Run and Program Run

MOD Functions

Tables and Overviews

iTNC 530 with Windows 2000 (Option)



1 Introduction 31

1.1 The iTNC 530 32
Programming: HEIDENHAIN conversational and ISO formats 32
Compatibility 32
1.2 Visual Display Unit and Keyboard 33
Visual display unit 33
Screen layout 34
Keyboard 35
1.3 Modes of Operation 36
Manual Operation and Electronic Handwheel 36
Positioning with Manual Data Input (MDI) 36
Programming and editing 37
Test Run 37
Program Run, Full Sequence and Program Run, Single Block 38
1.4 Status Displays 39
"General" status display 39
Additional status displays 40
1.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels 43
3-D touch probes 43
HR electronic handwheels 44

2 Manual Operation and Setup 45

2.1 Switch-On, Switch-Off 46
Switch-on 46
Switch-off 47
2.2 Moving the Machine Axes 48
Note 48
To traverse with the machine axis direction buttons: 48
Traversing with the HR 410 electronic handwheel 49
Incremental jog positioning 50
2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M 51
Function 51
Entering values 51
Changing the spindle speed and feed rate 51
2.4 Datum Setting (Without a 3-D Touch Probe) 52
Note 52
Preparation 52
Datum setting 53
Datum management with the preset table 54
2.5 Tilting the Working Plane (Software Option 1) 59
Application, function 59
Traversing the reference points in tilted axes 60
Setting the datum in a tilted coordinate system 61
Datum setting on machines with rotary tables 61
Datum setting on machines with spindle-head changing systems 61
Position display in a tilted system 61
Limitations on working with the tilting function 61
Activating manual tilting 62

3 Positioning with Manual Data Input (MDI) 63

3.1 Programming and Executing Simple Machining Operations 64
 Positioning with Manual Data Input (MDI) 64
 Protecting and erasing programs in \$MDI 67

4 Programming: Fundamentals of NC, File Management, Programming Aids, Pallet Management 69

```
4.1 Fundamentals ..... 70
        Position encoders and reference marks ..... 70
        Reference system ..... 70
        Reference system on milling machines ..... 71
        Polar coordinates ..... 72
        Absolute and incremental workpiece positions ..... 73
        Setting the datum ..... 74
4.2 File Management: Fundamentals ..... 75
        Files ..... 75
        Data backup ..... 76
4.3 Standard File Management ..... 77
        Note ..... 77
        Calling the file manager ..... 77
        Selecting a file ..... 78
        Deleting a file ..... 78
        Copying a file ..... 79
        Data transfer to or from an external data medium ..... 80
        Selecting one of the last 10 files selected ..... 82
        Renaming a file ..... 82
        Protecting a file / Canceling file protection ..... 83
4.4 Advanced File Management ..... 84
        Note ..... 84
        Directories ..... 84
        Paths ..... 84
        Overview: Functions of the expanded file manager ..... 85
        Calling the file manager ..... 86
        Selecting drives, directories and files ..... 87
        Creating a new directory (only possible on the drive TNC:\) ..... 88
        Copying a single file ..... 89
        Copying a directory ..... 90
        Choosing one of the last 10 files selected ..... 91
        Deleting a file ..... 91
        Deleting a directory ..... 91
        Tagging files ..... 92
        Renaming a file ..... 93
        Additional functions ..... 93
        Data transfer to or from an external data medium ..... 94
        Copying files into another directory ..... 95
        The TNC in a Network ..... 97
```

4.5 Creating and Writing Programs 98 Organization of an NC program in ISO format 98 Define blank form: G30/G31 98 Creating a new part program 99 Programming tool movements 101 Actual position capture 102 Editing a program 103 The TNC search function 107 4.6 Interactive Programming Graphics 109 To generate/not generate graphics during programming: 109 Generating a graphic for an existing program 109 Block number display ON/OFF 110 To erase the graphic: 110 Magnifying or reducing a detail 110 4.7 Structuring Programs 111 Definition and applications 111 Displaying the program structure window / Changing the active window 111 Inserting a structuring block in the (left) program window 111 Selecting blocks in the program structure window 111 4.8 Adding Comments 112 Function 112 Entering comments during programming 112 Inserting comments after program entry 112 Entering a comment in a separate block 112 Functions for editing of the comment 112 4.9 Creating Text Files 113 Function 113 Opening and exiting text files 113 Editing texts 114 Erasing and inserting characters, words and lines 115 Editing text blocks 115 Finding text sections 116 4.10 Integrated Pocket Calculator 117 Operation 117 4.11 Immediate Help for NC Error Messages 118 Displaying error messages 118 Displaying Help texts 118 4.12 Pallet Management 119 Function 119 Selecting a pallet table 121 Leaving the pallet file 121 Executing the pallet file 121

4.13 Pallet Operation with Tool-Oriented Machining 123 Function 123 Selecting a pallet file 128 Setting up the pallet file with the entry form 128 Sequence of tool-oriented machining 132 Leaving the pallet file 133 Executing the pallet file 133

5 Programming: Tools 135

5.1 Entering Tool-Related Data 136 Feed rate F 136 Spindle speed S 136 5.2 Tool Data 137 Requirements for tool compensation 137 Tool numbers and tool names 137 Tool length L 137 Tool radius R 138 Delta values for lengths and radii 138 Entering tool data into the program 138 Entering tool data in tables 139 Editing tool tables 142 Using an external PC to overwrite individual tool data 144 Pocket table for tool changer 145 Calling tool data 147 Tool change 148 5.3 Tool Compensation 150 Introduction 150 Tool length compensation 150 Tool radius compensation 151 5.4 Peripheral Milling: 3-D Radius Compensation with Workpiece Orientation 154 Function 154 5.5 Working with Cutting Data Tables 155 Note 155 Applications 155 Table for workpiece materials 156 Table for tool cutting materials 157 Table for cutting data 157 Data required for the tool table 158 Working with automatic speed / feed rate calculation 159 Changing the table structure 159 Data transfer from cutting data tables 161 Configuration file TNC.SYS 161

6 Programming: Programming Contours 163

7 Programming: Miscellaneous Functions 193

7.1 Entering Miscellaneous Functions M and G38 (STOP) 194
Fundamentals 194
7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant 195
Overview 195
7.3 Miscellaneous Functions for Coordinate Data 196
Programming machine-referenced coordinates: M91/M92 196
Activating the most recently entered datum: M104 198
Moving to positions in a non-tilted coordinate system with a tilted working plane: M130 198
7.4 Miscellaneous Functions for Contouring Behavior 199
Smoothing corners: M90 199
Insert rounding arc between straight lines: M112 200
Do not include points when executing non-compensated line blocks: M124 200
Machining small contour steps: M97 201
Machining open contours: M98 202
Feed rate factor for plunging movements: M103 202
Feed rate in millimeters per spindle revolution: M136 203
Feed rate at circular arcs: M109/M110/M111 204
Calculating the radius-compensated path in advance (LOOK AHEAD): M120 205
Superimposing handwheel positioning during program run: M118 206
Retraction from the contour in the tool-axis direction: M140 207
Suppressing touch probe monitoring: M141 208
Delete modal program information: M142 209
Delete basic rotation: M143 209
7.5 Miscellaneous Functions for Rotary Axes 210
Feed rate in mm/min on rotary axes A, B, C: M116 210
Shorter-path traverse of rotary axes: M126 211
Reducing display of a rotary axis to a value less than 360°: M94 212
Automatic compensation of machine geometry when working with tilted axes: M114 (software option 2) 213
Maintaining the position of the tool tip when positioning with tilted axes (TCPM*):
M128 (software option 2) 214
Exact stop at corners with nontangential transitions: M134 216
Selecting tilting axes: M138 216
Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block: M144 217
7.6 Miscellaneous Functions for Laser Cutting Machines 218
Principle 218
Output the programmed voltage directly: M200 218
Output voltage as a function of distance: M201 218
Output voltage as a function of speed: M202 219
Output voltage as a function of time (time-dependent ramp): M203 219
Output voltage as a function of time (time-dependent pulse): M204 219

8 Programming: Cycles 221

8.1 Working with Cycles 222 Defining a cycle using soft keys 222 Calling a cycle 224 Calling a cycle with G79 (CYCL CALL) 224 Calling a cycle with G79 PAT (CYCL CALL PAT) 224 Calling a cycle with G79:G01 (CYCL CALL POS) 225 Cycle call with M99/89 225 Working with the secondary axes U/V/W 225 8.2 Point Tables 226 Function 226 Creating a point table 226 Selecting a point table in the program 227 Calling a cycle in connection with point tables 228 8.3 Cycles for Drilling, Tapping and Thread Milling 230 Overview 230 PECKING (Cycle G83) 232 DRILLING (Cycle G200) 233 REAMING (Cycle G201) 235 BORING (Cycle G202) 237 UNIVERSAL DRILLING (Cycle G203) 239 BACK BORING (Cycle G204) 241 UNIVERSAL PECKING (Cycle G205) 244 BORE MILLING (Cycle G208) 247 TAPPING with a floating tap holder (Cycle G84) 249 TAPPING NEW with floating tap holder (Cycle G206) 250 RIGID TAPPING (Cycle G85) 252 RIGID TAPPING NEW (Cycle G207) 253 THREAD CUTTING (Cycle G86) 255 TAPPING WITH CHIP BREAKING (Cycle G209) 256 Fundamentals of thread milling 258 THREAD MILLING (Cycle G262) 260 THREAD MILLING/COUNTERSINKING (Cycle G263) 262 THREAD DRILLING/MILLING (Cycle G264) 265 HELICAL THREAD DRILLING/MILLING (Cycle G265) 269 OUTSIDE THREAD MILLING (Cycle G267) 272

8.4 Cycles for Milling Pockets, Studs and Slots 281 Overview 281 RECTANGULAR POCKET (Cycle G251) 283 CIRCULAR POCKET (Cycle G252) 288 SLOT MILLING (Cycle G253) 291 CIRCULAR SLOT (Cycle G254) 295 POCKET MILLING (Cycles G75, G76) 300 POCKET FINISHING (Cycle G212) 302 STUD FINISHING (Cycle G213) 304 CIRCULAR POCKET MILLING (Cycle G77, G78) 306 CIRCULAR POCKET FINISHING (Cycle G214) 308 CIRCULAR STUD FINISHING (Cycle G215) 310 SLOT MILLING (Cycle G74) 312 SLOT with reciprocating plunge-cut (Cycle G210) 314 CIRCULAR SLOT with reciprocating plunge-cut (Cycle G211) 317 8.5 Cycles for Machining Hole Patterns 323 Overview 323 CIRCULAR PATTERN (Cycle G220) 325 LINEAR PATTERN (Cycle G221) 327 8.6 SL Cycles Group I 331 Fundamentals 331 Overview of SL Cycles, Group I 332 CONTOUR GEOMETRY (Cycle G37) 333 PILOT DRILLING (Cycle G56) 334 ROUGH-OUT (Cycle G57) 335 CONTOUR MILLING (Cycle G58/G59) 336 8.7 SL Cycles Group II 337 Fundamentals 337 Overview of SL Cycles 338 CONTOUR GEOMETRY (Cycle G37) 339 Overlapping contours 339 CONTOUR DATA (Cycle G120) 342 PILOT DRILLING (Cycle G121) 343 ROUGH-OUT (Cycle G122) 344 FLOOR FINISHING (Cycle G123) 345 SIDE FINISHING (Cycle G124) 346 CONTOUR TRAIN (Cycle G125) 347 CYLINDER SURFACE (Cycle G127, software option 1) 349 CYLINDER SURFACE slot milling (Cycle G128, software option 1) 351 8.8 SL Cycles with Contour Formula 362 Fundamentals 362 Selecting a program with contour definitions 363 Defining contour descriptions 363 Entering a contour formula 364 Overlapping contours 364 Contour machining with SL Cycles 366 8.9 Cycles for Multipass Milling 370 Overview 370 RUN 3-D DATA (Cycle G60) 371 MULTIPLASS MILLING (Cycle G230) 372 RULED SURFACE (Cycle G231) 374 8.10 Coordinate Transformation Cycles 379 Overview 379 Effect of coordinate transformations 379 DATUM SHIFT (Cycle G54) 380 DATUM SHIFT with datum tables (Cycle G53) 381 DATUM SETTING (Cycle G247) 384 MIRROR IMAGE (Cycle G28) 385 ROTATION (Cycle G73) 387 SCALING FACTOR (Cycle G72) 388 WORKING PLANE (Cycle G80) 389 8.11 Special Cycles 396 DWELL TIME (Cycle G04) 396 PROGRAM CALL (Cycle G39) 397 ORIENTED SPINDLE STOP (Cycle G36) 398 TOLERANCE (Cycle G62) 399

9 Programming: Subprograms and Program Section Repeats 401

9.1 Labeling Subprograms and Program Section Repeats 402
Labels 402
9.2 Subprograms 403
Operating sequence 403
Programming notes 403
Programming a subprogram 403
Calling a subprogram 403
9.3 Program Section Repeats 404
Label G98 404
Operating sequence 404
Programming notes 404
Programming a program section repeat 404
Calling a program section repeat 404
9.4 Separate Program as Subprogram 405
Operating sequence 405
Programming notes 405
Calling any program as a subprogram 406
9.5 Nesting 407
Types of nesting 407
Nesting depth 407
Subprogram within a subprogram 407
Repeating program section repeats 408
Repeating a subprogram 409

10 Programming: Q Parameters 417

10.1 Principle and Overview 418
Programming notes 418
Calling Q parameter functions 419
10.2 Part Families—Q Parameters in Place of Numerical Values 420
Example NC blocks 420
Example 420
10.3 Describing Contours through Mathematical Operations 421
Function 421
Overview 421
Programming fundamental operations 422
10.4 Trigonometric Functions 424
Definitions 424
Programming trigonometric functions 425
10.5 If-Then Decisions with Q Parameters 426
Function 426
Unconditional jumps 426
Programming If-Then decisions 426
Abbreviations used: 427
10.6 Checking and Changing Q Parameters 428
Procedure 428
10.7 Additional Functions 429
Overview 429
D14: ERROR: Output error messages 430
D15: PRINT: Output of texts or Q parameter values 432
D19: PLC: Transferring values to the PLC 432
10.8 Entering Formulas Directly 433
Entering formulas 433
Rules for formulas 435
Programming example 436
10.9 Preassigned Q Parameters 437
Values from the PLC: Q100 to Q107 437
Active tool radius: Q108 437
Tool axis: Q109 437
Spindle status: Q110 438
Coolant on/off: Q111 438
Overlap factor: Q112 438
Unit of measurement for dimensions in the program: Q113 438
Tool length: Q114 438
Coordinates after probing during program run 439
Deviation between actual value and nominal value during automatic tool measurement with the TT 130 439
Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC 439
Results of measurements with touch probe cycles (also see the Touch Probe Cycles User's Manual) 440

11 Test Run and Program Run 449

11.1 Graphics 450
Function 450
Overview of display modes 450
Plan view 451
Projection in 3 planes 452
3-D view 453
Magnifying details 454
Repeating graphic simulation 456
Measuring the machining time 457
11.2 Functions for Program Display 458
Overview 458
11.3 Test Run 459
Function 459
11.4 Program Run 461
Function 461
Running a part program 462
Interrupting machining 463
Moving the machine axes during an interruption 464
Resuming program run after an interruption 465
Mid-program startup (block scan) 466
Returning to the contour 468
11.5 Automatic Program Start 469
Function 469
11.6 Optional Block Skip 470
Function 470
Erasing the "/" character 470
11.7 Optional Program Run Interruption 471
Function 471

12 MOD Functions 473

12.1 MOD Functions 474 Selecting the MOD functions 474 Changing the settings 474 Exiting the MOD functions 474 Overview of MOD functions 474 12.2 Software Numbers and Option Numbers 476 Function 476 12.3 Code Numbers 477 Function 477 12.4 Setting the Data Interfaces 478 Function 478 Setting the RS-232 interface 478 Setting the RS-422 interface 478 Setting the OPERATING MODE of the external device 478 Setting the BAUD RATE 478 Assign 479 Software for data transfer 480 12.5 Ethernet Interface 482 Introduction 482 Connection possibilities 482 Connecting the iTNC directly with a Windows PC 483 Configuring the TNC 485 12.6 Configuring PGM MGT 489 Function 489 Changing the PGM MGT setting 489 Changing the setting for dependent files 490 12.7 Machine-Specific User Parameters 491 Function 491 12.8 Showing the Workpiece in the Working Space 492 Function 492 12.9 Position Display Types 494 Function 494

- 12.10 Unit of Measurement 495 Function 495
- 12.11 Select the Programming Language for \$MDI 496 Function 496
- 12.12 Selecting the Axes for Generating L Blocks 497 Function 497
- 12.13 Enter the Axis Traverse Limits, Datum Display 498
 Function 498
 Working without additional traverse limits 498
 Find and enter the maximum traverse 499
 Datum display 499
- 12.14 Displaying HELP Files 500 Function 500 Selecting HELP files 500
- 12.15 Display Operating Times 501 Function 501
- 12.16 Teleservice 502 Function 502 Calling/exiting teleservice 502
- 12.17 External Access 503 Function 503

13 Tables and Overviews 505

13.1 General User Parameters 506

Input possibilities for machine parameters 506
Selecting general user parameters 506

13.2 Pin Layout and Connecting Cable for the Data Interfaces 519

RS-232-C/V.24 interface for HEIDENHAIN devices 519
Non-HEIDENHAIN devices 520
RS-422/V.11 interface 521
Ethernet interface RJ45 socket 522

13.3 Technical Information 523
13.4 Exchanging the Buffer Battery 529
13.5 Addresses (ISO) 530
G functions 530
Assigned addresses 533
Parameter functions 534

14 iTNC 530 with Windows 2000 (Option) 537

14.1 Introduction 538 General information 538 Specifications 539 14.2 Starting an iTNC 530 Application 540 Logging on to Windows 540 Logging on as a TNC user 540 Logging on as a local administrator 541 14.3 Switching Off the iTNC 530 542 Fundamentals 542 Logging a user off 542 Exiting the iTNC application 543 Shutting down Windows 544 14.4 Network Settings 545 Prerequisite 545 Adjusting the network settings 545 Controlling access 546 14.5 Specifics About File Management 547 The iTNC drive 547 Data transfer to the iTNC 530 548





Introduction

i

1.1 The iTNC 530

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable 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, as well as for machining centers. The iTNC 530 can control up to 9 axes. You can also change the angular position of the spindle under program control.

An integrated hard disk provides storage for as many programs as you like, even if they were created off-line. For quick calculations you can call up the on-screen pocket calculator at any time.

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

Programming: HEIDENHAIN conversational and ISO formats

HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the HEIDENHAIN FK free contour programming does the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining. It is also possible to program in ISO format or DNC mode.

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

Compatibility

The TNC can run all part programs that were written on HEIDENHAIN controls TNC 150 B and later.



1.2 Visual Display Unit and Keyboard

Visual display unit

The TNC is available with either a BF 150 color TFT flat-panel display or the BF 120 color TFT flat-panel display. The figure at top right shows the keys and controls on the BF 150, and the figure at center right shows those of the BF 120.

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 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 lines immediately above the softkey row indicate the number of soft-key rows that can be called with the black arrow keys to the right and left. The line representing the active soft-key row is highlighted.

- 3 Soft-key selection keys
- 4 Switches the soft-key rows
- 5 Sets the screen layout
- 6 Shift key for switchover between machining and programming modes
- 7 Soft-key selection keys for machine tool builders
- 8 Soft-key rows for machine tool builders





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. You could also display the program structure in the right window instead, or display only program blocks in one large window. The available screen windows depend on the selected operating mode.

To change the screen layout:



Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see "Modes of Operation," page 36).



Select the desired screen layout.

i

Keyboard

The TNC is available either with the TE 420 or TE 530 keyboard. The figure at upper right shows the operating elements of the TE 420 keyboard; the figure at center right shows the operating elements of the TE 530 keyboard:

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

Dual-processor version: Additional keys for Windows operation

- 2 File management
 - Pocket calculator
 - MOD function
 - HELP function
- 3 Programming modes
- 4 Machine operating modes
- 5 Initiation of programming dialog
- 6 Arrow keys and GOTO jump command
- 7 Numerical input and axis selection
- 8 Mouse pad: Only for operating the dual-processor version

The functions of the individual keys are described on the inside front cover. Machine panel buttons, e.g. NC START, are described in the manual for your machine tool.







1.3 Modes of Operation

Manual Operation and Electronic 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, set the datums, and tilt the working plane.

The Electronic 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)

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

Manual operation Programming and editing -194.306 DIST. X +1100.000 Y +253.910 Z +1000.000 +9 -69998.995 ACTL. Х +191.570+100.250 YZ +A +C +0.000 +0.000 A -90.0000 B +0.0000 C +0.0000 🔯 Basic rotation +0.0000 ∲-:**1** M 5/9 s 359.938 Z S 2612 T 5 104% S-OVR 14:38 I 93% F-OVR LIMIT 1 INCRE-MENT 3D ROT тоисн SET TOOL TABLE Μ s F they PROBE DATUM

Positioning with Manual Data Input (MDI)

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

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program blocks, right: status display	PGM + STATUS

%\$MDI G71 *				REF	-100.000					+
N10 T0 G19*				ž	+160.462				-	-
NZØ TZ5 619	*			B	+0.000					
N30 T "TAST	ER" G17*					R	+0.000	lo lo		
N20 G00 G90	Z+100*			64		ĉ	+0:000	10		-
N20 G00 G90	X+0 Y+0 Z+1	8 G40 M91*		Ва	asic rota	ion	+0.000	10		_
N30 G00 G90	C+0 B+0 M9	l*								
N40 683 P01	5 P02 -25 P	P03 5 P04 1	Ρ ≫							
N30 6200 DR	ILLING QZ00	+2 ;SE1	-U ≫							_
мээээээээ %	≇MDI G71 *									-
	150% S-0	VR 12:45								
		VR LIMIT 1							s	
X	-99.60	30 <u>Y</u>	:	177.1	337 2	2	-16	57.8	76	7 ₹
C	+0.01	30 B		+0.0	000					
									s	
			_							
HOTE.		ZZ	2		F	0		n 5/9		
			070			1				
STATUS	STATUS	STATUS	COD	ITUS	TOOL	STR	TUS OF			
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 add necessary information. If desired, you can have the programming graphics show the individual steps.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program, right: programming graphics	PGM + GRAPHICS
Left: program blocks, right: program structure	PGM + SECTS



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 work 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 38.



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

Screen windows	Soft key
Program	PGM
Left: program blocks, right: program structure	PGM + SECTS
Left: program, right: status	PGM + STATUS
Left: program, right: graphics	PGM + GRAPHICS
Graphics	GRAPHICS



Soft keys for selecting the screen layout for pallet tables

Screen windows	Soft key
Pallet table	PALLET
Left: program, right: pallet table	PGM + PALLET
Left: pallet table, right: status	PALLET + STATUS
Left: pallet table, right: graphics	PALLET + GRAPHICS

1.4 Status Displays

"General" status display

The status display **1** 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 mode and Electronic Handwheel mode the status display appears in the large window.

Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
XYZ	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
ES M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
*	Program run started
→	Axis locked
\bigcirc	Axis can be moved with the handwheel
	Axes are moving in a tilted working plane
	Axes are moving under a basic rotation



1.4 Status Displ<mark>ays</mark>

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:

\bigcirc							
	PGM						
	+						
	STATUS						

Call the soft-key row for screen layout.

Select the layout option for the additional status display.

To select an additional status display:



Shift the soft-key rows until the STATUS soft keys appear.

STATUS PGM

Select the desired additional status display, e.g. general program information.

You can choose between several additional status displays with the following soft keys:



- 1 Name of main program
- 2 Active programs
- 3 Active machining cycle
- 4 Circle center CC (pole)
- 5 Operating time
- 6 Dwell time counter



STATUS POS.

Positions and coordinates

- 1 Position display
- 2 Type of position display, e.g. actual position
- 3 Tilt angle of the working plane
- 4 Angle of a basic rotation





Information on tools

- T: Tool number and name
 RT: Number and name of a replacement tool
- 2 Tool axis
- 3 Tool lengths and radii
- 4 Oversizes (delta values) from TOOL CALL (PGM) and the tool table (TAB)
- 5 Tool life, maximum tool life (TIME 1) and maximum tool life for TOOL CALL (TIME 2)
- 6 Display of the active tool and the (next) replacement tool

1	Tool	data	T5	TASTI	ER	
	2 <mark>Z</mark> ∬			3 R R2	+3.0000 +3.0000 +0.0000	_
4	TAB PGM	DL +0.2 +0.2	1000 2500	DR +0.1000 +0.1000	DR2 +0.0250 +0.0500	
5	0	CUF 02	R.TIME	TIME1 04:10	TIME2 03:55	
6	TOOL RT ∓	CALL	5	TASTE	R	

STATUS COORD. TRANSF.

Coordinate transformations

- 1 Name of main program
- 2 Active datum shift (Cycle 7)
- 3 Active rotation angle (Cycle 10)
- 4 Mirrored axes (Cycle 8)
- 5 Active scaling factor(s) (Cycles 11 / 26)
- 6 Scaling datum

See "Coordinate Transformation Cycles" on page 379.



CALL LBL

Program section repeats/subprograms STATUS OF

- 1 Active program section repeats with block number, label number, and number of programmed repeats/repeats yet to be run
- 2 Active subprogram numbers with block number in which the subprogram was called and the label number that was called





- 1 Number of the tool to be measured
- 2 Display whether the tool radius or the tool length is being measured
- MIN and MAX values of the individual cutting edges and the 3 result of measuring the rotating tool (DYN = dynamic measurement)
- 4 Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the allowable tolerance in the tool table was exceeded

1	<u>T001</u>	data	Τ5		TASTER	
	10	2		MIN MAX DYN	3	
4						



STATUS OF Active miscellaneous functions M

- List of the active M functions with fixed meaning. 1
- 2 List of the active M functions with function assigned by machine manufacturer.

1	M-Functions M118	
2		



1.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels

3-D touch probes

With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run
- Measure and inspect tools

All of the touch probe functions are described in a separate manual. Please contact HEIDENHAIN if you require a copy of this User's Manual. Id. Nr.: 369 280-xx.

TS 220, TS 630 and TS 632 touch trigger probes

These touch probes are particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable and is a cost-effective alternative for applications where digitizing is not frequently required.

The TS 630 and TS 632 feature infrared transmission of the triggering signal to the TNC. This makes them highly convenient for use on machines with automatic tool changers.

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 TNC, which stores the current position of the stylus as an actual value.



TT 130 tool touch probe for tool measurement

The TT 130 is a triggering 3-D touch probe for tool measurement and inspection. Your TNC provides three cycles for this touch probe with which you can measure the tool length and radius automatically either with the spindle rotating or stopped. The TT 130 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.

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 integral handwheels, HEIDENHAIN also offers the HR 410 portable handwheel (see figure at lower right).











Manual Operation and Setup

2.1 Switch-On, Switch-Off

Switch-on

Switch-on and Traversing the Reference Points can vary depending on the machine tool. Refer to your machine manual.

Switch on the power supply for control and machine. The TNC automatically initiates the following dialog:

MEMORY TEST

The TNC memory is automatically checked.





TNC message that the power was interrupted—clear the message.

TRANSLATE PLC PROGRAM

The PLC program of the TNC is automatically compiled.

RELAY EXT. DC VOLTAGE MISSING

I

Ι

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

The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, 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 traverse the reference points later by pressing the PASS OVER REFERENCE soft key in the Manual Operation mode.

Traversing the reference point in a tilted working plane

The reference point of a tilted coordinate system can be traversed by pressing the machine axis direction buttons. The "tilting the working plane" function must be active in the Manual Operation mode, see "Activating manual tilting," page 62. The TNC then interpolates the corresponding axes.

The NC START button has no function. Pressing this button may result in an error message.



Make sure that the angle values entered in the menu for tilting the working plane match the actual angles of the tilted axis.

Switch-off



iTNC 530 with Windows 2000: See "Switching Off the iTNC 530," page 542.

To prevent data being lost at switch-off, you need to shut down the operating system as follows:

Select the Manual operating mode



Select the function for shutting down, confirm again with the YES soft key.

When the TNC displays the message Now you can switch off the TNC in a superimposed window, you may cut off the power supply to the TNC.



Inappropriate switch-off of the TNC can lead to data loss.

2.2 Mor Note

2.2 Moving the Machine Axes

Traversing with the machine axis direction buttons can vary depending on the machine tool. The machine tool manual provides further information.

To traverse with the machine axis direction buttons:



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 51).

2.2 Moving the Machine Axes

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
- 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.

It is also possible to move the machine axes with the handwheel during a program run.

To move an axis:





X

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.



INCREMENT soft key to ON

Enter the jog increment in millimeters, i.e. 8 mm.

Press the machine axis direction button as often as desired.



2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M

Function

In the Manual Operation and Electronic 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."

_	P]	

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	To enter the spindle speed, press the S soft key.
SPINDLE SPE	EDS =
1000	Enter the desired spindle speed and confirm your entry with the machine START button.
I	·

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 $\mathsf{F},$ 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 MP1020 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 dial for spindle speed is only functional on machines with infinitely variable spindle drive.





2.4 Datum Setting (Without a 3-D Touch Probe)

Note



For datum setting with a 3-D touch probe, refer to the Touch Probe Cycles Manual.

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

Preparation

- Clamp and, if required, align the workpiece.
- ▶ Insert the zero tool with known radius into the spindle.
- Ensure that the TNC is showing actual position values.



Datum setting



Y

 \bigcirc

0

Х

Х

Fragile workpiece?

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*.



呣

Select the Manual Operation mode.

Move the tool slowly until it touches the workpiece surface.

Select an axis (all axes can also be selected via the ASCII keyboard).

DATUM SET Z=



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.

Datum management with the preset table

You should definitely use preset tables if:

- Your machine is equipped with rotary axes (tilting table or swivel head) and you work with the tilted working plan function
- Your machine is equipped with a spindle-head changing system
- Up to now you have been working with older TNC controls with REF-based datum tables
- You wish to machine identical workpieces that are differently aligned

Preset tables 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.

Tabl Rota	e editi tion an	ng gle?				Prog and	ramming editing
File:	PRESET . PR					>>	
NB -	DOC	ROT	x	, a	2		
	Machine Datum	+0	+0	+0	+0		
1	WORKPiece 1	+12.55	+86.2676	+86.2676	+86.2676		
z	Workpiece Z	+5.555	+349.257	+86.2676	+86.2676		
3	Workpiece 3	+0	+100	+0	+442.680	18	
4	Offset Z-Axis	-1	-	-	-72.4641		
5	Workpiece 4	+3.47	+0.4	-70.7635	+2536.9		
6		+0	+86.2676	+86.2676	+86.2676		
7		+12.375	+3.4761	+86.2676	+86.2676	5	.
		10	4% 5-0	VR 14:	42		~
		9	3% F-0	VR LIM	IT 1		
× ₩ A	-190.90 +0.00	06 Y 00+C	+253. +0.	850 Z 000	-345	5.170	0
				S	359.9	138	s 🖡
ACTL.	÷:1	T 5	Z S 2	612 F (8	M 5/9	
BEGIN		PAGE	PAGE	EDIT OFF ON	SAVE PRESET	ACTIVATE PRESET	END

3-D Touch Probe) 2.4 Datum Setting (Without a

Saving the datums in the preset table

The preset table has the name PRESET.PR, and is saved in the directory TNC:\. PRESET.PR is editable only in the **Manual** and **E1**. **Handwhee1** modes. In the Programming and Editing mode you can only read the table, not edit it.

There are several methods for saving datums and/or basic rotations in the preset table:

- Through probing cycles in the Manual or E1. Handwheel modes (see User's Manual, Touch Probe Cycles, Chapter 2)
- Through the probing cycles 400 to 402 and 410 to 419 in automatic mode (see User's Manual, Touch Probe Cycles, Chapter 3)
- By adopting the current datum, which you set manually with the axis keys

Manual entry of values in the preset table is allowed only if there are no tilting devices on your machine. An exception to this rule is the entry of basic rotations in the **ROT** column. The reason is that the TNC compensates the geometry of the tilting device when it saves values in the preset table.

When setting a datum, the TNC checks whether the position of the tilting axes match the corresponding values of the 3D ROT menu (depending on Machine Parameter 7500 bit 5). Therefore:

- If the "Tilt working plane" function is not active, the position displays for the rotary axes must = 0° (zero the rotary axes if necessary).
- If the "Tilt working plane" function is active, the position displays for the rotary axes must match the angles entered in the 3D ROT menu.

The machine manufacturer can lock any lines in the preset table in order to place fixed datums there (e.g. a center point for a rotary table). Such lines in the preset table are shown in a different color (default: red).

Explanation of values saved in the preset table

- Simple machine with three axes without tilting device The TNC saves in the preset table the distance from the workpiece datum to the reference point (including algebraic sign, see figure at upper right)
- Machine with swivel head

ф

The TNC saves in the preset table the distance from the workpiece datum to the reference point (including algebraic sign, see figure at center right)

Machine with rotary table

The TNC saves in the preset table the distance from the workpiece datum to the center of the rotary table (including algebraic sign, see figure at lower right)







Ċ

Editing the preset table

Editing function in table mode	Soft key
Select beginning of table	BEGIN
Select end of table	
Select previous page in table	PAGE
Select next page in table	PAGE
Release/lock preset table for editing	EDIT EDIT
In the selected line of the preset table, save the datum that is active in the Manual operating mode	SAVE PRESET
Activate the datum of the selected line of the preset table	ACTIVATE PRESET
Add the entered number of lines to the end of the table (2nd soft-key row)	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD
Reset the selected line: The TNC enters – in all columns (2nd soft-key row)	INSERT LINE
Insert a single line at the end of the table (2nd soft-key row)	INSERT LINE
Delete a single line at the end of the table (2nd soft-key row)	DELETE

i

Activating the datum from the preset table in the Manual operating mode

Whe rese with	n activating a datum from the preset table, the TNC ts all coordinate transformations that were activated the following cycles:	
 Cycle G53, Datum shift in datum table Cycle G54, Datum shift in program Cycle G28, Mirroring Cycle G73, Rotation Cycle G72, Scaling 		
How Tilter	ever, the coordinate transformation from Cycle G80, d Working Plane, remains active.	
(¹)	Select the Manual Operation mode.	
SET DATUM	Call the function for setting the datum.	
DATUM SET X:	•	
PRESET	Call preset table.	
EDIT OFF ON	Release the preset table for editing: Set the EDIT OFF/ON soft key to ON.	
	With the arrow keys, select the datum number that you want to activate, or	
	With the GOTO key, select the datum number that you want to activate. Confirm with the ENT key.	



i

ACTIVATE PRESET	Activate preset
EXECUTE	Confirm activation of the datum. The TNC sets the display and—if defined—the basic rotation
	Leave the preset table

Activating the datum from the preset table in an NC program

Use Cycle G247 in order to activate datums from the preset table during program run. In Cycle G247 you simply define the number of the datum to be activated (see "DATUM SETTING (Cycle G247)" on page 384).

2.5 Tilting the Working Plane (Software Option 1)

Application, function

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the rotary axes or as angular components of a tilted plane. Refer to your machine manual.

The TNC supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are two functions available for tilting the working plane:

- 3-D ROT soft key in the Manual mode and Electronic Handwheel mode, see "Activating manual tilting," page 62.
- Tilting under program control, Cycle **680 WORKING PLANE** in the part program (see "WORKING PLANE (Cycle G80)" on page 389).

The TNC functions for "tilting the working plane" are coordinate transformations in which the working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the TNC differentiates between two machine types:

Machine with tilting tables

- You must tilt the workpiece into the desired position for machining by positioning the tilting table, for example with a G0 block.
- The position of the transformed tool axis does not change in relation to the machine-based coordinate system. Thus if you rotate the table—and therefore the workpiece—by 90° for example, the coordinate system does not rotate. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in Z+ direction.
- In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).



Machine with swivel head

- You must bring the tool into the desired position for machining by positioning the swivel head, for example with a G0 block.
- The position of the transformed tool axis changes in relation to the machine-based coordinate system. Thus if you rotate the swivel head of your machine—and therefore the tool—in the B axis by 90° for example, the coordinate system rotates also. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in X+ direction of the machine-based coordinate system.
- In calculating the transformed coordinate system, the TNC considers both the mechanically influenced offsets of the particular swivel head (the so-called "translational" components) and offsets caused by tilting of the tool (3-D tool length compensation).

Traversing the reference points in tilted axes

With tilted axes, you use the machine axis direction buttons to cross over the reference points. The TNC interpolates the corresponding axes. Be sure that the function for tilting the working plane is active in the Manual Operation mode and that the actual angle of the tilted axis was entered in the menu field.

Setting the datum in a tilted coordinate system

After you have positioned the rotary axes, set the datum in the same manner as for a non-tilted system. The behavior of the TNC during datum setting depends on machine parameter 7500:

MP 7500, bit 5=0

With an active tilted working plane, the TNC checks during datum setting in the X, Y and Z axes whether the current coordinates of the rotary axes agree with the tilt angles that you defined (3D-ROT menu). If the tilted working plane function is not active, the TNC checks whether the rotary axes are at 0° (actual positions). If the positions do not agree, the TNC will display an error message.

MP 7500, bit 5=1

The TNC does not check whether the current coordinates of the rotary axes (actual positions) agree with the tilt angles that you defined.

ſ	m	L
Ĭ	~	7

If your machine tool is not equipped with axis control, you must enter the actual position of the rotary axis in the menu for manual tilting: The actual positions of one or several rotary axes must match the entry. Otherwise the TNC will calculate an incorrect datum.

Datum setting on machines with rotary tables

If you use a rotary table to align the workpiece, for example with probing cycle G403, you must set the table position value to zero after alignment and before setting the datum in the linear axes X, Y and Z. The TNC will otherwise display an error message. Cycle G403 provides you with an input parameter for this purpose (see User's Manual for Touch Probe Cycles, "Basic Rotation Compensation via Rotary Axis").

Datum setting on machines with spindle-head changing systems

If your machine is equipped with a spindle head changer, you should use the preset table to manage your datums. Datums saved in preset tables account for the active machine kinematics (head geometry). If you exchange heads, the TNC accounts for the new head dimensions so that the active datum is retained.

Position display in a tilted system

The positions displayed in the status window (ACTL. and NOML.) are referenced to the tilted coordinate system.

Limitations on working with the tilting function

- The touch probe function Basic Rotation cannot be used.
- PLC positioning (determined by the machine tool builder) is not possible.

Activating manual tilting



To select manual tilting, press the 3-D ROT soft key. You can now select the desired menu items with the arrow keys.

Enter the tilt angle.

To set the desired operating mode in menu option "Tilt working plane" to Active, select the menu option and switch with the ENT key.



To conclude entry, press the END key.

To reset the tilting function, set the desired operating modes in the menu "Tilt working plane" to Inactive.

If the tilted working plane function is active and the TNC moves the machine axes in accordance with the tilted axes, the status display shows the symbol \underline{k} .

If you activate the "Tilt working plane" function for the Program Run operating mode, the tilt angle entered in the menu becomes active in the first block of the part program. If you are using **G80 WORKING PLANE** in the part program, the angular values defined in the cycle (starting at the cycle definition) are effective. Angle values entered in the menu will be overwritten.









Positioning with Manual Data Input (MDI)

3.1 Programming and Executing Simple Machining Operations

The Positioning with Manual Data Input mode of operation is particularly convenient for simple machining operations or prepositioning of the tool. It enables you to write a short program in HEIDENHAIN conversational programming or in ISO format, and execute it immediately. You can also call TNC cycles. The program is stored in the file \$MDI. In the Positioning with MDI operating mode, the additional status displays can also be activated.

Positioning with Manual Data Input (MDI)



- Interactive Programming graphics
- Program run graphics

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.



%\$MDI G71 *	
N10 G99 T1 L+0 R+5 *	Define tool: zero tool, radius 5
N20 T1 G17 S2000 *	Call tool: tool axis Z
	spindle speed 2000 rpm
N30 G00 G40 G90 Z+200 *	Retract tool (rapid traverse)
N40 X+50 Y+50 M3 *	Move the tool at rapid traverse to a position above
	the hole, spindle on
N50 G01 Z+2 F2000 *	Position tool to 2 mm above hole
N60 G200 DRILLING	Define Cycle G200 Drilling
Q200=2 ;SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q201=-20 ;DEPTH	Total hole depth (Algebraic sign=working direction)
Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for pecking
Q2O2=10 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time at top for chip release (in seconds)
Q2O3=+O ;SURFACE COORDINATE	Workpiece surface coordinate
Q204=50 ;2ND SET-UP CLEARANCE	Position after the cycle, with respect to Q203
Q211=0.5 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
N70 G79 *	Call Cycle G200 PECKING
N80 G00 G40 Z+200 M2 *	Retract the tool
N9999999 %\$MDI G71 *	End of program

Straight-line function **G00** (see "Straight line at rapid traverse G00 Straight line with feed rate G01 F..." on page 173), Cycle **G200** Drilling (see "DRILLING (Cycle G200)" on page 233).

i (

Example 2: Correcting workpiece misalignment on machines with rotary tables

Use the 3-D touch probe to rotate the coordinate system. See "Touch Probe Cycles in the Manual and Electronic Handwheel Operating Modes," section "Compensating workpiece misalignment," in the Touch Probe Cycles User's Manual.

Write down the rotation angle and cancel the Basic Rotation.

	Select operating mode: Positioning with MDI.
5° IV	Select the axis of the rotary table, enter the rotation angle you wrote down previously and set the feed rate. For example: G00 G40 G90 C+2.561 F50
	Conclude entry.
I	Press the machine START button: The rotation of the table corrects the misalignment.

i

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:

I	Select the Programming and Editing mode of operation.
PGM MGT	To call the file manager, press the PGM MGT key (program management).
ł	Move the highlight to the \$MDI file.
	To select the file copying function, press the COPY soft key.
TARGET FILE	-
BOREHOLE	Enter the name under which you want to save the current contents of the \$MDI file.
ENT	Press the EXECUTE soft key to start copying.
END	To close the file manager, press the END soft key.

Erasing the contents of the \$MDI file is done in a similar way: Instead of copying the contents, however, you erase them with the DELETE soft key. The next time you select the operating mode Positioning with MDI, the TNC will display an empty \$MDI file.



If you wish to delete \$MDI, then

- you must not have selected the Positioning with MDI mode (not even in the background).
- you must not have selected the \$MDI file in the Programming and Editing mode.

For further information, see "Copying a single file," page 89.





Programming: Fundamentals of NC, File Management, Programming Aids, Pallet Management

4.1 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 the axes pass over them. From the 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.







4.1 F<mark>un</mark>damentals

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 "right-hand rule" is illustrated 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.

The iTNC 530 can control up to 9 axes. 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.





Polar coordinates

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

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

- Polar Radius, the distance from the pole to the position, and the
- Polar Angle, the size of the angle between the reference axis and the line that connects the pole with the position.

See figure at upper right.

Definition of pole and angle reference axis

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

Coordinates of the pole (plane)	Reference axis of the angle
l and J	+X
J and K	+Y
K and I	+Z





i
.1 F<mark>un</mark>damentals

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

Incremental workpiece positions

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

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

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole 4

X = 10 mm Y = 10 mm

Hole 5, referenced to 4	Hole 5, referenced to 4
G91 X= 20 mm	G91 X= 20 mm
G91 Y= 10 mm	G91 Y= 10 mm

Absolute and incremental polar coordinates

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

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







Setting the datum

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. Before setting the datum, you align the workpiece with the machine axes and move the tool in each axis to a known position relative to the workpiece. You then set the TNC display either to zero or to a predetermined position value. 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 "Coordinate Transformation Cycles" on page 379).

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece which is suitable for deducing the dimensions of the remaining workpiece positions.

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 at right shows holes (1 to 4) whose dimensions are shown with respect to an absolute datum with the coordinates X=0 Y=0. The 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 the holes (5 to 7) without further calculations.





4.2 File Management: Fundamentals

Files



Using the MOD function PGM MGT (see "Configuring PGM MGT" on page 489), select between standard and advanced file management.

If the TNC is connected to a network, then use file management with additional functions.

Files in the TNC	Туре	
Programs In HEIDENHAIN format In ISO format	.H .I	
Tables forToolsTool changersPalletsDatumsPointsPresetsCutting dataCutting materials, workpiece materialsDependent data (such as structure items)	.T .TCH .P .D .PNT .PR .CDT .TAB .DEP	
Texts as ASCII files	A	-

When you write a part program on the TNC, you must first enter a file name. The TNC saves the program 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 manage an almost unlimited number of files with the TNC, at least **2000 MB**.

File names

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

PROG20		
File name	File type	

Data backup

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

You can do this with the free backup program TNCBACK.EXE from HEIDENHAIN. Your machine tool builder can provide you with a copy of TNCBACK.EXE.

In addition, you need a floppy disk on which all machine-specific data, such as PLC program, machine parameters, etc., are stored. Please contact your machine tool builder for more information on both the backup program and the floppy disk.



Saving the contents of the entire hard disk (> 2 GB) can take up to several hours. In this case, it is a good idea to save the data outside of working hours, (e.g. overnight), or to use the PARALLEL EXECUTE function to copy in the background while you work.



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.

1

4.3 Standard File Management

Note



The standard file management is best if you wish to save all files in one directory, or if you are well practiced in the file management of old TNC controls.

To use the standard file management, set the MOD function **PGM MGT** (see "Configuring PGM MGT" on page 489) to **Standard.**

Calling the file manager

PGM MGT Press the PGM MGT key: The TNC displays the file management window (see figure at right)

The window shows you all of the files that are stored in the TNC. Each file is shown with additional information:

Display	Meaning
FILE NAME	Name with up to 16 characters and file type
ВҮТЕ	File size in bytes
STATUS	File properties:
E	Program is selected in the Programming and Editing mode of operation.
S	Program is selected in the Test Run mode of operation.
Μ	Program is selected in a program run mode of operation.
Р	File is protected against editing and erasure.
+	This file has dependent files (see "Changing the setting for dependent files" on page 490)

Manual operation		Programming and editing						
		Fil	e name	= <mark>%</mark> T0	CHPRNT.	A		
TNC:\	*.*							
Fil	e n	ame			Bytes	Stat	us	
%TCH	PRN	Т		.A	132			
CVRE	POR	Т		.Α	672			\rightarrow
LOGB	оок			.А	21699			
FRAE	S_2			.CDT	10882			
FRAE	S _ G	в		.CDT	10882			
TEST				.D	959K	S		
521.	н.s	ЕC		.DEP	1472			
522.	н.s	ЕC		.DEP	1472			H
999.	h.S	ЕC		.DEP	1472			
\$MDI				.н	374			S I
1				.н	898		+	0 🕇
31 f	ile	(s)	37842	96 kt	oyte va	cant		-
PAGE	Р	AGE	SELECT	DELETE	COPY		LAST	
1		↓	<u></u>			EXT	FILES	END

Selecting a file

PGM MGT	Call the file manager.
Use the arrov file you wish	v keys or the arrow soft keys to move the highlight to the to select:
t t	Moves the highlight up or down file by file in the window.
PAGE PAGE	Moves the highlight up or down page by page in the
, ↓ T	window.
Oľ	To select the file: Press the SELECT soft key or the ENT key.
ENT	
Deleting	a file
PGM MGT	Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to delete:



Moves the highlight up or down **file by file** in the window.



Moves the highlight up or down **page by page** in the window.



To delete the file: Press the DELETE soft key.



i

Copying a file



Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to copy:



Moves the highlight up or down **file by file** in the window.



Moves the highlight up or down **page by page** in the window.



To copy the file: Press the COPY soft key.

TARGET FILE =

Enter the new name, and confirm your entry with the EXECUTE soft key or the ENT key. A status window appears on the TNC, informing you about the copying progress. As long as the TNC is copying, you can no longer work, or

If you wish to copy very long programs, enter the new file name and confirm with the PARALLEL EXECUTE soft key. The file will now be copied in the background, so you can continue to work while the TNC is copying.



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

Data transfer to or from an external data medium

PGM MGT

EXT

Before you can transfer data to an external data medium, you must set up the data interface (see "Setting the Data Interfaces" on page 478).

Call the file manager.

Activate data transfer: Press the EXT soft key. In the left half of the screen (1) the TNC shows all files saved on its hard disk. In the right half of the screen (2) it shows all files saved on the external data medium.

peration	Pro Fil	grammi e name	.ng ? = <mark>2</mark>	and e	diti NT.A	ng		I
TNC:∖*.* File name		Bytes State	15	RSZ3Z:*. INO DIR1		2	2	
XTCHPRNT	. A	132						
CUREPORT	.A	672						-
LOGBOOK	.A	21699						
FRAES_2	. CDT	10882						
FRAES_GB	. CDT	10332						
TEST	. D	959K S						
521.H.SEC	.DEP	1472						-
522.H.SEC	.DEP	1472						
999.h.SEC	.DEP	1472						
\$MDI	.н	374						S
1	.н	898	۰,					
81 file(s) 3	784296 kb	yte vacant						s J
PAGE	PAGE	COPY		EXT	TAG	TNC		END

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



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

Moves the highlight up and down within a window.

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.

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.

Tagging functions	Soft key
Tag a single file	TAG FILE
Tag all files	TAG ALL FILES
Untag a single file	UNTAG FILE
Untag all files	UNTAG ALL FILES
Copy all tagged files	COPY THE





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

If you wish to transfer more than one file or longer files, press the PARALLEL EXECUTE soft key. The TNC then copies the file in the background.



To stop transfer, press the TNC soft key. The standard file manager window is displayed again.

Selecting one of the last 10 files selected





Renaming a file



Or Ent

Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to rename:



Moves the highlight up or down **file by file** in the window.



Moves the highlight up or down **page by page** in the window.



Press the RENAME soft key to select the renaming function

TARGET FILE =

Enter the name of the new file and confirm your entry with the ENT key or EXECUTE soft key.

i

Protecting a file / Canceling file protection

Call the file manager. PGM MGT Use the arrow keys or arrow soft keys to move the highlight to the file you wish to protect or whose protection you wish to cancel: Moves the highlight up or down file by file in the ¥ window. Moves the highlight up or down page by page in the window. To enable file protection: Press the PROTECT soft PROTECT) 🔒 key. The file now has status P, or Press the UNPROTECT soft key to cancel file UNPROTECT ີ 🖬 protection. The P status is canceled.

4.4 Advanced File Management

Note

G

Use the advanced file manager if you wish to keep your files in individual directories.

To use it, set the MOD function PGM MGT (see "Configuring PGM MGT" on page 489).

See also "File Management: Fundamentals" on page 75.

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.



The TNC can manage up to 6 directory levels!

If you save more than 512 files in one directory, the TNC no longer sorts them alphabetically!

Directory names

The name of a directory can contain up to 16 characters and does not have an extension. If you enter more than 16 characters for the directory name, the TNC will display an error message.

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 "\".

Example

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

TNC:\AUFTR1\WCPROG\PROG1.H

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





Overview: Functions of the expanded file manager

Function	Soft key
Copy (and convert) individual files	
Select target directory	E.
Display a specific file type	
Display the last 10 files that were selected	FILES
Erase a file or directory	DELETE
Tag a file	ТАБ
Rename a file	RENAME RBC = XYZ
Protect a file against editing and erasure	PROTECT
Cancel file protection	
Manage network drives	NET
Copy a directory	COPY DIR ←
Display all the directories of a particular drive	
Delete directory with all its subdirectories	



Calling the file manager



PGM MGT Press the PGM MGT soft key: The TNC displays the file management window. (The figure at right shows the basic settings. If the TNC shows a different screen layout, press the WINDOW soft key.)

The narrow window on the left **1** 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, RS422, Ethernet), which can be used, for example, to connect a personal computer. A directory is always identified by a folder symbol to the left and the directory name to the right. The control displays a subdirectory to the right of and below its parent directory. A box with the + symbol in front of the folder symbol indicates that there are further subdirectories, which can be shown with the -/+ key or ENT.

The wide window on the right 2 shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

Manual operation	Prog File	rammi name	ng and = <mark>BLK.</mark>	l edi H	ting		
	1	TNC:\SCRE	ENDUMP*.*	Bytes S	2 Status Date	Time	Ţ
RS422:\	- C.	1E	.н	478	+ 10-05-2002	08:29:22	
Albert		1F	.н	470	+ 10-05-2002	2 08:29:10	\rightarrow
B Muell		168	.н	468	+ 10-06-2002	08:29:12	
		11	.н	450	+ 19-07-2003	2 08:39:54	
PROSPEKT PROSPERT		1NL	.н	484	+ 10-06-2003	08:29:12	
- Screendump		15	.н	542	+ 10-05-2003	2 08:29:12	
B WORLD : N		3507	.н	1102	10-05-2003	08:29:12	-
		35071	.н	542	+ 10-05-2003	2 08:29:12	
		3516	.н	1306	+ 10-06-2003	2 08:29:12	
		3DJOINT	.н	634	10-06-2002	08:29:20	S L
		BLK	.н	72	E + 10-06-2003	08:29:12	
		50 file(5) 3784296 kl	byte vaca	int		s I
PAGE PI	AGE	SELECT			WINDOW		END

Display	Meaning
FILE NAME	Name with up to 16 characters and file type
ВҮТЕ	File size in bytes
STATUS	File properties:
E	Program is selected in the Programming and Editing mode of operation.
S	Program is selected in the Test Run mode of operation.
Μ	Program is selected in a program run mode of operation.
Р	File is protected against editing and erasure.
DATE	Date the file was last changed
TIME	Time the file was last changed

1

Selecting drives, directories and files

PGM MGT Call the file manager.

With the arrow keys or the soft keys, you can move the highlight to the desired position on the screen:



1st step: Select a drive

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



Select a drive: Press the SELECT soft key or the ENT key.

2nd step: 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.

3rd step: select a file



Creating a new directory (only possible on the drive TNC:\)

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



j

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 displays a soft-key row with soft keys for different functions.
- Press the "Select target directory" soft key to select the desired directory in a pop-up window. After the target directory has been selected, the corresponding path is indicated in the header. Use the Backspace key to position the cursor directly at the end of the path name and enter the name of the destination file.
- EXECUTE
- Enter the name of the destination file and confirm your entry with the ENT key or EXECUTE soft key: The TNC copies the file into the active directory or into the selected destination directory. The original file is retained, or
- PARALLEL

Press the PARALLEL EXECUTE soft key to copy the file in the background. Copying in the background permits you to continue working while the TNC is copying. This can be useful if you are copying very large files that take a long time. While the TNC is copying in the background you can press the INFO PARALLEL EXECUTE soft key (under MORE FUNCTIONS, second soft-key row) to check the progress of copying.

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

Copying a table

If you are copying tables, you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:

- The target table must exist.
- The file to be copied must only contain the columns or lines you want to replace.



The **REPLACE FIELDS** soft key does not appear when you want to overwrite the table in the TNC with an external data transfer software, such as TNCremoNT. Copy the externally created file into a different directory, and then copy the desired fields with the TNC file management.

Example

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

- Tool number (column T)
- Tool length (column L)
- Tool radius (column R)

Copy this file to a directory other than the one containing the previous TOOL.T. If you wish to copy this file over 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 this copying process the new TOOL.T table consists of 10 lines. The only remaining columns in the table are tool number, tool length and tool radius.
- Or, if you press the REPLACE FIELDS soft key, the TNC merely overwrites the first 10 lines of the columns number, length and radius in the TOOL.T file. The TNC does not change the data in the other lines and columns.

Copying a directory

Move the highlight in the left window onto the directory you want to copy. Instead of the COPY soft key, press the COPY DIR soft key. Subdirectories are also copied at the same time.

Choosing one of the last 10 files selected

PGM MGT	Call the file manager.
LAST FILES	Display the last 10 files selected: Press the LAST FILES soft key.
Use the arrow k	xeys to move the highlight to the file you wish to select: Moves the highlight up and down within a window.
Or ENT	Select a drive: Press the SELECT soft key or the ENT key.



Deleting a file

▶ Move the highlight to the file you want to delete.

- DELETE
- To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to erase the file.
- ▶ To confirm, press the YES soft key;
- ▶ To abort erasure, press the NO soft key.

Deleting a directory

- Delete all files and subdirectories stored in the directory that you wish to erase.
- ▶ 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 erase the directory.
 - ▶ To confirm, press the YES soft key;
- ▶ To abort erasure, press the NO soft key.

Tagging files

Tagging functions	Soft key
Tag a single file	TAG FILE
Tag all files in the directory	TRG ALL FILES
Untag a single file	UNTAG FILE
Untag all files	UNTAG ALL FILES
Copy all tagged files	
Some functions, such as copying or erasing files, ca for individual files, but also for several files at once. T proceed as follows:	n not only be used o tag several files,

Move the highlight to the first file.

TAG	To display the tagging functions, press the TAG soft key.
TAG FILE	Tag a file by pressing the TAG FILE soft key.
Move the high	light to the next file you wish to tag:

TAG FILE	To mark more files, press the TAG FILE soft key.
COPY TAG	To copy the tagged files, press the COPY TAG soft key, or
END	Delete the tagged files by pressing END to end the marking function, and then the DELETE soft key to delete the tagged files.

i

Renaming a file

▶ Move the highlight to the file you want to rename.



- Select the renaming function.
- Enter the new file name; the file type cannot be changed.
- ▶ To execute renaming, press the ENT key.

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.
- PROTECT
- ► To enable file protection, press the PROTECT soft key. The file now has status P.
- To cancel file protection, proceed in the same way using the UNPROTECT soft key.

Erasing a directory together with all its subdirectories and files

Move the highlight in the left window onto the directory you want to erase.

	MORE
l	FUNCTIONS

- To select the additional functions, press the MORE FUNCTIONS soft key.
- Press DELETE ALL to erase the directory together with its subdirectories.
- To confirm, press the YES soft key. To abort erasure, press the NO soft key.

Data transfer to or from an external data medium

<u>í</u>

PGM MGT

Before you can transfer data to an external data medium, you must set up the data interface (see "Setting the Data Interfaces" on page 478).

Call the file manager.

Select the screen layout for data transfer: press the WINDOW soft key. In the left half of the screen (1) the TNC shows all files saved on its hard disk. In the right half of the screen (2) it shows all files saved on the external data medium.

Manual operation	Pro Fil	ogramm le nam	ing e = <mark>:</mark>	and edi [.] LK.H	ting		
TNC:\SCREEN	DUMP*.*			TNC: *.*			-
File name		Bytes Sta	tus	File name	Bytes	Status	
1E	.н	478	•	*TCHPRNT	.A 132		
1F	.н	470	+	CUREPORT	.A 672		\rightarrow
168	.н	468	+	LOGBOOK	.A 21699		
1I	.н	450	+	FRAES_2	.CDT 10882		
1NL	.н	484	+	FRAES_GB	.CDT 10882		
15	.н	542	+	TEST	.D 959K	s	
3507	.н	1102		521.H.SEC	.DEP 1472		_
35071	.н	542	+	522.H.SEC	.DEP 1472		A
3516	.н	1306	+	999.h.SEC	.DEP 1472		
BUJOINT	.н	634		SMDI	.н 374		S
BLK	.н	72 E	+	1	.н 898	+	0 T
50 file(s)	3784296 kl	oyte vacant		31 file(s) 37	84296 kbyte va	icant	
	1				2		
PAGE	PAGE	SELECT	CO ABC	PY SELECT	WINDOW	РАТН	END

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



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

Moves the highlight up and down within a window.

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.

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.

	Transfer a single file: Press the COPY soft key, or
TAG	Transfer several files: Press the TAG soft key (in the second soft-key row, see "Tagging files," page 92), or



Transfer all files: Press the TNC => EXT soft key.

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

If you wish to transfer more than one file or longer files, press the PARALLEL EXECUTE soft key. The TNC then copies the file in the background.



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 PATH soft key. Select the desired directory in the pop-up window by using the arrow keys and the ENT key.

Copying files into another directory

- Select the screen layout with the 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 with the files that you wish to copy and press ENT to display them.



Display 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 92.

If you have marked files in 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, press the YES soft key, or
- ▶ To overwrite no files, press the NO soft key, or
- To confirm each file separately before overwriting it, press the CONFIRM soft key.

If you wish to overwrite a protected file, this must also be confirmed or aborted separately.

i

The TNC in a Network



To connect the iTNC with Windows 2000 to your network, see "Network Settings," page 545.

The TNC logs error messages during network operation (see "Ethernet Interface" on page 482).

If the TNC is connected to a network, the directory window 1 displays up to 7 drives (see figure at right). All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been given the corresponding rights.

Connecting and disconnecting a network drive

PGM MGT

NET

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.

To manage the network drives: Press the NETWORK soft key (second soft-key row). In the right-hand window 2 the TNC shows the network drives available for access. With the soft keys described below you can define the connection for each drive.

Function	Soft key
Establish network connection. If the connection is active, the TNC shows an M in the Mnt column. You can connect up to 7 additional drives with the TNC.	MOLINT DEVICE
Delete network connection.	
Automatically establish network connection whenever the TNC is switched on. The TNC shows an A in the Auto column if the connection is established automatically.	AUTO MOLINT

Do not establish network connection	
automatically when the TNC is switched on.	

It may take some time to mount a network device. At the upper right of the screen the TNC displays **[READ DIR]** to indicate that a connection is being established. The maximum transmission speed is 2 to 5 MB/s, depending on the type of file being transferred and how busy the network is.

NO AUTO MOLINT

Manager	-							
operation	Prog	rammi	ng and	i ed:	1 t 1	ng		
	Patr	ם = 🛄 ב נ	06:\					l
		TNC : \SCREE	ENDUMP*.*		2			-
₽ ₽C2862:\		File nar	ne	Bytes	Statu	s Date	Tine	
	1	1E	.н	478	+	10-05-200	2 08:29:22	
BHB520		1F	.н	470	+	10-06-200	2 08:29:10	-
Muell		1GB	.н	468	+	10-05-2003	2 08:29:12	
		11	.н	450	+	19-07-200	2 08:39:54	
		1NL	.н	484	+	10-06-200	2 08:29:12	
- Screendung	,	15	.н	542	+	10-05-200	2 08:29:12	
⊕ ∰ WORLD:\		3507	.н	1102		10-06-200	2 08:29:12	-
		35071	.н	542	+	10-05-2003	2 08:29:12	
		3516	.н	1306	+	10-05-2003	2 08:29:12	
		SDJOINT	.н	634		10-05-200	2 08:29:20	S I
		BLK	.н	72	E +	10-06-200	2 08:29:12	
		50 file(s	5) 3784296 k	byte vac	ant			5
								Ť,
PAGE P	AGE	DELETE					MORE	END
T	¥ ∣		E TREE			INE I	FUNCTIONS	END

4.5 Creating and Writing Programs

Organization of an NC program in ISO 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 of a part program automatically depending on MP7220. MP7220 defines the block number increment.

The first block of a program is identified by %, the program name and the active unit of measure (G70/G71).

The subsequent blocks contain information on:

- The workpiece blank
- Tool definitions, tool calls
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

The last block of a program is identified by **N99999999 %**, the program name and the active unit of measure (G70/G71).

Define blank form: G30/G31

Immediately after initiating a new program, you define a cuboid workpiece blank. 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 blank form is defined by two of its corner points:

- MIN point G30: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point G31: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values (with G91).



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

Blocks					
N10 G00	G40	X+10	Y+5	F100	M3
Pa	ath fu	nction		V	Vords
l Block nu	mber				

Creating a new part program

You always enter a part program in the $\ensuremath{\text{Programming}}$ and $\ensuremath{\text{Editing}}$ mode of operation:

 \Rightarrow

Select the **Programming and Editing** mode of operation.



To call the file manager, press the PGM MGT key.

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





690 691 Define absolute/incremental separately for each coordina

Define absolute/incremental input; can be defined separately for each coordinate.



COORDINATES ?



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 blank form in the NC program

%NEW G71 *	Program begin, name, unit of measure
N10 G30 G17 X+0 Y+0 Z-40 *	Spindle axis, MIN point coordinates
N20 G31 G90 X+100 Y+100 Z+0 *	MAX point coordinates
N9999999 %NEW G71 *	Program end, name, unit of measure

The TNC automatically generates the first and last blocks of the program.

If you do not wish to define a blank form, cancel the dialog at Spindle axis Z – XY plane 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.

i

Programming tool movements

To program a block, select an ISO function key on the alphabetic keyboard. You can also use the gray contouring keys to get the corresponding G code.



You only need to make sure that capitalization is active.

Example of a positioning block

G 1 ENT	Start block.
COORDINATES	?
X 10	Enter the target coordinate for the X axis.
Y 5 ENT	Enter the target coordinate for the Y axis, and go to the next question with ENT.
PATH OF THE	CUTTER CENTER
G 40	Select tool movement without radius compensation: Confirm with the ENT key or
641 642	To move the tool to the left or to the right of the contour, select function G41 (to the left) or G42 (to the right) by soft key.
FEED RATE ?	F=
750 ENT	Enter a feed rate of 750 mm/min for this path contour and confirm with the ENT key.
MISCELLANEOU	S FUNCTION M?
3 END	Enter the desired miscellaneous function (e.g. M3 Spindle ON) and press the END key to terminate and save the block.
M120	Select the miscellaneous function the TNC displays in the soft-key row.
The program-bl	ock window displays the following line:

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

Actual position capture

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

- Positioning-block programming.
- Cycle programming.
- Tool definition with function G99.
- 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 softkey 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.

1



Editing a program

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/key
Go to previous page	PAGE
Go to next page	PAGE
Go to beginning of program	BEGIN
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.	T
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	
Function	Soft key/key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	
Delete the selected block	DEL
Insert the block that was last edited or deleted	INSERT LAST NC BLOCK



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 plainlanguage 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

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

-	To 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.

i

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 using the marking function.
- 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 superimposes the soft key CANCEL SELECTION.
- 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, and 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. For reasons of clarity, the text you have inserted remains highlighted.
- To end the marking function, press the CANCEL SELECTION soft key.

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





Changing the block number increment

If you have deleted, moved or added program sections, you can have the TNC renumber the blocks by pressing the ORDER BLOCK NUMBERS soft key:



To renumber the blocks, press the ORDER BLOCK NUMBERS soft key. The TNC superimposes a window where you can enter the block number increment.

Enter the desired block number increment and confirm your entry with the ENT key. The TNC renumbers the complete program.



When the TNC inserts a new NC block, it uses the block number increment that is defined in MP7220.

i

The TNC search function

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

Searching for texts

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

FIND	Select the search function: The TN the search window and displays th functions in the soft-key row (see t functions).	C superimposes e available search table of search
G 40	Enter the text to be searched for. Pl search is case-sensitive.	ease note that the
EXECUTE	Start the search process: The TNC available search options in the soft table of search options on the next	displays the -key row (see the t page).
WHOLE WORD	If required, change the search optic	ons.
EXECUTE	Start the search process: The TNC block containing the text you are set.	moves to the next earching for.
EXECUTE	Repeat the search process: The TN next block containing the text you	IC moves to the are searching for.
END	▶ End the search function.	
Search fu	inctions	Soft key
Search fu Show the last searc search ite	superimposed window containing the h items. Use the arrow keys to select a m and confirm with the ENT key.	Soft key LAST SERRCH ELEMENTS
Search fu Show the last searc search ite Show the possible s the arrow confirm w	superimposed window containing the h items. Use the arrow keys to select a m and confirm with the ENT key. superimposed window containing search items of the current block. Use keys to select a search item and vith the ENT key.	Soft key LAST ELEMENTS
Show the last search search ite Show the possible s the arrow confirm w Show the selection Use the a confirm w	superimposed window containing the h items. Use the arrow keys to select a im and confirm with the ENT key. superimposed window containing search items of the current block. Use keys to select a search item and with the ENT key. superimposed window containing a of the most important NC functions. rrow keys to select a search item and with the ENT key.	Soft key LAST BERCH ELEMENTS
Search fu Show the last searc search ite Show the possible s the arrow confirm w Show the selection Use the a confirm w	superimposed window containing the h items. Use the arrow keys to select a m and confirm with the ENT key. superimposed window containing search items of the current block. Use keys to select a search item and vith the ENT key. superimposed window containing a of the most important NC functions. rrow keys to select a search item and vith the ENT key. he Find/Replace function.	Soft key LAST SEARCH ELEMENTS ULOCK ELEMENTS



Search options	Soft key
Define the search direction.	UPWARD DOWNWARD DOWNWARD
Define the end of the search: With COMPLETE, the search starts at the current block and is continued until the current block is reached again.	COMPLETE BEGIN/END BEGIN/END
Start a new search.	NEW SEARCH

Find/Replace any text

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

P II loquilou,	scient 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.
SEARCH + REPLACE	Activate the Replace function: The TNC superimposes a window for entering the text to be inserted.
G 02	Enter the text to be searched for. Please note that the search is case-sensitive. Then confirm with the ENT key.
G 03	Enter the text to be inserted. Please note that the entry is case-sensitive.
EXECUTE	Start the search process: The TNC displays the available search options in the soft-key row (see the table of search options).
WHOLE WORD OFF ON	▶ If required, change the search options.
EXECUTE	Start the search process: The TNC moves to the next occurrence of the text you are searching for.
EXECUTE	If you wish to replace the text and then move to the next position where the text was found, press the

- to the the REPLACE soft key. If you do not want to replace the text, but move to the next position where the text was found, press the DO NOT REPLACE soft key.
- ▶ End the search function.

1
4.6 Interactive Programming Graphics

To generate/not generate 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 PGM + 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 graphics generated 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 interactive 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



RESET + START

Block number display ON/OFF



Shift the soft-key row.

- ▶ To show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW.
- ► To omit block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT.

To erase the graphic:



Shift the soft-key row.

▶ Delete graphic: Press CLEAR GRAPHIC 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).

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.	$\begin{array}{c c} \leftarrow & \rightarrow \\ \hline \downarrow & \uparrow \end{array}$
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.

With the WINDOW BLK FORM soft key, you can restore the original section.

1

4.7 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 244 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 way.

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.

Displaying the program structure window / Changing the active window



To display the program structure window, select the screen display PGM+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.

INSERT	
SECTION	

〓

HEIDENHAIN iTNC 530

Press the INSERT STRUCTURE soft key or the * key on the ASCII keyboard.

Enter the structuring text with the alphabetic keyboard.

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.





4.8 Adding Comments

Function

You can add comments to any desired block in the part program to explain program steps or make general notes. There are three possibilities for adding comments:

Entering comments during programming

- Enter the data for a program block, then press the semicolon key (;) on the alphabetic keyboard—the TNC displays the dialog prompt COMMENT ?
- Enter your comment and conclude the block by pressing the END key.

Inserting comments after program entry

- Select the block to which a comment is to be added.
- Select any word in the block with the right arrow key, then press the ";" (semicolon) key on the alphabetic keyboard: The TNC displays the dialog prompt Comment?
- Enter your comment and conclude the block by pressing the END key.

Entering a comment in a separate block

- Select the block after which the comment is to be inserted.
- Initiate the programming dialog with the semicolon key (;) on the alphabetic keyboard.
- Enter your comment and conclude the block by pressing the END 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.	MOVE WORD <<
Jump to the end of a word. Words must be separated by a space.	MOVE WORD >>
Switch between insert mode and overwrite mode	INSERT

Manual operation	Programming and editing Comment?	
XNEU G71 N10 G30 N20 G31 * :TOOL N50 G00 N50 G00 N70 G01 N80 G01 N80 G41 N100 G42 N100 K43	<pre> . * . * . * . * . * . * . * . * . * . *</pre>	1
N120 X+5 N130 G26 N140 X+6	(0 Y+0*) 5 R15* 1 Y+50* → HOVE HOVE HOVE UGRO VGRO VGRUTTE	



4.9 Creating Text Files

Function

You can use the TNC's text editor to write and edit texts. Typical applications:

- Recording test results
- Documenting working procedures
- Creating formularies

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.
- ▶ To call the file manager, press the PGM MGT key.
- To 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 one word to the right	MOVE WORD >>
Move one word to the left	MOVE WORD <<
Go to next screen page	PAGE
Go to previous screen page	PAGE
Go to beginning of file	BEGIN
Go to end of file	

Manual operation	Programmi	ng and	d edit.	ing		
File: 3516.A		ine: Ø	Column: 1	INSERT		-
S BEGIN PGM 3516	a mm					
1 BLK FORM 0.1 Z	Z X-90 Y-90 Z-40					
2 BLK FORM 0.2 X	(+90 Y+90 Z+0					
3 TOOL DEF 50						
4 TOOL CALL 1 Z	51400					
L Z+50 R0 F MAX						
4 L X+0 Y+100 R0	€F MAX M3					
7 L Z-20 R0 F MA	ях					
8 L X+0 Y+80 RL	F250					
9 FPOL X+0 Y+0						1
10 FC DR- R80 CC	X+0 CCY+0					_
11 FCT DR- R7,5						
12 FCT DR+ R90 C	CX+69,282 CCY-40					
13 FSELECT 2						
14 FCI DR+ R10 P	UX+0 PDY+0 D20					
						1
UNSERT WO	RD WORD	PAGE	PHGE	BEGIN	END	FIND

Editing functions	Кеу
Begin a new line	RET
Erase the character to the left of the cursor	X
Insert a blank space	SPACE
Switch between upper and lower case letters	SHIFT SPACE

Editing texts

The first line of the text editor is an information headline displaying the file name, and the location and writing mode of the cursor:

File:	Name of the text file
Line:	Line in which the cursor is presently located
Column:	Column in which the cursor is presently located
INSERT:	Insert new text, pushing the existing text to the right
OVERWRITE:	Write over the existing text, erasing it by replacing it with new text

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. A line can have up to 77 characters. To start a new line, press the RET key or the ENT key.

Move the cursor to the word or line that you wish to erase and insert at a different place in the text.

With the text editor, you can erase words and even lines, and insert

Press the DELETE WORD or DELETE LINE soft key: The text is placed in the buffer memory.

Erasing and inserting characters, words and

them at any desired location in the text.

Move the cursor to the location where you wish insert the text, and press the RESTORE LINE/WORD soft key.

Function	Soft key
Delete and temporarily store a line	DELETE
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 carrying out any of these editing functions, you must first select the desired text block:

To select a text block, move the cursor to the first character of the text you wish to select.

SELECT BLOCK

lines

- ▶ 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 text and store temporarily	DELETE BLOCK
Store marked 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.

To transfer the selected text 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 target file. The TNC appends the selected text to the end of the specified file. If no target file with the specified name is found, the TNC creates a new file with the selected text.

To insert 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.
- ▶ To select the search function, press the FIND soft key.
- ▶ Press the FIND CURRENT WORD soft key.
- ▶ To leave the search function, press the END soft key.

Finding any text

- To 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.
- ▶ To find the text, press the EXECUTE soft key.
- ▶ To leave the search function, press the END soft key.



4.10 Integrated Pocket Calculator

Operation

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

- ▶ Use the CALC key to show and hide the on-line pocket 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	:
Sine	S
Cosine	С
Tangent	Т
Arc sine	AS
Arc cosine	AC
Arc tangent	AT
Powers	٨
Square root	Q
Inversion	1
Parenthetic calculations	()
p (3.14159265359)	Р
Display result	=

To transfer the calculated value into the program,

- Select the word into which the calculated value is to be transferred by using the arrow keys.
- Superimpose the on-line calculator by using 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.

Positioning P: with mdi	rogramming and editing	
XNEU G71 N10 G30 G N20 G31 G N40 T1 G1 N50 G00 G N50 X-30 N70 G01 Z N80 G01 X	* 17 X+0 Y+0 Z-40* 90 X+100 Y+100 Z+0* 7 S5000* 40 690 Z+250* Y+50* -5 F20 +5 G=	~
N90 X+50 X N100 G42 (N110 X+10(N120 X+50 N130 G26 F	Y+100*	-
N140 X+0 N150 G00 (BEGIN END	Y+50* 540 X-20* PRCE PRCE FIND	s I -

HELP

4.11 Immediate Help for NC Error Messages

Displaying error messages

The TNC automatically 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 the touch probe system

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. The TNC error messages can be canceled with the CE key, after the cause of the error has been removed.

If you require more information on a particular error message, press the HELP key. A window is then superimposed where the cause of the error is explained and suggestions are made for correcting the error.

Displaying Help texts

- ▶ To display Help, press the HELP key.
- Read the description of the error and the possibilities for correcting it. Close the Help window with the CE key, thus canceling the error message.
- Remove the cause of the error as described in the Help window.

The TNC displays the Help text automatically if the error message is blinking. The TNC needs to be restarted after blinking error messages. To restart the TNC, press and hold the END key for two seconds.

Manual operation Error de	Path c	omp wron	gly st	arted	110	
N40 Counting and Counting and Counter	mroo: radiuscreen radiuscreen radiuscreen radiuscreen radiuscreen z-5 F2 x+0 Y+1 3 Y+100 2 G25 R 100 Y+5 50 Y+0* 50 Y+50* 3 G40 X 100 M2* 39 %NEU	ram a corner rad ram 100* 50 F750* 20* 0* -20* 671 *	us before s	tarting e tool		
			START SINGLE	STOP AT	START	RESET + START

4.12 Pallet Management

Function

Pallet table management is a machine-dependent function. The standard functional range will be described below. Refer to your machine manual for more information.

Pallet tables are used for machining centers with pallet changers: The pallet table calls the part programs that are required for the different pallets, and activates datum shifts or datum tables.

You can also use pallet tables to run in succession several programs that have different datums.

Pallet tables contain the following information:

PAL/PGM (entry obligatory):

Identification for pallet or NC program (select with ENT or NO ENT)

NAME (entry obligatory):

Pallet or program name. The machine tool builder determines the pallet name (see Machine Manual). The program name must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the program.

DATUM (entry optional):

Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle G53 **DATUM SHIFT.**

X, **Y**, **Z** (entry optional, other axes also possible):

For pallet names, the programmed coordinates are referenced to the machine datum. For NC programs, the programmed coordinates are referenced to the pallet datum. These entries overwrite the datum that you last set in the Manual mode of operation. With the miscellaneous function M104 you can reactivate the datum that was last set. With the actual-position-capture key, the TNC opens a window that enables you to have the TNC enter various points as datums (see table below):

Position	Meaning
Actual values	Enter the coordinates of the current tool position referenced to the active coordinate system.
Reference values	Enter the coordinates of the current tool position referenced to the machine datum.
ACTL measured values	Enter the coordinates referenced to the active coordinate system of the datum last probed in the Manual operating mode.
REF measured values	Enter the coordinates referenced to the machine datum of the datum last probed in the Manual operating mode.



With the arrow keys and ENT, select the position that you wish to confirm. Then press the ALL VALUES soft key so that the TNC saves the respective coordinates of all active axes in the pallet table. With the PRESENT VALUE soft key, the TNC saves the coordinates of the axis on which the highlight in the pallet table is presently located.



If you have not defined a pallet before an NC program, the programmed coordinates are then referenced to the machine datum. If you do not define an entry, the datum that was set manually remains active.

Editing function	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	
Insert as last line in the table	INSERT LINE
Delete the last line in the table	DELETE
Go to beginning of next line	NEXT LINE
Add the entered number of lines at the end of the table	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD

Selecting a pallet table

- Call the file manager in the Programming and Editing or Program Run mode: Press the PGM MGT key.
- ▶ To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- Select a pallet table with the arrow keys, or enter a new file name to create a new table.
- Confirm your entry with the ENT key.

Leaving the pallet file

- ▶ To call the file manager, press the PGM MGT soft key.
- To select a different type of file, press the SELECT TYPE soft key and the soft key for the desired file type, for example SHOW.H.
- Select the desired file.

Executing the pallet file



Programs executed over the pallet file must not contain M30 (M02).

In MP7683, set whether the pallet table is to be executed blockwise or continuously (see "General User Parameters" on page 506).

- Select the file manager in the Program Run, Full Sequence or Program Run, Single Block operating modes: Press the PGM MGT key.
- ► To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- ▶ Select the pallet table with the arrow keys and confirm with ENT.
- To execute the pallet table: Press the NC Start button. The TNC executes the pallets as set in MP7683.

Screen layout for executing pallet tables

You can have the TNC display the program contents and pallet file contents on the screen together by selecting the screen layout PGM + PALLET. During execution, the TNC then shows program blocks to the left and the pallet to the right. To check the program contents before execution, proceed as follows:

- ▶ Select a pallet table.
- ▶ With the arrow keys, choose the program you would like to check.
- Press the OPEN PGM soft key: The TNC displays the selected program on the screen. You can now page through the program with the arrow keys.
- ▶ To return to the pallet table, press the END PGM soft key.





4.13 Pallet Operation with Tool-Oriented Machining

Function

Pallet management in combination with tool-oriented machining is a machine-dependent function. The standard functional range will be described below. Refer to your machine manual for more information.

Pallet tables are used for machining centers with pallet changers: The pallet table calls the part programs that are required for the different pallets, and activates datum shifts or datum tables.

You can also use pallet tables to run in succession several programs that have different datums.

Pallet tables contain the following information:

PAL/PGM (entry obligatory):

The entry **PAL** identifies the pallet, **FIX** marks the fixture level and **PGM** is used to enter the workpiece.

W-STATE:

Current machining status. The machining status is used to determine the current stage of machining. Enter **BLANK** for an unmachined (raw) workpiece. During machining, the TNC changes this entry to **INCOMPLETE**, and after machining has finished, to **ENDED**. The entry **EMPTY** is used to identify a space at which no workpiece is to be clamped or where no machining is to take place.

METHOD (entry obligatory):

Entry that determines the method of program optimization. Machining is workpiece-oriented if **WPO** is entered. Machining of the piece is tool-oriented if **TO** is entered. In order to include subsequent workpieces in the tool-oriented machining, you must enter **CTO** (continued tool oriented). Tool-oriented machining is also possible with pallet fixtures, but not for multiple pallets.

NAME (entry obligatory):

Pallet or program name. The machine tool builder determines the pallet name (see Machine Manual). Programs must be stored in the same directory as the pallet table. Otherwise you must enter the full path and name for the program.





DATUM (entry optional):

Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle G53 **DATUM SHIFT.**

X, Y, Z (entry optional, other axes also possible): For pallets and fixtures, the programmed coordinates are referenced to the machine datum. For NC programs, the programmed coordinates are referenced to the pallet or fixture datum. These entries overwrite the datum that you last set in the Manual mode of operation. With the miscellaneous function M104 you can reactivate the datum that was last set. With the actual-position-capture key, the TNC opens a window that enables you to have the TNC enter various points as datums (see table below):

Position	Meaning
Actual values	Enter the coordinates of the current tool position referenced to the active coordinate system.
Reference values	Enter the coordinates of the current tool position referenced to the machine datum.
ACTL measured values	Enter the coordinates referenced to the active coordinate system of the datum last probed in the Manual operating mode.
REF measured values	Enter the coordinates referenced to the machine datum of the datum last probed in the Manual operating mode.

With the arrow keys and ENT, select the position that you wish to confirm. Then press the ALL VALUES soft key so that the TNC saves the respective coordinates of all active axes in the pallet table. With the PRESENT VALUE soft key, the TNC saves the coordinates of the axis on which the highlight in the pallet table is presently located.



If you have not defined a pallet before an NC program, the programmed coordinates are then referenced to the machine datum. If you do not define an entry, the datum that was set manually remains active.

SP-X, SP-Y, SP-Z (entry optional, other axes also possible): Safety positions can be entered for the axes. These positions can be read with SYSREAD FN18 ID510 NR 6 from NC macros. SYSREAD FN18 ID510 NR 5 can be used to determine if a value was programmed in the column. The positions entered are only approached if these values are read and correspondingly programmed in the NC macros.

1

CTID (entered by the TNC):

The context ID number is assigned by the TNC and contains instructions about the machining progress. Machining cannot be resumed if the entry is deleted or changed.

Editing function in table mode	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Insert as last line in the table	INSERT LINE
Delete the last line in the table	DELETE
Go to beginning of next line	NEXT LINE
Add the entered number of lines at the end of the table	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD

Editing function in entry-form mode	Soft key
Select previous pallet	
Select next pallet	
Select previous fixture	FIXTURE
Select next fixture	FIXTURE
Select previous workpiece	WORKPIECE
Select next workpiece	



Editing function in entry-form mode	Soft key
Switch to pallet plane	VIEW PALLET PLANE
Switch to fixture plane	VIEW FIXTURE PLANE
Switch to workpiece plane	VIEW WORKPIECE PLANE
Select standard pallet view	PALLET DETAIL OF PALLET
Select detailed pallet view	PALLET DETAIL OF PALLET
Select standard fixture view	FIXTURE DETAIL OF FIXTURE
Select detailed fixture view	FIXTURE DETAIL OF FIXTURE
Select standard workpiece view	WORKPIECE DETAIL OF WORKPIECE
Select detailed workpiece view	WORKPIECE DETAIL OF WORKPIECE
Insert pallet	INSERT PALLET
Insert fixture	INSERT
Insert workpiece	INSERT WORKPIECE
Delete pallet	DELETE PALLET
Delete fixture	DELETE FIXTURE
Delete workpiece	DELETE WORKPIECE
Copy all fields to clipboard	COPY ALL FIELDS
Copy highlighted field to clipboard	COPY SELECTED FIELD
Insert the copied field	PASTE FIELDS
Delete clipboard contents	ERASE INTERMED. MEMORY

Editing function in entry-form mode	Soft key
Tool-optimized machining	TOOL ORIENTAT.
Workpiece-optimized machining	WORKPIECE ORIENTAT.
Connect or separate the types of machining	CONNECTED DIS- CONNECTED
Mark plane as being empty	EMPTY POSITION
Mark plane as being unmachined	BLANK

Selecting a pallet file

- Call the file manager in the Programming and Editing or Program Run mode: Press the PGM MGT key.
- ▶ To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- Select a pallet table with the arrow keys, or enter a new file name to create a new table.
- ▶ Confirm your entry with the ENT key.

Setting up the pallet file with the entry form

Pallet operation with tool- or workpiece-oriented machining is divided into three levels:

Pallet level PAL

- Fixture level FIX
- Workpiece level PGM

You can switch to a detail view in each level. Set the machining method and the statuses for the pallet, fixture and workpiece in the standard view. If you are editing an existing pallet file, the updated entries are displayed. Use the detail view for setting up the pallet file.

> Set up the pallet file according to the machine configuration. If you only have one fixture with multiple workpieces, then defining one fixture **FIX** with the workpieces **PGM** is sufficient. However, if one pallet contains several fixtures, or if a fixture is machined from more than one side, you must define the pallet **PAL** with the corresponding fixture levels **FIX**.

Use the screen layout button to switch between table view and form view.

Graphic support for form entry is not yet available.

The various levels of the entry form can be reached with the appropriate soft keys. The current level is highlighted in the status line of the entry form. When you switch to table view with the screen layout button, the cursor is placed in the same level as it was in the form view.



Setting up the pallet plane

- Pallet Id: The pallet name is displayed
- Method: You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. The selected method is assumed for the workpiece level and overwrites any existing entries. In tabular view, WORKPIECE ORIENTED appears as WPO, and TOOL ORIENTED appears as TO.

The TO-/WP-ORIENTED entry cannot be made via soft key. It only appears when different machining methods were chosen for the workpieces in the workpiece or machining level.

If the machining method was determined in the fixture level, the entries are transferred to the workpiece level, where they overwrite any existing entries.

Status: The soft key BLANK identifies the pallet and the corresponding fixtures and workpieces as not yet having been machined, and enters BLANK in the Status field. Use the soft key EMPTY POSITION if you want to skip the pallet during machining. EMPTY appears in the Status field.

Setting up details in the pallet level

- Pallet ID: Enter the pallet name.
- **Datum:** Enter the pallet datum.
- **Datum table:** Enter the name and path of the datum table of the workpiece. The data is transferred to the fixture and workpiece levels.
- **Safe height:** (optional): Safe position for the individual axes referenced to the pallet. The positions entered are only approached if these values were read and correspondingly programmed in the NC macros.



Manual operation	Program † Pallet /	able e NC pro	diting gram?	I	
File:TNC	SCREEND	UMP\PAI FIXI	LETTE. PGM	Р	
Pallet I	D: PAL4	-206-4			
X	Y		z		
Datum ta	ble: TNC:	\RK\TE	ST\TAB	LE01.	4
Cl. heig	iht:		7		
			2		0
					s 🖡
PALLET PA		VIEW FIXTURE PLANE	PALLET DETAIL OF PALLET	INSERT PALLET	DELETE WORKPIECE

Setting up the fixture level

- **Fixture:** The number of the fixture is displayed. The number of fixtures within this level is shown after the slash.
- Method: You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. The selected method is assumed for the workpiece level and overwrites any existing entries. In tabular view, WORKPIECE ORIENTED appears as WPO, and TOOL ORIENTED appears as TO.

Use the **CONNECT/SEPARATE** soft key to mark fixtures that are to be included for calculating the machining process for tool-oriented machining. Connected fixtures are marked with a dashed line, whereas separated fixtures are connected with a solid line. Connected workpieces are marked in tabular view with the entry **CT0** in the METHOD column.

The TO-/WP-ORIENTED entry cannot be made via soft key. It only appears when different machining methods were chosen for the workpieces in the workpiece level.

If the machining method was determined in the fixture level, the entries are transferred to the workpiece level, where they overwrite any existing entries.

Status: The soft key **BLANK** identifies the fixture and the corresponding workpieces as not yet having been machined, and enters BLANK in the Status field. Use the soft key **EMPTY POSITION** if you want to skip the fixture during machining. **EMPTY** appears in the Status field.

Setting up details in the fixture level

- **Fixture:** The number of the fixture is displayed. The number of fixtures within this level is shown after the slash.
- **Datum:** Enter the fixture datum.
- **Datum table:** Enter the name and path of the datum table valid for machining the workpiece. The data is transferred to the workpiece level.
- **NC macro:** In tool-oriented machining, the macro TCTOOLMODE is carried out instead of the normal tool-change macro.
- **Safe height:** (optional): Safe position for the individual axes referenced to the fixture.
- Safety positions can be entered for the axes. These positions can be read with SYSREAD FN18 ID510 NR 6 from NC macros. SYSREAD FN18 ID510 NR 5 can be used to determine if a value was programmed in the column. The positions entered are only approached if these values are read and correspondingly programmed in the NC macros.

Manual operation	Progra Machin	m table (ing metho	editin⊆ od?	1		
Pallet I	D:PAL4	-206-4 Al <mark>FIX</mark>	_P G M			<u>, t</u>
Fixtu	re:	1/4				
Status	s:	BLANK	E-ORIE	NIEU		
Fixtu	re:	2/4				
Method Status	1: s:	WORKPIEC Blank	CE-ORIE	NTED		4
Fixtur	re: d:	3/4 Workpie(E-ORIE	NTED		s 🖡
Status	5:	BLANK			»	<u>в</u> .,
FIXTURE FI	XTURE VI	EW VIEW	FIXTURE DETAIL OF	INSERT		DELETE
	♦ PLF	NE PLANE	FIXTURE	FIXTURE		FIXTURE



Setting up the workpiece level

- Workpiece: The number of the workpiece is displayed. The number of workpieces within this fixture level is shown after the slash.
- Method: You can choose between the WORKPIECE ORIENTED and TOOL ORIENTED machining methods. In tabular view, WORKPIECE ORIENTED appears as WPO, and TOOL ORIENTED appears as TO.

Use the **CONNECT/SEPARATE** soft key to mark workpieces that are to be included for calculating the machining process for tool-oriented machining. Connected workpieces are marked with a dashed line, whereas separated workpieces are connected with a solid line. Connected workpieces are marked in tabular view with the entry **CT0** in the METHOD column.

Status: The soft key **BLANK** identifies the workpiece as not yet having been machined, and enters BLANK in the Status field. Use the soft key **EMPTY POSITION** if you want to skip the workpiece during machining. EMPTY appears in the Status field.



Enter the method and status in the pallet or fixture level. Then the entry will be assumed for all corresponding workpieces.

For several workpiece variants within one level, the workpieces of one variant should be entered together. This way, the workpieces of each variant can be marked with the CONNECT/SEPARATE soft key, and can be machined in groups.

Setting up details in the workpiece level

- Workpiece: The number of the workpiece is displayed. The number of workpieces within this fixture or pallet level is shown after the slash.
- **Datum:** Enter the workpiece datum.
- **Datum table:** Enter the name and path of the datum table valid for machining the workpiece. If you use the same datum table for all workpieces, enter the name and path in the pallet or fixture levels. The data is automatically transferred to the workpiece level.
- **NC program:** Enter the path of the NC program that is necessary for machining the workpiece.
- Safe height: (optional): Safe position for the individual axes referenced to the workpiece. The positions entered are only approached if these values were read and correspondingly programmed in the NC macros.



nerval operation Program table editing Datum?	
Pallet ID:PAL4-206-4 Fixture:1 PALFIXGM Workpiece: 1/4	
Datum: X84.502 Y20.957 Z86.5362	
Datum table: TNC:\RK\TEST\TABLE01.D NC program: TNC:\RK\TEST\FK1.H	4
X Y Z100	s •
WORKPIECE WORKPIECE VIEW WORKPIECE INSERT	
	WORKPIECE



Sequence of tool-oriented machining

The TNC only carries out tool-oriented machining if the TOOL ORIENTED method was selected, and TO or CTO is entered in the table.

- The entry TO or CTO in the Method field tells the TNC that the oriented machining is valid beyond these lines.
- The pallet management starts the NC program given in the line with the entry TO.
- The first workpiece is machined until the next tool call is pending. Departure from the workpiece is coordinated by a special toolchange macro.
- The entry in the column W-STATE is changed from BLANK to INCOMPLETE, and the TNC enters a hexadecimal value in the field CTID.

The value entered in the field CTID is a unique identifier of the machining progress for the TNC. If this value is deleted or changed, machining cannot be continued, nor is midprogram startup or resumption of machining possible.

- All lines in the pallet file that contain the entry CTO in the Method field are machined in the same manner as the first workpiece. Workpieces in several fixtures can be machined.
- The TNC uses the next tool for the following machining steps again from the line with the entry TO if one of the following situations applies:
- If the entry PAL is in the PAL/PGM field in the next line.
- If the entry TO or WPO is in the Method field in the next line.
- If in the lines already machined there are entries under Method which do not have the status EMPTY or ENDED.
- The NC program is continued at the stored location based on the value entered in the CTID field. Usually the tool is changed for the first piece, but the TNC suppresses the tool change for the following workpieces.
- The entry in the CTID field is updated after every machining step. If an END PGM or M02 is executed in an NC program, then an existing entry is deleted and ENDED is entered in the Machining Status field.

- If the entries TO or CTO for all workpieces within a group contain the status ENDED, the next lines in the pallet file are run.

In mid-program startup, only one tool-oriented machining operation is possible. Subsequent pieces are machined according to the method entered.

The value entered in the CTID field is stored for a maximum of one week. Within this time the machining process can be continued at the stored location. After this time the value is deleted, in order to prevent large amounts of unnecessary data on the hard disk.

The operating mode can be changed after executing a group of entries with TO or CTO.

The following functions are not permitted:

- Switching the traverse range
- PLC datum shift
- M118

Leaving the pallet file

- ▶ To call the file manager, press the PGM MGT soft key.
- To select a different type of file, press the SELECT TYPE soft key and the soft key for the desired file type, for example SHOW.H.
- ▶ Select the desired file.

Executing the pallet file



In MP7683, set whether the pallet table is to be executed blockwise or continuously (see "General User Parameters" on page 506).

- Select the file manager in the Program Run, Full Sequence or Program Run, Single Block operating modes: Press the PGM MGT key.
- ▶ To display all type .P files, press the soft keys SELECT TYPE and SHOW .P.
- ▶ Select the pallet table with the arrow keys and confirm with ENT.
- To execute the pallet table: Press the NC Start button. The TNC executes the pallets as set in MP7683.

Screen layout for executing pallet tables

You can have the TNC display the program contents and pallet file contents on the screen together by selecting the screen layout PGM + PALLET. During execution, the TNC then shows program blocks to the left and the pallet to the right. To check the program contents before execution, proceed as follows:

- ▶ Select a pallet table.
- ▶ With the arrow keys, choose the program you would like to check.
- Press the OPEN PGM soft key: The TNC displays the selected program on the screen. You can now page through the program with the arrow keys.
- ▶ To return to the pallet table, press the END PGM soft key.











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 moves. The maximum feed rates can be different for each machine axis, and are set in machine parameters.

Input

You can enter the feed rate in every positioning block or in a separate block. Press the F key on the alphabetic keyboard.

Rapid traverse

If you wish to program rapid traverse, enter GOO.

Duration of effect

A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. If the new feed rate is **G00** (rapid traverse), the last programmed feed rate is once again valid after the next block with **G01**.

Changing during program run

You can adjust the feed rate during program run with the feed-rate override knob.

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in any block (e.g. during tool call).

Programmed change

In the part program, you can change the spindle speed with an S block:



Press the S key on the alphabetic keyboard.

Enter the new spindle speed.

Changing during program run

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



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 **G99** or separately in tool tables. 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 254. If you are working with tool tables, you can use higher numbers and you can also enter a tool name for each tool.

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 0 should also be defined with L=0 and R=0.

Tool length L

There are two ways to determine the tool length L:

Determining the difference between the length of the tool and that of a zero tool L0

For the algebraic sign:

- L>L0: The tool is longer than the zero tool
- L<L0: The tool is shorter than the zero tool

To determine the length:

- Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).
- Set the datum in the tool axis to 0 (datum setting).
- ▶ Insert the desired tool.
- ▶ Move the tool to the same reference position as the zero tool.
- ▶ The TNC displays the difference between the current tool and the zero tool.
- Enter the value in the G99 block or in the tool table by pressing the actual-position-capture key.

Determining the length L with a tool presetter

Enter the determined value directly in the **G99** tool definition block or in the tool table without further calculations.







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>0). If you are programming the machining data with an allowance, enter the oversize value with T.

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

Delta values are usually entered as numerical values. In a ${\rm T}$ 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 ${\bf G99}$ block of the part program.

▶ To select 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.

Resulting NC block:

N40 G99 T5 L+10 R+5 *



Entering tool data in tables

You can define and store up to 32 767 tools and their tool data in a tool table. In Machine Parameter 7260, you can define how many tools are to be stored by the TNC when a new table is set up. Also see the Editing Functions later in this Chapter. In order to be able to assign various compensation data to a tool (indexing the tool number), MP7262 must not be equal to 0.

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 measure tools automatically with the TT 130 touch probe (see the Touch Probe Cycles User's Manual, Chapter 4),
- you want to rough-mill the contour with Cycle G122 (see "ROUGH-OUT (Cycle G122)" on page 344),
- you want to work with automatic cutting data calculations.

Tool table: Standard tool data

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

Abbr.	Input	Dialog		
TIME2	Maximum tool life in minutes during a tool call: If the current tool age exceeds this value, the TNC changes the tool during the next tool call (see also CUR.TIME).	Maximum tool age for TOOL CALL?		
CUR.TIME	Time in minutes the tool has been in use: The TNC automatically counts the current tool age. A starting value can be entered for used tools.	Current tool life?		
DOC	Comment on tool (up to 16 characters)	Tool description?		
PLC	Information on this tool that is to be sent to the PLC	PLC status?		
PLC VAL	Value of this tool that is to be sent to the PLC	PLC value?		
РТҮР	Tool type for evaluation in the pocket table	Tool type for pocket table?		
NMAX	Limits the spindle speed for this tool. The programmed value is monitored (error message) as well as a shaft speed increase via the potentiometer. Function inactive: Enter "–"	Maximum speed [rpm]?		

Tool table: Tool data required for automatic tool measurement

For a description of the cycles governing automatic tool measurement, see the Touch Probe Cycles Manual, Chapter 4.

Abbr.	Input	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 = -)?
TT:R-OFFS	For tool length measurement: tool offset between stylus center and tool center. Preset value: Tool radius R (NO ENT means ${f R}$).	Tool offset: radius?
TT:L-OFFS	Tool radius measurement: tool offset in addition to MP6530 (see "General User Parameters" on page 506) 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?

Tool table: Tool data for automatic speed/feed rate calculation.

Abbr.	Input	Dialog		
ТҮРЕ	Tool type (MILL for milling, DRILL for drilling or boring, TAP for tapping): Press the SELECT TYPE soft key (3rd soft-key row): The TNC superimposes a window where you can select the type of tool you want.	Tool type?		
TMAT	Tool material: Press the SELECT MATERIAL soft key (3rd soft-key row): The TNC superimposes a window where you can select the type of material you want.	Tool material?		
CDT	Cutting data table: Press the SELECT CDT soft key (3rd soft-key row): The TNC superimposes a window where you can select a cutting data table.	Name of cutting data table?		

Tool table: Tool data for 3-D touch trigger probe (only when bit 1 is set in MP7411 = 1, also see the Touch Probe Cycles Manual)

Abbr.	Input	Dialog		
CAL-OF1	During calibration, the TNC stores in this column the center misalignment in the reference axis of the 3-D probe, if a tool number is indicated in the calibration menu.	Center misalignmt. in ref. axis?		
CAL-OF2	During calibration, the TNC stores in this column the center misalignment in the minor axis of the 3-D probe, if a tool number is indicated in the calibration menu.	Center misalignment minor axis?		
CAL-ANG	During calibration, the TNC stores in this column the spindle angle at which the 3-D probe was calibrated, if a tool number is indicated in the calibration menu.	Spindle angle for calibration?		

1

Editing tool tables

The tool table that is active during execution of the part program is designated as TOOL.T. TOOL.T must be saved in the directory TNC:\ and can only be edited in one of the machine operating modes. Other tool tables that are used for archiving or test runs are given different file names with the extension ".T".

To open the tool table TOOL.T:

- Select any machine operating mode.
- TOOL TABLE

OFF ON

5.2 Tool Data

key. ▶ Set the EDIT soft key to ON.

To open any other tool table:

Select the Programming and Editing mode of operation.



- Call the file manager.
- ► To select the file type, press the SELECT TYPE soft key.

▶ To select the tool table, press the TOOL TABLE soft

- ▶ To 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 symbol ">>" or "<<".

Editing functions for tool tables	Soft key
Select beginning of table	BEGIN
Select end of table	
Select previous page in table	PAGE
Select next page in table	
Look for the tool name in the table	FIND TOOL NAME
Show tool information in columns or show all information on one tool on one screen page	FORM

Tool T <mark>ool</mark>	table length	editin ?	9			Pros and	ramming editing
File: TOO	0L.T	MM				>>	+
t nar	18	L	R	R2	DL		
1		+12.5	+3	+0	+0		
z		+85.35	+25	+0	+0		
		+23.15	+3.5	+0	+0		
. 50		+47.5	+3	+0	+0		
, IH2		+0	+1.5	+0	+0.1		
,		+0	+2.5	+0	+0		
3		+25	+4	+0	+0		
							Ā
		10	4% S-0)VR 14:	42		• •
		9	3% F-0	OVR LIM	IT 1		S
×A ◆A	-29.2	76 Y 30+C	+242.	904 Z 000	-36	5.413	0
				s	359.9	38	s I
ACTL.	¢+÷1	© T 5	ZS	2612 F	0	M 5/9	
BEGIN		PAGE	PAGE	EDIT OFF ON	FIND TOOL NAME	POCKET	END

5 Programming: Tools



Editing functions for tool tables	Soft key
Move to beginning of line	BEGIN LINE
Move to end of line	
Copy highlighted field	COPY FIELD
Insert copied field	PASTE FIELD
Add the entered number of lines (tools) at the end of the table.	APPEND N LINES
Insert a line for the indexed tool number after the active line. The function is only active if you are permitted to store multiple compensation data for a tool (MP7262 not equal to 0). The TNC inserts a copy of the tool data after the last available index and increases the index by 1. Application: e.g. stepped drill with more than one length compensation value.	INSERT
Delete current line (tool).	DELETE
Display / Do not display pocket numbers.	POCKET H DISPLAY HIDE
Display all tools / only those tools that are stored in the pocket table.	TOOLS DISPLAY HIDE

Leaving the tool table

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

Additional notes on tool tables

MP7266.x defines which data can be entered in the tool table and in which sequence the data is displayed.



You can overwrite individual columns or lines of a tool table with the contents of another file. Prerequisites:

- The target file must exist.
- The file to be copied must contain only the columns (or lines) you want to replace.

To copy individual columns or lines, press the REPLACE FIELDS soft key (see "Copying a single file" on page 89).



Using an external PC to overwrite individual tool data

The HEIDENHAIN data transfer software TNCremoNT provides an especially convenient way to use an external PC to overwrite tool data (see "Software for data transfer" on page 480). This applies when you measure tool data on an external tool presetter and then want to transfer the data to the TNC. Follow this procedure:

- Copy the tool table TOOL.T to the TNC, for example to TST.T
- Start the data transfer software TNCremoNT on the PC
- Establish a connection with the TNC
- ▶ Transfer the copied tool table TST.T to the PC
- Use any text editor to reduce TST.T to the lines and columns to be changed (see figure at upper right). Make sure that the header is not changed and the data is always flush in the column. The tool numbers (column T) need not be consecutive
- ▶ In TNCremoNT, select the menu item <Extras> and <TNCcmd>: This starts TNCcmd
- To transfer TST.T to the TNC, enter the following command and confirm with the return key (see figure at lower right): put tst.t tool.t /m

During transfer, only the tool data defined in the subfile (e.g. TST.T) is overwritten. All other tool data of the table TOOL.T remains unchanged.

BEGIN	TST	.т	MM		
Т	NAME			L	R
1				+12.5	+9
3				+23.15	+3.5
[END]					
Pocket table for tool changer

P

The machine tool builder adapts the functional range of the pocket table to the requirements of your machine. The machine tool manual provides further information.

For automatic tool changing you need the pocket table TOOL_P.TCH. The TNC can manage several pocket tables with any file names. To activate a specific pocket table for program run you must select it in the file management of a Program Run mode of operation (status M). In order to be able to manage various magazines in a tool-pocket table (indexing the pocket number), Machine Parameters 7261.0 to 7261.3 must not be equal to 0.

Editing a pocket table in a Program Run operating mode



To select the tool table, press the TOOL TABLE soft key.

▶ To select the pocket table, press the POCKET TABLE

- POCKET TABLE EDIT
- Set the EDIT soft key to ON.

Selecting a pocket table in the Programming and Editing operating mode

soft key.



- ▶ Call the file manager.
- To select the file type, press the SELECT TYPE soft key.
- ▶ To show files of the type .TCH, press the soft key TCH FILES (second soft-key row).
- Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key.

Abbr.	Input	Dialog
Р	Pocket number of the tool in the tool magazine	-
Т	Tool number	Tool number?
ST	Special tool with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L).	Special tool?
F	Fixed tool number. The tool is always returned to the same pocket in the tool magazine	Fixed pocket? Yes = ENT / No = NO ENT
L	Locked pocket (see also column ST)	Pocket locked Yes = ENT / No = NO ENT
PLC	Information on this tool pocket that is to be sent to the PLC	PLC status?
TNAME	Display of the tool name from TOOL.T	-
DOC	Display of the comment to the tool from TOOL.T	-





Pocket table editing

(

Editing functions for pocket tables	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Reset pocket table	RESET POCKET TABLE
Go to beginning of next line	NEXT LINE
Reset tool number column T	RESET COLUMN T

i

Calling tool data

TOOL

To call a tool in the machining program, press the TOOL CALL key:

- ▶ Tool number: Enter the number or name of the tool. The tool must already be defined in a G99 block or in the tool table. To call a tool by the tool name, enter the 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.
 - Spindle axis Z XY plane: Enter the tool axis. To transfer the default setting G17, press the ENT key. If you wish to select a different tool axis, use the soft keys.
 - Spindle speed S: Enter the spindle speed directly or allow the TNC to calculate the spindle speed if you are working with cutting data tables. Press the S CALCULATE AUTOMAT. soft key. The TNC limits the spindle speed to the maximum value set in MP 3515. Confirm the speed you have entered by pressing the ENT key.
 - Feed rate F: Enter the feed rate directly or allow the TNC to calculate the feed rate if you are working with cutting data tables. Press the F CALCULATE AUTOMAT. soft key. The TNC limits the feed rate to the maximum feed rate of the slowest axis (set in MP1010). F is effective until you program a new feed rate in a positioning block or a T block. Confirm the feed rate you have entered by pressing the ENT key.
 - ▶ Tool length oversize: Enter the delta value for the tool length and confirm your entry with the ENT key.
 - Tool radius oversize: Enter the delta value for the tool radius and confirm your entry with the ENT key.
 - **Tool radius oversize 2**: Enter the delta value for the tool radius 2 and confirm with the ENT key.

Example: Tool call

Call tool number 5 in the tool axis Z with a spindle speed 2500 rpm. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius with an undersize of 1 mm.

N20 T 5.2 G17 S2500 DL+0.2 DR-1

The character ${\bf D}$ preceding ${\bf L}$ and ${\bf R}$ designates delta values.

Tool preselection with tool tables

When you use tool tables, enter a **G51** block to preselect the next tool to be selected. 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 tool manual provides further information.

Tool change position

The tool change position must be approachable without collision. With the miscellaneous functions **M91** and **M92**, you can enter machinereferenced (rather than workpiece-referenced) coordinates for the tool change position. If **T0** 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 463).
- ▶ Change the tool.
- Resume program run (see "Resuming program run after an interruption," page 465).

Automatic tool change

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

Automatic tool change if the tool life expires: M101



The function of **M101** can vary depending on the individual machine tool. The machine tool manual provides further information.

The TNC automatically changes the tool if the tool life **TIME2** expires during program run. To use this miscellaneous function, activate **M101** at the beginning of the program. **M101** is reset with **M102**.

The tool is not always changed immediately, but, depending on the workload of the control, a few NC blocks later.

Prerequisites for standard NC blocks with radius compensation R0, RR, RL

The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

5.3 Tool Compensation

Introduction

The TNC adjusts the spindle path in the tool 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. The TNC accounts for the compensation value in up to five axes including the rotary axes.

Tool length compensation

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



If you cancel a positive length compensation with **T0**, the distance between tool and workpiece will be reduced.

After a 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 ${\bf T}$ block and the tool table into account.

Compensation value = L + \textbf{DL}_{T} + \textbf{DL}_{TAB} , where

- L: is the tool length L from the G99 block or tool table DL _{T1} is the oversize for length DL in the T block (not
- taken into account by the position display)

 $\mathbf{DL}_{\mathsf{TAB}}$ is the oversize for length \mathbf{DL} in the tool table.



Tool radius compensation

The NC block for programming a tool movement contains:

- **G41** or **G42** for radius compensation,
- **G43** or **G44**, for radius compensation with axis-parallel traverse,
- **G40** if there is no radius compensation.

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with G41 or G42.



- program a positioning block with G40,
- program a program call with %...,
- select a new program with PGM MGT.

For tool radius compensation, the TNC takes the delta values from both the ${\bf T}$ block and the tool table into account.

Compensation value = $\mathbf{R} + \mathbf{D}\mathbf{R}_{T} + \mathbf{D}\mathbf{R}_{TAB}$, where

- **R** is the tool radius **R** from the **G99** block or tool table
- **DR**_T is the oversize for radius **DR** in the **T** block (not taken into account by the position display)
- $\mathbf{DR}_{\mathsf{TAB}}$ is the oversize for radius \mathbf{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.







G42

Contouring with radius compensation: G41 and G42

- The tool moves to the right of the programmed contour
- **G41** The tool moves to the left of the programmed contour

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

Betv com one

Between two program blocks with different radius compensations (**G42** and **G41**) you must program at least one traversing block in the working plane without radius compensation (that is, with **G40**).

Radius compensation does not take effect until the end of the block in which it is first programmed.

You can also activate the radius compensation for secondary axes in the working plane. Program the secondary axes as well in each following block, since otherwise the TNC will execute the radius compensation in the principal axis again.

Whenever radius compensation is activated with **G42/G41** or canceled with G40, the TNC positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.

Entering radius compensation

Radius compensation is entered in a G01 block:

641	To select tool movement to the left of the contour, select function G41, or
G 4 2	To select tool movement to the right of the contour, select function G42, or
G 4 Ø	To select tool movement without radius compensation or to cancel radius compensation, select function G40.
END	To terminate the block, press the END key.





Radius compensation: Machining corners

Outside corners

If you program radius compensation, the TNC moves the tool around outside corners either on a transitional arc or on a spline (selectable via MP7680). If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.

Inside corners

The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.



To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.

Machining corners without radius compensation

If you program the tool movement without radius compensation, you can change the tool path and feed rate at workpiece corners with the miscellaneous function **M90.** See "Smoothing corners: M90," page 199.





5.4 Peripheral Milling: 3-D Radius Compensation with Workpiece Orientation

Function

P

ᇞ

With peripheral milling, the TNC displaces the tool perpendicular to the direction of movement and perpendicular to the tool direction by the sum of the delta values **DR** (tool table and **T** block). Determine the compensation direction with radius compensation **G41/G42** (see figure at right, traverse direction Y+).

For the TNC to be able to reach the set tool orientation, you need to activate the function **M128** (see "Maintaining the position of the tool tip when positioning with tilted axes (TCPM*):

M128 (software option 2)" on page 214) and subsequently the tool radius compensation. The TNC then positions the rotary axes automatically so that the tool can reach the orientation defined by the coordinates of the rotary axes with the active compensation.

This function is possible only on machines for which you can define spatial angles for the tilting axis configuration. Refer to your machine manual.

The TNC is not able to automatically position the rotary axes on all machines. Refer to your machine manual.



Danger of collision

On machines whose rotary axes only allow limited traverse, sometimes automatic positioning can require the table to be rotated by 180°. In this case, make sure that the tool head does not collide with the workpiece or the clamps.

You can define the tool orientation in a G01 block as described below.

Example: Definition of the tool orientation with M128 and the coordinates of the rotary axes

N10 G00 G90 X-20 Y+0 Z+0 B+0 C+0 *	Pre-position
N20 M128 *	Activate M128
N30 G01 G42 X+0 Y+0 Z+0 B+0 C+0 F1000 *	Activate radius compensation
N40 X+50 Y+0 Z+0 B-30 C+0 *	Position rotary axis (tool orientation)

1

5.5 Working with Cutting Data Tables

Note

The TNC must be specially prepared by the machine tool builder for the use of cutting data tables.

Some functions or additional functions described here may not be provided on your machine tool. Refer to your machine manual.

Applications

In cutting data tables containing various workpiece and cutting material combinations, the TNC can use the cutting speed V_C and the tooth feed f_Z to calculate the spindle speed S and the feed rate F. This calculation is only possible if you defined the workpiece material in the program and various tool-specific features in the tool table.

ſ

Before you let the TNC automatically calculate the cutting data, the tool table from which the TNC is to take the tool-specific data must be first be activated in the Test Run mode (status S).

Editing function for cutting data tables	Soft key
Insert line	INSERT LINE
Delete line	DELETE LINE
Go to beginning of next line	NEXT LINE
Sort the table	ORDER N
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE
Edit the table format (2nd soft-key row)	EDIT FORMAT





Table for workpiece materials

Workpiece materials are defined in the table WMAT.TAB (see figure at right). WMAT.TAB is stored in the TNC:\ directory and can contain as many materials as you want. The name of the material type can have up to 32 characters (including spaces). The TNC displays the contents of the NAME column when you are defining the workpiece material in the program (see the following section).

If you change the standard workpiece material table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data. Define the path in the TNC.SYS file with the code word WMAT= (see "Configuration file TNC.SYS," page 161).

To avoid losing data, save the WMAT.TAB file at regular intervals.

Defining the workpiece material in the NC program

In the NC program select the workpiece material from the WMAT.TAB table using the WMAT soft key:

- UMAT
- Program the workpiece material: In the Programming and Editing operating mode, press the WMAT soft key.
- SELECTION WINDOW

- The WMAT.TAB table is superimposed: Press the SELECTION WINDOW soft key, and in a second window the TNC displays the list of materials that are stored in the WMAT.TAB table.
- Select your workpiece material by using the arrow keys to move the highlight onto the material you wish to select and confirming with the ENT key. The TNC transfers the selected material to the WMAT block.
- ▶ To terminate the dialog, press the END key.

If you change the WMAT block in a program, the TNC outputs a warning. Check whether the cutting data stored in the T block are still valid.

Manua: opera	tion Pr	ogram t IME ?	able	editing	3		
Fil	e: WMAT.TAB						-
NR	NAME	DOC					- Inter
0	110 WCrV 5	WerkzStahl	1.2519				
1	14 NiCr 14	Einsatz-Stah	1.5752				
2	142 WV 13	WerkzStahl	1.2562				
3	15 CrNi 6	Einsatz-Stah	1.5919				
4	16 CrMo 4 4	Baustahl 1.73	37				
5	16 MnCr 5	Einsatz-Stah	1.7131				
6	17 MoV 8 4	Baustahl 1.54	106				
7	18 CrNi 8	Einsatz-Stah	1.5920				
8	19 Mn 5	Baustahl 1.04	82				Ξ.
9	21 MnCr 5	WerkzStahl	1.2162				
10	26 CrMo 4	Baustahl 1.72	19				
11	28 NiCrMo 4	Baustahl 1.65	513				5
12	30 CrMoV 9	VergStahl :	. 7707				0 🕇
13	30 CrNiMo 8	VergStahl :	.6580				,
							s I
BEG		PAGE	PAGE	INSERT LINE	DELETE	NEXT	ORDER

Table for tool cutting materials

Tool cutting materials are defined in the TMAT.TAB table. TMAT.TAB is stored in the TNC:\ directory and can contain as many material names as you want (see figure at upper right). The name of the cutting material type can have up to 16 characters (including spaces). The TNC displays the NAME column when you are defining the tool cutting material in the TOOL.T tool table.

If you change the standard tool cutting material table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data. Define the path in the TNC.SYS file with the code word TMAT= (see "Configuration file TNC.SYS," page 161).

To avoid losing data, save the TMAT.TAB file at regular intervals.

Table for cutting data

Define the workpiece material / cutting material combinations with the corresponding cutting data in a file table with the file name extension .CDT; see figure at center right. You can freely configure the entries in the cutting data table. Besides the obligatory columns NR, WMAT and TMAT, the TNC can also manage up to four cutting speed (V_C) / feed rate (F) combinations.

The standard cutting data table FRAES_2.CDT is stored in the directory TNC:\. You can edit FRAES_2.CDT, or add as many new cutting-data tables as you wish.

If you change the standard cutting data table, you must copy it into a new directory. Otherwise your changes will be overwritten during a software update by the HEIDENHAIN standard data (see "Configuration file TNC.SYS," page 161).

All of the cutting data tables must be stored in the same directory. If the directory is not the standard directory TNC:\, then behind the code word PCDT= you must enter the path in which your cutting data is stored.

To avoid losing data, save your cutting data tables at regular intervals.



Manual operation Program table editing Workpiece material?							
- 30	e: FRAES_2.CDT						
NR	WMAT	THAT	Vc1	F1	Vc2 F2		
0	St 33-1	HSSE/T IN	40	0,016	55 0,0	320	
1	St 33-1	HSSE/TiCN	40	0,016	55 0,0	320	
2	St 33-1	HC-P25	100	0,200	130 0,0	250	
з	St 37-2	HSSE-Co5	20	0,025	45 0,0	930	
4	St 37-2	HSSE/TiCN	40	0,016	55 0,0	920	
5	St 37-2	HC-P25	100	0,200	130 0,3	250	
6	St 50-2	HSSE/T IN	40	0,016	55 0,0	920	
7	St 50-2	HSSE/TiCN	40	0,016	55 0,0	320	
8	St 50-2	HC-P25	100	0,200	130 0,0	250	T.
9	St 60-2	HSSE/T IN	40	0,016	55 0,0	320	
10	St 60-2	HSSE/TiCN	40	0,016	55 0,0	920	
11	St 60-2	HC-P25	100	0,200	130 0,3	250	5
12	C 15	HSSE-Co5	20	0,040	45 0,0	050	0 👕
13	C 15	HSSE/TiCN	26	0,040	35 0,0	050	
							s 🖡
BEC		PAGE		INSERT	DELETE	NEXT	ORDER

ſ

Creating a new cutting data table

- Select the Programming and Editing mode of operation.
- Press the PGM MGT key to select the file manager.
- Select the directory where the cutting data table is to be stored.
- Enter any file name with file name extension .CDT, and confirm with ENT.
- ▶ On the right half of the screen, the TNC displays various table formats (machine-dependent, see example in figure at right). These tables differ from each other in the number of cutting speed / feed rate combinations they allow. Use the arrow keys to move the highlight onto the table format you wish to select and confirm with ENT. The TNC generates a new, empty cutting data table.

Data required for the tool table

- Tool radius—column R (DR)
- Number of teeth (only with tools for milling)—column CUT
- Tool type—column TYPE
- The tool type influences the calculation of the feed rate:
- Milling tool: $F = S \cdot f_Z \cdot z$
- All other tools: $F = S \cdot f_U$
- S: Spindle speed
- f_Z: Feed per tooth
- f_U: Feed per revolution
- z: Number of teeth
- Tool cutting material—column TMAT
- Name of the cutting data table for which this tool will be used column CDT
- In the tool table, select the tool type, tool cutting material and the name of the cutting data table via soft key (see "Tool table: Tool data for automatic speed/feed rate calculation.," page 141).



Working with automatic speed / feed rate calculation

- 1 If it has not already been entered, enter the type of workpiece material in the file WMAT.TAB.
- 2 If it has not already been entered, enter the type of cutting material in the file TMAT.TAB.
- **3** If not already entered, enter all of the required tool-specific data in the tool table:
 - Tool radius
 - Number of teeth
 - Tool type
 - Tool material
 - The cutting data table for each tool
- 4 If not already entered, enter the cutting data in any cutting data table (CDT file).
- **5** Test Run operating mode: Activate the tool table from which the TNC is to take the tool-specific data (status S).
- 6 In the NC program, set the workpiece material by pressing the WMAT soft key.
- 7 In the NC program, let the TOOL CALL block automatically calculate spindle speed and feed rate via soft key.

Changing the table structure

Cutting data tables constitute so-called "freely-definable tables" for the TNC. You can change the format of freely definable tables by using the structure editor.

The TNC can process up to 200 characters per line and up to 30 columns.

If you insert an additional column into an existing table, the TNC does not automatically shift the values that have been entered.

Calling the structure editor

「白

Press the EDIT FORMAT soft key (2nd soft-key level). The TNC opens the editing window (see figure at right), in which the table structure is shown "rotated by 90° ." In other words, a line in the editing window defines a column in the associated table. The meanings of the structure commands (header entries) are shown in the table at right.



Exiting the structure editor

Press the END key. The TNC changes data that was already in the table into the new format. Elements that the TNC could not convert into the new structure are indicated with a hash mark # (e.g., if you have narrowed the column width).

Structure command	Meaning
NR	Column number
NAME	Overview of columns
TYPE	N: Numerical input C: Alphanumeric input
WIDTH	Width of column. For type N including algebraic sign, comma, and decimal places.
DEC	Number of decimal places (max. 4, effective only for type N)
ENGLISH to HUNGARIAN	Language-dependent dialogs (max. 32 characters)

i

Data transfer from cutting data tables

If you output a file type .TAB or .CDT via an external data interface, the TNC also transfers the structural definition of the table. The structural definition begins with the line #STRUCTBEGIN and ends with the line #STRUCTEND. The meanings of the individual code words are shown in the table "Structure Command" (see "Changing the table structure," page 159). Behind #STRUCTEND the TNC saves the actual content of the table.

Configuration file TNC.SYS

You must use the configuration file TNC.SYS if your cutting data tables are not stored in the standard directory TNC:\. In TNC.SYS you must then define the paths in which you have stored your cutting data tables.



The TNC.SYS file must be stored in the root directory TNC:\.

Entries in TNC.SYS	Meaning
WMAT=	Path for workpiece material table
TMAT=	Path for cutting material table
PCDT=	Path for cutting data tables

Example of TNC.SYS

WMAT=TNC:\CUTTAB\WMAT_GB.TAB
TMAT=TNC:\CUTTAB\TMAT_GB.TAB
PCDT=TNC:\CUTTAB\









Programming: Programming Contours

i

6.1 Tool Movements

Path functions

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

Miscellaneous functions M

With the miscellaneous functions of the TNC you can control:

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring 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 9.

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 10.





6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

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

The TNC moves all axes programmed in a single block simultaneously.

Movement parallel to the machine axes

The program block contains only one coordinate. The TNC thus moves the tool parallel to the programmed axis.

Depending on the 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 moves and the workpiece remains stationary.

Example:

N50 G00 X+100 *

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

The tool retains the Y and Z coordinates and moves to the position X=100 (see figure at upper right).

Movement in the main planes

The program block contains two coordinates. The TNC thus moves the tool in the programmed plane.

Example:

N50 G00 X+70 Y+50 *

The tool retains the Z coordinate and moves in the XY plane to the position X=70, Y=50 (see figure at center right).

Three-dimensional movement

The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example:

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







Entering more than three coordinates

The TNC can control up to 5 axes simultaneously. Machining with 5 axes, for example, moves 3 linear and 2 rotary axes simultaneously.

Such programs are too complex to program at the machine, however, and are usually created with a CAD system.

Example:

N G01 G40 X+20 Y+10 Z+2 A+15 C+6 F100 M3 *

The TNC graphics cannot simulate movements in more than three axes.

Circles and circular arcs

The TNC moves two axes simultaneously in a circular path relative to the workpiece. You can define a circular movement by entering a circle center.

When you program a circle, the TNC assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a tool call:

Tool axis	Main plane	Circle center
Z (G17)	XY, also UV, XV, UY	I, J
Y (G18)	ZX, also WU, ZU, WX	К, І
X (G19)	YZ, also VW, YW, VZ	J, K

You can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see "WORKING PLANE (Cycle G80)," page 389) or Q parameters (see "Principle and Overview," page 418).

Direction of rotation for circular movements

If a circular path has no tangential transition to another contour element, enter the direction of rotation with the following functions:

- Clockwise direction of rotation: G02/G12
- Counterclockwise direction of rotation: G03/G13







Radius compensation

The radius compensation must be in the block in which you move to the first contour element. You cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line block (see "Path Contours—Cartesian Coordinates," page 172).

Pre-positioning

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



6.3 Contour Approach and Departure

Starting point and end point

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

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

Example

Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.

First contour point

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

Approaching the starting point in the spindle axis

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

Example NC blocks

N30 G00 G40 X+20 Y+30 *

N40 Z-10 *







End point

The end point should be selected so that it is:

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

Example

Figure at upper right: If you set the ending point in the dark gray area, the contour will be damaged when the end point is approached.

Depart the end point in the spindle axis:

Program the departure from the end point in the spindle axis separately. See figure at center right.

Example NC blocks

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





Common starting and end points

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

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

Example

Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.

Tangential approach and departure

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

Starting point and end point

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

Approach

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

Departure

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



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







6.3 Contour App<mark>roa</mark>ch and Departure

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



6.4 Path Contours—Cartesian Coordinates

Overview of path functions

Tool movement	Function	Required input
Straight line at feed rate Straight line at rapid traverse	G00 G01	Coordinates of the end points of the straight line
Chamfer between two straight lines	G24	Length of chamfer R
-	I, J, K	Coordinates of the circle center
Circular path in clockwise direction Circular path in counterclockwise direction	G02 G03	Coordinates of the arc end point in connection with ${\bf I},{\bf J},{\bf K}{\rm or}$ additional circular radius ${\bf R}$
Circular path corresponding to active direction of rotation	G05	Coordinates of the arc end point and circular radius ${\bf R}$
Circular arc with tangential connection to the preceding contour element	G06	Coordinates of the arc end point
Circular arc with tangential connection to the preceding and subsequent contour elements	G25	Rounding-off radius R

i

6.4 Path Contours—Ca<mark>rte</mark>sian Coordinates

Straight line at rapid traverse G00 Straight line with feed rate G01 F...

The TNC moves the tool in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.

Programming



 \blacktriangleright Coordinates of the end point of the straight line

Further entries, if necessary:

Radius compensation G40/G41/G42

▶ Feed rate F

Miscellaneous function M

Example NC blocks

N70 G01 G41 X+10 Y+40 F200 M3 *	
N80 G91 X+20 Y-15 *	
N90 G90 X+60 G91 Y-10 *	

Actual position capture

With the actual-position-capture function, you can transfer any desired axis position into a block:

- In the Manual Operation mode, move the tool to the position you wish to capture.
- Switch the screen display to Programming and Editing.
- Select the program block into which you want to take over an axis position.



Select the actual position capture function: In the softkey row the TNC displays the axes whose positions can be transferred.



Select the axis, e.g. X: The TNC writes the current position of the selected axis into the active input box.



Inserting a chamfer CHF between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

- The blocks before and after the **G24** block must be in the same working plane.
- The radius compensation before and after the **G24** block must be the same.
- An inside chamfer must be large enough to accommodate the current tool.

Programming

G 24

- **Chamfer side length:** Length of the chamfer
- Further entries, if necessary: Feed rate F (only effective in G24 block)

Example NC blocks

N70 G01 G41 X+0 Y+30 F300 M3 *
N80 X+40 G91 Y+5 *
N90 G24 R12 F250 *
N100 G91 X+5 G90 Y+0 *

You cannot start a contour with a **G24** block.

A chamfer is possible only in the working plane.

The corner point is cut off by the chamfer and is not part of the contour.

A feed rate programmed in the **G24** block is effective only in that block. After the **G24** block, the previous feed rate becomes effective again.





1

Rounding corners G25

The G25 function is used for rounding off corners.

The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

The rounding arc must be large enough to accommodate the tool.

Programming



Rounding-off radius: Enter the radius

Further entries, if necessary: Feed rate F (only effective in G25 block)

Example NC blocks

N50 G01 G41 X+10 Y+40 F300 M3 *	
N60 X+40 Y+25 *	
N70 G25 R5 F100 *	
N80 X+10 Y+5 *	

In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

The corner point is cut off by the rounding arc and is not part of the contour.

A feed rate programmed in the **G25** block is effective only in that block. After the **G25** block, the previous feed rate becomes effective again.

You can also use a **G25** block for a tangential contour approach (see "Tangential approach and departure," page 170).





Circle center I, J

You can define a circle center for circles that are programmed with the functions G02, G03 or G05. This is done in the following ways:

- Entering the Cartesian coordinates of the circle center, or
- Using the last programmed circle center (G29),
- Transferring the coordinates with the actual-position-capture function.

Programming



Enter the coordinates for the circle center, or if you want to use the last programmed position, enter G29.

Example NC blocks

N50 I+25 J+25 *

or

N10 G00 G40 X+25 Y+25 *	
N20 G29 *	

The program blocks N10 and N20 do not refer to the illustration.

Duration of effect

The circle center definition remains in effect until a new circle center is programmed. You can also define a circle center for the secondary axes U, V and W.

Entering incremental values for the circle center I, J

If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

The only effect of **I** and **J** is to define a position as a circle center—the tool does not move to the position.

The circle center is also the pole for polar coordinates.

If you wish to define the pole in parallel axes, first press the ${\bf I}$ (J) key on the ASCII keyboard, and then the orange axis key for the corresponding parallel axis.



6.4 Path Contours—Cartesian Coordinates

Х

Circular path G02/G03/G05 around circle center I, J

Before programming a circular arc, you must first enter the circle center I, J. The last programmed tool position will be the starting point of the arc.

Direction

- In clockwise direction: G02
- In counterclockwise direction: G03
- Without programmed direction: **G05**. The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

Move the tool to the circle starting point.



Enter the coordinates of the circle center.



Enter the coordinates of the arc end point.

Further entries, if necessary: ▶ Feed rate F

Miscellaneous function M

Example NC blocks



Full circle

For the end point, enter the same point that you used for the starting point.



HEIDENHAIN TNC iTNC 530

The starting and end points of the arc must lie on the circle.

Input tolerance: up to 0.016 mm (selected with MP7431).



Y

Ē

S



Circular path G02/G03/G05 with defined radius

The tool moves on a circular path with the radius R.

Direction

G³

- In clockwise direction: G02
- In counterclockwise direction: G03
- Without programmed direction: **G05.** The TNC traverses the circular arc with the last programmed direction of rotation.

Programming

- Enter the coordinates of the arc end point.
 - Radius R Note: The algebraic sign determines the size of the arc!
 - Further entries, if necessary:
 - ► Feed rate F
 - Miscellaneous function M

Full circle

For a full circle, program two CR blocks in succession:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.



i

6.4 Path Contours—Cartesian Coordinates

Central angle CCA and arc radius R

The starting and end points on the contour can be connected with four arcs of the same radius:

Smaller arc: CCA<180° Enter the radius with a positive sign R>0 $\,$

Larger arc: CCA>180° Enter the radius with a negative sign R<0 $\,$

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

Convex: Direction of rotation ${\bf G02}$ (with radius compensation ${\bf G41})$

Concave: Direction of rotation G03 (with radius compensation G41)

Example NC blocks

N100 G01 G41 X+40 Y+40 F200 M3 * N110 G02 X+70 Y+40 R+20 * (ARC 1)

or

N110 G03 X+70 Y+40 R+20 * (ARC 2)

or

N110 G02 X+70 Y+40 R-20 * (ARC 3)

or

N110 G03 X+70 Y+40 R-20 * (ARC 4)

The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.

The maximum radius is 99.9999 m.

You can also enter rotary axes A, B and C.







Circular path G06 with tangential approach

The tool moves on an arc that starts tangentially to the previously programmed contour element.

A transition between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

The contour element to which the tangential arc connects must be programmed immediately before the **G06** block. This requires at least two positioning blocks.

Programming

G 6

Enter the coordinates of the arc end point.

Further entries, if necessary: ▶ Feed rate F

Miscellaneous function M

Example NC blocks

N70 G01 G41 X+0 Y+25 F300 M3 *
N80 X+25 Y+30 *
N90 G06 X+45 Y+20 *
G01 Y+0 *

G

A tangential arc is a two-dimensional operation: the coordinates in the **G06** block and in the contour element preceding it must be in the same plane of the arc.



i
Example: Linear movements and chamfers with Cartesian coordinates



%LINEAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define blank form for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define tool in the program
N40 T1 G17 S4000 *	Call tool in the spindle axis and with the spindle speed S
N50 G00 G40 G90 Z+250 *	Retract tool in the spindle axis at rapid traverse
N60 X-10 Y-10 *	Pre-position the tool
N70 G01 Z-5 F1000 M3 *	Move to working depth at feed rate F = 1000 mm/min
N80 G01 G41 X+5 Y+5 F300 *	Approach the contour at point 1, activate radius compensation G41
N90 G26 R5 F150 *	Tangential approach
N100 Y+95 *	Move to point 2
N110 X+95 *	Point 3: first straight line for corner 3
N120 G24 R10 *	Program chamfer with length 10 mm
N130 Y+5 *	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
N140 G24 R20 *	Program chamfer with length 20 mm
N150 X+5 *	Move to last contour point 1, second straight line for corner 4
N160 G27 R5 F500 *	Tangential departure
N170 G40 X-20 Y-20 F1000 *	Retract tool in the working plane, cancel radius compensation
N180 G00 Z+250 M2 *	Retract in the tool axis, end program
N999999 %LINEAR G71 *	



Example: Circular movements with Cartesian coordinates



%CIRCULAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define blank form for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define tool in the program
N40 T1 G17 S4000 *	Call tool in the spindle axis and with the spindle speed S
N50 G00 G40 G90 Z+250 *	Retract tool in the spindle axis at rapid traverse
N60 X-10 Y-10 *	Pre-position the tool
N70 G01 Z-5 F1000 M3 *	Move to working depth at feed rate F = 1000 mm/min
N80 G01 G41 X+5 Y+5 F300 *	Approach the contour at point 1, activate radius compensation G41
N90 G26 R5 F150 *	Tangential approach
N100 Y+85 *	Point 2: first straight line for corner 2
N110 G25 R10 *	Insert radius with $R = 10$ mm, feed rate: 150 mm/min
N120 X+30 *	Move to point 3: Starting point of the arc
N130 G02 X+70 Y+95 R+30 *	Move to point 4: end point of the arc with G02, radius 30 mm
N140 G01 X+95 *	Move to point 5
N150 Y+40 *	Move to point 6
N160 G06 X+40 Y+5 *	Move to point 7: End point of the arc, radius with tangential
	connection to point 6, TNC automatically calculates the radius

N170 G01 X+5 *	Move to last contour point 1
N180 G27 R5 F500 *	Depart the contour on a circular arc with tangential connection
N190 G40 X-20 Y-20 F1000 *	Retract tool in the working plane, cancel radius compensation
N200 G00 Z+250 M2 *	Retract tool in the tool axis, end of program
N999999 %CTRCIIIAR G71 *	



Example: Full circle with Cartesian coordinates



%C-CC G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+12.5 *	Define the tool
N40 T1 G17 S3150 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 I+50 J+50 *	Define the circle center
N70 X-40 Y+50 *	Pre-position the tool
N80 G01 Z-5 F1000 M3 *	Move to working depth
N90 G41 X+0 Y+50 F300 *	Approach starting point, radius compensation G41
N100 G26 R5 F150 *	Tangential approach
N110 G02 X+0 *	Move to the circle end point (= circle starting point)
N120 G27 R5 F500 *	Tangential departure
N130 G01 G40 X-40 Y-50 F1000 *	Retract tool in the working plane, cancel radius compensation
N140 G00 Z+250 M2 *	Retract tool in the tool axis, end of program
N999999 %C-CC G71 *	

6.5 Path Contours—Polar Coordinates

6.5 Path Contours—Polar Coordinates

Overview of path functions with polar coordinates

With polar coordinates you can define a position in terms of its angle **H** and its distance **R** relative to a previously defined pole **I**, **J** (see "Definition of pole and angle reference axis," page 72).

Polar coordinates are useful with:

Positions on circular arcs

Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Tool movement	Function	Required input
Straight line at feed rate Straight line at rapid traverse	G10 G11	Polar radius, polar angle of the straight-line end point
Circular path in clockwise direction Circular path in counterclockwise direction	G12 G13	Polar angle of the circle end point
Circular path corresponding to active direction of rotation	G15	Polar angle of the circle end point
Circular arc with tangential connection to the preceding contour element	G16	Polar radius, polar angle of the arc end point

Zero point for polar coordinates: pole I, J

You can set the pole **I**, **J** at any point in the machining program, before indicating points in polar coordinates. Set the pole in the same way as you would program the circle center.

Programming



Enter Cartesian coordinates for the pole, or if you want to use the last programmed position, enter **G29.** Before programming polar coordinates, define the pole. You can only define the pole in Cartesian coordinates. The pole remains in effect until you define a new pole.

Example NC blocks

N120 I+45 J+45 *



Straight line at rapid traverse G10 Straight line with feed rate G11 F . . .

The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.

Programming



- Polar coordinates radius R: Enter distance from the straight line end point to the pole I, J
 - Polar-coordinates angle H: Angular position of the straight-line end point between -360° and +360°

The sign of ${\bf H}$ depends on the angle reference axis:

- Angle from angle reference axis to **R** is counterclockwise: **H** >0
- Angle from angle reference axis to **R** is clockwise: **H** <0

Example NC blocks

N120 I+45 J+45 *	
N130 G11 G42 R+30 H+0 F300 M3 *	
N140 H+60 *	
N150 G91 H+60 *	
N160 G90 H+180 *	



Circular path G12/G13/G15 around pole I, J

The polar coordinate radius **R** is also the radius of the arc. It is defined by the distance from the starting point to the pole **I**, **J**. The last programmed tool position before the **G12**, **G13** or **G15** block is the starting point of the arc.

Direction

- In clockwise direction: G12
- In counterclockwise direction: G13
- Without programmed direction: G15. The TNC traverses the circular arc with the last programmed direction of rotation.

Programming



▶ Polar-coordinates angle **H**: Angular position of the arc end point between -5400° and +5400°

Example NC blocks

N180 I+25 J+25 * N190 G11 G42 R+20 H+0 F250 M3 * N200 G13 H+180 *



6.5 Path Contours – Polar Coordinates

Х

Circular arc with tangential connection

The tool moves on a circular path, starting tangentially from a preceding contour element.

Programming



- Polar coordinates radius R: Distance from the arc end point to the pole I, J
- Polar coordinates angle H: Angular position of the arc end point

Example NC blocks

N120 I+40 J+35 * N130 G01 G42 X+0 Y+35 F250 M3 * N140 G11 R+25 H+120 * N150 G16 R+30 H+30 * N160 G01 Y+0 *



The pole is **not** the center of the contour arc!

Helical interpolation

A helix is a combination of a circular movement in a main plane and a linear movement perpendicular to this plane.

A helix is programmed only in polar coordinates.

Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

For calculating a helix that is to be cut in an upward direction, you need the following data:

Thread revolutions <i>n</i>	Thread revolutions + thread overrun at the start and end of the thread
Total height <i>h</i>	Thread pitch P times thread revolutions <i>n</i>
Incremental total angle H	Number of revolutions times 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Pitch P times (thread revolutions + thread overrun at start of thread)



120

40 = 1

30

Y

35=J

Shape of the helix

The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction	Radius comp.
Right-handed	Z+	G13	G41
Left-handed	Z+	G12	G42
Right-handed	Z	G12	G42
Left-handed	Z	G13	G41

External thread				
Right-handed	Z+	G13	G42	
Left-handed	Z+	G12	G41	
Right-handed	Z–	G12	G41	
Left-handed	Z–	G13	G42	

Programming a helix

Always enter the same algebraic sign for the direction of rotation and the incremental total angle **G91 H.** The tool may otherwise move in a wrong path and damage the contour.

For the total angle **G91 H**, you can enter a value from -5400° to +5400°. If the thread has more than 15 revolutions, program the helix in a program section repeat (see "Program Section Repeats," page 404)

G 12

 Polar coordinates angle H: Enter the total angle of tool traverse along the helix in incremental dimensions.
After entering the angle, specify the tool axis with an axis selection key.

- Enter the coordinate for the height of the helix in incremental dimensions.
- Enter the radius compensation G41/G42 according to the table above.

Example NC blocks: Thread M6 x 1 mm with 5 revolutions

N120 I+40 J+25 *
N130 G01 Z+0 F100 M3 *
N140 G11 G41 R+3 H+270 *
N150 G12 G91 H-1800 Z+5 *



6.5 Path Contours—Polar Coordinates

Example: Linear movement with polar coordinates



%LINEARPO G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+7.5 *	Define the tool
N40 T1 G17 S4000 *	Tool call
N50 G00 G40 G90 Z+250 *	Define the datum for polar coordinates
N60 I+50 J+50 *	Retract the tool
N70 G10 R+60 H+180 *	Pre-position the tool
N80 G01 Z-5 F1000 M3 *	Move to working depth
N90 G11 G41 R+45 H+180 F250 *	Approach the contour at point 1
N110 G26 R5 *	Approach the contour at point 1
N120 H+120 *	Move to point 2
N130 H+60 *	Move to point 3
N140 H+0 *	Move to point 4
N150 H-60 *	Move to point 5
N160 H-120 *	Move to point 6
N170 H+180 *	Move to point 1
N180 G27 R5 F500 *	Tangential departure
N190 G40 R+60 H+180 F1000 *	Retract tool in the working plane, cancel radius compensation
N200 G00 Z+250 M2 *	Retract in the spindle axis, end of program
N999999 %LINEARPO G71 *	

ĺ

Example: Helix



%HELIX G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+5 *	Define the tool
N40 T1 G17 S1400 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 X+50 Y+50 *	Pre-position the tool
N70 G29 *	Transfer the last programmed position as the pole
N80 G01 Z-12.75 F1000 M3 *	Move to working depth
N90 G11 G41 R+32 H+180 F250 *	Approach first contour point
N100 G26 R2 *	Tangential connection
N110 G13 G91 H+3240 Z+13.5 F200 *	Helical interpolation
N120 G27 R2 F500 *	Tangential departure
N170 G01 G40 G90 X+50 Y+50 F1000 *	Retract in the tool axis, end program
N180 G00 Z+250 M2 *	

6.5 Path Contours-Polar Coordinates

To cut a thread with more than 16 revolutions

N80 G01 Z-12.75 F1000 M3 *	
N90 G11 G41 H+180 R+32 F250 *	
N100 G26 R2 *	Tangential approach
N110 G98 L1 *	Identify beginning of program section repeat
N120 G13 G91 H+360 Z+1.5 F200 *	Enter pitch directly as incremental Z value
N130 L1.24 *	Program the number of repeats (thread revolutions)
N999999 %HELIX G71 *	







Programming: Miscellaneous Functions

7.1 Entering Miscellaneous Functions M and G38 (STOP)

Fundamentals

With the TNC's miscellaneous functions—also called M functions—you can influence:

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring 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. The TNC displays the following dialog question:

Miscellaneous function M?

You usually enter only the number of the M function. 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 Electronic 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.

M functions come into effect in the block in which they are called. Unless the M function is only effective blockwise, it is canceled in a subsequent block or at the end of the program. Some M functions are effective only in the block in which they are called.

Entering an M function in a G38 block

If you program a G38 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 miscellaneous function M.

Example NC blocks

87 G38 M6

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

Overview

М	Effect E	ffective at block	start	end
M00	Stop program run Spindle STOP Coolant OFF			
M01	Optional program	STOP		-
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status dis MP7300)	splay (depends on		
M03	Spindle ON clockw	vise	-	
M04	Spindle ON counte	erclockwise	-	
M05	Spindle STOP			-
M06	Tool change Spindle STOP Program run stop (MP7440)	(depends on		
M08	Coolant ON		-	
M09	Coolant OFF			
M13	Spindle ON clockw Coolant ON	vise		
M14	Spindle ON counte Coolant ON	erclockwise		
M30	Same as M02			



7.3 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 52).

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.

The coordinate values on the TNC screen are shown with respect to the machine datum. Switch the display of coordinates in the status display to REF (see "Status Displays," page 39).

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.



7.3 Miscellaneous Functi<mark>ons</mark> for Coordinate Data

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 (see "General User Parameters" on page 506).

If datum setting is inhibited for all axes, the TNC no longer displays the soft key DATUM SET in the Manual Operation mode.

The figure at right 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 in the Working Space," page 492).





Activating the most recently entered datum: M104

Function

When processing pallet tables, the TNC may overwrite your most recently entered datum with values from the pallet table. With M104 you can reactivate the original datum.

Effect

M104 is effective only in the blocks in which it is programmed.

M104 becomes effective at the end of block.

Moving to positions in a non-tilted coordinate system with a tilted working plane: M130

Standard behavior with a tilted working plane

The TNC places the coordinates in the positioning blocks in the tilted coordinate system.

Behavior with M130

The TNC places coordinates in straight line blocks in the untilted coordinate system.

The TNC then positions the (tilted) tool to the programmed coordinates of the untilted system.



Subsequent positioning blocks or fixed cycles are carried out in a tilted coordinate system. This can lead to problems in fixed cycles with absolute pre-positioning.

The function M130 is allowed only if the tilted working plane function is active.

Effect

M130 functions only in straight-line blocks without tool radius compensation and in blocks in which M130 is programmed.

7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

Standard behavior

The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an exact stop.

In program blocks with radius compensation (**G41/G42**), the TNC automatically inserts a transition arc at outside corners.

Behavior with M90

The tool moves at corners with constant speed: This provides a smoother, more continuous surface. Machining time is also reduced. See figure at center right.

Application example: Surface consisting of a series of straight line segments.

Effect

M90 is effective only in the blocks in which it is programmed with M90.

M90 becomes effective at the start of block. Operation with servo lag must be active.







Insert rounding arc between straight lines: M112

Compatibility

For reasons of compatibility, the M112 function is still available on the iTNC 530 control. However, to define the tolerance for fast contour milling, HEIDENHAIN recommends the use of the TOLERANCE cycle for these TNCs (see "TOLERANCE (Cycle G62)," page 399).

Do not include points when executing noncompensated line blocks: M124

Standard behavior

The TNC runs all line blocks that have been entered in the active program.

Behavior with M124

When running **non-compensated line blocks** with very small point intervals, you can use parameter **E** to define a minimum point interval up to which the TNC will not include points during execution.

Effect

M124 becomes effective at the start of block.

The TNC automatically resets M124 if you select a new program.

Programming M124

If you enter M124 in a positioning block, the TNC continues the dialog for this block by asking you the minimum distance between points ${f E}$.

You can also define **E** through Q parameters (see "Programming: Q Parameters" on page 417).

7.4 Miscellaneous Functions f<mark>or C</mark>ontouring Behavior

Machining small contour steps: M97

Standard behavior

The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour.

In such cases the TNC interrupts program run and generates the error message "Tool radius too large."

Behavior with M97

The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point.

Program M97 in the same block as the outside corner.

Effect

M97 is effective only in the blocks in which it is programmed.



A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.





Example NC blocks

N50 G99 G01 R+20 *	Large tool radius
N130 X Y F M97 *	Move to contour point 13
N140 G91 Y-0.5 F *	Machine small contour step 13 to 14
N150 X+100 *	Move to contour point 15
N160 Y+0.5 F M97 *	Machine small contour step 15 to 16
N170 G90 X Y *	Move to contour point 17

Machining open contours: M98

Standard behavior

The TNC calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining.

Behavior with M98

With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined.

Effect

M98 is effective only in the blocks in which it is programmed.

M98 takes effect at the end of block.

Example NC blocks

Move to the contour points 10, 11 and 12 in succession:

N100 G01 G41 X	Y F *
N110 X G91 Y	M98 *
N120 X+ *	

Feed rate factor for plunging movements: M103

Standard behavior

The 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 x F%

Programming M103

If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor $\ensuremath{\mathsf{F}}.$

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):
N107 G01 G41 X+20 Y+20 F500 M103 F20 *	500
N180 Y+50 *	500
N190 G91 Z-2.5 *	100
N200 Y+5 Z-5 *	141
N210 X+50 *	500
N220 G90 Z+5 *	500

Feed rate in millimeters per spindle revolution: M136

Standard behavior

The TNC moves the tool at the programmed feed rate F in mm/min.

Behavior with M136

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.

Feed rate at circular arcs: M109/M110/M111

Standard behavior

The TNC applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109

The TNC adjusts the feed rate for circular arcs at inside and outside contours so that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110

The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.

Ê	M110 is also effective for the inside machining of circular arcs using contour cycles. If you define M109 or M110 before calling a machining cycle, the adjusted feed rate is also effective for circular arcs within machining cycles. The initial state is restored after finishing or aborting a machining cycle
	machining cycle.

Effect

M109 and M110 become effective at the start of block. To cancel M109 and M110, enter M111.

7.4 Miscellaneous Functions for Contouring Behavior

Calculating the radius-compensated path in advance (LOOK AHEAD): M120

Standard behavior

If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97 (see "Machining small contour steps: M97" on page 201) can be used to prohibit the error message, but this will result in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour.

Behavior with M120

The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block. Areas of the contour that might be damaged by the tool are not machined (dark areas in figure at right). You can also use M120 to calculate the radius compensation for digitized data or data created on an external programming system. This means that deviations from the theoretical tool radius can be compensated.

Use LA (Look Ahead) after M120 to define the number of blocks (maximum: 99) that you want the TNC to calculate in advance. Note that the larger the number of blocks you choose, the higher the block processing time will be.

Input

If you enter M120 in a positioning block, the TNC continues the dialog for this block by asking you the number of blocks LA that are to be calculated in advance.

Effect

M120 must be located in an NC block that also contains radius compensation G41 or G42. M120 is then effective from this block until

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA, or
- Call another program with %...

M120 becomes effective at the start of block.

Limitations

- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS. AT N.
- If you are using the path functions G25 and G24, the blocks before and after G25 or CHF must contain only coordinates of the working plane.



Superimposing handwheel positioning during program run: M118

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M118

M118 permits manual corrections by handwheel during program run. You can use this miscellaneous function by entering axis-specific values X, Y and Z (in mm) behind M118.

Programming M118

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. The coordinates are entered with the orange axis direction buttons or the ASCII keyboard.

Effect

Cancel handwheel positioning by programming M118 once again without X, Y and Z.

M118 becomes effective at the start of block.

Example NC blocks

If you want to be able to use the handwheel during program run to move the tool in the working plane X/Y by $\pm 1~\text{mm}$ from the programmed value:

G01 G41 X+0 Y+38.5 F125 M118 X1 Y1 *



M118 is always effective in the original coordinate system, even if the working plane is tilted.

M118 also functions in the Positioning with MDI mode of operation.

If M118 is active, the MANUAL OPERATION function is not available after a program interruption.

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 M104

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 MAX soft key to move to the limit of the traverse range.

Effect

M140 is effective only in the block in which it is programmed.

M140 becomes effective at the start of the 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.

N45	G01	X+0	Y+38.5	F125	M140	MB	50
N55	G01	X+0	Y+38.5	F125	M140	MB	MAX

M140 is also effective if the tilted-working-plane function, M114 or M128 is active. On machines with tilting heads, the TNC then moves the tool in the tilted coordinate system.

With the **FN18: SYSREAD ID230 NR6** function you can find the distance from the current position to the limit of the traverse range in the positive tool axis.

With M140 MB MAX you can only retract in positive direction.

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.



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 the block.

Delete modal program information: M142

Standard behavior

The TNC resets modal program information in the following situations:

- Select a new program.
- Execute a miscellaneous function M02, M30, or an N999999 %... block (depending on Machine Parameter 7300).
- Defining cycles for basic behavior with a new value

Behavior with M142

All modal program information except for basic rotation, 3-D rotation and $\ensuremath{\mathsf{Q}}$ parameters are reset.

Effect

M142 is effective only in the block in which it is programmed.

M142 becomes effective at the start of the block.

Delete basic rotation: M143

Standard behavior

The basic rotation remains in effect until it is reset or is overwritten with a new value.

Behavior with M143

The TNC erases a programmed basic rotation from the NC program.

Effect

M143 is effective only in the block in which it is programmed.

M143 becomes effective at the start of the block.

7.5 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116

Standard behavior

The TNC interprets the programmed feed rate in a rotary axis in degrees per minute. The contouring feed rate therefore depends on the distance from the tool center to the center of the rotary axis.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116



The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

The TNC interprets the programmed feed rate in a rotary axis in mm/ min. With this miscellaneous function, the TNC calculates the feed rate for each block at the start of the block. With a rotary axis, the feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane. With M117 you can reset M116. M116 is also canceled at the end of the program.

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior

The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° depends on Machine Parameter 7682. In MP7682 is set whether the TNC should consider the difference between nominal and actual position, or whether the TNC should always (even without M126) choose the shortest path traverse to the programmed position. Examples:

Actual position	Nominal position	Traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With M126, the TNC will move the axis on the shorter path of traverse if you reduce display of a rotary axis to a value less than 360°. Examples:

Actual position	Nominal position	Traverse
350°	10°	+20°
10°	340°	–30°

Effect

M126 becomes effective at the start of block. To cancel M126, enter M127. At the end of program, M126 is automatically canceled.

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:

N50 M94 *

To reduce display of the C axis only:

N50 M94 C *

To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

N50 G00 C+180 M94 *

Effect

M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.

Automatic compensation of machine geometry when working with tilted axes: M114 (software option 2)

The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated by a postprocessor and traversed in a positioning block. As the machine geometry is also relevant, the NC program must be calculated separately for each machine tool.

Behavior with M114

If the position of a controlled tilted axis changes in the program, the TNC automatically compensates the tool offset by a 3-D length compensation. As the geometry of the individual machine tools is set in machine parameters, the TNC also compensates machine-specific offsets automatically. Programs only need to be calculated by the postprocessor once, even if they are being run on different machines with TNC control.

If your machine tool does not have controlled tilted axes (head tilted manually or positioned by the PLC), you can enter the current valid swivel head position after M114 (e.g. M114 B+45, Q parameters permitted).

The radius compensation must be calculated by a CAD system or by a postprocessor. A programmed radius compensation G41/G42 will result in an error message.

If the tool length compensation is calculated by the TNC, the programmed feed rate refers to the point of the tool. Otherwise it refers to the tool datum.



If your machine tool is equipped with a swivel head that can be tilted under program control, you can interrupt program run and change the position of the tilted axis, for example with the handwheel.

With the RESTORE POS. AT N function, you can then resume program run at the block at which the part program was interrupted. If M114 is active, the TNC automatically calculates the new position of the tilted axis.

If you wish to use the handwheel to change the position of the tilted axis during program run, use M118 in conjunction with M128.

Effect

M114 becomes effective at the start of block, M115 at the end of block. M114 is not effective when tool radius compensation is active.

To cancel M114, enter M115. At the end of program, M114 is automatically canceled.





Maintaining the position of the tool tip when positioning with tilted axes (TCPM*): M128 (software option 2)



The machine geometry must be entered in Machine Parameters 7510 and following by the machine tool builder.

Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated and traversed in a positioning block (see figure for M114).

Behavior with M128

If the position of a controlled tilted axis changes in the program, the position of the tool tip to the workpiece remains the same.

If you wish to use the handwheel to change the position of the tilted axis during program run, use M118 in conjunction with M128. Handwheel positioning in a machine-based coordinate system is possible when M128 is active.



For tilted axes with Hirth coupling: Do not change the position of the tilted axis until after retracting the tool. Otherwise you might damage the contour.

After M128 you can program another feed rate, at which the TNC will carry out the compensation movements in the linear axes. If you program no feed rate here, or if you program a larger feed rate than is defined in MP7471, the feed rate from MP7471 will be effective.



Reset M128 before positioning with M91 or M92 and before a T block.

To avoid contour gouging you must use only spherical cutters with M128.

The tool length must refer to the spherical center of the tool tip.

If M128 is active, the TNC shows in the status display the following symbol: \bigotimes .

M128 on tilting tables

If you program a tilting table movement while M128 is active, the TNC rotates the coordinate system accordingly. If, for example, you rotate the C axis by 90° (through a positioning command or datum shift) and then program a movement in the X axis, the TNC executes the movement in the machine axis Y.

The TNC also transforms the defined datum, which has been shifted by the movement of the rotary table.



M128 with 3-D tool compensation

If you carry out a 3-D tool compensation with active M128 and active radius compensation G41/G42, the TNC will automatically position the rotary axes for certain machine geometries (Peripheral milling, see "Peripheral Milling: 3-D Radius Compensation with Workpiece Orientation," page 154).

Effect

M128 becomes effective at the start of block, M129 at the end of block. M128 is also effective in the manual operating modes and remains active even after a change of mode. The feed rate for the compensation movement will be effective until you program a new feed rate or until you reset M128 with M129.

To cancel M128, enter M129. The TNC also resets M128 if you select a new program in a program run operating mode.

Example NC blocks

Feed rate of 1000 mm/min for compensation movements.

G01 G41 X+0 Y+38.5 F125 M128 F1000 *

Exact stop at corners with nontangential transitions: M134

Standard behavior

The standard behavior of the TNC during positioning with rotary axes is to insert a transitional element in nontangential contour transitions. The contour of the transitional element depends on the acceleration, the rate of acceleration (jerk), and the defined tolerance for contour deviation.



With MP7440 you can change the standard behavior of the TNC so that M134 becomes active automatically whenever a program is selected (see "General User Parameters," page 506).

Behavior with M134

The TNC moves the tool during positioning with rotary axes so as to perform an exact stop at nontangential contour transitions.

Effect

M134 becomes effective at the start of block, M135 at the end of block.

You can reset M134 with M135. The TNC also resets M134 if you select a new program in a program run operating mode.

Selecting tilting axes: M138

Standard behavior

The TNC performs M114 and M128, and tilts the working plane, only in those axes for which the machine tool builder has set the appropriate machine parameters.

Behavior with M138

The TNC performs the above functions only in those tilting axes that you have defined using M138.

Effect

M138 becomes effective at the start of block.

You can reset M138 by reprogramming it without entering any axes.

Example NC blocks

Perform the above-mentioned functions only in the tilting axis C:

G00 G40 Z+100 M138 C *
Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block: M144

Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated, and traversed in a positioning block.

Behavior with M144

The TNC calculates into the position value any changes in the machine's kinematic configuration which result, for example, from adding a spindle attachment. If the position of a controlled tilted axis changes, the position of the tool tip to the workpiece is also changed. The resulting offset is calculated in the position display.



Positioning blocks with M91/M92 are permitted if M144 is active.

The position display in the operating modes FULL SEQUENCE and SINGLE BLOCK does not change until the tilting axes have reached their final position.

Effect

M144 becomes effective at the start of the block. M144 does not function in connection with M114, M128 or a tilted working plane.

You can cancel M144 by programming M145.

P	
	Γ

The machine geometry must be entered in Machine Parameters 7502 and following by the machine tool builder. The machine tool builder determines the behavior in the automatic and manual operating modes. Refer to your machine manual.

7.6 Miscellaneous Functions for Laser Cutting Machines

Principle

The TNC can control the cutting efficiency of a laser by transferring voltage values through the S-analog output. You can influence laser efficiency during program run through the miscellaneous functions M200 to M204.

Entering miscellaneous functions for laser cutting machines

If you enter an M function for laser cutting machines in a positioning block, the TNC continues the dialog by asking you the required parameters for the programmed function.

All miscellaneous functions for laser cutting machines become effective at the start of the block.

Output the programmed voltage directly: M200

Behavior with M200

The TNC outputs the value programmed after M200 as the voltage V.

Input range: 0 to 9999 V

Effect

M200 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of distance: M201

Behavior with M201

M201 outputs the voltage in dependence on the distance to be covered. The TNC increases or decreases the current voltage linearly to the value programmed for V.

Input range: 0 to 9999 V

Effect

M201 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of speed: M202

Behavior with M202

The TNC outputs the voltage as a function of speed. In the machine parameters, the machine tool builder defines up to three characteristic curves FNR in which specific feed rates are assigned to specific voltages. Use miscellaneous function M202 to select the curve FNR from which the TNC is to determine the output voltage.

Input range: 1 to 3

Effect

M202 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (timedependent ramp): M203

Behavior with M203

The TNC outputs the voltage V as a function of the time TIME. The TNC increases or decreases the current voltage linearly to the value programmed for V within the time programmed for TIME.

Input range

Voltage V: 0 to 9.999 Volt TIME: 0 to 1.999 seconds

Effect

M203 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (timedependent pulse): M204

Behavior with M204

The TNC outputs a programmed voltage as a pulse with a programmed duration TIME.

Input range

Voltage V: 0 to 9.999 Volt TIME: 0 to 1.999 seconds

Effect

M204 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.







Programming: Cycles

i

8.1 Working with Cycles

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and other special cycles are also provided as standard cycles (see table on next page).

Fixed cycles with numbers 200 and above 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.

In order to avoid erroneous entries during cycle definition, you should run a graphical program test before machining (see "Test Run" on page 459).

Defining a cycle using soft keys



- 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.
- 200 7

THREAD

- Select a cycle, e.g. DRILLING. The TNC initiates the programming dialog and asks 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 asked for in the dialog prompt is highlighted.
- Enter all parameters asked by the TNC and conclude each entry with the ENT key.
- The TNC ends the dialog when all required data has been entered.



N10 G200 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



Group of cycles	Soft key
Cycles for pecking, reaming, boring, counterboring, tapping and thread milling	DRILLING/ THREAD
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS
Cycles for producing hole patterns, such as circular or linear patterns	PATTERN
SL (Subcontour List) cycles which allow the contour- parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation	SL CYCLES
Cycles for face milling of flat or twisted surfaces	MULTIPASS
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Special cycles such as dwell time, program call, oriented spindle stop and tolerance	SPECIAL CYCLES

If you use indirect parameter assignments in fixed cycles with numbers greater than 200 (e.g. **D00 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. **D00 Q210**) directly in such cases.

In order to be able to run cycles G83 to G86, G74 to G78 and G56 to G59 on older TNC models, you must program an additional negative sign before the values for setup clearance and plunging depth.



Calling a cycle

8.1 Working with Cycles

Prerequisites

The following data must always be programmed before a cycle call:

- G30/G31 for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Define cycle

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:

- Cycle G220 for circular and Cycle G221 for linear hole patterns
- SL Cycle G14 CONTOUR GEOMETRY
- SL Cycle G20 CONTOUR DATA
- Cycle G62 TOLERANCE
- Coordinate transformation cycles
- Cycle G04 DWELL TIME

You can call all other cycles with the functions described as follows.

Calling a cycle with G79 (CYCL CALL)

The **G79** function calls the last defined fixed cycle once. The starting point of the cycle is the position that was programmed last before the G79 block.

- CYCL CALL
- ► 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 G79 PAT (CYCL CALL PAT)

The **G79 PAT** function calls the most recently defined fixed cycle at all positions defined in a point table (see "Point Tables" on page 226).

Calling a cycle with G79:G01 (CYCL CALL POS)

The **G79:G01** function calls the fixed cycle that was last defined. The starting point of the cycle is the position that you defined in the **G79:G01** block.



The TNC first moves the tool to the defined position and then calls the fixed cycle most recently defined.

The feed rate most recently defined in the **G79:G01** block applies only for traverse to the start position programmed in this block.

As a rule, the TNC moves without radius compensation (R0) to the position defined in the **G79:G01** block.

If you use **G79:GO1** to call a cycle in which a start position is defined (for example Cycle 212), then the TNC uses the position defined in the **G79:GO1** block as starting position.

Cycle call 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 execute the cycle automatically after every positioning block, program the first cycle call with **M89** (depending on machine parameter 7440).

To cancel the effect of **M89**, program:

- **M99** in the positioning block in which you move to the last starting point, or
- **G79**, or
- Define with CYCL DEF a new fixed cycle

Working with the secondary axes U/V/W

The TNC performs infeed movements in the axis that was defined in the TOOL CALL block as the spindle axis. It performs movements in the working plane only in the principal axes X, Y or Z. Exceptions:

- You program secondary axes for the side lengths in cycles G74 SLOT MILLING and G75/G76 POCKET MILLING.
- You program secondary axes in the contour geometry subprogram of an SL cycle.



8.2 Point Tables

Function

You should create a point table 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	To call the file manager, press the PGM MGT key.
FILE NAME ?	
	Enter the name and file type of the point table and confirm your entry with the ENT key.
ММ	To select the unit of measure, press the MM or INCH soft key. The TNC changes 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.



With the soft keys X OFF/ON, Y OFF/ON, Z OFF/ON (second soft-key row), you can specify which coordinates you want to enter in the point table.

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 $\ensuremath{\mathsf{ENT}}$ key.

Example NC block

N72 %:PAT: "NAMES"*



8.2 Point Tables

CYCL

Calling a cycle in connection with point tables

With **G79 PAT** the TNC runs the point table that you last defined (even if you have defined the point table in a program that was nested with %).

The TNC uses the coordinate in the spindle axis as the clearance height, where the tool is located during cycle call. A clearance height or 2nd set-up clearance that is defined separately in a cycle must not be greater than the clearance height defined in the global pattern.

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 **G79 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).
- If required, enter a miscellaneous function M, then confirm with the END key.

The TNC moves the tool back to the clearance height over each successive starting point (clearance height = the spindle axis coordinate for cycle call). To use this procedure for cycles above Cycle 199, you must define the 2nd set-up clearance (Ω 204) to equal 0.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103 (see "Feed rate factor for plunging movements: M103" on page 202).

Effect of the point tables with Cycles G83, G84 and G74 to G78

The TNC interprets the points of the working plane as coordinates of the hole centers. The coordinate of the spindle axis defines the upper surface of the workpiece, so the TNC can pre-position automatically (first in the working plane, then in the spindle axis).

Effect of the point tables with SL Cycles and Cycle G39

The TNC interprets the points as an additional datum shift.

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 G210 to G215

The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0.

Effect of the point tables with Cycles G251 to G254

The TNC interprets the points of the working plane as coordinates of the 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 ($\Omega 203$) as 0.

8.3 Cycles for Drilling, Tapping and Thread Milling

Overview

The TNC offers 19 cycles for all types of drilling operations:

Cycle	Soft key
G83 PECKING Without automatic pre-positioning	83 (
G200 DRILLING With automatic pre-positioning, 2nd set-up clearance	200 7
G201 REAMING With automatic pre-positioning, 2nd set-up clearance	201
G202 BORING With automatic pre-positioning, 2nd set-up clearance	202
G203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrementing	203 7
G204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	204
G205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	205 (111
G208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	208

i

Cycle	Soft key
G84 TAPPING With a floating tap holder	84
G85 RIGID TAPPING Without a floating tap holder	85 R T
G86 THREAD CUTTING For integration into OEM cycles	86
G206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	206
G207 RIGID TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	207 🛔 RT
G209 TAPPING W/ CHIP BRKG Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	209 🛔 RT
G262 THREAD MILLING Cycle for milling a thread in pre-drilled material	262
G263 THREAD MLLNG/CNTSNKG Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	263
G264 THREAD DRILLING/MLLNG Cycle for drilling into the solid material with subsequent milling of the thread with a tool	264
G265 HEL.THREAD DRLG/MLG Cycle for milling the thread into the solid material	265
G267 OUTSIDE THREAD MLLNG Cycle for milling an external thread and machining a countersunk chamfer	267



PECKING (Cycle G83)

- 1 The tool drills from the current position to the first plunging depth at the programmed feed rate F.
- 2 When it reaches the first plunging depth, the tool retracts at rapid traverse to the starting position and advances again to the first plunging depth minus the advanced stop distance t.
- **3** The advanced stop distance is automatically calculated by the control:
 - At a total hole depth of up to 30 mm: t = 0.6 mm
 - At a total hole depth exceeding 30 mm: t = hole depth / 50
 - Maximum advanced stop distance: 7 mm
- 4 The tool then advances with another infeed at the programmed feed rate F.
- **5** The TNC repeats this process (1 to 4) until the programmed depth is reached.
- 6 After a dwell time at the hole bottom, the tool is returned to the starting position at rapid traverse for chip breaking.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.



- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Total hole depth 2 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- Plunging depth 3 (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
 - the plunging depth is equal to the depth
 - the plunging depth is greater than the total hole depth
- Dwell time in seconds: Amount of time the tool remains at the total hole depth for chip breaking.
- **Feed rate F:** Traversing speed of the tool during drilling in mm/min.





Example: NC block

N10 G83 P01 2 P02 -20 P03 -8 P04 0 P05 500*



8.3 Cycles for Drilling, <mark>Tap</mark>ping and Thread Milling

DRILLING (Cycle G200)

- **1** The TNC positions the tool in the tool axis at rapid traverse to the setup 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 rapid traverse to the setup clearance, dwells there (if a dwell time was entered), and then moves at rapid traverse to the setup 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 rapid traverse.



ф,

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!





8.3 Cycles for Drilling, <mark>Tap</mark>ping and Thread Milling

200 🎸

- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- ► Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min.
- Plunging depth Q202 (incremental value): Infeed per cut. 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 release.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.

Example: NC blocks

N100 G00 Z+100 G40)
N110 G200 DRILLING	ì
Q200=2	;SET-UP CLEARANCE
Q291=-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
N120 X+30 Y+20 M3	M99
N130 X+80 Y+50 M99)
N140 Z+100 M2	

REAMING (Cycle G201)

ᇞ

- **1** The TNC positions the tool in the tool axis at rapid traverse to the programmed setup 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 rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!







- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool during reaming in mm/min.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- 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.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

Example: NC blocks

N100 G00 Z+100 G40
N110 G201 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
N120 X+30 Y+20 M3 M99
N130 X+80 Y+50 M99
N140 G00 Z+100 M2

201

BORING (Cycle G202)



ф,

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle G202.

- **1** The TNC positions the tool in the tool axis at rapid traverse to the setup clearance above the workpiece surface.
- **2** The tool drills 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 rapid traverse. If **Q214=0** the tool point remains on the wall of the hole.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

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.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!





- ► Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- ► Feed rate for plunging Q206: Traversing speed of the tool during boring in mm/min.
- Dwell time at depth Q211: Time in seconds that the tool remains at the hole bottom.
- 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.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- 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

ᇞ

- 1: Retract tool in the negative reference axis direction
- 2: Retract tool in the negative secondary axis direction
- 3: Retract tool in the positive reference axis direction
- 4: Retract tool in the positive secondary axis direction

Danger of collision

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.

Angle for spindle orientation Q336 (absolute value): Angle at which the TNC positions the tool before retracting it.

Example:

N100 G00 Z+100 G4	0
N110 G202 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
N120 X+30 Y+20 M3	
N130 G79	
N140 L X+80 Y+50	FMAX M99

202

.3 Cycles for Drilling, <mark>Tap</mark>ping and Thread Milling

UNIVERSAL DRILLING (Cycle G203)

- **1** The TNC positions the tool in the tool axis at rapid traverse to the programmed setup 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 set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at rapid traverse 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 you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

ar f

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



Example: NC blocks

N110 G203 UNIVERS	AL 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	; 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

- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- ▶ Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min.
- Plunging depth Q202 (incremental value): Infeed per cut. 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 release.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Decrement Q212 (incremental value): Value by which the TNC decreases the plunging depth Q202 after each infeed.
- ▶ No. of breaks before retracting Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip release. For chip breaking, the TNC retracts the tool each time by the value in Q256.
- Minimum plunging depth Q205 (incremental value): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- 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.
- Retraction rate for chip breaking Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.

8.3 Cycles for Drilling, <mark>Tap</mark>ping and Thread Milling



BACK BORING (Cycle G204)



Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Special boring bars for upward cutting are required for this cycle.

This cycle allows holes to be bored from the underside of the workpiece.

- **1** The TNC positions the tool in the tool axis at rapid traverse 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 setup clearance and then, if entered, to the 2nd setup clearance at rapid traverse.



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

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.







- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth of counterbore Q249 (incremental value): 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.
- ► Material thickness Q250 (incremental value): Thickness of the workpiece.
- Off-center distance Q251 (incremental value): Offcenter distance for the boring bar; value from tool data sheet.
- ▶ Tool edge height Q252 (incremental value): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet.
- Feed rate for pre-positioning Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- ▶ Feed rate for counterboring Q254: Traversing speed of the tool during counterboring in mm/min.
- **Dwell time** Q255: Dwell time in seconds at the top of the bore hole.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- 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).
 - 1 Retract tool in the negative ref. axis direction
 - 2 Retract tool in the neg. secondary axis direction
 - 3 Retract tool in the positive ref. axis direction
 - 4 Retract tool in the pos. secondary axis direction

Example: NC blocks

N110 G204 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 COUNTERBORING
Q255=0	;DWELL TIME
Q203=+20	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN
Q336=0	;ANGLE OF SPINDLE

204 👖

8 Programming: Cycles

Danger of collision

ᇞ

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.

Angle for spindle orientation Q336 (absolute value): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole.



UNIVERSAL PECKING (Cycle G205)

- 1 The TNC positions the tool in the tool axis at rapid traverse to the programmed 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 is moved at rapid traverse to set-up clearance and then at rapid traverse to the entered starting position 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 you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

ᇞ

8 Programming: Cycles



- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- ▶ Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min.
- Plunging depth Q202 (incremental value): Infeed per cut. 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
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ **Decrement** Q212 (incremental value): Value by which the TNC decreases the plunging depth Q202.
- Minimum plunging depth Q205 (incremental value): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205.
- ▶ Upper advanced stop distance Q258 (incremental value): 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.
- ▶ Lower advanced stop distance Q259 (incremental value): 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.
- If you enter Q258 not equal to Q259, the TNC will change the advance stop distances between the first and last plunging depths at the same rate.



Example: NC blocks

N110 G205 UNIVERS	AL 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 POSITION
Q253=750	;F PRE-POSITIONING

- Infeed depth for chip breaking Q257 (incremental value): Depth at which the TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- Retraction rate for chip breaking Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- 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.
- 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.

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.



BORE MILLING (Cycle G208)

- 1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface and then moves the tool to the bore hole circumference on a rounded arc (if enough space is available).
- **2** The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate.
- **3** When the drilling depth is reached, the TNC once again traverses a full circle to remove the material remaining after the initial plunge.
- 4 The TNC then positions the tool at the center of the hole again.
- **5** Finally the TNC returns to the set-up clearance at rapid traverse. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.



al

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

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 have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface! 208

- Set-up clearance Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool during helical drilling in mm/min.
- ▶ Infeed per helix Q334 (incremental value): Depth of the tool plunge with each helix (=360°).

Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the **ANGLE** column of the tool table (see "Tool Data," page 137). The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Nominal diameter Q335 (absolute value): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.
- Roughing diameter Q342 (absolute value): As soon as you enter a value greater than 0 in Q342, the TNC no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter.





Example: NC blocks

N120 G208 BORE	MILLING
Q200=2	;SET-UP CLEARANCE
Q201=-80	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q334=1.5	;PLUNGING DEPTH
Q203=+100	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q335=25	;NOMINAL DIAMETER
Q342=0	;ROUGHING DIAMETER

8 Programming: Cycles

8.3 Cycles for Drilling, <mark>Tap</mark>ping and Thread Milling

TAPPING with a floating tap holder (Cycle G84)

- 1 The tool drills to the total hole depth in one movement.
- **2** Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the dwell time.
- **3** At the starting position, the direction of spindle rotation reverses once again.



84 🛔

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

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 ${\tt M3},$ for left-hand threads use ${\tt M4.}$

- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- Total hole depth 2 (thread length, incremental value): Distance between workpiece surface and end of thread.
- Dwell time in seconds: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.
- Feed rate F: Traversing speed of the tool during tapping.

The feed rate is calculated as follows: F = S x 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 block

N13 G84 P01 2 P02 -20 P03 0 P04 100 *



TAPPING NEW with floating tap holder (Cycle G206)

- **1** The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
- 2 The tool drills 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 you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.
- **4** At the set-up clearance, the direction of spindle rotation reverses once again.

Before	progra	amn	ning,	not	e the	e tollov	ving:
_							

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

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.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

ᇞ



- Set-up clearance Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch.
- Total hole depth Q201 (thread length, incremental value): Distance between workpiece surface and end of thread.
- ► Feed rate F Q206: Traversing speed of the tool during tapping.
- Dwell time at bottom Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

The feed rate is calculated as follows: F = S x 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

N250 G206 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

RIGID TAPPING (Cycle G85)



Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The TNC cuts the thread without a floating tap holder in one or more passes.

Rigid tapping offers the following advantages over tapping with a floating tap holder:

- Higher machining speeds possible.
- Repeated tapping of the same thread is possible; repetitions are enabled via spindle orientation to the 0° position during cycle call (depending on MP7160).
- Increased traverse range of the spindle axis due to absence of a floating tap holder.



P

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

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 spindle speed override is used during tapping, the feed rate is automatically adjusted.

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**).



- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- ▶ Total hole depth 2 (incremental value): Distance between workpiece surface (beginning of thread) and end of thread
- ▶ Pitch 3:

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

- + = right-hand thread
- = left-hand thread

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.



Example: NC block

N18 G85 P01 2 P02 -20 P03 +1 *


RIGID TAPPING NEW (Cycle G207)



al

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The TNC cuts the thread without a floating tap holder in one or more passes.

See "RIGID TAPPING (Cycle G85)," page 252, for the advantages that rigid tapping offers over tapping with a floating tap holder.

- **1** The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.
- 2 The tool drills 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 you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.
- 4 The TNC stops the spindle rotation at the set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

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 spindle speed override is used during tapping, the feed rate is automatically adjusted.

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).

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



207 | RT

- Set-up clearance Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- ▶ Total hole depth Q201 (incremental value): Distance between workpiece surface and end of thread.
- ▶ Pitch Q239

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

- + = right-hand thread
- = left-hand thread
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

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 the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.



Example: NC blocks

N26 G207	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q239=+1	;PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

1



THREAD CUTTING (Cycle G86)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Cycle G86 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped. Tool approach and departure must be programmed separately. The most convenient way to do this is by using OEM cycles. The machine tool builder can give you further information.



Before programming, note the following:

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during thread cutting, the feed rate is automatically adjusted.

The feed-rate override knob is disabled.

The TNC automatically activates and deactivates spindle rotation. Do not program **M3** or **M4** before cycle call.



Total hole depth 1: Distance between current tool position and end of thread.

The algebraic sign for the total hole depth determines the working direction (a negative value means a negative working direction in the tool axis).

Pitch 2:

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads: += right-hand thread (M3 with negative depth)

- = left-hand thread (M4 with negative depth)



Example: NC block

N22 G86 P01 -20 P02 +1 *



TAPPING WITH CHIP BREAKING (Cycle G209)



Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The tool machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

- 1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface. There it carries out an oriented spindle stop.
- 2 The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition.
- It then reverses the direction of spindle rotation again and advances to the next infeed depth.
- The TNC repeats this process (2 to 3) until the programmed thread 4 depth is reached.
- The tool is then retracted to the set-up clearance. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.
- The TNC stops the spindle rotation at the set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation G40.

The algebraic sign for the parameter thread depth determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

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).

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a positive depth is entered. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- Set-up clearance Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Thread depth Q201 (incremental value): Distance between workpiece surface and end of thread.
- ▶ Pitch Q239

Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:

- + = right-hand thread
- = left-hand thread
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Infeed depth for chip breaking Q257 (incremental value): Depth at which TNC carries out chip breaking
- ▶ Retraction rate for chip breaking Q256: The TNC multiplies the pitch Q239 by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter Q256 = 0, the TNC retracts the tool completely from the hole (to the set-up clearance) for chip release.
- ▶ Angle for spindle orientation Q336 (absolute value): Angle at which the TNC positions the tool before machining the thread. This allows you to regroove the thread, if required.

Retracting after a program interruption

HEIDENHAIN iTNC 530

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 the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.



Example: NC blocks

N260 G207 RIGID	TAPPING NEW
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q239=+1	;PITCH
Q203=+25	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE

Fundamentals of thread milling

Prerequisites

- Your machine tool should feature internal spindle cooling (cooling lubricant at least 30 bar, compressed air supply at least 6 bar).
- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius DR in the tool call.
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265 you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread / – = lefthand thread) and milling method Q351 (+1 = climb / –1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/Up- cut	Work direction
Right-handed	+	+1(RL)	Z+
Left-handed	_	–1(RR)	Z+
Right-handed	+	–1(RR)	Z–
Left-handed	_	+1(RL)	Z–

External thread	Pitch	Climb/Up- cut	Work direction
Right-handed	+	+1(RL)	Z–
Left-handed	-	–1(RR)	Z–
Right-handed	+	–1(RR)	Z+
Left-handed	_	+1(RL)	Z+

Danger of collision

ф

Always program the same algebraic sign for the infeeds: Cycles comprise several sequences of operation that are independent of each other. The order of precedence according to which the work direction is determined is described with the individual cycles. For example, if you only want to repeat the countersinking process of a cycle, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

Procedure in case of a tool break

If a tool break occurs during thread cutting, stop the program run, change to the Positioning with MDI operating mode and move the tool in a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

The TNC references the programmed feed rate during thread milling to the tool cutting edge. Since the TNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRRORING in only one axis.

THREAD MILLING (Cycle G262)

- 1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.
- 2 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
- **3** The tool then approaches the thread diameter tangentially in a helical movement. Before the helical approach, a compensating motion of the tool axis is carried out in order to begin at the programmed starting plane for the thread path.
- 4 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset movements or in one continuous movement.
- **5** After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
- 6 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign for the cycle parameter thread depth determines the working direction. If you program the thread DEPTH = 0, the cycle will not be executed.

The nominal thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the pitch of the tool diameter is four times smaller than the nominal thread diameter.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!







ф

- 262 💧
- ▶ Nominal diameter Q335: Nominal thread diameter.
- Thread pitch Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 - + = right-hand thread
 - = left-hand thread
- Thread depth Q201 (incremental value): Distance between workpiece surface and root of thread.
- Threads per step Q355: Number of thread revolutions by which the tool is offset, see figure at lower right
 - $\mathbf{0}$ = one 360° helical path to the depth of thread.
 - $\boldsymbol{1}$ = continuous helical path over the entire length of the thread

>1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch.

- ▶ Feed rate for pre-positioning Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- ▶ C1imb or up-cut Q351: Type of milling operation with M03.
 - **+1** = climb milling
 - **-1** = up-cut milling
- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

N250 G262 THREAD	MILLING
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;PITCH
Q201=-20	;THREAD DEPTH
Q355=0	;THREADS PER STEP
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q207=500	;FEED RATE FOR MILLING



THREAD MILLING/COUNTERSINKING (Cycle G263)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Countersinking

- **2** The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up clearance, and then at the feed rate for countersinking to the countersinking depth.
- **3** If a safety clearance to the side has been entered, the TNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth.
- **4** Then, depending on the available space, the TNC makes a tangential approach to the core diameter, either tangentially from the center or with a pre-positioning move to the side, and follows a circular path.

Countersinking at front

- **5** The tool moves at the feed rate for pre-positioning to the sinking depth at front.
- 6 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
- 7 The tool then moves on a semicircle to the hole center.

Thread milling

- **8** The TNC moves the tool at the programmed feed rate for prepositioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).
- **9** Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
- **10** After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
- **11** At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.



al a

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:

1st: Depth of thread 2nd: Countersinking depth 3rd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you wish to countersink with the front of the tool, define the countersinking depth as 0.

Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **Nominal diameter** Q335: Nominal thread diameter.
- Thread pitch Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 - + = right-hand thread
 - = left-hand thread
- Thread depth Q201 (incremental value): Distance between workpiece surface and root of thread.
- Countersinking depth Q356 (incremental value): Distance between tool point and the top surface of the workpiece.
- ▶ Feed rate for pre-positioning Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- Climb or up-cut Q351: Type of milling operation with M03.
 - **+1** = climb milling
 - **-1** = up-cut milling
- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Set-up clearance to the side Q357 (incremental value): Distance between tool tooth and the wall.
- Depth at front Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- Countersinking offset at front Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.







263

8 Programming: Cycles

1

- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ Feed rate for counterboring Q254: Traversing speed of the tool during counterboring in mm/min.
- ► Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

N250 G263 THREAD	MLLNG/CNTSNKG
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;PITCH
Q201=-16	;THREAD DEPTH
Q356=-20	;COUNTERSINKING DEPTH
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q357=0.2	;CLEARANCE TO SIDE
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
0207=500	;FEED RATE FOR MILLING

THREAD DRILLING/MILLING (Cycle G264)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed setup clearance above the workpiece surface.

Drilling

- **2** The tool drills to the first plunging depth at the programmed feed rate for plunging.
- **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 is moved at rapid traverse to set-up clearance and then at rapid traverse to the entered starting position above the first plunging depth.
- **4** The tool then advances with another infeed at the programmed feed rate.
- **5** The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

Countersinking at front

- 6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
- 7 The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
- 8 The tool then moves on a semicircle to the hole center.



Thread milling

- **9** The TNC moves the tool at the programmed feed rate for prepositioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).
- **10** Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
- **11** After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
- **12** At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence: 1st: Depth of thread

2nd: Total hole depth 3rd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

ᇞ



- ▶ Nominal diameter Q335: Nominal thread diameter.
- Thread pitch Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 - + = right-hand thread
 - = left-hand thread
- Thread depth Q201 (incremental value): Distance between workpiece surface and root of thread.
- ▶ Total hole depth Q356 (incremental value): Distance between workpiece surface and bottom of hole.
- ▶ Feed rate for pre-positioning Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- Climb or up-cut Q351: Type of milling operation with M03.
 - +1 = climb milling
 - -1 = up-cut milling
- Plunging depth Q202 (incremental value): Infeed per cut. 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
- ▶ Upper advanced stop distance Q258 (incremental value): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole.
- Infeed depth for chip breaking Q257 (incremental value): Depth at which TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- Retraction rate for chip breaking Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.
- Depth at front Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- Countersinking offset at front Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.







- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

N250 G264 THREAD	DRILLING/MILLING
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;PITCH
Q201=-16	;THREAD DEPTH
Q356=-20	;TOTAL HOLE DEPTH
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q2O2=5	;PLUNGING DEPTH
Q258=0.2	;ADVANCED STOP DISTANCE
Q257=5	;DEPTH FOR CHIP BRKNG
Q256=0.2	;DIST. FOR CHIP BRKNG
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q206=150	;FEED RATE FOR PLNGNG
Q207=500	;FEED RATE FOR MILLING

1

HELICAL THREAD DRILLING/MILLING (Cycle G265)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Countersinking at front

- **2** If countersinking is before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking is after thread milling, the tool moves at the feed rate for pre-positioning to the countersinking depth.
- **3** The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
- 4 The tool then moves on a semicircle to the hole center.

Thread milling

- **5** The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.
- **6** The tool then approaches the thread diameter tangentially in a helical movement.
- 7 The tool moves on a continuous helical downward path until it reaches the thread depth.
- **8** After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
- **9** At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation **G40**.

The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence: 1st: Depth of thread

2nd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

呣

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **Nominal diameter** Q335: Nominal thread diameter.
- Thread pitch Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 - + = right-hand thread
 - = left-hand thread
- Thread depth Q201 (incremental value): Distance between workpiece surface and root of thread.
- ▶ Feed rate for pre-positioning Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- Depth at front Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- Countersinking offset at front Q359 (incremental value): Distance by which the TNC moves the tool center away from the hole center.
- ▶ Countersink Q360: Execution of the chamfer
 - **0** = before thread machining
 - 1 = after thread machining
- Set-up clearance Ω200 (incremental value): Distance between tool tip and workpiece surface.



Х

- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ Feed rate for counterboring Q254: Traversing speed of the tool during counterboring in mm/min.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

N250 G265 HEL.	THREAD DRLG/MLG
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;PITCH
Q201=-16	;THREAD DEPTH
Q253=750	;F PRE-POSITIONING
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q360=0	;COUNTERSINKING
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLING

OUTSIDE THREAD MILLING (Cycle G267)

1 The TNC positions the tool in the tool axis at rapid traverse to the programmed set-up clearance above the workpiece surface.

Countersinking at front

- 2 The TNC moves in the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.
- **3** The tool moves at the feed rate for pre-positioning to the sinking depth at front.
- **4** The TNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
- 5 The tool then moves on a semicircle to the starting point.

Thread milling

- 6 The TNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.
- 7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
- 8 The tool then approaches the thread diameter tangentially in a helical movement.
- **9** Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset movements or in one continuous movement.
- **10** After this, the tool departs the contour tangentially and returns to the starting point in the working plane.



11 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.



ф

Before programming, note the following:

Program a positioning block for the starting point (stud center) in the working plane with radius compensation **G40**.

The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. The working direction is defined in the following sequence:

1st: Depth of thread 2nd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

The algebraic sign for the cycle parameter thread depth determines the working direction.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **Nominal diameter** Q335: Nominal thread diameter.
- Thread pitch Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
 - + = right-hand thread
 - = left-hand thread
- Thread depth Q201 (incremental value): Distance between workpiece surface and root of thread.
- ▶ Threads per step Q355: Number of thread revolutions by which the tool is offset (see figure at lower right):
 - **0** = one helical line to the thread depth
 - $\mathbf{1}$ = continuous helical path over the entire length of the thread

>1 = several helical paths with approach and departure; between them, the TNC offsets the tool by Q355, multiplied by the pitch.

- ▶ Feed rate for pre-positioning Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min.
- Climb or up-cut Q351: Type of milling operation with M03.
 - **+1** = climb milling
 - -1 = up-cut milling







267 💧

- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth at front Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- Countersinking offset at front Q359 (incremental value): Distance by which the TNC moves the tool center away from the stud center.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Feed rate for counterboring Ω254: Traversing speed of the tool during counterboring in mm/min.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

N250 G267 OUTSIDE	THREAD MLLNG
Q335=10	;NOMINAL DIAMETER
Q239=+1.5	;PITCH
Q201=-20	;THREAD DEPTH
Q355=0	;THREADS PER STEP
Q253=750	;F PRE-POSITIONING
Q351=+1	;CLIMB OR UP-CUT
Q200=2	;SET-UP CLEARANCE
Q358=+0	;DEPTH AT FRONT
Q359=+0	;OFFSET AT FRONT
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q254=150	;F COUNTERBORING
Q207=500	;FEED RATE FOR MILLING

Example: Drilling cycles



%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 DRILLING	Define cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q2O3=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	

N70 X+10 Y+10 M3 *	Approach hole 1, spindle ON
N80 Z-8 M99 *	Pre-position in the spindle axis, cycle call
N90 Y+90 M99 *	Approach hole 2, call cycle
N100 Z+20 *	Retract in the spindle axis
N110 X+90 *	Approach hole 3
N120 Z-8 M99 *	Pre-position in the spindle axis, cycle call
N130 Y+10 M99 *	Approach hole 4, call cycle
N140 G00 Z+250 M2 *	Retract in the tool axis, end program
N999999 %C200 G71 *	Call the cycle



Example: Drilling cycles

Program sequence

- Program the drilling cycle in the main program
- Program machining within a subprogram (see "Subprograms," page 403)



%C18 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G86 P01 +30 P02 -1.75 *	Define THREAD CUTTING cycle
N70 X+20 Y+20 *	Approach hole 1
N80 L1.0 *	Call subprogram 1
N90 X+70 Y+70 *	Approach hole 2
N100 L1.0 *	Call subprogram 1
N110 G00 Z+250 M2 *	Retract tool, end of main program
N120 G98 L1 *	Subprogram 1: Thread cutting
N130 G36 S0 *	Define angle of spindle orientation
N140 M19 *	Orient spindle (makes it possible to cut repeatedly)
N150 G01 G91 X-2 F1000 *	Tool offset to prevent collision during tool infeed (depends
	on core diameter and tool)
N160 G90 Z-30 *	Move to starting depth
N170 G91 X+2 *	Reset the tool to hole center
N180 G79 *	Call Cycle 18
N190 G90 Z+5 *	Retract tool
N200 G98 L0 *	End of subprogram 1
N999999 %C18 G71 *	

Example: Calling drilling cycles in connection with point tables

The drill hole coordinates are stored in the point table TAB1.PNT and are called by the TNC with G79 PAT.

The tool radii are selected so that all work steps can be seen in the test graphics.

Program sequence

- Centering
- Drilling
- Tapping



%1 G71*		
N10 G30 G17 X+0 Y+0 Z-	-20 *	Define the workpiece blank
N20 G31 X+100 Y+100	Z+0 *	
N30 G99 1 L+0 R+4 *		Tool definition of center drill
N40 G99 2 L+0 R+2.4 *		Define tool: drill
N50 G99 3 L+0 R+3 *		Tool definition of tap
N60 T1 G17 S5000 *		Tool call of centering drill
N70 G01 G40 Z+10 F5000) *	Move tool to clearance height (Enter a value for F.
		The TNC positions to the clearance height after every cycle.)
N80 %:PAT: "TAB1" *		Defining point tables
N90 G200 DRILLING		Cycle definition: Centering
Q200=2 ;SET	T-UP CLEARANCE	
Q201=-2 ;DE	PTH	
Q206=150 ; FEI	ED RATE FOR PLNGNG	
Q202=2 ;PLU	UNGING DEPTH	
Q210=0 ;DWI	ELL TIME AT TOP	
Q203=+0 ;SUF	RFACE COORDINATE	0 must be entered here, effective as defined in point table
Q204=0 ;2NI	D SET-UP CLEARANCE	0 must be entered here, effective as defined in point table
Q211=0.2 ;DWH	ELL TIME AT DEPTH	

1

N100 G79 "PAT" F5000 M3 *	Cycle call in connection with point table TAB1.PNT
	Feed rate between points: 5000 mm/min
N110 G00 G40 Z+100 M6 *	Retract the tool, change the tool
N120 T2 G17 S5000 *	Call toll: drill
N130 G01 G40 Z+10 F5000 *	Move tool to clearance height (enter a value for F)
N140 G200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-25 ;DEPTH	
Q206=150 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q2O3=+O ;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	
N150 G79 "PAT" F5000 M3 *	Cycle call in connection with point table TAB1.PNT
N160 G00 G40 Z+100 M6 *	Retract the tool, change the tool
N170 T3 G17 S200 *	Tool call for tap
N180 G00 G40 Z+50 *	Move tool to clearance height
N190 G84 P01 +2 P02 -15 P030 P04 150 *	Cycle definition for tapping
N200 G79 "PAT" F5000 M3 *	Cycle call in connection with point table TAB1.PNT
N210 G00 G40 Z+100 M2*	Retract in the tool axis, end program
N99999 %1 G71*	

Point table TAB1.PNT

	TAB1.	PNT	MM
NR	X	Y	Z
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]			

8.4 Cycles for Milling Pockets, Studs and Slots

Overview

Cycle	Soft key
G251 RECTANGULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	251
G252 CIRCULAR POCKET Roughing/finishing cycle with selection of machining operation and helical plunging	252
G253 SLOT MILLING Roughing/finishing cycle with selection of machining operation and reciprocal/helical plunging	253
G254 CIRCULAR SLOT Roughing/finishing cycle with selection of machining operation and reciprocal/helical plunging	254
G75/G76 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning G75: In clockwise direction G76: In counterclockwise direction	75 1 78 1
G212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	212
G213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	213
G77/G78 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning G77: In clockwise direction	77
G78: In counterclockwise direction	
G214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	214
G215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	215
G74 SLOT MILLING Roughing/finishing cycle without automatic pre- positioning, vertical depth infeed	74



Cycle	Soft key
G210 SLOT RECIP. PLNG Roughing/finishing cycle with automatic pre- positioning, with reciprocating plunge infeed	210
G211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre- positioning, with reciprocating plunge infeed	211

RECTANGULAR POCKET (Cycle G251)

Use Cycle G251 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 process

- 1 The tool penetrates the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with Parameter Q366.
- 2 The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowance (Parameter Q368) into account.
- **3** This process is repeated until the programmed pocket depth is reached.

Finishing process

- **4** If finishing allowances have been defined, the TNC first finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.
- **5** The TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.



Before programming, note the following:

Pre-position the tool in the machining plane to the starting position with radius compensation R0. Note Parameter Q367 (pocket position).

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **G79:G01 X... Y...** or in U and V if you programmed **G79:G01 U... V...**

The TNC automatically pre-positions the tool in the tool axis. Note Parameter $\Omega 204$ (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.



吵

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



defined.

- 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 allowances (Q368, Q369) have been
- First side length Q218 (incremental value): Pocket length, parallel to the reference axis of the working plane.
- Second side length Q219 (incremental value): Pocket length, parallel to the minor axis of the working plane.
- ▶ **Corner radius** Q220: Radius of the pocket corner: If you make no entry here, the TNC assumes that the corner radius is equal to the tool radius.
- ▶ Finishing allowance for side Q368 (incremental value): Finishing allowance in the working plane.
- Angle of rotation Q224 (absolute): Angle by which the entire pocket is rotated. The center of rotation is the position at which the tool is located when the cycle is called.
- Pocket position Q367: Position of the pocket in reference to the position of the tool when the cycle is called (see figure at center right):
 - **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
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Climb or up-cut Q351: Type of milling operation with M03.
 - **+1** = climb milling
 - **-1** = up-cut milling







1

- Depth Q201 (incremental value): Distance between workpiece surface and pocket floor.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- ▶ Finishing allowance for floor Q369 (incremental value): Finishing allowance in the tool axis.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool while moving to depth in mm/min.
- ▶ Infeed for finishing Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Absolute coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.





- Path overlap factor Q370: Q370 x tool radius = stepover factor k.
- ▶ Plunging strategy Q366: Type of plunging strategy.
 - 0 = vertical plunging. In the tool table, the plunging angle ANGLE for the active tool must also be defined as 0. The TNC will otherwise display an error message.
 - 1 = helical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined not equal to 0. The TNC will otherwise display an error message.

Example: NC blocks

N80 G251 RECTANG	JLAR POCKET
Q215=0	;MACHINING OPERATION
Q218=80	;FIRST SIDE LENGTH
Q219=60	;SECOND SIDE LENGTH
Q220=5	;CORNER RADIUS
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;POCKET POSITION
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q370=1	;TOOL PATH OVERLAP
Q366=1	;PLUNGING
N90 G79:G01 X+50	Y+50 F10000 M3



CIRCULAR POCKET (Cycle G252)

Use Cycle G252 CIRCULAR POCKET to completely machine circular 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 process

- 1 The tool penetrates the workpiece at the pocket center and advances to the first plunging depth. Specify the plunging strategy with Parameter Q366.
- 2 The TNC roughs out the pocket from the inside out, taking the overlap factor (Parameter Q370) and the finishing allowance (Parameter Q368) into account.
- **3** This process is repeated until the programmed pocket depth is reached.

Finishing process

- **4** If finishing allowances have been defined, the TNC first finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.
- **5** The TNC then finishes the pocket walls, in multiple infeeds if so specified. The pocket wall is approached tangentially.

Before programming, note the following:

Pre-position the tool in the machining plane to the starting position (circle center) with radius compensation R0.

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **G79:G01 X... Y...** or in U and V if you programmed **G79:G01 U... V...**

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.
Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



ᇞ

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 allowances (Ω 368, Ω 369) have been defined.

- Circle diameter Q223: Diameter of the finished pocket.
- ▶ Finishing allowance for side Q368 (incremental value): Finishing allowance in the working plane.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Climb or up-cut Q351: Type of milling operation with M03.

+1 = climb milling

-1 = up-cut milling

- Depth Q201 (incremental value): Distance between workpiece surface and pocket floor.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- ▶ Finishing allowance for floor Q369 (incremental value): Finishing allowance in the tool axis.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool while moving to depth in mm/min.
- ▶ Infeed for finishing Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.





8.4 Cycles for Milling Pockets, Studs and Slots

- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Absolute coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Path overlap factor Q370: Q370 x tool radius = stepover factor k.
- ▶ Plunging strategy Q366: Type of plunging strategy.
 - 0 = vertical plunging. In the tool table, the plunging angle ANGLE for the active tool must also be defined as 0. The TNC will otherwise display an error message.
 - 1 = helical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined not equal to 0. The TNC will otherwise display an error message.



Example: NC blocks

N80 G252 CIRCULAR	POCKET
Q215=0	;MACHINING OPERATION
Q223=60	;CIRCLE DIAMETER
Q368=0.2	;ALLOWANCE FOR SIDE
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q370=1	;TOOL PATH OVERLAP
Q366=1	;PLUNGING
N90 G79:G01 X+50 Y	(+50 F10000 M3

SLOT MILLING (Cycle G253)

Use Cycle G253 to completely machine a slot. 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 process

- 1 The tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. If space permits, the TNC plunges helically instead of reciprocatingly. Specify the plunging strategy with Parameter Q366.
- 2 The TNC roughs the slot at the active infeed depth.
- 3 This process is repeated until the slot depth is reached.

Finishing process

- 4 If finishing allowances have been defined, the TNC first roughs the slot floor. The slot floor is approached tangentially.
- **5** The TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially.



Before programming, note the following:

Pre-position the tool in the machining plane to the starting position with radius compensation R0. Note Parameter Q367 (slot position).

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **G79:G01 X... Y...** or in U and V if you programmed **G79:G01 U... Y...**

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.

衂

253

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- 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 allowances (Q368, Q369) have been defined.

- Slot length Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot
- Slot width Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).
- ▶ Finishing allowance for side Q368 (incremental value): Finishing allowance in the working plane.
- Angle of rotation Q224 (absolute): Angle by which the entire slot is rotated. The center of rotation is the position at which the tool is located when the cycle is called.
- Slot position (0/1/2/3/4) Q367: Position of the slot in reference to the position of the tool when the cycle is called (see figure at center right):
 - **0:** Tool position = Center of slot
 - **1:** Tool position = Left end of slot
 - **2:** Tool position = Center of left slot circle
 - 3: Tool position = Center of right slot circle
 - **4:** Tool position = Right end of slot
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- ▶ Climb or up-cut Q351: Type of milling operation with M03.
 - **+1** = climb milling
 - **-1** = up-cut milling





8.4 Cycles for Milling Pockets, Studs and Slots

- Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- ▶ Finishing allowance for floor Q369 (incremental value): Finishing allowance in the tool axis.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool while moving to depth in mm/min.
- Infeed for finishing Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.



- 8.4 Cycles for Milling Pockets, Studs and Slots
- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Absolute coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ Plunging strategy Q366: Type of plunging strategy.
 - 0 = vertical plunging. In the tool table, the plunging angle ANGLE for the active tool must also be defined as 0. The TNC will otherwise display an error message.
 - 1 = helical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined not equal to 0. The TNC will otherwise display an error message.



Example: NC blocks

N80 G253 SLOT M	ILLING
Q215=0	;MACHINING OPERATION
Q218=80	;SLOT LENGTH
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q224=+0	;ANGLE OF ROTATION
Q367=0	;SLOT POSITION
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=1	;PLUNGING
N90 G79:G01 X+5	0 Y+50 F10000 M3

CIRCULAR SLOT (Cycle G254)

Use Cycle G254 to completely machine a circular slot. 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 process

- 1 The tool moves in a reciprocating motion at the plunging angle defined in the tool table to the first infeed depth. If space permits, the TNC plunges helically instead of reciprocatingly. Specify the plunging strategy with Parameter Q366.
- 2 The TNC roughs the slot at the active infeed depth.
- 3 This process is repeated until the slot depth is reached.

Finishing process

- **4** If finishing allowances have been defined, the TNC first roughs the slot floor. The slot floor is approached tangentially.
- **5** The TNC then finishes the slot walls, in multiple infeeds if so specified. The slot side is approached tangentially.



Before programming, note the following:

Pre-position the tool in the machining plane with radius compensation R0. Define Parameter Q367 (**Reference for slot position**) appropriately.

The TNC runs the cycle in the axes (machining plane) with which you approached the starting position. For example, in X and Y if you programmed **G79:G01 X... Y...** or in U and V if you programmed **G79:G01 U... Y...**

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.



吵

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

i



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 allowances (Q368, Q369) have been defined.

- Slot width Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).
- ▶ Finishing allowance for side Q368 (incremental value): Finishing allowance in the working plane.
- Pitch circle diameter Q375: Enter the diameter of the pitch circle.
- Reference for slot position (0/1/2/3) Q367: Position of the slot in reference to the position of the tool when the cycle is called (see figure at center right):

0: The tool position is not taken into account. The slot position is determined from the entered pitch circle center and the starting angle.

1: Tool position = Center of left slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.

2: Tool position = Center of center line. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.

3: Tool position = Center of right slot circle. Starting angle Q376 refers to this position. The entered pitch circle center is not taken into account.

- Center in 1st axis Q216 (absolute value): Center of the pitch circle in the reference axis of the working plane. Only effective if Q367 = 0.
- Center in 2nd axis Q217 (absolute value): Center of the pitch circle in the minor axis of the working plane. Only effective if Q367 = 0.
- Starting angle Q376 (absolute value): Enter the polar angle of the starting point.
- Angular length Q248 (incremental value): Enter the angular length of the slot.





- Angle increment Q378 (incremental): Angle by which the entire slot is rotated. The center of rotation is at the center of the pitch circle.
- Number of repetitions Q377: Number of machining operations on the pitch circle.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Climb or up-cut Q351: Type of milling operation with M03.
 - +1 = climb milling
 - -1 = up-cut milling
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- ▶ Finishing allowance for floor Q369 (incremental value): Finishing allowance in the tool axis.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool while moving to depth in mm/min.
- ▶ Infeed for finishing Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.





- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Absolute coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- ▶ Plunging strategy Q366: Type of plunging strategy.
 - 0 = vertical plunging. In the tool table, the plunging angle ANGLE for the active tool must also be defined as 0. The TNC will otherwise display an error message.
 - 1 = helical plunging. In the tool table, the plunging angle ANGLE for the active tool must be defined not equal to 0. The TNC will otherwise display an error message.



Example: NC blocks

N80 G254 CIRCULAR	SLOT
Q215=0	;MACHINING OPERATION
Q219=12	;SLOT WIDTH
Q368=0.2	;ALLOWANCE FOR SIDE
Q375=80	;PITCH CIRCLE DIA.
Q367=0	;REF. SLOT POSITION
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q376=+45	;STARTING ANGLE
Q248=90	;ANGULAR LENGTH
Q378=0	;STEPPING ANGLE
Q377=1	;NR OF REPETITIONS
Q207=500	;FEED RATE FOR MILLING
Q351=+1	;CLIMB OR UP-CUT
Q201=-20	;DEPTH
Q202=5	;PLUNGING DEPTH
Q369=0.1	;ALLOWANCE FOR FLOOR
Q206=150	;FEED RATE FOR PLNGNG
Q338=5	;INFEED FOR FINISHING
Q200=2	;SET-UP CLEARANCE
Q203=+0	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q366=1	;PLUNGING
G90 G79:G01 X+50	(+50 F10000 M3

) (

POCKET MILLING (Cycles G75, G76)

- 1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first plunging depth.
- 2 The cutter begins milling in the positive axis direction of the longer side (on square pockets, always starting in the positive Y direction) and then roughs out the pocket from the inside out.
- **3** This process (1 to 2) is repeated until the depth is reached.
- **4** At the end of the cycle, the TNC retracts the tool to the starting position.

Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

Pre-position over the pocket center with radius compensation **G40**.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The following prerequisite applies for the 2nd side length: 2nd side length greater than [(2 x rounding radius) + stepover factor k].

Direction of rotation during rough-out

- In clockwise direction: G75 (DR-)
- In counterclockwise direction: G76 (DR+)



- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Milling depth 2 (incremental value): Distance between workpiece surface and bottom of pocket
- ▶ Plunging depth 3 (incremental value): Infeed per cut 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
- ► Feed rate for plunging: Traversing speed of the tool during penetration
- First side length 4 (incremental value): Pocket length, parallel to the reference axis of the working plane





N27	G75	P01 2	P02 -20	P03 5 P04 100
	P05	X+80	P06 Y+40	P07 275 P08 5 *
N35	G76	P01 2	P02 -20	P03 5 P04 100
	P05	X+80	P06 Y+40	P07 275 P08 5 *

- > 2nd side length 5: Pocket width
- **Feed rate F:** Traversing speed of the tool in the working plane.
- Rounding off radius: Radius for the pocket corners. If radius = 0 is entered, the pocket corners will be rounded with the radius of the cutter.

Calculations:

Stepover factor $k = K \times R$

- K: is the overlap factor, preset in MP7430, and
- R: is the cutter radius

POCKET FINISHING (Cycle G212)

- 1 The TNC automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
- 2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the allowance and tool radius into account for calculating the starting point. If necessary, the TNC penetrates at the pocket center.
- **3** If the tool is at the 2nd set-up clearance, it moves at rapid traverse to the set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
- **4** The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
- **5** The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
- 6 This process (3 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).

Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

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 want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Minimum size of the pocket: 3 times the tool radius.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!







砚

8.4 Cycles for Milling Pockets, Studs and Slots



212

- Depth Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a value lower than that defined in Q207.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the pocket in the reference axis of the working plane.
- Center in 2nd axis Q217 (absolute value): Center of the pocket in the minor axis of the working plane.
- First side length Q218 (incremental value): Pocket length, parallel to the reference axis of the working plane.
- Second side length Q219 (incremental value): Pocket length, parallel to the minor axis of the working plane
- ▶ **Corner radius** Q220: Radius of the pocket corner: If you make no entry here, the TNC assumes that the corner radius is equal to the tool radius.
- Allowance in 1st axis Q221 (incremental value): Allowance for pre-positioning in the reference axis of the working plane referenced to the length of the pocket.

N350 G212 POCKET	FINISHING
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q207=500	;FEED RATE FOR MILLING
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q218=80	;FIRST SIDE LENGTH
Q219=60	;SECOND SIDE LENGTH
Q220=5	;CORNER RADIUS
Q221=0	;OVERSIZE

STUD FINISHING (Cycle G213)

- 1 The TNC moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
- 2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3 times the tool radius.
- **3** If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
- **4** The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
- **5** The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
- 6 This process (3 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).

Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

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 want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!







8.4 Cycles for Milling Pockets, Studs and Slots



213 📗

- Depth Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the stud in the reference axis of the working plane.
- Center in 2nd axis Q217 (absolute value): Center of the stud in the minor axis of the working plane.
- ▶ First side length Q218 (incremental value): Length of stud parallel to the reference axis of the working plane.
- Second side length Q219 (incremental value): Length of stud parallel to the secondary axis of the working plane.
- **Corner radius** Q220: Radius of the stud corner.
- Allowance in 1st axis Q221 (incremental value): Allowance for pre-positioning in the reference axis of the working plane referenced to the length of the stud.

N350 G213 STUD	FINISHING
Q200=2	;SET-UP CLEARANCE
Q291=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q207=500	;FEED RATE FOR MILLING
Q203=+30	;SURFACE COORDINATE
Q294=50	;2ND SET-UP CLEARANCE
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q218=80	;FIRST SIDE LENGTH
Q219=60	;SECOND SIDE LENGTH
Q220=5	;CORNER RADIUS
Q221=0	;OVERSIZE

CIRCULAR POCKET MILLING (Cycle G77, G78)

- 8.4 Cycles for Milling Pockets, Studs and Slots
- 1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first plunging depth.
- 2 The tool subsequently follows a spiral path at the feed rate F see figure at right. For calculating the stepover factor k, see "POCKET MILLING (Cycles G75, G76)," page 300.
- **3** This process is repeated until the depth is reached.
- **4** At the end of the cycle, the TNC retracts the tool to the starting position.

Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

Pre-position over the pocket center with radius compensation **G40**.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Direction of rotation during rough-out

In clockwise direction: G77 (DR-)

In counterclockwise direction: G78 (DR+)

78

G

- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Milling depth 2: Distance between workpiece surface and bottom of pocket.
- ▶ Plunging depth 3 (incremental value): Infeed per cut 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





- Feed rate for plunging: Traversing speed of the tool during penetration
- **Circular radius:** Radius of the circular pocket
- ► Feed rate F: Traversing speed of the tool in the working plane.



Example: NC blocks

N26	G77 P05	P01 40	2 PO2 -20 PO6 250 *	P035	P04 100
•••					
N48	G78	P01	2 P02 -20	P03 5	P04 100
	P05	40	P06 250 *		

i

CIRCULAR POCKET FINISHING (Cycle G214)

- 1 The TNC automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
- 2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point. If you enter a workpiece blank diameter of 0, the TNC plunge-cuts into the pocket center.
- **3** If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
- **4** The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
- **5** After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
- 6 This process (3 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, the TNC retracts the tool at rapid traverse to set-up clearance, or, if programmed, to the 2nd set-up clearance and then to the center of the pocket (end position = starting position).



8.4 Cycles for Milling Pockets, Studs and Slots

Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

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 want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!









214

- Depth Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a value lower than that defined in Q207.
- Plunging depth Q202 (incremental value): Infeed per cut.
- ► Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the pocket in the reference axis of the working plane.
- Center in 2nd axis Q217 (absolute value): Center of the pocket in the minor axis of the working plane.
- Workpiece blank diameter Q222: Diameter of the premachined pocket for calculating the pre-position. Enter the workpiece blank diameter to be less than the diameter of the finished part.
- ▶ Finished part diameter Q223: Diameter of the finished pocket. Enter the diameter of the finished part to be greater than the workpiece blank diameter and greater than the tool diameter.

N420 G214 CIRCULAR POCKET FINISHING
Q200=2 ;SET-UP CLEARANCE
Q201=-20 ;DEPTH
Q206=150 ;FEED RATE FOR PLNGNG
Q202=5 ;PLUNGING DEPTH
Q207=500 ;FEED RATE FOR MILLING
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q216=+50 ;CENTER IN 1ST AXIS
Q217=+50 ;CENTER IN 2ND AXIS
Q222=79 ;WORKPIECE BLANK DIA.
Q223=80 ;FINISHED PART DIA.



8.4 Cycles for Milling Pockets, Studs and Slots

CIRCULAR STUD FINISHING (Cycle G215)

- 1 The TNC automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
- **2** From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 2 times the tool radius.
- **3** If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
- **4** The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
- **5** The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
- 6 This process (3 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).

Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

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 want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!







ᇞ

- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- Plunging depth Q202 (incremental value): Infeed per cut. Enter a value greater than 0.
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the stud in the reference axis of the working plane.
- **Center in 2nd axis** Q217 (absolute value): Center of the stud in the minor axis of the working plane.
- ▶ Workpiece blank diameter Q222: Diameter of the premachined stud for calculating the pre-position. Enter the workpiece blank diameter to be greater than the diameter of the finished part.
- Diameter of finished part Q223: Diameter of the finished stud. Enter the diameter of the finished part to be less than the workpiece blank diameter.

N430 G215 C. STUD	FINISHING
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLNGNG
Q202=5	;PLUNGING DEPTH
Q207=500	;FEED RATE FOR MILLING
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q222=81	;WORKPIECE BLANK DIA.
Q223=80	;FINISHED PART DIA.



SLOT MILLING (Cycle G74)

Roughing process

- 1 The TNC moves the tool inward by the milling allowance (half the difference between the slot width and the tool diameter). From there it plunge-cuts into the workpiece and mills in the longitudinal direction of the slot.
- 2 After downfeed at the end of the slot, milling is performed in the opposite direction. This process is repeated until the programmed milling depth is reached.

Finishing process

- **3** The TNC advances the tool at the slot bottom on a tangential arc to the outside contour. The tool subsequently climb mills the contour (with M3).
- **4** At the end of the cycle, the tool is retracted at rapid traverse to the set-up clearance. If the number of infeeds was odd, the tool returns to the starting position at the level of the set-up clearance.



Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point.

Pre-position to the center of the slot and offset by the tool radius into the slot with radius compensation **G40**.

The cutter diameter must be not be larger than the slot width and not smaller than half the slot width.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.





- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Milling depth 2 (incremental value): Distance between workpiece surface and bottom of pocket
- ▶ Plunging depth 3 (incremental value): Infeed per cut. The tool will drill to the depth in one operation if:
 - the plunging depth is equal to the depth
 - the plunging depth is greater than the depth
- **Feed rate for plunging:** Traversing speed during penetration.
- ▶ 1st side length 4: Slot length; specify the sign to determine the first milling direction.
- > 2nd side length 5: Slot width.
- ▶ Feed rate F: Traversing speed of the tool in the working plane.





Example: NC block

N44 G74 P01 2 P02 -20 P0 5 P04 100 P05 X+80 P06 Y+12 P07 275 *

SLOT with reciprocating plunge-cut (Cycle G210)

Roughing process

- 1 At rapid traverse, the TNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the left circle. From there, the TNC positions the tool to the set-up clearance above the workpiece surface.
- 2 The tool moves at the feed rate for milling to the workpiece surface. From there, the cutter advances in the longitudinal direction of the slot—plunge-cutting obliquely into the material—until it reaches the center of the right circle.
- **3** The tool then moves back to the center of the left circle, again with oblique plunge-cutting. This process is repeated until the programmed milling depth is reached.
- **4** For the purpose of face milling, the TNC moves the tool at the milling depth to the other end of the slot and then back to the center of the slot.

Finishing process

- 5 The TNC positions the tool in the center of the left circle and then moves it tangentially to the left end of the slot. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed.
- **6** When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the left circle.
- 7 At the end of the cycle, the tool is retracted at rapid traverse to the set-up clearance and—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

During roughing the tool plunges into the material with a sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.







8.4 Cycles for Milling Pockets, Studs and Slots

Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



al,

Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.

- Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Plunging depth Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.
- Machining operation (0/1/2) Q215: Define the machining operation:
 0: Roughing and finishing
 - 1: Only roughing
 - 2: Only finishing
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the slot in the reference axis of the working plane.
- Center in 2nd axis Q217 (absolute value): Center of the slot in the minor axis of the working plane.
- First side length Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot.
- Second side length Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).

N510 G210 SLOT	RECIP. PLNG
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q207=500	;FEED RATE FOR MILLING
Q202=5	;PLUNGING DEPTH
Q215=0	;MACHINING OPERATION
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q218=80	;FIRST SIDE LENGTH
Q219=12	;SECOND SIDE LENGTH
Q224=+15	;ANGLE OF ROTATION
Q338=5	;INFEED FOR FINISHING
Q206=150	;FEED RATE FOR PLNGNG



- Angle of rotation Q224 (absolute value): Angle by which the entire slot is rotated. The center of rotation lies in the center of the slot.
- ▶ Infeed for finishing Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- Feed rate for plunging Q206: Traversing speed of the tool while moving to depth in mm/min. Only in effect during finishing if an infeed for finishing has been entered.

i

CIRCULAR SLOT with reciprocating plunge-cut (Cycle G211)

Roughing process

- 1 At rapid traverse, the TNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the right circle. From there, the tool is positioned to the programmed set-up clearance above the workpiece surface.
- 2 The tool moves at the milling feed rate to the workpiece surface. From there, the cutter advances—plunge-cutting obliquely into the material—to the other end of the slot.
- **3** The tool then moves at a downward angle back to the starting point, again with oblique plunge-cutting. This process (steps 2 to 3) is repeated until the programmed milling depth is reached.
- **4** For the purpose of face milling, the TNC moves the tool at the milling depth to the other end of the slot.

Finishing process

- **5** The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed. The starting point for the finishing process is the center of the right circle.
- **6** When the tool reaches the end of the contour, it departs the contour tangentially.
- 7 At the end of the cycle, the tool is retracted at rapid traverse to the set-up clearance and—if programmed—to the 2nd set-up clearance.



Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

During roughing the tool plunges into the material with a helical sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.







Enter in MP7441 bit 2 whether the TNC should output an error message (bit 2=1) or not (bit 2=0) if a positive depth is entered.

Danger of collision!

Keep in mind that the TNC reverses the calculation for prepositioning when a **positive depth is entered**. This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

211

ᇞ

- Set-up clearance Ω200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Plunging depth Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement.
- Machining operation (0/1/2) Q215: Define the machining operation:
 0: Roughing and finishing
 - 1: Only roughing
 - 2: Only finishing
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the slot in the reference axis of the working plane.
- Center in 2nd axis Q217 (absolute value): Center of the slot in the minor axis of the working plane.
- Pitch circle diameter Q244: Enter the diameter of the pitch circle.
- Second side length Q219: Enter the slot width. If you enter a slot width that equals the tool diameter, the TNC will carry out the roughing process only (slot milling).

N520 G211 CIRCULAR	SLOT
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q207=500	;FEED RATE FOR MILLING
Q202=5	;PLUNGING DEPTH
Q215=0	;MACHINING OPERATION
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	CENTER IN 2ND AXIS
Q244=80	;PITCH CIRCLE DIA.
Q219=12	;SECOND SIDE LENGTH
Q245=+45	;STARTING ANGLE
Q248=90	;ANGULAR LENGTH
Q338=5	;INFEED FOR FINISHING
Q206=150	;FEED RATE FOR PLNGNG

- Starting angle Q245 (absolute value): Enter the polar angle of the starting point.
- ► Angular length Q248 (incremental value): Enter the angular length of the slot.
- ▶ Infeed for finishing Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- ▶ Feed rate for plunging Q206: Traversing speed of the tool while moving to depth in mm/min. Only in effect during finishing if an infeed for finishing has been entered.



Example: Milling pockets, studs and slots



%C210 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Define the tool for roughing/finishing
N40 G99 T2 L+0 R+3 *	Define slotting mill
N50 T1 G17 S3500 *	Call the tool for roughing/finishing
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G213 STUD FINISHING	Define cycle for machining the contour outside
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q2O2=5 ;PLUNGING DEPTH	
Q207=250 ;FEED RATE FOR MILLING	i de la companya de l
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=90 ;FIRST SIDE LENGTH	
Q219=80 ;SECOND SIDE LENGTH	
Q220=0 ;CORNER RADIUS	
Q221=5 ;OVERSIZE	

i

N80 G79 M03 *	Call cycle for machining the contour outside
N90 G252 CIRCULAR POCKET	Define CIRCULAR POCKET MILLING cycle
Q215=0 ;MACHINING OPERATION	
Q223=50 ;CIRCLE DIAMETER	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q207=500 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-30 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	
Q206=150 ;FEED RATE FOR PLNGNG	
Q338=5 ;INFEED FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q370=1 ;TOOL PATH OVERLAP	
Q366=1 ;PLUNGING	
N100 G00 G40 X+50 Y+50 *	
N110 Z+2 M99 *	Call CIRCULAR POCKET MILLING cycle
N120 Z+250 M06 *	Tool change
N130 T2 G17 S5000 *	Call slotting mill
N140 G254 CIRCULAR SLOT	Define SLOT cycle
Q215=0 ;MACHINING OPERATION	
Q219=8 ;SLOT WIDTH	
Q368=0.2 ;ALLOWANCE FOR SIDE	
Q375=70 ;PITCH CIRCLE DIA.	
Q367=0 ;REF. SLOT POSITION	No pre-positioning in X/Y required
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q376=+45 ;STARTING ANGLE	
Q248=90 ;ANGULAR LENGTH	
Q378=180 ;STEPPING ANGLE	Starting point for second slot
Q377=2 ;NR OF REPETITIONS	
Q207=500 ;FEED RATE FOR MILLING	
Q351=+1 ;CLIMB OR UP-CUT	
Q201=-20 ;DEPTH	
Q202=5 ;PLUNGING DEPTH	
Q369=0.1 ;ALLOWANCE FOR FLOOR	



0206=150 :FFFD RATE FOR PINGNG	
Q550-5 ,INIELD FOR FINISHING	
Q200=2 ;SET-UP CLEARANCE	
Q2O3=+O ;SURFACE COORDINATE	
Q204=50 ;2ND SET-UP CLEARANCE	
Q366=1 ;PLUNGING	
N150 G79:G01 X+50 Y+50 F10000 M03 *	Call SLOT cycle
N160 G00 Z+250 M02 *	Retract in the tool axis, end program
N999999 %C210 G71 *	

i

8.5 Cycles for Machining Hole Patterns

Overview

The TNC provides two cycles for machining hole patterns directly:

Cycle	Soft key
G220 CIRCULAR PATTERN	220
G221 LINEAR PATTERN	221

You can combine Cycle G220 and Cycle G221 with the following fixed cycles:



If you have to machine irregular hole patterns, use **G79 "PAT"** to develop point tables (see "Point Tables" on page 226).

Cycle G74	SLOT MILLING
Cycle G75/G76	POCKET MILLING
Cycle G77/G78	CIRCULAR POCKET MILLING
Cycle G83	PECKING
Cycle G84	TAPPING with a floating tap holder
Cycle G85	RIGID TAPPING without a floating tap holder
Cycle G86	THREAD CUTTING
Cycle G200	DRILLING
Cycle G201	REAMING
Cycle G202	BORING
Cycle G203	UNIVERSAL DRILLING
Cycle G204	BACK BORING
Cycle G205	UNIVERSAL PECKING
Cycle G206	TAPPING NEW with a floating tap holder
Cycle G207	RIGID TAPPING NEW without a floating tap holder
Cycle G208	BORE MILLING
Cycle G209	TAPPING WITH CHIP BREAKING
Cycle G212	POCKET FINISHING
Cycle G213	STUD FINISHING
Cycle G214	CIRCULAR POCKET FINISHING
Cycle G215	CIRCULAR STUD FINISHING
Cycle G251	RECTANGULAR POCKET
Cycle G252	CIRCULAR POCKET MILLING
Cycle G253	SLOT MILLING
Cycle G254	CIRCULAR SLOT (cannot be combined with Cycle 220)
Cycle G262	THREAD MILLING

Cycle G263	THREAD MILLING/COUNTERSINKING
Cycle G264	THREAD DRILLING/MILLING
Cycle G265	HELICAL THREAD DRILLING/MILLING
Cycle G267	OUTSIDE THREAD MILLING

8 Programming: Cycles
8.5 Cycles f<mark>or Machining Hole Patterns</mark>

CIRCULAR PATTERN (Cycle G220)

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 spindle axis.
- 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 for the next machining operation on a straight line at set-up clearance (or 2nd set-up clearance).
- **4** This process (1 to 3) is repeated until all machining operations have been executed.



Before programming, note the following:

Cycle G220 is DEF active, which means that Cycle G220 automatically calls the last defined fixed cycle.

If you combine Cycle G220 with one of the fixed cycles G200 to G209, G212 to G215 and G262 to G267, the setup clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G220 will be effective for the selected fixed cycle.

- Center in 1st axis Q216 (absolute value): Center of the pitch circle in the reference axis of the working plane.
- Center in 2nd axis Q217 (absolute value): Center of the pitch circle in the minor axis of the working plane.
- Pitch circle diameter Q244: Diameter of the pitch circle.
- Starting angle Q245 (absolute value): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle.
- Stopping angle Q246 (absolute value): 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 complete 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.





N530 G220 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	;NR OF REPETITIONS
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q203=1	;MOVE TO CLEARANCE
Q365=0	;TYPE OF TRAVERSE

- 8.5 Cycles f<mark>or</mark> Machining Hole Patterns
- Stepping angle Q247 (incremental value): 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 (– = clockwise).
- Number of repetitions Q241: Number of machining operations on a pitch circle.
- Set-up clearance Ω200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Moving to clearance height Q301: Definition of how the tool is to move between machining processes.
 O: Move to the set-up clearance between operations.
 1: Move to the 2nd set-up clearance between the measuring points.
- Type of traverse? Line=0/Arc=1 Q365: Definition of the path function with which the tool is to move between machining operations.
 O: Move between operations on a straight line
 - 1: Move between operations on the pitch circle



Before programming, note the following:

Cycle G221 is DEF active, which means that Cycle G221 automatically calls the last defined fixed cycle.

If you combine Cycle G221 with one of the fixed cycles G200 to G209, G212 to G215 and G262 to G267, the setup clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G221 will be effective for the selected fixed cycle.

1 The TNC automatically moves the tool 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 spindle axis.
- 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 for 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 carries out the machining operation.
- **6** From this position the tool approaches the starting point for 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 processed in a reciprocating movement.







221

- Starting point 1st axis Q225 (absolute value): Coordinate of the starting point in the reference axis of the working plane.
- Starting point 2nd axis Q226 (absolute value): Coordinate of the starting point in the minor axis of the working plane.
- Spacing in 1st axis Q237 (incremental value): Spacing between each point on a line.
- Spacing in 2nd axis Q238 (incremental value): Spacing between each line.
- Number of columns Q242: Number of machining operations on a line.
- ▶ Number of lines Q243: Number of passes.
- Angle of rotation Q224 (absolute value): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point.
- Set-up clearance Ω200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface.
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Moving to clearance height Q301: Definition of how the tool is to move between machining processes.
 O: Move to the set-up clearance between operations.
 1: Move to the 2nd set-up clearance between the measuring points.

N540 G221 CARTESIA	NN PATTERN
Q225=+15	;STARTING PNT 1ST AXIS
Q226=+15	;STARTING PNT 2ND AXIS
Q237=+10	;SPACING IN 1ST AXIS
Q238=+8	;SPACING IN 2ND AXIS
Q242=6	;NUMBER OF COLUMNS
Q243=4	;NUMBER OF LINES
Q224=+15	;ANGLE OF ROTATION
Q200=2	;SET-UP CLEARANCE
Q203=+30	;SURFACE COORDINATE
Q204=50	;2ND SET-UP CLEARANCE
Q301=1	;MOVE TO CLEARANCE

Example: Circular hole patterns



%PATTERN G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define the tool
N40 T1 G17 S3500 *	Tool call
N50 G00 G40 G90 Z+250 M03 *	Retract the tool
N60 G200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=4 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME	
Q203=+0 ;SURFACE COORDINATE	
Q204=0 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	

N70 G220 POLAR PA	TTERN	Define cycle for circular pattern 1, CYCL 200 is called automatically,
Q216=+30	;CENTER IN 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+70	;CENTER IN 2ND AXIS	
Q244=50	;PITCH CIRCLE DIA.	
Q245=+0	;STARTING ANGLE	
Q246=+360	;STOPPING ANGLE	
Q247=+0	;STEPPING ANGLE	
Q241=10	;NR OF REPETITIONS	
Q200=2	;SET-UP CLEARANCE	
Q203=+0	;SURFACE COORDINATE	
Q204=100	;2ND SET-UP CLEARANCE	
Q301=1	;MOVE TO CLEARANCE	
Q365=1	;TYPE OF TRAVERSE	
N80 G220 POLAR PA	TTERN	Define cycle for circular pattern 2, CYCL 200 is called automatically,
Q216=+90	;CENTER IN 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+25	;CENTER IN 2ND AXIS	
Q244=70	;PITCH CIRCLE DIA.	
Q245=+90	;STARTING ANGLE	
Q246=+360	;STOPPING ANGLE	
Q247=30	;STEPPING ANGLE	
Q241=5	;NR OF REPETITIONS	
Q200=2	;SET-UP CLEARANCE	
Q203=+0	;SURFACE COORDINATE	
Q204=100	;2ND SET-UP CLEARANCE	
Q301=1	;MOVE TO CLEARANCE	
Q365=1	;TYPE OF TRAVERSE	
N90 G00 G40 Z+250) MO2 *	Retract in the tool axis, end program
N000000 %DATTEDN	671	

1

8.6 SL Cycles Group

8.6 SL Cycles Group I

Fundamentals

SL Cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **G37** CONTOUR GEOMETRY.

The memory capacity for programming an SL cycle (all contour subprograms) is limited to 48 kilobytes. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. For example, you can program up to approx. 256 line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation 642.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation **G41**.
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the starting position in the machining plane before a cycle. In the spindle axis the tool must be pre-positioned to set-up clearance.
- Each level of infeed depth is roughed-out axis-parallel or at a preset angle (angle defined in Cycle 657). In the standard setting, islands are traversed at safety clearance. In MP7420.1 you can also define that the TNC should rough-out individual pockets separately, plunging only once for each pocket.
- The TNC takes the entered finishing allowance (Cycle 657) into consideration.



With MP7420 you can determine where the tool is positioned at the end of Cycles 21 to 24.

Example: Program structure: Machining with SL Cycles

%SL G71 *

N12 G37 P01
N16 G56 P01
N17 G79 *
N18 G57 P01
N19 G79 *
N26 G59 P01
N27 G79 *
•••
N50 G00 G40 G90 Z+250 M2 *
N51 G98 L1 *
N60 G98 L0 *
N61 G98 L2 *
N62 G98 L0 *
N999999 %SL G71 *

Overview of SL Cycles, Group I

Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	37 LBL 1N
G56 PILOT DRILLING (optional)	56 (
G57 ROUGH-OUT (essential)	57
G58/G59 CONTOUR MILLING (optional) G58: In clockwise direction G59: In counterclockwise direction	58 +-
	59 ┃

CONTOUR GEOMETRY (Cycle G37)

All subprograms that are superimposed to define the contour are listed in Cycle G37 CONTOUR GEOMETRY.



Before programming, note the following:

Cycle **G37** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subroutines (subcontours) in Cycle **G37**.



Label numbers for the contour: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.

Overlapping contours: (see "Overlapping contours" on page 339)





Example: NC blocks

N54 G37 P01 1 P02 5 P03 7 P04 8 *

PILOT DRILLING (Cycle G56)

Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

Process

Same as Cycle **683** Pecking (see "Cycles for Drilling, Tapping and Thread Milling," page 230).

Application

Cycle **G56** is for PILOT DRILLING of the cutter infeed points. It accounts for the finishing allowance. The cutter infeed points also serve as starting points for roughing.



Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.

- Total hole depth 2 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- Plunging depth 3 (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
 - the plunging depth is equal to the depth
 - the plunging depth is greater than the total hole depth
- Feed rate for plunging: Traversing speed in mm/min for drilling
- Finishing allowance: Allowance in the machining plane





Example: NC blocks

N54 G56 P01 2 P02 -15 P03 5 P04 250 P05 +0.5*

ROUGH-OUT (Cycle G57)

Process

- 1 The TNC positions the tool in the working plane above the first cutting point, taking the finishing allowance into consideration.
- **2** The TNC moves the tool at the feed rate for plunging to the first plunging depth.

The contour is fully rough-milled (see figure at top right):

- **1** The tool mills the first subcontour at the programmed feed rate, taking the finishing allowance in the machining plane into consideration.
- 2 Further depths and further subcontours are milled by the TNC in the same way.
- **3** The TNC moves the tool in the spindle axis to the set-up clearance and then positions it above the first cutter infeed point in the machining plane.

Rough out pocket (see figure at center right):

- 1 After reaching the first plunging depth, the tool mills the contour at the programmed feed rate paraxially or at the entered roughing angle.
- 2 The island contours (here: C/D) are traversed at set-up clearance.
- **3** This process is repeated until the programmed milling depth is reached.



Before programming, note the following:

With MP7420.0 and MP7420.1 you define how the TNC should machine the contour (see "General User Parameters" on page 506).

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.







==

- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Milling depth 2 (incremental value): Distance between workpiece surface and bottom of pocket
- Plunging depth 3 (incremental value): Infeed per cut The milling 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 milling depth
- ▶ Feed rate for plunging: Traversing speed of the tool in mm/min during penetration
- **Finishing allowance:** Allowance in the machining plane
- Rough-out angle: Direction of the roughing-out movement The rough-out angle is referenced to the reference axis of the machining plane. Enter the angle so that the cuts can be as long as possible.
- **Feed rate:** Feed rate for milling in mm/min

CONTOUR MILLING (Cycle G58/G59)



Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

Application

Cycle G58/G59 CONTOUR MILLING serves for finishing the contour pocket.

Direction of rotation during contour milling

- In clockwise direction: G58
- In counterclockwise direction: G59



- Set-up clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- Milling depth 2 (incremental value): Distance between workpiece surface and bottom of pocket
- Plunging depth 3 (incremental value): Infeed per cut The milling 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 milling depth
- ▶ Feed rate for plunging: Traversing speed of the tool in mm/min during penetration
- **Feed rate:** Feed rate for milling in mm/min



Example: NC block

154	G57	P01 2	P02 -15	P03 5	P04 250
	P05	+0.5	P06 +30	P07 500	*



N54	G58 P05	P01 2 500*	P02	-15	P03	5	P04	250
•••								
N71	G59 P05	P01 2 500*	P02	-15	P03	5 I	P04	250

8.7 SL Cycles Group I

8.7 SL Cycles Group II

Fundamentals

SL Cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that you enter in Cycle **G37** CONTOUR GEOMETRY.

The memory capacity for programming an SL cycle (all contour subprograms) is limited. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of subcontours. For example, you can program up to approx. 1024 line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation 642.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation **G41**.
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

With MP7420 you can determine where the tool is positioned at the end of Cycles G121 to G124.

The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle ${\bf G120}$.

Example: Program structure: Machining with SL Cycles

%SL2 G71 *
N120 G37 *
N130 G120 *
N160 G121 *
N170 G79 *
N180 G122 *
N190 G79 *
N220 G123 *
N230 G79 *
N260 G124 *
N270 G79 *
N500 G00 G40 Z+250 M2 *
N510 G98 L1 *
N550 G98 L0 *
N560 G98 L2 *
N600 G98 L0 *
N99999 %SL2 G71 *



Overview of SL Cycles

Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	37 LBL 1N
G120 CONTOUR DATA (essential)	120 CONTOUR DATA
G121 PILOT DRILLING (optional)	121
G122 ROUGH-OUT (essential)	122
G123 FLOOR FINISHING (optional)	123
G124 SIDE FINISHING (optional)	124

Enhanced cycles:

Cycle	Soft key
G125 CONTOUR TRAIN	125
G127 CYLINDER SURFACE	127
G128 CYLINDER SURFACE slot milling	128

CONTOUR GEOMETRY (Cycle G37)

All subprograms that are superimposed to define the contour are listed in Cycle **G37** CONTOUR GEOMETRY.



Before programming, note the following:

Cycle **G37** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subroutines (subcontours) in Cycle ${\bf G37.}$



Label numbers for the contour: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.







Example: NC blocks

N120 G37 P01 1 P02 5 P03 7 P04 8 *

Overlapping contours

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: Overlapping pockets

The subsequent programming examples are contour subprograms that are called by Cycle **G37** CONTOUR GEOMETRY in a main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Pocket A

N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50 *
N550 G98 LO *

Subprogram 2: Pocket B

N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50 *
N590 G02 X+90 Y+50 *
N600 G98 L0 *

Area of inclusion

Both surfaces A and B are to be machined, including the overlapping area:

■ The surfaces A and B must be pockets.

The first pocket (in Cycle G37) must start outside the second pocket.

Surface A:

N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50 *
N550 G98 LO *

Surface B:

N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50 *
N590 G02 X+90 Y+50 *
N600 G98 L0 *



Area of exclusion

Surface A is to be machined without the portion overlapped by B:

Surface A must be a pocket and B an island.

■ A must start outside of B.

Surface A:

N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50 *
N550 G98 L0 *

Surface B:

N560 G98 L2 *
N570 G01 G41 X+90 Y+50 *
N580 I+65 J+50 *
N590 G02 X+90 Y+50 *
N600 G98 L0 *

Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

■ A and B must be pockets.

■ A must start inside of B.

Surface A:

N510 G98 L1 *
N520 G01 G42 X+60 Y+50 *
N530 I+35 J+50 *
N540 G02 X+60 Y+50 *
N550 G98 LO *

Surface B:

N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50 *
N590 G02 X+90 Y+50 *
N600 G98 L0 *





CONTOUR DATA (Cycle G120)

Machining data for the subprograms describing the subcontours are entered in Cycle G120.



Before programming, note the following:

Cycle G120 is DEF active which means that Cycle G120 becomes effective as soon as it is defined in the part program.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the TNC does not execute that next cycle.

The machining data entered in Cycle G120 are valid for Cycles G121 to G124.

If you are using the SL Cycles in Q parameter programs, the Cycle Parameters Q1 to Q19 cannot be used as program parameters.



- ▶ Milling depth Q1 (incremental value): Distance between workpiece surface and bottom of pocket.
- **Path overlap** factor Q2: Q2 x tool radius = stepover factor k.
- ▶ Finishing allowance for side Q3 (incremental value): Finishing allowance in the working plane
- Finishing allowance for floor Q4 (incremental) value): Finishing allowance in the tool axis.
- ▶ Workpiece surface coordinate Q5 (absolute value): Absolute coordinate of the workpiece surface
- **Set-up clearance** Q6 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Clearance height Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle).
- ▶ Inside corner radius Q8: Inside "corner" rounding radius; entered value is referenced to the tool midpoint path.
- ▶ Direction of rotation ? Clockwise = -1 Q9: Machining direction for pockets.
 - Clockwise (Q9 = -1 up-cut milling for pocket and island)
 - Counterclockwise (Q9 = +1 climb milling for pocket and island)

You can check the machining parameters during a program interruption and overwrite them if required.





Example: NC block

N57 G120 CONTOUR	DATA
Q1=-20	;MILLING DEPTH
Q2=1	;TOOL PATH OVERLAP
Q3=+0.2	;ALLOWANCE FOR SIDE
Q4=+0.1	;ALLOWANCE FOR FLOOR
Q5=+30	;SURFACE COORDINATE
Q6=2	;SET-UP CLEARANCE
Q7=+80	;CLEARANCE HEIGHT
Q8=0.5	;ROUNDING RADIUS
Q9=+1	;DIRECTION OF ROTATION

PILOT DRILLING (Cycle G121)



When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **T** block.

In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

Process

Same as Cycle **G83** Pecking (see "Cycles for Drilling, Tapping and Thread Milling," page 230).

Application

Cycle **G121** is for PILOT DRILLING of the cutter infeed points. It accounts for the allowance for side and the allowance for floor as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for roughing.



Plunging depth Q10 (incremental value): Dimension by which the tool drills in each infeed (negative sign for negative working direction).

- ▶ Feed rate for plunging Q11: Traversing speed in mm/min during drilling.
- Rough-out tool number Q13: Tool number of the roughing mill.



N58 G121 PILOT	DRILLING
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q13=1	;ROUGH-OUT TOOL

ROUGH-OUT (Cycle G122)

- 1 The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
- 2 In the first plunging depth, the tool mills the contour from the inside outward at the milling feed rate Q12.
- **3** The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B).
- **4** Then the TNC rough-mills the pocket contour and retracts the tool to the clearance height.



Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle **G121.**

If you define a plunge angle in the ANGLE column of the tool table for the roughing tool, the TNC moves on a helical path to the respective roughing depth (see "Tool table: Standard tool data" on page 139).

- 122
- ▶ Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool in mm/min during penetration.
- Feed rate for milling Q12: Traversing speed for milling in mm/min.
- Coarse roughing tool number Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If there was no coarse roughing, enter "0"; if you enter a value other than zero, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool.

If the portion that is to be roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut; For this purpose you must enter the tool length LCUTS in the tool table TOOL.T (see "Tool Data," page 137) and define the maximum plunging ANGLE of the tool. The TNC will otherwise generate an error message.

Reciprocation feed rate Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut.



N59 G122 ROUGH-OUT	
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q18=1	;COARSE ROUGHING TOOL
Q19=150	;RECIPROCATION FEED RATE

FLOOR FINISHING (Cycle G123)



The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.

The tool approaches the machining plane smoothly (in a vertically tangential arc). The tool then clears the finishing allowance remaining from rough-out.



► Feed rate for plunging Q11: Traversing speed of the tool during penetration.

► Feed rate for milling Q12: Traversing speed for milling.



N60	G123	FLOOR	FINISHING	i			
	Q11=	100	;FEED	RATE	FOR	PLNGNG	
	Q12=	350	;FEED	RATE	FOR	MILLING	



SIDE FINISHING (Cycle G124)

The subcontours are approached and departed on a tangential arc. Each subcontour is finish-milled separately.



Before programming, note the following:

The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle **G120**) and the radius of the rough mill.

This calculation also holds if you run Cycle **G124** without having roughed out with Cycle **G122**; in this case, enter "0" for the radius of the rough mill.

The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.



- Direction of rotation ? Clockwise = -1 Q9: Machining direction:
 - **+1**: Counterclockwise
 - -1: Clockwise
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- ► Feed rate for milling Q12: Traversing speed for milling.
- Finishing allowance for side Q14 (incremental value): Enter the allowed material for several finishmilling operations. If you enter Q14 = 0, the remaining finishing allowance will be cleared.



N61 G124 SIDE	FINISHING
Q9=+1	;DIRECTION OF ROTATION
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q14=+0	;ALLOWANCE FOR SIDE

CONTOUR TRAIN (Cycle G125)

In conjunction with Cycle **G37** CONTOUR GEOMETRY, this cycle facilitates the machining of open contours (i.e. where the starting point of the contour is not the same as its end point).

Cycle **G125** CONTOUR TRAIN offers considerable advantages over machining an open contour using positioning blocks:

- The TNC monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked.
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are mirrored.
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining.
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

Before programming, note the following:

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The TNC takes only the first label of Cycle **G37** CONTOUR GEOMETRY into account.

The memory capacity for programming an SL cycle is limited. For example, you can program up to 1024 straightline blocks in one SL cycle.

Cycle G120 CONTOUR DATA is not required.

Positions that are programmed in incremental dimensions immediately after Cycle **G125** are referenced to the position of the tool at the end of the cycle.



Danger of collision!

To avoid collisions,

- Do not program positions in incremental dimensions immediately after Cycle G125, since they are referenced to the position of the tool at the end of the cycle.
- Move the tool to defined (absolute) positions in all main axes, since the position of the tool at the end of the cycle is not identical to the position of the tool at the start of the cycle.





8.7 SL Cycles Group II

125

- Milling depth Q1 (incremental value): Distance between workpiece surface and contour floor.
- ▶ Finishing allowance for side Q3 (incremental value): Finishing allowance in the working plane.
- Workpiece surface coordinate Q5 (absolute value): Absolute coordinate of the workpiece surface referenced to the workpiece datum.
- Clearance height Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle.
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- Feed rate for plunging Q11: Traversing speed of the tool in the tool axis.
- ▶ Feed rate for milling Q12: Traversing speed of the tool in the working plane.
- Climb or up-cut ? Up-cut = -1 Q15: Climb milling: Input value = +1 Up-cut milling: Input value = -1 To enable climb milling and up-cut milling alternately in several infeeds:Input value = 0

Example: NC block

N62 G125 CONTOUR	TRAIN
Q1=-20	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q5=+0	;SURFACE COORDINATE
Q7=+50	;CLEARANCE HEIGHT
Q10=+5	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q15=-1	;CLIMB OR UP-CUT

CYLINDER SURFACE (Cycle G127, software option 1)

P

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle **G128** if you wish to mill guide notches onto the cylinder surface.

The contour is described in a subprogram identified in Cycle **G37** CONTOUR GEOMETRY.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, is parallel to the Z axis. The available path functions are G1, G11, G24, G25 and G2/G3/G12/G13 with R.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition.

- **1** The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
- **2** At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.
- **3** At the end of the contour, the TNC returns the tool to the setup clearance and returns to the point of penetration.
- 4 Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.
- **5** Then the tool moves to the set-up clearance.





Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 1024 straight-line blocks in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

The TNC checks whether the compensated and noncompensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message "Contour programming error" is output, set MP 810.x = 0.



Milling depth Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.

- ▶ Finishing allowance for side Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.
- Set-up clearance Q6 (incremental value): Distance between the tool tip and the cylinder surface.
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- Feed rate for plunging Q11: Traversing speed of the tool in the tool axis.
- Feed rate for milling Q12: Traversing speed of the tool in the working plane.
- ▶ Cylinder radius Q16: Radius of the cylinder on which the contour is to be machined.
- Dimension type ? ang./lin. Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).

N63 G127 CYLINDER	SURFACE
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q16=25	;RADIUS
Q17=0	;DIMENSION TYPE

8.7 SL Cycles Group

CYLINDER SURFACE slot milling (Cycle G128, software option 1)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

This cycle enables you to program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle G127, with this cycle the TNC adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the midpoint path of the contour together with the tool radius compensation. With the radius compensation you specify whether the TNC cuts the slot with climb milling or up-cut milling:

- **1** The TNC positions the tool over the cutter infeed point.
- 2 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate Q12 while respecting the finishing allowance for the side.
- 3 At the end of the contour, the TNC moves the tool to the opposite wall and returns to the infeed point.
- 4 Steps 2 and 3 are repeated until the programmed milling depth Q1 is reached.
- **5** Then the tool moves to the set-up clearance.



Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 1024 straightline blocks in one SL cycle.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

The TNC checks whether the compensated and noncompensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message "Contour programming error" is output, set MP 810.x = 0.





128 |

- Milling depth Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour.
- ▶ Finishing allowance for side Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.
- ▶ Set-up clearance Q6 (incremental value): Distance between the tool tip and the cylinder surface.
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- Feed rate for plunging Q11: Traversing speed of the tool in the tool axis.
- Feed rate for milling Q12: Traversing speed of the tool in the working plane.
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined.
- Dimension type ? ang./lin. Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1).
- Slot width Q20: Width of the slot to be machined.

N63 G128 CYLINDER	SURFACE
Q1=-8	;MILLING DEPTH
Q3=+0	;ALLOWANCE FOR SIDE
Q6=+0	;SET-UP CLEARANCE
Q10=+3	;PLUNGING DEPTH
Q11=100	;FEED RATE FOR PLNGNG
Q12=350	;FEED RATE FOR MILLING
Q16=25	;RADIUS
Q17=0	;DIMENSION TYPE
Q20=12	;SLOT WIDTH

Example: Pilot drilling, roughing-out and finishing overlapping contours



%C21 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Define tool: drill
N40 G99 T2 L+0 R+6 *	Define the tool for roughing/finishing
N50 T1 G17 S4000 *	Call toll: drill
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 P02 2 P03 3 P04 4 *	Define contour subprogram
N80 G120 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q4=+0 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION OF ROTATION	



N90 G121 PILOT DR	RILLING	Cycle definition: Pilot drilling
Q10=5	;PLUNGING DEPTH	
Q11=250	;FEED RATE FOR PLNGNG	
Q13=0	;ROUGH-OUT TOOL	
N100 G79 M3 *		Cycle call: Pilot drilling
N110 Z+250 M6 *		Tool change
N120 T2 G17 S3000) *	Call the tool for roughing/finishing
N130 G122 ROUGH-0)UT	Cycle definition: Coarse roughing
Q10=5	;PLUNGING DEPTH	
Q11=100	;FEED RATE FOR PLNGNG	
Q12=350	;FEED RATE FOR MILLING	
Q18=0	;COARSE ROUGHING TOOL	
Q19=150	;RECIPROCATION FEED RATE	
N140 G79 M3 *		Cycle call: Rough-out
N150 G123 FLOOR F	INISHING	Cycle definition: Floor finishing
Q11=100	;FEED RATE FOR PLNGNG	
Q12=200	;FEED RATE FOR MILLING	
N160 G79 *		Cycle call: Floor finishing
N170 G124 SIDE FI	INISHING	Cycle definition: Side finishing
Q9=+1	;DIRECTION OF ROTATION	
Q10=-5	;PLUNGING DEPTH	
Q11=100	;FEED RATE FOR PLNGNG	
Q12=400	;FEED RATE FOR MILLING	
Q14=0	;ALLOWANCE FOR SIDE	
N180 G79 *		Cycle call: Side finishing
N190 G00 Z+250 M2	*	Retract in the tool axis, end program

-
0
_
4 -
C T
$\mathbf{\nabla}$
-
C D
– – – – – – – – – –
•
A N
U
~
6.7
\smile
-
10
UJ.
\sim
ω

N200 G98 L1 *	Contour subprogram 1: left pocket
N210 I+35 J+50 *	
N220 G01 G42 X+10 Y+50 *	
N230 G02 X+10 *	
N240 G98 L0 *	
N250 G98 L2 *	Contour subprogram 2: right pocket
N260 I+65 J+50 *	
N270 G01 G42 X+90 Y+50 *	
N280 G02 X+90 *	
N290 G98 LO *	
N300 G98 L3 *	Contour subprogram 3: square left island
N310 G01 G41 X+27 Y+50 *	
N320 Y+58 *	
N330 X+43 *	
N340 Y+42 *	
N350 X+27 *	
N360 G98 L0 *	
N370 G98 L4 *	Contour subprogram 4: triangular right island
N380 G01 G41 X+65 Y+42 *	
N390 X+57 *	
N400 X+65 Y+58 *	
N410 X+73 Y+42 *	
N420 G98 L0 *	
N999999 %C21 G71 *	



Example: Contour train



%C25 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define the tool
N50 T1 G17 S2000 *	Tool call
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 *	Define contour subprogram
N80 G125 CONTOUR TRAIN	Define machining parameters
Q1=-20 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q5=+0 ;SURFACE COORDINATE	
Q7=+250 ;CLEARANCE HEIGHT	
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR MILLING	
Q15=+1 ;CLIMB OR UP-CUT	
N90 G79 M3 *	Call the cycle
N100 G00 G90 Z+250 M2 *	Retract in the tool axis, end program

N110 G98 L1 *	Contour subprogram	
N120 G01 G41 X+0 Y+15 *		
N130 X+5 Y+20 *		Ş
N140 G06 X+5 Y+75 *		Ū
N150 G01 Y+95 *		U
N160 G25 R7.5 *		
N170 X+50 *		
N180 G25 R7.5 *		C
N190 X+100 Y+80 *		Ū
N200 G98 L0 *		
N999999 %C25 G71 *		Q



Example: Cylinder surface with Cycle G127

Note:

Cylinder centered on rotary table

Datum at center of rotary table



%C27 G71 *	
N10 G99 T1 L+0 R+3.5 *	Define the tool
N20 T1 G18 S2000 *	Call tool, tool axis is Y
N30 G00 G40 G90 Y+250 *	Retract the tool
N40 G37 P01 1 *	Define contour subprogram
N70 G127 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE FOR MILLING	
Q16=25 ;RADIUS	
Q17=1 ;DIMENSION TYPE	
N60 C+0 M3 *	Pre-position rotary table
N70 G79 *	Call the cycle
N80 G00 G90 Y+250 M2 *	Retract in the tool axis, end program

N90 G98 L1 *	Contour subprogram
N100 G01 G41 C+91.72 Z+20 *	Data for the rotary axis entered in degrees
N110 C+114.65 Z+20 *	Drawing dimensions are converted from mm to degrees (157 mm = 360°)
N120 G25 R7.5 *	
N130 G91 Z+40 *	
N140 G90 G25 R7.5 *	
N150 G91 C-45.86 *	
N160 G90 G25 R7.5 *	
N170 Z+20 *	
N180 G25 R7.5 *	
N190 C+91.72 *	
N200 G98 L0 *	
N999999 %C27 G71 *	



8.7 SL Cycles Group II

Example: Cylinder surface with Cycle G128

Notes:

- Cylinder centered on rotary table
- Datum at center of rotary table
- Description of the midpoint path in the contour subprogram



%C28 G71 *	
N10 G99 T1 L+0 R+3.5 *	Define the tool
N20 T1 G18 S2000 *	Call tool, tool axis is Y
N30 G00 G40 G90 Y+250 *	Retract the tool
N40 G37 P01 1 *	Define contour subprogram
N50 X+0 *	Position tool on rotary table center
N60 G128 CYLINDER SURFACE	Define machining parameters
Q1=-7 ;MILLING DEPTH	
Q3=+0 ;ALLOWANCE FOR SIDE	
Q6=2 ;SET-UP CLEARANCE	
Q10=-4 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=250 ;FEED RATE FOR MILLING	
Q16=25 ;RADIUS	
Q17=1 ;DIMENSION TYPE	
Q2O=10 ;SLOT WIDTH	
N70 C+0 M3 *	Pre-position rotary table
N80 G79 *	Call the cycle
N90 G00 G40 Y+250 M2 *	Retract in the tool axis, end program
N100 G98 L1 *	Contour subprogram, description of the midpoint path
-------------------------	--
N100 G01 G41 C+40 Z+0 *	Data for the rotary axis are entered in mm (Q17=1)
N110 Z+35 *	
N120 C+60 Z+52.5 *	
N130 Z+70 *	
N140 G98 L0 *	
N999999 %C28 G71 *	



8.8 SL Cycles with Contour Formula

Fundamentals

SL Cycles and the contour formula enable you to form complex contours by combining subcontours (pockets or islands). You define the individual subcontours (geometry data) as separate programs. In this way, any subcontour can be used any number of times. The TNC calculates the complete contour from the selected subcontours, which you link together through a contour formula.

Which

The memory capacity for programming an SL cycle (all contour description programs) is limited to 32 contours. The number of possible contour elements depends on the type of contour (inside or outside contour) and the number of contour descriptions. For example, you can program up to approx. 1024 line blocks.

The SL Cycles with contour formula presuppose a structured program layout and enable you to save frequently used contours in individual programs. Using the contour formula, you can connect the subcontours to a complete contour and define whether it applies to a pocket or island.

In its present form, the "SL Cycles with contour formula" function requires input from several areas in the TNC's user interface. This function is to serve as a basis for further development.

Properties of the subcontours

- By default, the TNC assumes that the contour is a pocket. Do not program a radius compensation. In the contour formula you can convert a pocket to an island by making it negative.
- The TNC ignores feed rates F and miscellaneous functions M.
- Coordinate transformations are allowed. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- Although the subprograms can contain coordinates in the spindle axis, such coordinates are ignored.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the Rough-out and Side Finishing cycles).
- The contour is approached in a tangential arc for side finishing.

Example: Program structure: Machining with SL Cycles and contour formula

%CONTOUR G71
N50 %:CNT: "MODEL"
N60 G120 Q1=
N70 G122 Q10=
N80 G79
N120 G123 Q11=
N130 G79
N160 G124 Q9=
N170 G79
N180 G00 G40 G90 Z+250 M2
N99999999 %CONTOUR G71

Example: Program structure: Calculation of the subcontours with contour formula

%MODEL G71
N10 DECLARE CONTOUR QC1 = "ARC1"
N20 DECLARE CONTOUR QC2 = "ARC31XY"
N30 DECLARE CONTOUR QC3 = "TRIANGLE"
N40 DECLARE CONTOUR QC4 = "SQUARE"
N50 QC10 = (QC1 QC3 QC4) \ QC2
N99999999 %MODEL G71
%ARC1 G71
N10 I+75 J+50
N20 G11 R+45 H+0 G40
N30 G13 G91 H+360
N99999999 %ARC1 G71
%ARC31XY G71

- For floor finishing, the tool again approaches the workpiece on a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.



With MP7420 you can determine where the tool is positioned at the end of Cycles G121 to G124.

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle G120.

Selecting a program with contour definitions

With the **%: CNT** function you select a program with contour definitions, from which the TNC takes the contour descriptions:

- PGM CALL
- ▶ To select the functions for program call, press the PGM CALL key.



G

- Press the SELECT CONTOUR soft key.
- Enter the full name of the program with the contour definition and confirm with the END key.

Program a %:CNT block before the SL Cycles. Cycle 14 CONTOUR GEOMETRY is no longer necessary if you use %:CNT.

Defining contour descriptions

With the **DECLARE CONTOUR** function you enter in a program the path for programs from which the TNC draws the contour descriptions:



- Press the DECLARE soft key.
- Press the CONTOUR soft key.
- Enter the number for the contour designator QC, and confirm with the ENT key.
- Enter the full name of the program with the contour description and confirm with the END key.



With the given contour designators QC you can include the various contours in the contour formula.

With the **DECLARE STRING** function you define a text. For the time being, this function is not evaluated.



Entering a contour formula

You can use soft keys to interlink various contours in a mathematical formula.

- Select a Q parameter function: Press the Q key (in the numerical keypad at right). The Q parameter functions are displayed in a softkey row.
- To select the function for entering the contour formula, press the CONTOUR FORMULA soft key. The TNC then shows the following soft keys:

Logic command	Soft key
Intersected with e.g. QC10 = QC1 & QC5	
Joined with e.g. QC25 = QC7 QC18	
Joined without intersection e.g. QC12 = QC5 ^ QC25	
Joined with complement of e.g. QC25 = QC1 \ QC2	
Complement of contour area e.g. Q12 = #Q11	
Opening parenthesis e.g. QC12 = QC1 * (QC2 + QC3)	C
Closing parenthesis e.g. QC12 = QC1 * (QC2 + QC3)	>

Overlapping contours

By default, the TNC considers a programmed contour to be a pocket. With the functions of the contour formula, you can convert a contour from a pocket to an island.

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: Overlapping pockets

The subsequent programming examples are contour description programs that are defined in a contour definition program. The contour definition program is called through the **%: CNT** function in the actual main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.



Contour description program 1: Pocket A

%POCKET_A G71
N10 G01 X+10 Y+50 G40
N20 I+35 J+50
N30 G02 X+10 Y+50
N999999999 %POCKET A G71

Contour description program 2: Pocket B

%POCKET_B G71
N10 G01 X+90 Y+50 G40
N20 I+65 J+50
N30 G02 X+90 Y+50
N999999999 %POCKET_B G71

Area of inclusion

Both surfaces A and B are to be machined, including the overlapping area:

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "joined with" function.

Contour definition program:

N50
N60
N70 DECLARE CONTOUR QC1 = "POCKET_A.H"
N80 DECLARE CONTOUR QC2 = "POCKET_B.H"
N90 QC10 = QC1 QC2
N100
N110



Area of exclusion

Surface A is to be machined without the portion overlapped by B:

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surface B is subtracted from the surface A with the "joined with complement of" function.

Contour definition program:

N50
N60
N70 DECLARE CONTOUR QC1 = "POCKET_A.H"
N80 DECLARE CONTOUR QC2 = "POCKET_B.H"
N90 QC10 = QC1 \ QC2
N100
N110



Area of intersection

Only the area where A and B overlap is to be machined. (The areas covered by A or B alone are to be left unmachined.)

- The surfaces A and B must be programmed in separate programs without radius compensation.
- In the contour formula, the surfaces A and B are processed with the "intersection with" function.

Contour definition program:

N50
N60
N70 DECLARE CONTOUR QC1 = "POCKET_A.H"
N80 DECLARE CONTOUR QC2 = "POCKET_B.H"
N90 QC10 = QC1 & QC2
N100
N110

Contour machining with SL Cycles

The complete contour is machined with the SL Cycles G120 to G124 (see "SL Cycles Group II" on page 337).



Example: Roughing and finishing superimposed contours with the contour formula



%C21 G	71 *		
N10 G30 G17 X+0 Y+0 Z-40 *		0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *		Y+100 Z+0 *	
N30 G99	9 T1 L+0 R+2	.5 *	Tool definition of roughing cutter
N40 G99 T2 L+0 R+3 *		*	Tool definition of finishing cutter
N50 T1 G17 S2500 *			Tool call of roughing cutter
N60 G00 G40 G90 Z+250 *		250 *	Retract the tool
N70 %:CNT: "MODEL" *		*	Specify contour definition program
N80 G120 CONTOUR DATA		ATA	Define general machining parameters
Q1	1=-20	;MILLING DEPTH	
Q2	2=1	;TOOL PATH OVERLAP	
Q3	3=+0.5	;ALLOWANCE FOR SIDE	
Q4	4=+0.5	;ALLOWANCE FOR FLOOR	
Q 5	5=+0	;SURFACE COORDINATE	
Q 6	6=2	;SET-UP CLEARANCE	
Q7	7=+100	;CLEARANCE HEIGHT	
Q8	8=0.1	;ROUNDING RADIUS	
Q 9	9=-1	;DIRECTION OF ROTATION	

N90 G122 ROUGH-OUT	Cycle definition: Rough-out
Q10=5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=350 ;FEED RATE FOR MILLING	
Q18=0 ;COARSE ROUGHING TOOL	
Q19=150 ;RECIPROCATION FEED RATE	
N100 G79 M3 *	Cycle call: Rough-out
N110 T2 G17 S5000 *	Tool call of finishing cutter
N150 G123 FLOOR FINISHING	Cycle definition: Floor finishing
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=200 ;FEED RATE FOR MILLING	
N160 G79 *	Cycle call: Floor finishing
N170 G124 SIDE FINISHING	Cycle definition: Side finishing
Q9=+1 ;DIRECTION OF ROTATION	
Q10=-5 ;PLUNGING DEPTH	
Q11=100 ;FEED RATE FOR PLNGNG	
Q12=400 ;FEED RATE FOR MILLING	
Q14=0 ;ALLOWANCE FOR SIDE	
N180 G79 *	Cycle call: Side finishing
N190 G00 Z+250 M2 *	Retract in the tool axis, end program
N999999 %C21 G71 *	

Contour definition program with contour formula:

%MODEL G71 *	Contour definition program
N10 DECLARE CONTOUR QC1 = "ARC1" *	Definition of the contour designator for the program "CIRCLE1"
N20 D00 Q1 P01 +35 *	Assignment of values for parameters used in PGM "CIRCLE31XY"
N30 D00 Q2 P01 +50 *	
N40 D00 Q3 P01 +25 *	
N50 DECLARE CONTOUR QC2 = "ARC31XY" *	Definition of the contour designator for the program "CIRCLE31XY"
N60 DECLARE CONTOUR QC3 = "TRIANGLE" *	Definition of the contour designator for the program "TRIANGLE"
N70 DECLARE CONTOUR QC4 = "SQUARE" *	Definition of the contour designator for the program "SQUARE"
N80 QC10 = (QC 1 QC 2) \ QC 3 \ QC 4 *	Contour formula
N99999999 %MODEL G71 *	

Contour description programs:

%ARC1 G71 *	Contour description program: circle at right
N10 I+65 J+50 *	
N20 G11 R+25 H+0 G40 *	
N30 CP IPA+360 DR+ *	
N99999999 %ARC1 G71 *	

%ARC31XY G71 *	Contour description program: circle at left
N10 I+Q1 J+Q2 *	
N20 G11 R+Q3 H+O G40 *	
N30 G13 G91H+360 *	
N99999999 %ARC31XY G71 *	

%TRIANGLE G71 *	Contour description program: triangle at right
N10 G01 X+73 Y+42 G40 *	
N20 G01 X+65 Y+58 *	
N30 G01 X+42 Y+42 *	
N40 G01 X+73 *	
N99999999 %TRIANGLE G71 *	

%SQUARE G71 *	Contour description program: square at left
N10 G01 X+27 Y+58 G40 *	
N20 G01 X+43 *	
N30 G01 Y+42 *	
N40 G01 X+27 *	
N50 G01 Y+58 *	
N9999999%SQUARE G71 *	



8.9 Cycles for Multipass Milling

Overview

The TNC offers three cycles for machining the following surface types:

- Created with a CAD/CAM system
- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

Cycle	Soft key
G60 RUN 3-D DATA For multipass milling of 3-D data in several infeeds	80 MILLING PNT FILE
G230 MULTIPASS MILLING For flat rectangular surfaces	230
G231 RULED SURFACE For oblique, inclined or twisted surfaces	231

8.9 Cycles for Multipass Milling

RUN 3-D DATA (Cycle G60)

- **1** From the current position, the TNC positions the tool in rapid traverse in the tool axis to the set-up clearance above the MAX point that you have programmed in the cycle.
- **2** The tool then moves in rapid traverse in the working plane to the MIN point you have programmed in the cycle.
- **3** From this point, the tool advances to the first contour point at the feed rate for plunging.
- 4 The TNC subsequently processes all points that are stored in the 3-D data file at the feed rate for milling. If necessary, the TNC retracts the tool between machining operations to set-up clearance if specific areas are to be left unmachined.
- **5** At the end of the cycle, the tool is retracted in rapid traverse to setup clearance.



Before programming, note the following:

Cycle G60 allows you to run 3-D data using several infeeds which have been created with an off-line programming system.

- 60 MILLING PNT FILE
- PGM Name 3-D data: Enter the name of the file in which the data is stored. If the file is not stored in the current directory, enter the complete path.
- Min. point of range: Lowest coordinates (X, Y and Z coordinates) in the range to be milled.
- Max. point of range: Largest coordinates (X, Y and Z coordinates) in the range to be milled.
- Set-up clearance 1 (incremental value): Distance between tool tip and workpiece surface for tool movements in rapid traverse.
- ▶ Plunging depth 2 (incremental value): Infeed per cut.
- **Feed rate for plunging 3**: Traversing speed of the tool in mm/min during penetration.
- **Feed rate for milling 4**: Traversing speed of the tool in mm/min while milling.
- Miscellaneous function M: Optional entry of a miscellaneous function, for example M13.





Example: NC block

N64	G60	P01 BSP.I P01 X+0 P02 Y+0
	P03	Z-20 P04 X+100 P05 Y+100 P06 Z+0
	P07	2 P08 +5 P09 100 P10 350 M13 *

MULTIPLASS MILLING (Cycle G230)

- 1 From the current position in the working plane, the TNC positions the tool at rapid traverse to the starting point 1; the TNC moves the tool by its radius to the left and upward.
- **2** The tool then moves in rapid traverse in the tool axis to set-up clearance. From there it approaches the programmed starting position in the tool axis at the feed rate for plunging.
- **3** The tool then moves at the programmed feed rate for milling to the end point **2**. The TNC calculates the end point from the programmed starting point, the program length, and the tool radius.
- **4** The TNC offsets the tool to the starting point in the next pass at the stepover feed rate. The offset is calculated from the programmed width and the number of cuts.
- **5** The tool then returns in the negative direction of the first axis.
- **6** Multipass milling is repeated until the programmed surface has been completed.
- 7 At the end of the cycle the tool is retracted in rapid traverse to setup clearance.



Before programming, note the following:

From the current position, the TNC positions the tool at the starting point 1, first in the working plane and then in the tool axis.

Pre-position the tool in such a way that no collision between tool and clamping devices can occur.





- Starting point in 1st axis Q225 (absolute value): Minimum point coordinate of the surface to be multipass-milled in the reference axis of the working plane.
- Starting point in 2nd axis Q226 (absolute value): Minimum-point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- Starting point in 3rd axis Q227 (absolute value): Height in the spindle axis at which multipass-milling is carried out.
- First side length Q218 (incremental value): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in the 1st axis.
- Second side length Q219 (incremental value): Length of the surface to be multipass-milled in the minor axis of the working plane, referenced to the starting point in the 2nd axis.
- ▶ Number of cuts Q240: Number of passes to be made over the width.
- Feed rate for plunging 206: Traversing speed of the tool in mm/min while penetrating from the set-up clearance to the milling depth.
- ► Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Stepover feed rate Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207. If you are moving it transversely in the open, Q209 may be greater than Q207.
- Set-up clearance Q200 (incremental value): Distance between tool tip and milling depth for positioning at the start and end of the cycle.





Example: NC block

N71 G230 MULTIPASS	5 MILLING
Q225=+10	;STARTING PNT 1ST AXIS
Q226=+12	;STARTING PNT 2ND AXIS
Q227=+2.5	;STARTING PNT 3RD AXIS
Q218=150	;FIRST SIDE LENGTH
Q219=75	;SECOND SIDE LENGTH
Q240=25	;NUMBER OF CUTS
Q206=150	;FEED RATE FOR PLNGNG
Q207=500	;FEED RATE FOR MILLING
Q209=200	;STEPOVER FEED RATE
Q200=2	;SET-UP CLEARANCE

RULED SURFACE (Cycle G231)

- 1 From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1.
- 2 The tool subsequently advances to the stopping point 2 at the feed rate for milling.
- **3** From this point, the tool moves at rapid traverse by the tool diameter in the positive tool axis direction, and then back to starting point **1**.
- **4** At the starting point **1** the TNC moves the tool back to the last traversed Z value.
- **5** Then the TNC moves the tool in all three axes from point **1** in the direction of point **4** to the next line.
- 6 From this point, the tool moves to the stopping point on this pass. The TNC calculates the end point from point 2 and a movement in the direction of point 3.
- 7 Multipass milling is repeated until the programmed surface has been completed.
- 8 At the end of the cycle, the tool is positioned above the highest programmed point in the tool axis, offset by the tool diameter.

Cutting motion

The starting point, and therefore the milling direction, is selectable because the TNC always moves from point 1 to point 2 and in the total movement from point 1/2 to point 3/4. You can program point 1 at any corner of the surface to be machined.

If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

- A shaping cut (spindle axis coordinate of point 1 greater than spindle-axis coordinate of point 2) for slightly inclined surfaces.
- A drawing cut (spindle axis coordinate of point 1 smaller than spindle-axis coordinate of point 2) for steep surfaces.
- When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) parallel to the direction of the steeper inclination.

If you are using a spherical cutter for the machining operation, you can optimize the surface finish in the following way:

When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) perpendicular to the direction of the steepest inclination.

Before programming, note the following:

From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

The TNC moves the tool with radius compensation **G40** to the programmed positions.

If required, use a center-cut end mill (ISO 1641).









- Starting point in 1st axis Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the reference axis of the working plane.
- Starting point in 2nd axis Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane.
- Starting point in 3rd axis Q227 (absolute value): Starting point coordinate of the surface to be multipass-milled in the tool axis.
- 2nd point in 1st axis Q228 (absolute value): Stopping point coordinate of the surface to be multipass milled in the reference axis of the working plane.
- 2nd point in 2nd axis Q229 (absolute value): Stopping point coordinate of the surface to be multipass milled in the minor axis of the working plane.
- 2nd point in 3rd axis Q230 (absolute value): Stopping point coordinate of the surface to be multipass milled in the tool axis.
- 3rd point in 1st axis Q231 (absolute value): Coordinate of point 3 in the reference axis of the working plane.
- 3rd point in 2nd axis Q232 (absolute value): Coordinate of point 3 in the minor axis of the working plane.
- **3rd point in 3rd axis** Q233 (absolute value): Coordinate of point **3** in the tool axis





(

- 4th point in 1st axis Q234 (absolute value): Coordinate of point 4 in the reference axis of the working plane.
- 4th point in 2nd axis Q235 (absolute value): Coordinate of point 4 in the minor axis of the working plane.
- ▶ **4th point in 3rd axis** Q236 (absolute value): Coordinate of point **4** in the tool axis.
- Number of cuts Q240: Number of passes to be made between points 1 and 4, 2 and 3.
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling. The TNC performs the first step at half the programmed feed rate.

Example: NC blocks

N72 G231 RULED	SURFACE
Q225=+0	;STARTING PNT 1ST AXIS
Q226=+5	;STARTING PNT 2ND AXIS
Q227=-2	;STARTING PNT 3RD AXIS
Q228=+100	;2ND POINT IN 1ST AXIS
Q229=+15	;2ND POINT IN 2ND AXIS
Q230=+5	;2ND POINT IN 3RD AXIS
Q231=+15	;3RD POINT IN 1ST AXIS
Q232=+125	;3RD POINT IN 2ND AXIS
Q233=+25	;3RD POINT IN 3RD AXIS
Q234=+15	;4TH POINT IN 1ST AXIS
Q235=+125	;4TH POINT IN 2ND AXIS
Q236=+25	;4TH POINT IN 3RD AXIS
Q240=40	;NUMBER OF CUTS
Q207=500	;FEED RATE FOR MILLING

1



%C230 G71	
N10 G30 G17 X+0 Y+0 Z+0 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+40 *	
N30 G99 T1 L+0 R+5 *	Define the tool
N40 T1 G17 S3500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G230 MULTIPASS MILLING	Cycle definition: MULTIPASS MILLING
N60 G230 MULTIPASS MILLING	Cycle definition: MULTIPASS MILLING
Q225=+0 ;STARTING PNT 1ST AXIS	
Q226=+0 ;STARTING PNT 2ND AXIS	
Q227=+35 ;STARTING PNT 3RD AXIS	
Q218=100 ;FIRST SIDE LENGTH	
Q219=100 ;SECOND SIDE LENGTH	
Q240=25 ;NUMBER OF CUTS	
Q206=250 ;FEED RATE FOR PLNGNG	
Q207=400 ;FEED RATE FOR MILLING	
Q209=150 ;STEPOVER FEED RATE	
Q200=2 ;SET-UP CLEARANCE	

N70 X-25 Y+0 M03 *	Pre-position near the starting point
N80 G79 *	Call the cycle
N90 G00 G40 Z+250 M02 *	Retract in the tool axis, end program
N999999 %C230 G71 *	

1

8.10 Coordinate Transformation Cycles

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
G53/G54 DATUM SHIFT For shifting contours directly within the program or from datum tables	53 54 54
G247 DATUM SETTING Datum setting during program run	247
G28 MIRROR IMAGE Mirroring contours	28
G73 ROTATION For rotating contours in the working plane	73
G72 SCALING FACTOR For increasing or reducing the size of contours	72
G80 WORKING PLANE For executing machining operations in a tilted coordinate system for machines with tilting heads and/or rotary tables	80

Effect of coordinate transformations

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called. 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 M02, M30, or an N999999 %... block (depending on MP7300).
- Select a new program.
- Program miscellaneous function M142 Erasing modal program information.



DATUM SHIFT (Cycle G54)

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.

Effect

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.



▶ 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.

Cancellation

A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Graphics

If you program a new workpiece blank after a datum shift, you can use Machine Parameter 7310 to determine whether the blank is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.

Status displays

- The actual position values are referenced to the active (shifted) datum.
- All of the position values shown in the additional status display are referenced to the manually set datum.





Example: NC blocks

N7 2	G 54	G90	X+25	Y-1	2.5	Z+1(00 *		
•••									
N78	G 54	G90	REF	X+25	Y- 1	12.5	Z+100	*	

1

DATUM SHIFT with datum tables (Cycle G53)



Datums from a datum table are **always and exclusively** referenced to the current datum (preset).

MP7475, which earlier defined whether datums are referenced to the machine datum or the workpiece datum, now serves only as a safety measure. If MP7475 = 1, the TNC outputs an error message if a datum shift is called from a datum table.

Datum tables from the TNC 4xx whose coordinates are referenced to the machine datum (MP7475 = 1) cannot be used in the iTNC 530.

If you are using datum shifts with datum tables, then use the Select Table function to activate the desired datum table from the NC program.

If you work without the Select Table block **%:TAB:**, you must activate the desired datum table before the test run or the program run (This applies also for 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 a 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.

Function

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.



Shift: Table row? P01: 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 found in the Q parameter.

Cancellation

- 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.





Example: NC blocks

N72 G53 P01 12 *



Selecting a datum table in the part program

With the Select Table (**:TAB:**) function, you select the datum table from which the TNC takes the datums:



PGM MGT ► To select the functions for program call, press the PGM CALL key.

- Press the TOOL TABLE soft key.
 - Enter the complete path name of the datum table and confirm your entry with the END key.



Program a **%:TAB:** block before Cycle **G53** Datum Shift.

A datum table selected with Select Table remains active until you select another datum table with **%:TAB:** or through PGM MGT.

Editing a datum table

Select the datum table in the $\ensuremath{\text{Programming}}$ and $\ensuremath{\text{Editing}}$ mode of operation.

- To call the file manager, press the PGM MGT key (see "File Management: Fundamentals," page 75).
- Display the datum tables: Press the soft keys SELECT TYPE and SHOW .D.
- 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
Select beginning of table	BEGIN
Select end of table	
Go to previous page	PAGE
Go to next page	
Insert line (only possible at end of table)	INSERT LINE
Delete line	DELETE
Confirm entered line and go to the beginning of next line	NEXT LINE
Add the entered number of lines (reference points) to the end of the table	APPEND N LINES
Select list view (standard) or form view.	LIST



Editing a pocket table in a Program Run operating mode

In a program run mode you can select the active datum table. Press the DATUM TABLE soft key. You can then use the same editing functions as in the **Programming and Editing** mode of operation.

Transferring the actual values into the datum table

You can enter the current tool position or the last probed position in the datum table by pressing the "actual-position-capture" key:

Place the text box on the line of the column in which you want to enter the position.



Select the actual-position-capture function: The TNC opens a pop-up window that asks whether you want to enter the current tool position or the last probed values.

- Select the desired function with the arrow keys and confirm your selection with the ENT key.
- To enter the values in all axes, press the ALL VALUES soft key.
- ALL VALUES PRESENT VALUE
- To enter the value in the axis where the text box is located, press the CURRENT VALUE soft key.

Configuring the datum table

In the second and third soft-key rows you can define for each datum table the axes for which you wish to set the datums. In the standard setting all of the axes are active. If you wish to exclude an axis, set the corresponding soft key to OFF. The TNC then deletes that column from the datum table.

If you do not wish to define a datum table for an active axis, press the NO ENT key. The TNC then enters a dash in that column.

To leave a datum table

Select a different type of file in file management and choose the desired file.

Status displays

In the additional status display, the following data from the datum table are shown (see "Coordinate transformations" on page 41):

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

File	NULLTAB.D		MM			>>	-
	x	Ŷ	z	Ð	c		
	+5	+0	+0	+0	+0		
	+25	+37.5	+0	-45	-90		
	-1.8258	-179.7921	+125.7103	+0.0034	+0		
	+0	+0	+150	+0	+0		
	+27.25	+12.5	+0	-10	+0		
i	+250	+325	+10	+0	+90		
	+350	-248	+15	+0	+0		
	+1200	+0	+0	+0	+0		
	+1700	+0	+0	+0	+0		T
	-1700	+0	+0	+0	+0		
0	+0	+0	+0	+0	+0		
1	+0	+0	+0	+0	+0		s
2	+0	+0	+0	+0	+0		0 1
3	+0	+0	+0	+0	+0		
							s I



DATUM SETTING (Cycle G247)

With the Cycle DATUM SETTING, you can activate a datum defined in a preset table as the new datum.

Effect

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.



Number for datum?: Enter the number of the datum to be activated from the preset table



When activating a datum from the preset table, the TNC resets all coordinate transformations that were activated with the following cycles:

- Cycle G53/G54, Datum Shift
- Cycle G28, Mirroring
- Cycle G73, Rotation
- Cycle G72, Scaling

However, the coordinate transformation from Cycle G80, Tilted Working Plane, remains active.

The TNC sets the preset only in the axes that are defined with values in the preset table. The datums of axes marked with – remain unchanged.

Cycle G247 is not functional in Test Run mode.

Status displays

In the additional status display, the following data from the datum table are shown (see "Coordinate transformations" on page 41):

- Name and path of the active datum table
- Active datum number
- Comment from the DOC column of the active datum number

In addition, the active preset number is shown in the large status window behind to the datum symbol.



Example: NC block

N13 G247 DATUM	SETTING
Q339=4	;DATUM NUMBER

8.10 Coordinate Transformation Cycles

MIRROR IMAGE (Cycle G28)

The TNC can machine the mirror image of a contour in the working plane.

Effect

The mirror image 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 mirror image 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.



If you mirror only one axis, the machining direction is reversed for the new machining cycles (cycles 2xx). The machining direction remains the same for older machining cycles, such as Cycle G75/G76 POCKET MILLING.









Mirrored axis?: Enter the axes to be mirrored. You can mirror all axes, including rotary axes, except for the spindle axis and its auxiliary axes. You can enter up to three axes.

Reset

Program the MIRROR IMAGE cycle once again with NO ENT.



Example: NC block

N72 G28 X Y *

8.10 Coordinate Transformation Cycles

ROTATION (Cycle G73)

The TNC can rotate the coordinate system about the active datum in the working plane within a program.

Effect

The ROTATION 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 rotation angle is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane X axis
- Y/Z plane Y axis
- Z/X plane Z axis

Before programming, note the following:

An active radius compensation is canceled by defining Cycle **G73** and must therefore be reprogrammed, if required.

After defining Cycle **G73**, you must move both axes of the working plane to activate rotation for all axes.



Rotation: Enter the rotation angle in degrees (°). Input range: -360° to +360° (absolute G90 before H or incremental G91 before H).

Cancellation

Program the ROTATION cycle once again with a rotation angle of 0°.





Example: NC block

N72 G73 G90 H+25 *

SCALING FACTOR (Cycle G72)

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

Effect

The SCALING FACTOR 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

- the working plane, or on all three coordinate axes at the same time (depending on MP7410)
- the dimensions in cycles
- the parallel axes U,V,W

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.



Scaling factor?: Enter the scaling factor F. The TNC multiplies the coordinates and radii by the F factor (as described under "Effect" above).

Enlargement: F greater than 1 (up to 99.999 999)

Reduction: F less than 1 (down to 0.000 001)

Cancellation

Program the SCALING FACTOR cycle once again with a scaling factor of 1 for the same axis.





Example: NC blocks

N72 G72 F0.750000 *

٦

WORKING PLANE (Cycle G80)



The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the rotary axes or as mathematical angles of a tilted plane. Refer to your machine manual.

The working plane is always tilted around the active datum.

For fundamentals, see "Tilting the Working Plane (Software Option 1)," page 59. Please read this section completely.

Effect

In Cycle **G80** you define the position of the working plane—i.e. the position of the tool axis referenced to the machine coordinate system—by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the position of the tilting axes directly.
- Describe the position of the working plane using up to 3 rotations (spatial angle) of the **machine-referenced** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt. With two spatial angles, every tool position in space can be defined exactly.

Note that the position of the tilted coordinate system, and therefore also all movement in the tilted system, are dependent on your description of the tilted plane.

If you program the position of the working plane via spatial angles, the TNC will calculate the required angle positions of the tilted axes automatically and will store these in the parameters Q120 (A axis) to Q122 (C axis). If two solutions are possible, the TNC will choose the shorter path from the zero position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle 19 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You must move all axes to activate compensation for all axes.







If you set the function TILTING program run to ACTIVE in the Manual Operation mode (see "Tilting the Working Plane (Software Option 1)," page 59), the angular value entered in this menu is overwritten by Cycle **680** WORKING PLANE.



Tilt axis and tilt angle?: Enter the axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys.

If the TNC automatically positions the rotary axes, you can enter the following parameters:

- ► Feed rate ? F=: Traverse speed of the rotary axis during automatic positioning.
- Set-up clearance ?: (incremental value): The TNC positions the tilting head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece.

Cancellation

To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle again, without defining an axis, to disable the function.

Positioning the axis of rotation

	ĥ	
5		Γ

The machine tool builder determines whether Cycle **G80** positions the axes of rotation automatically or whether they must be pre-positioned in the program. Refer to your machine manual.

- If the axes are positioned automatically in Cycle G80:
- The TNC can position only controlled axes.
- In order for the tilted axes to be positioned, you must enter a feed rate and a set-up clearance in addition to the tilting angles, during cycle definition.
- You can use only preset tools (with the full tool length defined in the G99 block or in the tool table).
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting.
- The TNC tilts the working plane at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

If the axes are not positioned automatically in Cycle **G80**, position them before defining the cycle, for example with a G01 block.

Example NC blocks:

N50 G00 G40 Z+100 *	
N60 X+25 Y+10 *	
N70 G01 A+15 F1000 *	Position the axis of rotation
N80 G80 A+15 *	Define the angle for calculation of the compensation
N90 G00 G40 Z+80 *	Activate compensation for the tool axis
N100 X-7.5 Y-10 *	Activate compensation for the working plane

Position display in the tilted system

On activation of Cycle **G80**, the displayed positions (**ACTL** and **NOML**) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition may not be the same as the coordinates of the last programmed position before Cycle **G80**.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If necessary, the TNC outputs an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system (see "Miscellaneous Functions for Coordinate Data," page 196).

Positioning movements with straight lines that are referenced to the machine coordinate system (blocks with M91 or M92) can also be executed in a tilted working plane. Constraints:

- Positioning is without length compensation.
- Positioning is without machine geometry compensation.
- Tool radius compensation is not permitted.



Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle **G80**. In this case, you are shifting the "machine-based coordinate system."

If you program a datum shift after having activated Cycle **G80**, you are shifting the "tilted coordinate system."

Important: When resetting the cycles, use the reverse sequence used for defining them:

1st Activate the datum shift. 2nd: Activate tilting function. 3rd: Activate rotation.

Machining

. . .

1st: Reset the rotation.2nd: Reset the tilting function.3rd: Reset the datum shift.

Automatic workpiece measurement in the tilted system

The TNC measuring cycles enable you to have the TNC measure a workpiece in a tilted system automatically. The TNC stores the measured data in Q parameters for further processing (for example, for printout).

Procedure for working with Cycle G80 WORKING PLANE

1 Write the program

- Define the tool (not required if TOOL.T is active), and enter the full tool length.
- ▶ Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece (clamping devices) during tilting.
- ▶ If required, position the tilt axis or axes with a **G01** block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- ▶ Define Cycle **G80** WORKING PLANE. Enter the angular values for the tilt axes.
- Traverse all main axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- ▶ If required, define Cycle **G80** WORKING PLANE with other angular values to execute machining in a different axis position. In this case, it is not necessary to reset Cycle **G80**. You can define the new angular values directly.
- ▶ Reset Cycle **G80** WORKING PLANE. Program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle G80, without defining an axis.
- Reset datum shift if required.
- ▶ Position the tilt axes to the 0° position if required.



2 Clamp the workpiece

3 Preparations in the Positioning with Manual Data Input (MDI) operating mode

Pre-position the rotary axis/axes to the corresponding angular value(s) for setting the datum. The angular value depends on the selected reference plane on the workpiece.

4 Preparations in the Manual Operation operating mode

Use the 3D-ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the Manual Operating mode. Enter the angular values for the tilt axes into the menu if the axes are not controlled.

If the axes are not controlled, the angular values entered in the menu must correspond to the actual position(s) of the rotary axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Set the datum

- Manually by touching the workpiece with the tool in the untilted coordinate system (see "Datum Setting (Without a 3-D Touch Probe)," page 52).
- Controlled with a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles Manual, chapter 2).
- Automatically by using a HEIDENHAIN 3-D touch probe (see the Touch Probe Cycles Manual, chapter 3).

6 Start the part program in the operating mode Program Run, Full Sequence

7 Manual Operation mode

Use the 3-D ROT soft key to set the TILT WORKING PLANE function to INACTIVE. Enter an angular value of 0° for each axis in the menu (see "Activating manual tilting," page 62).



Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- For subprograms within a subprogram, see "Subprograms," page 403.



%KOUMR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+130 Y+130 Z+0 *	
N30 G99 T1 L+0 R+1 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G54 X+65 Y+65 *	Shift datum to center
N70 L1.0 *	Call milling operation
N80 G98 L10 *	Set label for program section repeat
N90 G73 G91 H+45 *	Rotate by 45° (incremental)
N100 L1.0 *	Call milling operation
N110 L10.6 *	Return jump to LBL 10; repeat the milling operation six times
N120 G73 G90 H+0 *	Reset the rotation
N130 G54 X+0 Y+0 *	Reset the datum shift
N140 G00 Z+250 M2 *	Retract in the tool axis, end program
N150 G98 L1 *	Subprogram 1:
N160 G00 G40 X+0 Y+0 *	Define milling operation
N170 Z+2 M3 *	
N180 G01 Z-5 F200 *	
N190 G41 X+30 *	
N200 G91 Y+10 *	

N210 G25 R5 *	
N220 X+20 *	
N230 X+10 Y-10 *	
N240 G25 R5 *	
N250 X-10 Y-10 *	
N260 X-20 *	
N270 Y+10 *	
N280 G40 G90 X+0 Y+0 *	
N290 G00 Z+20 *	
N300 G98 L0 *	
N999999 %KOUMR G71 *	



8.11 Special Cycles

DWELL TIME (Cycle G04)

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.

Effect

Cycle 9 becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



Dwell time in seconds: Enter the dwell time in seconds.

Input range: 0 to 3600 s (1 hour) in steps of 0.001 seconds



Example: NC block

N74 G04 F1.5 *
PROGRAM CALL (Cycle G39)

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.



Before programming, note the following:

If you want to define an ISO program to be a cycle, enter the file type .I behind the program name.

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 to 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.I.



Program name: Enter the name of the program you want to call and, if necessary, the directory it is located in.

Call the program with

- **G79** (separate block) or
- M99 (blockwise) or
- **M89** (executed after every positioning block)

Example: Program call

A callable program 50 is to be called into a program via a cycle call.



Example: NC blocks

- N550 G39 P01 50 *
- N560 G00 X+20 Y+50 M9 9*



ORIENTED SPINDLE STOP (Cycle G36)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Cycle 13 is used internally for machining 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.

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

Effect

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 G36, the TNC positions the machine tool spindle to an angle that has been set in a machine parameter (see your machine manual).



Angle of orientation: Enter the angle according to the reference axis of the working plane.

Input range: 0 to 360°

Input resolution: 0.001°



Example: NC block

N76 G36 S25*

TOLERANCE (Cycle G62)

[Ŷ	1	
Γ			

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The input parameters **Finishing/roughing** and **Tolerance for rotary axes** are effective only if the HSC filter (**software option 2**) is active on your machine. The TNC will otherwise display an error message. If necessary, contact your machine tool builder.

The TNC automatically smoothes the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without short pauses for computing time. As a result the surface quality is improved and the machine is protected.

A contour deviation results from the smoothing. The size of this deviation **(tolerance value)** is set in a machine parameter by the machine manufacturer. With Cycle **G62**, you can change the pre-set tolerance value and select different filter settings.



Before programming, note the following:

Cycle **G62** is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can reset Cycle **G62** by defining Cycle **G62** again and confirming the dialog question after the **Tolerance value** with NO ENT. Resetting Cycle 32 reactivates the pre-set tolerance:



Example: NC block

N78 G62 T0.05 P01 0 P02 5*

- Tolerance for contour deviation: Permissible contour deviation in mm (for inch programs in inches)
- **Finishing=0, Roughing=1:** Activate filter:
 - Input value 0:

Milling with increased contour accuracy. The TNC uses the filter settings that your machine tool builder has defined for finishing operations.

Input value 1:

Milling at an increased feed rate. The TNC uses the filter settings that your machine tool builder has defined for roughing operations.

▶ Tolerance for rotary axes: Permissible position error of rotary axes in degrees when M128 is active. The TNC always reduces the feed rate in such a way that—if more than one axis is traversed—the slowest axis moves at its maximum feed rate. Rotary axes are usually much slower than linear axes. You can significantly reduce the machining time for programs for more than one axis by entering a large tolerance value (e.g. 10°), since the TNC does not always have to move the rotary axis to the given nominal position. The contour will not be damaged by entering a tolerance value. Only the position of the rotary axis with respect to the workpiece surface will change.

^{в2}







Programming: Subprograms and Program Section Repeats

9.1 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 desired.

Labels

Subprograms and program section repeats begin with the function **G98** L in the part program. The letter L stands for "label."

A label is identified by a number between 1 and 254. Each label number can be set only once with **G98** in a program.



If a label is set more than once, the TNC sends an error message at the end of the **G98** block.

With very long programs, you can limit the number of blocks to be checked for repeated labels with MP7229.

Label 0 (G98 L0) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

9.2 Subprograms

Operating sequence

- **1** The TNC executes the part program up to the block in which a subprogram is called with **LN.0**. *n* can be any label number.
- 2 The subprogram is then executed from beginning to end. The subprogram end is marked **G98 L0**.
- **3** The TNC then resumes the part program from the block after the subprogram call **LN.0**.

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 M02 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 and confirm with the END key.
- To mark the end, press the LBL SET key and enter the label number "0".

Calling a subprogram



LBL SET

- ▶ To call a subprogram, press the LBL CALL key.
- Label number: Enter the label number of the subprogram to be called, then confirm with the ENT key.
- **Repeat REP:** Enter ".0", then confirm with the ENT key.



L0.0 is not permitted, as it corresponds to the program end call.



9.3 Program Section Repeats

Label G98

The beginning of a program section repeat is marked by the label **G98 L**. A program section repeat ends with Ln,m, where m is the number of repeats.

Operating sequence

- 1 The TNC executes the part program up to the end of the program section (L1.2).
- 2 Then the program section between the called label and the label call L 1.2 is repeated the number of times entered after the decimal point.
- **3** The TNC then resumes the part program after the last repetition.

Programming notes

- Vou can repeat a program section up to 65 534 times in succession.
- The TNC always executes the program section once more than the programmed number of repeats.

Programming a program section repeat



- To mark the beginning, press the LBL SET key, then confirm with the ENT key.
- Enter a label number for the program section to be repeated, then confirm with the ENT key.

Calling a program section repeat



- ▶ Press the LBL CALL key.
- ► Label number: Enter the label number of the subprogram to be called, then confirm with the ENT key.
- Repeat REP: Enter the number of repeats, then confirm with the ENT key.



9.4 Separate Program as Subprogram

Operating sequence

- 1 The TNC executes the part program up to the block in which another program is called with %.
- 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.
- The called program must not contain a call with % into the calling program (endless loop).





PGM CALL

PROGRAM

Calling any program as a subprogram

▶	To select the functions	for	program	call,	press
	PGM CALL key.				

- Press the PROGRAM soft key.
- Enter the complete path name of the program you want to call and confirm your entry with the END key.

the

The program you are calling must be stored on the hard disk of your TNC.

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

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 conversational dialog program, enter the file type .H behind the program name.

You can also call a program with Cycle G39.

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.

9.5 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: 8
- Maximum nesting depth for calling main programs: 4
- Vou can nest program section repeats as often as desired.

Subprogram within a subprogram

Example NC blocks

%UPGMS G71 *	
····	
N170 L1.0 *	Subprogram at label G98 L1 is called.
N350 G00 G40 Z+100 M2 *	Last program block of the
	main program (with M2)
N360 G98 L1 *	Beginning of subprogram 1
N390 L2.0 *	Subprogram at label G98 L2 is called.
N450 G98 LO *	End of subprogram 1
N460 G98 L2 *	Beginning of subprogram 2
N620 G98 L0 *	End of subprogram 2
N999999 %UPGMS G71*	



Program execution

- 1 Main program UPGMS is executed up to block N170.
- **2** Subprogram 1 is called, and executed up to block N390.
- **3** Subprogram 2 is called, and executed up to block N620. End of subprogram 2 and return jump to the subprogram from which it was called.
- 4 Subprogram 1 is executed from block N400 up to block N450. End of subprogram 1 and return jump to the main program SUBPGMS.
- Main program UPGMS is executed from block N180 up to block N350. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

%REPS G71 *	
····	
N150 G98 L1 *	Beginning of program section repeat 1
····	
N200 G98 L2 *	Beginning of program section repeat 2
····	
N270 L2.2 *	Program section between this block and G98 L2
····	(block N200) is repeated twice.
N350 L1.1 *	Program section between this block and G98 L1
	(block N150) is repeated once.
N999999 %REPS G71 *	

Program execution

- 1 Main program REPS is executed up to block N270.
- 2 Program section between block N270 and block N200 is repeated twice.
- **3** Main program REPS is executed from block N280 to block N350.
- 4 Program section between block N350 and block N150 is repeated once (including the program section repeat between block N200 and block N270).
- 5 Main program REPS is executed from block N360 to block N999999 (end of program).

Repeating a subprogram

Example NC blocks

%SUBPGREP G71 *	
N100 G98 L1 *	Beginning of program section repeat 1
N110 L2.0 *	Subprogram call
N120 L1.2 *	Program section between this block and G98 L1
····	(block N100) is repeated twice.
N190 G00 G40 Z+100 M2*	Last block of the main program with M2
N200 G98 L2 *	Beginning of subprogram
····	
N280 G98 L0 *	End of subprogram
N999999 %SUBPGREP G71 *	

Program execution

- **1** Main program SUBPGREP is executed up to block N110.
- **2** Subprogram 2 is called and executed.
- **3** Program section between block N120 and block N100 is repeated twice. Subprogram 2 is repeated twice.
- **4** Main program SUBPGREP is executed once from block N130 to block N190. End of program.



Example: Milling a contour in several infeeds

Program sequence

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Mill the contour
- Repeat downfeed and contour-milling



%PGMWDH G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+7.5 *	Define the tool
N40 T1 G17 S4000 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 I+50 J+50 *	Set pole
N70 G10 R+60 H+180 *	Pre-position in the working plane
N80 G01 Z+0 F1000 M3 *	Pre-position to the workpiece surface

N90 G98 L1 *	Set label for program section repeat
N100 G91 Z-4 *	Infeed depth in incremental values (in space)
N110 G11 G41 G90 R+45 H+180 F250 *	First contour point
N120 G26 R5 *	Approach contour
N130 H+120 *	
N140 H+60 *	
N150 H+O *	
N160 H-60 *	
N170 H-120 *	
N180 H+180 *	
N190 G27 R5 F500 *	Depart contour
N200 G40 R+60 H+180 F1000 *	Retract tool
N210 L1.4 *	Return jump to label 1; section is repeated a total of 4 times
N220 G00 Z+250 M2 *	Retract in the tool axis, end program
N9999999 %PGMWDH G71 *	

1

9.6 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



%UP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+2.5 *	Define the tool
N40 T1 G17 S5000 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=300 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=2 ;2ND SET-UP CLEARANCE	
Q211=0 ;DWELL TIME AT DEPTH	

zamples
Programming
9
_
0)

N70 X+15 Y+10 M3 *	Move to starting point for group 1
N80 L1.0 *	Call the subprogram for the group
N90 X+45 Y+60 *	Move to starting point for group 2
N100 L1.0 *	Call the subprogram for the group
N110 X+75 Y+10 *	Move to starting point for group 3
N120 L1.0 *	Call the subprogram for the group
N130 G00 Z+250 M2 *	End of main program
N140 G98 L1 *	Beginning of subprogram 1: Group of holes
N150 G79 *	Call cycle for 1st hole
N160 G91 X+20 M99 *	Move to 2nd hole, call cycle
N170 Y+20 M99 *	Move to 3rd hole, call cycle
N180 X-20 G90 M99 *	Move to 4th hole, call cycle
N180 X-20 G90 M99 * N190 G98 L0 *	Move to 4th hole, call cycle End of subprogram 1



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



%UP2 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+4 *	Define tool: center drill
N40 G99 T2 L+0 R+3 *	Define tool: drill
N50 G99 T3 L+0 R+3.5 *	Define tool: reamer
N60 T1 G17 S5000 *	Call tool: center drill
N70 G00 G40 G90 Z+250 *	Retract the tool
N80 G200 DRILLING	Cycle definition: Centering
Q200=2 ;SET-UP CLEARANCE	
Q201=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q2O2=3 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q2O3=+O ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	
N90 L1.0 *	Call subprogram 1 for the entire hole pattern

les
D
Ξ
xa
ш
δ
<u> </u>
Ξ
Ξ
ra
D
2
Δ
ອ
σ

N100 G00 7+250 M6 *	Tool change
N110 T2 G17 S4000 *	Call toll: drill
N120 D0 0201 P01 -25 *	New depth for drilling
N130 D0 0202 P01 +5 *	New plunging depth for drilling
N140 1 . 0 *	Call subprogram 1 for the entire hole pattern
N150 G00 Z+250 M6 *	
N160 T3 G17 S500 *	Call tool: reamer
N80 G201 REAMING	Cvcle 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	
N180 L1.0 *	Call subprogram 1 for the entire hole pattern
N190 G00 Z+250 M2 *	End of main program
N200 G98 L1 *	Beginning of subprogram 1: Entire hole pattern
N210 G00 G40 G90 X+15 Y+10 M3 *	Move to starting point for group 1
N220 L2.0 *	Call subprogram 2 for the group
N230 X+45 Y+60 *	Move to starting point for group 2
N240 L2.0 *	Call subprogram 2 for the group
N240 L2.0 * N250 X+75 Y+10 *	Call subprogram 2 for the group Move to starting point for group 3
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 * N280 G98 L2 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1 Beginning of subprogram 2: Group of holes
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 * N280 G98 L2 * N290 G79 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1 Beginning of subprogram 2: Group of holes Call cycle for 1st hole
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 * N280 G98 L2 * N290 G79 * N300 G91 X+20 M99 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1 Beginning of subprogram 2: Group of holes Call cycle for 1st hole Move to 2nd hole, call cycle
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 * N280 G98 L2 * N290 G79 * N300 G91 X+20 M99 * N310 Y+20 M99 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1 Beginning of subprogram 2: Group of holes Call cycle for 1st hole Move to 2nd hole, call cycle Move to 3rd hole, call cycle
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 * N280 G98 L2 * N290 G79 * N300 G91 X+20 M99 * N310 Y+20 M99 * N320 X-20 G90 M99 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1 Beginning of subprogram 2: Group of holes Call cycle for 1st hole Move to 2nd hole, call cycle Move to 3rd hole, call cycle Move to 4th hole, call cycle
N240 L2.0 * N250 X+75 Y+10 * N260 L2.0 * N270 G98 L0 * N280 G98 L2 * N290 G79 * N300 G91 X+20 M99 * N310 Y+20 M99 * N320 X-20 G90 M99 * N330 G98 L0 *	Call subprogram 2 for the group Move to starting point for group 3 Call subprogram 2 for the group End of subprogram 1 Beginning of subprogram 2: Group of holes Call cycle for 1st hole Move to 2nd hole, call cycle Move to 3rd hole, call cycle Move to 4th hole, call cycle End of subprogram 2







Programming: Q Parameters

10.1 Principle and Overview

You can program an entire family 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.

Q parameters are designated by the letter Q and a number between 0 and 299. They are grouped according to three ranges:

Meaning	Range
Freely applicable parameters, globally effective for all programs stored in the TNC memory	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, globally effective for all programs that are stored in the TNC memory	Q200 to Q399

Programming notes

You can mix $\ensuremath{\Omega}$ parameters and fixed numerical values within a program.

Q parameters can be assigned numerical values between -99 999.9999 and +99 999.9999. Internally, the TNC can calculate up to a width of 57 bits before and 7 bits after the decimal point (32-bit data width corresponds to a decimal value of 4 294 967 296).



Some Q 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 437).

If you are using the parameters Q60 to Q99 in encoded OEM cycles, define via MP7251 whether the parameters are only to be used locally in the OEM cycles, or may be used globally.



Calling Q parameter functions

When you are writing a part program, press the " Ω " 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
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.
Trigonometric functions	TRIGO- NOMETRY
lf/then conditions, jumps	JUMP
Other functions	DIVERSE
Entering formulas directly	FORMULA
Function for machining complex contours (see "Entering a contour formula," page 364)	CONTOUR



10.2 Part Families—Q Parameters in Place of Numerical Values

The Q parameter function D0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

N150 D00 Q10 P01 +25*	Assign		
	Q10 contains the value 25		
N250 G00 X +Q10*	corresponds to G00 X +25		

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual $\boldsymbol{\Omega}$ 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
	$U_{2} = +30$



10.3 Describing Contours through Mathematical Operations

Function

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 softkey row.
- To select the mathematical functions: Press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Overview

Function	Soft key
D00: ASSIGN Example: D00 Q5 P01 +60 * Assigns a numerical value.	D0 X = Y
D01: ADDITION Example: D01 Q1 P01 -Q2 P02 -5 * Calculates and assigns the sum of two values.	D1 X + Y
D02: SUBTRACTION Example: D02 Q1 P01 +10 P02 +5 * Calculates and assigns the difference of two values.	D2 X - Y
D03: MULTIPLICATION Example: D03 Q2 P01 +3 P02 +3 * Calculates and assigns the product of two values.	D3 X * Y
D04: DIVISION Example: D04 Q4 P01 +8 P02 +Q2 * Calculates and assigns the quotient of two values. Not permitted: division by 0	D4 X / Y
D05: SQUARE ROOT Example: D05 Q50 P01 4 * Calculates and assigns the square root of a number. Not permitted: Square root of a negative number	D5 SORT

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

Programming example 1:

Q	Call the Q parameter functions by pressing the Q key $% \mathcal{Q}$
BASIC ARITHM.	To select the mathematical functions: Press the BASIC ARITHMETIC soft key.
De X = Y	To select the Q parameter function ASSIGN, press the D0 X = Y soft key.
PARAMETER M	IO. FOR RESULT?
5 ENT	Enter the number of the Q parameter, e.g. 5.
1ST VALUE (DR PARAMETER?
10 ENT	Assign the value 10 to Q5.

Example: NC block

N16 D00 P01 +10 *

Programming example 2:

Q	Call the Q parameter functions by pressing the Q key.					
BASIC Arithm.	To select the mathematical functions: Press the BASIC ARITHMETIC soft key.					
D3 X * Y	To select the Q parameter function MULTIPLICATION, press the D03 X * Y soft key.					
PARAMETER N	0. FOR RESULT?					
12 ENT	Enter the number of the Q parameter, e.g. 12.					
1ST VALUE O	R PARAMETER?					
Q5 ENT	Enter Q5 for the first value.					
2ND VALUE O	R PARAMETER?					
7 ENT	Enter 7 for the second value.					

Example: NC block

N17 D03 Q12 P01 +Q5 P02 +7 *



10.4 Trigonometric Functions

10.4 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 α
b is the third side.
The TNC can find the angle from the tangent

 α = arctan (a / b) = arctan (sin α / cos α)

Example:

a = 10 mm

b = 10 mm

 α = arctan (a / b) = arctan 1 = 45°

Furthermore:

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

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



Programming trigonometric functions

Press the TRIGONOMETRY soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table below.

Programming: Compare "Example: Programming fundamental operations."

Function	Soft key
D06: SINE Example: D06 Q20 P01 -Q5 * Calculate the sine of an angle in degrees (°) and assign it to a parameter.	D6 SIN(X)
D07: COSINE Example: D07 Q21 P01 -Q5 * Calculate the cosine of an angle in degrees (°) and assign it to a parameter.	07 COS(X)
D08: ROOT SUM OF SQUARES Example: D08 Q10 P01 +5 P02 +4 * Calculate and assign length from two values.	DS X LEN Y
D13: ANGLE Example: D13 Q20 P01 +10 P02 -Q1 * Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter.	D13 X ANG Y



10.5 lf-Then Decisions with Q Parameters

Function

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 402). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter a program call with % after label G98.

Unconditional jumps

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

D09 P01 +10 P02 +10 P03 1 *

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
D09: IF EQUAL, JUMP Example: D09 P01 +Q1 P02 +Q3 P03 5 * If the two values or parameters are equal, jump to the given label.	D9 IF X EQ Y SOTO
D10: IF NOT EQUAL, JUMP Example: D10 P01 +10 P02 -Q5 P03 10 * If the two values or parameters are not equal, jump to the given label.	D10 IF X NE Y GOTO
D11: IF GREATER THAN, JUMP Example: D11 P01 +Q1 P02 +10 P03 5 * If the first parameter or value is greater than the second value or parameter, jump to the given label.	D11 IF X GT Y GOTO
D12: IF LESS THAN, JUMP Example: D12 P01 +05 P02 +0 P03 1 * If the first value or parameter is less than the second value or parameter, jump to the given label.	D12 IF X LT Y GOTO

Abbreviations used:

IF	:	lf
EQU	:	Equals
NE	:	Not equal
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to



10.6 Checking and Changing Q Parameters

Procedure

You can check and edit Q parameters when writing, testing and running programs in the Programming and Editing, Test Run, Program Run Full Sequence, and Program Run Single Block 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.
- Q

To call Q parameter functions: Press the Q key or the Q INFO soft key in the Programming and Editing mode of operation.

- The TNC lists all parameters and their current values. With the arrow keys or the soft keys, go pagewise to the desired parameters.
- If you would like to change the value, enter a 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 (parameter numbers > 100) used by the TNC are provided with comments.

Manua opera	al ation	Tes	t run			
00	=+0.000	99				-
01	=+12.00	000				
02	=+0.000	00				
QЗ	=-7.500	00				-
Q4	=+123.8	9000				
Q5	=+256.0	0000				
QB	=+0.000	00				
Q7	=+0.000	00				
Q 8	=+1250.	00000				
Q9	=+53.00	000				
Q10	=-2.500	00				T.
Q11	=+0.000	00				
Q12	=+15.00	000				
Q13	=+0.000	00				S
Q14	=+0.000	00				0 T
Q15	=+0.000	00				
						s I
BE			PAGE	PAGE	PRESENT	END

10.7 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
D14:ERROR	D14
Output error messages	ERROR=
D15:PRINT	D15
Unformatted output of texts or Q parameter values	PRINT
D19:PLC	D19
Transfer values to the PLC	PLC=



D14: ERROR: Output error messages

Example NC block

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

N180 D14 P01 254 *

With the function D14: ERROR you can call messages under program control. The messages were preprogrammed by the machine tool builder or by HEIDENHAIN. If the TNC encounters a block with D 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error numbers are listed in the table below.

Range of error numbers	Standard dialog text
0 299	D 14: Error number 0 299
300 999	Machine-dependent dialog
1000 1099	Internal error messages (see table at right)

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Slot width too large
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not
	permitted
1008	MIRRORING not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Entry value incorrect
1012	Wrong sign programmed
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory entry
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 subprogramming
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	Enter Q218 greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Enter Q222 greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be < 360°
1040	Enter Q223 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 0				
1067	Enter Q247 greater than 5				
1068	Datum table?				
1069	Enter direction Q351 unequal 0				
1070	Thread depth too large				
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 a negative value				
1078	Q303 not defined in measuring cycle				
1079	Tool axis not allowed				
1080	Calculated values incorrect				
1081	Contradictory measuring points				



D15: PRINT: Output of texts or Q parameter values

G

Setting the data interface: In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters (see "Assign," page 479).

The function D15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %FN 15RUN.A (output in program run mode) or in the file %FN15SIM.A (output in test run mode). The data are transmitted from a buffer. Data output begins at the latest by program end or when you stop the program. In the Single Block mode of operation, data transfer begins at block end.

Output dialog texts and error messages with D15: PRINT "numerical value"

Numerical values from 0 to 99: Dialog texts for OEM cycles Numerical values exceeding 100: PLC Error Messages

Example: Output of dialog text 20

N67 D15 P01 20 *

Output dialog texts and error messages with D15: PRINT "Q parameter"

Application example: Recording workpiece measurement.

You can transfer up to six Q parameters and numerical values simultaneously.

Example: Output of dialog text 1 and numerical value for Q1

N70 D15 P01 1 P02 Q1 *

D19: PLC: Transferring values to the PLC

The function D19: PLC transfers up to two numerical values or Q parameter contents to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 μm or 0.001°) to the PLC

N56 D19 P01 +10 P02 +Q3 *

Manual operation	Pro	grammi	ng and	l edit	ing		
RS232	inter	face	RS4	22 in	terfac	e	Ļ
Mode o	f op.	: FE1	Mod	e of	op.: F	E 1	
Baud r	ate		Bau	d rat	e		\rightarrow
FE :	96	00	FE	:	9600		
EXT1 :	96	00	EXT	1:	9600		
EXT2 :	96	00	EXT	2:	9600		
LSV-2:	11	5200	LSV	-2:	11520	0	
Assign	:						~ 4
Print		:					S
Print-	test	:					0
PGM MG	т:			Enha	nced		-
Depend	ent f	iles:		Auto	matic		
0-11	RS232 RS422 SETUR	DIAGNOSIS	USER PARAMETER	HELP			END
10.8 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 formula functions. The TNC displays the following soft keys in several soft-key rows:

Logic command	Soft key
Addition Example: Q10 = Q1 + Q5	+
Subtraction Example: Q25 = Q7 - Q108	-
Multiplication Example: Q12 = 5 * Q5	*
Division Example: Q25 = Q1 / Q2	,
Opening parenthesis Example: Q12 = Q1 * (Q2 + Q3)	(
Closing parenthesis Example: Q12 = Q1 * (Q2 + Q3)	>
Square of a value Example: Q15 = SQ 5	50
Square root Example: Q22 = SQRT 25	SORT
Sine of an angle Example: Q44 = SIN 45	SIN
Cosine of an angle Example: Q45 = COS 45	COS
Tangent of an angle Example: Q46 = TAN 45	TAN
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: Q10 = ASIN 0.75	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: 011 = ACOS 040	ACOS

10.8 Entering Formulas Directly



Logic command	Soft key
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: Q12 = ATAN Q50	ATAN
Powers of values Example: Q15 = 3^3	^
Constant "pi" (3.14159) Example: Q15 = PI	PI
Natural logarithm (LN) of a number Base 2.7183 Example: Q15 = LN Q11	LN
Logarithm of a number, base 10 Example: Q33 = LOG Q22	LOG
Exponential function, 2.7183 to the power of n Example: Q1 = EXP Q12	EXP
Negate (multiplication by -1) Example: Q2 = NEG Q1	NEG
Truncate decimal places Form an integer Example: Q3 = INT Q42	INT
Absolute value of a number Example: Q4 = ABS Q22	ABS
Truncate places before the decimal point Form a fraction Example: Q5 = FRAC Q23	FRAC
Check algebraic sign of a number Example: Q12 = SGN Q50 If result for Q12 = 1: Q50 >= 0 If result for Q12 = 0: Q50 < 0	SGN
Calculate modulo value 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 (multiplication and division before addition and subtraction)

N112 Q1 = 5 * 3 + 2 * 10 = 35

1st calculation: 5 * 3 = 15 **2nd** calculation: 2 * 10 = 20 **3rd** calculation: 15 +20 = 35

or

N113 Q2 = SQ 10 - 3³ = 73

1st calculation: 10 squared = 100 **2nd** calculation: 3 to the power of 3 = 27**3rd** calculation: 100 - 27 = 73

Distributive 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.

Q	Call the Q parameter functions by pressing the Q key.
FORMULA	For formula input, press the FORMULA soft key.
PARAMETER NO). FOR RESULT?
ENT 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.
Q 12	Enter Q parameter number 12.
,	Select division.
Q 13	Enter Q parameter number 13.
, END	Close parentheses and conclude formula entry.

Example NC block

N37 Q25 = ATAN (Q12/Q13)

10.9 Preassigned Q Parameters

The Q parameters Q100 to Q122 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

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 G99 block)
- Delta value DR from the tool table
- Delta value DR from the TOOL CALL block

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 Q110 depends on which M function was last programmed for the spindle:

M Function	Parameter value
No spindle status defined	Q110 = -1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M Function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112

The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with %...) is programmed in millimeters or inches.

Dimensions 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.

j

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 are referenced to the datum that is currently active in the Manual operating mode.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis dependent on MP100	Q118
Vth axis dependent on MP100	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 130

Actual-nominal deviation	Parameter value
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC

coordinates	Parameter value
A axis	Q120
B axis	Q121
C axis	Q122



Results of measurements with touch probe

cycles

(also see the Touch Probe Cycles User's Manual)

Measured actual values	Parameter value
Angle of a straight line	Q150
Center in reference axis	Q151
Center in minor axis	Q152
Diameter	Q153
Length of pocket	Q154
Width of pocket	Q155
Length in the axis selected in the cycle	Q156
Position of the center line	Q157
Angle of the A axis	Q158
Angle of the B axis	Q159
Coordinate of the axis selected in the cycle	Q160

Measured deviation	Parameter value
Center in reference axis	Q161
Center in minor axis	Q162
Diameter	Q163
Length of pocket	Q164
Width of pocket	Q165
Measured length	Q166
Position of the center line	Q167

Measured solid angle	Parameter value
Rotation about the A axis	Q170
Rotation about the B axis	Q171
Rotation about the C axis	Q172

Workpiece status	Parameter value
Good	Q180
Re-work	Q181
Scrap	Q182

Measured deviation with cycle 440	Parameter value
X axis	Q185
Y axis	Q186
Z axis	Q187

Reserved for internal use	Parameter value
Markers for cycles (point patterns)	Q197
Number of the active touch probe cycle	Q198

Status during tool measurement with TT	Parameter value
Tool within tolerance	Q199 = 0.0
Tool is worn (LTOL/RTOL exceeded)	Q199 = 1.0
Tool is broken (LBREAK/RBREAK exceeded)	Q199 = 2.0



Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculation steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane: Clockwise machining direction: starting angle > end angle Counterclockwise machining direction: starting angle < end angle
- The tool radius is not taken into account.



%ELLIPSE G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q3 P01 +50 *	Semiaxis in X
N40 D00 Q4 P01 +30 *	Semiaxis in Y
N50 D00 Q5 P01 +0 *	Starting angle in the plane
N60 D00 Q6 P01 +360 *	End angle in the plane
N70 D00 Q7 P01 +40 *	Number of calculation steps
N80 D00 Q8 P01 +30 *	Rotational position of the ellipse
N90 D00 Q9 P01 +5 *	Milling depth
N100 D00 Q10 P01 +100 *	Feed rate for plunging
N110 D00 Q11 P01 +350 *	Feed rate for milling
N120 D00 Q12 P01 +2 *	Set-up clearance for pre-positioning
N130 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+2.5 *	Define the tool
N160 T1 G17 S4000 *	Tool call
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 G00 Z+250 M2 *	Retract in the tool axis, end program
N200 G98 L10 *	Subprogram 10: Machining operation

les
mp
Exal
ng
imi
ran
rog
10 P
10.

N210 G54 X+Q1 Y+Q2 *	Shift datum to center of ellipse
N220 G73 G90 H+Q8 *	Account for rotational position in the plane
N230 Q35 = (Q6 - Q5) / Q7	Calculate angle increment
N240 D00 Q36 P01 +Q5 *	Copy starting angle
N250 D00 Q37 P01 +0 *	Set counter
N260 Q21 = Q3 * COS Q36	Calculate X coordinate for starting point
N270 Q22 = Q4 * SIN Q36	Calculate Y coordinate for starting point
N280 G00 G40 X+Q21 Y+Q22 M3 *	Move to starting point in the plane
N290 Z+Q12 *	Pre-position in tool axis to set-up clearance
N300 G01 Z-Q9 FQ10 *	Move to working depth
N310 G98 L1 *	
N320 Q36 = Q36 + Q35	Update the angle
N330 Q37 = Q37 + 1	Update the counter
N340 Q21 = Q3 * COS Q36	Calculate the current X coordinate
N350 Q22 = Q4 * SIN Q36	Calculate the current Y coordinate
N360 G01 X+Q21 Y+Q22 FQ11 *	Move to next point
N370 D12 P01 +Q37 P02 +Q7 P03 1 *	Unfinished? If not finished return to label 1
N380 G73 G90 H+0 *	Reset the rotation
N390 G54 X+0 Y+0 *	Reset the datum shift
N400 G00 G40 Z+Q12 *	Move to set-up clearance
N410 G98 LO *	End of subprogram
N999999 %ELLIPSE G71 *	



Example: Concave cylinder machined with spherical cutter

Program sequence

- Program functions only with a spherical cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The machining direction can be altered by changing the entries for the starting and end angles in space: Clockwise machining direction:
- starting angle > end angle
- Counterclockwise machining direction: starting angle < end angle
- The tool radius is compensated automatically.



%CYLIN G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +0 *	Center in Y axis
N30 D00 Q3 P01 +0 *	Center in Z axis
N40 D00 Q4 P01 +90 *	Starting angle in space (Z/X plane)
N50 D00 Q5 P01 +270 *	End angle in space (Z/X plane)
N60 D00 Q6 P01 +40 *	Radius of the cylinder
N70 D00 Q7 P01 +100 *	Length of the cylinder
N80 D00 Q8 P01 +0 *	Rotational position in the X/Y plane
N90 D00 Q10 P01 +5 *	Allowance for cylinder radius
N100 D00 Q11 P01 +250 *	Feed rate for plunging
N110 D00 Q12 P01 +400 *	Feed rate for milling
N120 D00 Q13 P01 +90 *	Number of cuts
N130 G30 G17 X+0 Y+0 Z-50 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+3 *	Define the tool
N160 T1 G17 S4000 *	Tool call
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 D00 Q10 P01 +0 *	Reset allowance
N200 L10.0 *	Call machining operation

N210 G00 G40 Z+250 M2 *	Retract in the tool axis, end program
N220 G98 L10 *	Subprogram 10: Machining operation
N230 Q16 = Q6 - Q10 - Q108	Account for allowance and tool, based on the cylinder radius
N240 D00 Q20 P01 +1 *	Set counter
N250 D00 Q24 P01 +Q4 *	Copy starting angle in space (Z/X plane)
N260 Q25 = (Q5 - Q4) / Q13	Calculate angle increment
N270 G54 X+Q1 Y+Q2 Z+Q3 *	Shift datum to center of cylinder (X axis)
N280 G73 G90 H+Q8 *	Account for rotational position in the plane
N290 G00 G40 X+0 Y+0 *	Pre-position in the plane to the cylinder center
N300 G01 Z+5 F1000 M3 *	Pre-position in the tool axis
N310 G98 L1 *	
N320 I+0 K+0 *	Set pole in the Z/X plane
N330 G11 R+Q16 H+Q24 FQ11 *	Move to starting position on cylinder, plunge-cutting obliquely into the material
N340 G01 G40 Y+Q7 FQ12 *	Longitudinal cut in Y+ direction
N350 D01 Q20 P01 +Q20 P02 +1 *	Update the counter
N360 D01 Q24 P01 +Q24 P02 +Q25 *	Update solid angle
N370 D11 P01 +Q20 P02 +Q13 P03 99 *	Finished? If finished, jump to end
N380 G11 R+Q16 H+Q24 FQ11 *	Move in an approximated "arc" for the next longitudinal cut
N390 G01 G40 Y+0 FQ12 *	Longitudinal cut in Y- direction
N400 D01 Q20 P01 +Q20 P02 +1 *	Update the counter
N410 D01 Q24 P01 +Q24 P02 +Q25 *	Update solid angle
N420 D12 P01 +Q20 P02 +Q13 P03 1 *	Unfinished? If not finished, return to LBL 1
N430 G98 L99 *	
N440 G73 G90 H+0 *	Reset the rotation
N450 G54 X+0 Y+0 Z+0 *	Reset the datum shift
N460 G98 L0 *	End of subprogram
N999999 %CYLIN G71 *	



Example: Convex sphere machined with end mill

Program sequence

- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically.



%SPHERE G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q4 P01 +90 *	Starting angle in space (Z/X plane)
N40 D00 Q5 P01 +0 *	End angle in space (Z/X plane)
N50 D00 Q14 P01 +5 *	Angle increment in space
N60 D00 Q6 P01 +45 *	Radius of the sphere
N70 D00 Q8 P01 +0 *	Starting angle of rotational position in the X/Y plane
N80 D00 Q9 P01 +360 *	End angle of rotational position in the X/Y plane
N90 D00 Q18 P01 +10 *	Angle increment in the X/Y plane for roughing
N100 D00 Q10 P01 +5 *	Allowance in sphere radius for roughing
N110 D00 Q11 P01 +2 *	Set-up clearance for pre-positioning in the tool axis
N120 D00 Q12 P01 +350 *	Feed rate for milling
N130 G30 G17 X+0 Y+0 Z-50 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+7.5 *	Define the tool
N160 T1 G17 S4000 *	Tool call
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 D00 Q10 P01 +0 *	Reset allowance
N200 D00 Q18 P01 +5 *	Angle increment in the X/Y plane for finishing

N210 L10.0 *	Call machining operation
N220 G00 G40 Z+250 M2 *	Retract in the tool axis, end program
N230 G98 L10 *	Subprogram 10: Machining operation
N240 D01 Q23 P01 +Q11 P02 +Q6 *	Calculate Z coordinate for pre-positioning
N250 D00 Q24 P01 +Q4 *	Copy starting angle in space (Z/X plane)
N260 D01 Q26 P01 +Q6 P02 +Q108 *	Compensate sphere radius for pre-positioning
N270 D00 Q28 P01 +Q8 *	Copy rotational position in the plane
N280 D01 Q16 P01 +Q6 P02 -Q10 *	Account for allowance in the sphere radius
N290 G54 X+Q1 Y+Q2 Z-Q16 *	Shift datum to center of sphere
N300 G73 G90 H+Q8 *	Account for starting angle of rotational position in the plane
N310 G98 L1 *	Pre-position in the tool axis
N320 I+0 J+0 *	Set pole in the X/Y plane for pre-positioning
N330 G11 G40 R+Q26 H+Q8 FQ12 *	Pre-position in the plane
N340 I+Q108 K+0 *	Set pole in the Z/X plane, offset by the tool radius
N350 G01 Y+0 Z+0 FQ12 *	Move to working depth
N360 G98 L2 *	
N370 G11 G40 R+Q6 H+Q24 FQ12 *	Move upward in an approximated "arc"
N380 D02 Q24 P01 +Q24 P02 +Q14 *	Update solid angle
N390 D11 P01 +Q24 P02 +Q5 P03 2 *	Inquire whether an arc is finished. If not finished, return to LBL 2.
N400 G11 R+Q6 H+Q5 FQ12 *	Move to the end angle in space
N410 G01 G40 Z+Q23 F1000 *	Retract in the tool axis
N420 G00 G40 X+Q26 *	Pre-position for next arc
N430 D01 Q28 P01 +Q28 P02 +Q18 *	Update rotational position in the plane
N440 D00 Q24 P01 +Q4 *	Reset solid angle
N450 G73 G90 H+Q28 *	Activate new rotational position
N460 D12 P01 +Q28 P02 +Q9 P03 1 *	Unfinished? If not finished, return to label 1
N470 D09 P01 +Q28 P02 +Q9 P03 1 *	
N480 G73 G90 H+0 *	Reset the rotation
N490 G54 X+0 Y+0 Z+0 *	Reset the datum shift
N500 G98 LO *	End of subprogram
N999999 %SPHERE G71 *	





AL PAC

Test Run and Program Run

1

11.1 Graphics

Function

In the Program Run modes of operation as well as in the Test Run mode, the TNC provides the following display modes. Using soft keys, select whether you desire:

- Plan view
- Projection in 3 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 blank form definition

no program is selected

With Machine Parameters 7315 to 7317 you can have the TNC display a graphic even if no tool axis is defined or moved.

A graphic simulation is not possible for program sections or programs in which rotary axis movements or a tilted working plane are defined. In this case, the TNC will display an error message.

The TNC graphic does not show a radius oversize ${\rm DR}$ that has been programmed in the ${\rm T}$ block.

Overview of display modes

The TNC displays the following soft keys in the Program Run and Test Run modes of operation:

Display mode	Soft key
Plan view	
Projection in 3 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

Plan view is the fastest of the three 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. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

Details can be isolated in this display mode for magnification (see "Magnifying details," page 454).

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 and select the soft key for sectional planes.
- ▶ The TNC then displays 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.

Coordinates of the line of intersection

At the bottom of the graphics window, the TNC displays the coordinates of the line of intersection, referenced to the workpiece datum. Only the coordinates of the working plane are shown. This function is activated with MP7310.

3-D view

The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

You can rotate the 3-D display about the vertical and horizontal axes. The shape of the workpiece blank 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 454.



Press the soft key for 3-D view.

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	ų,
Rotate in 5° steps about the horizontal axis	
Magnify the graphic stepwise. If the view is magnified, the TNC shows the letter Z in the footer of the graphic window.	*
Reduce the graphic stepwise If the view is magnified, the TNC shows the letter Z in the footer of the graphic window.	$\widehat{}$
Reset image to programmed size	1:1

Switch the frame overlay display for the workpiece blank on/off:

Shift the soft-key row until the soft key for the rotating and magnification/reduction appears.



- Select functions for rotating and magnifying/reducing:
- Show the frame for the BLK FORM: Set the highlight in the soft key to SHOW
- BLK FORM DISPLAY HIDE
- Hide the frame for the BLK FORM: Set the highlight in the soft key to OMIT



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 enlargement appears.



- Select the functions for section 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 blank 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	t∎ (
Shift the sectional plane to reduce or magnify the blank form	- +
Select the isolated detail	TRANSFER DETAIL



Cursor position during detail magnification

During detail magnification, the TNC displays the coordinates of the axis that is currently being isolated. The coordinates describe the area determined for magnification. To the left of the slash is the smallest coordinate of the detail (MIN point), to the left is the largest (MAX point).

If a graphic display is magnified, this is indicated with **MAGN** at the lower right of the graphics window.

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 blank to the detail magnification in which it was last shown.	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as the blank form was programmed.	WINDOW 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.



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 approximate time that the TNC calculates from the duration of tool movements. The time calculated by the TNC cannot 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 rows until the TNC displays the following soft keys with the stopwatch functions:

Stopwatch functions	Soft key
Store displayed time	STORE
Display the sum of stored time and displayed time	
Clear displayed time	RESET 00:00:00





The soft keys available to the left of the stopwatch functions depend on the selected screen layout.

The time is reset when a new BLK form is entered.



11.2 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 n in nagao.

program in pages:	%3813 G71 * N10 D00 Q1 P01 +0*
Function	oft key N20 D00 D2 P01 +0*
Go back in the program by one screen	PAGE N35 D00 D6 P01 +40* N36 D00 D16 P01 +10* N40 D00 D7 P01 +10* N40 D00 D17 P01 +270*
Go forward in the program by one screen	PAGE N50 D00 D8 P01 +0* N70 D00 D18 P01 +90* N80 D00 D9 P01 +0*
Go to beginning of program	BEGIN N100 D00 012 P01 +0* 0 N100 D00 020 P01 +500* Image: 1 to 1 t
Go to end of program	

Positioning with mdi

Test run

1

11.3 Test Run

Function

In the Test Run mode of operation you can simulate programs and program sections to prevent errors from occurring 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
- Interrupt test at any block
- Optional block skip
- Functions for graphic simulation
- Measuring the machining time
- Additional status display

Running a program test

If the central tool file is active, a tool table must be active (status S) to run a program test. Select a tool table via the file manager (PGM MGT) in the Test Run mode of operation.

With the MOD function BLANK IN WORK SPACE, you can activate work space monitoring for the test run (see "Showing the Workpiece in the Working Space," page 492).



- 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:

Function	Soft key
Test the entire program	START
Test each program block individually	START SINGLE
Show the blank form and test the entire program	RESET + START
Interrupt the test run	STOP



11.3 Test Run

Run a program test up to a certain block

With the STOP AT N function the TNC does a test run up to the block with block number N.

- ▶ Go to the beginning of program in the Test Run mode of operation.
- To run a program test up to a specific block, press the STOP AT N soft key.



Stop at N: Enter the block number at which you wish the test to stop.

- ▶ **Program:** Enter the name of the program that contains the block with the selected block number. The TNC displays the name of the selected program. If the test run is to be interrupted in a program that was called with %, you must enter this name.
- Repetitions: If N is located in a program section repeat, enter the number of repeats that you want to run.
- To test a program section, press the START soft key. The TNC will test the program up to the entered block.

Positioning with mdi	Test r	ΠU				
%3813 G	71 *					_
N10 D00	Q1 P01	+0*				
N20 D00	Q2 P01	+0*				
N30 D00	Q3 P01	+0*				\rightarrow
N35 D00	Q6 P01	+40*				
N36 D00	Q16 P0:	l +10*				
N40 D00	Q7 P01	+90*				
N50 D00	Q17 P0:	l +270∗				
N60 D00	Q8 P01	+0*				
N70 D00	Q18 P0:	L +90*				-T-I
N80 D00	Q9 P01	+0*				
N90 D00	Q10 Fsto	n am test termina ⊳at: N - №	tion @			S I
N100 D00	Q12	aram = 38 stitions = 1	13.I			0 🕈
N110*						
N120 D00	0 Q 2 Ø P Ø	01 +500*				1
			START SINGLE	END	START	RESET +

11.4 Program Run

Function

In the Program Run, Full Sequence mode 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 can be used in the program run modes of operation:

- Interrupt program run
- Start 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 necessary tables and pallet files (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 rapid traverse speed when starting the NC program using the FMAX soft key. The entered value remains in effect even after the machine has been turned off and on again. In order to re-establish the original rapid traverse speed, you need to re-enter the corresponding value.

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
- 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:

- 🔳 G38
- Miscellaneous function M0, M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

Interrupting the machining process with the machine STOP button

- Press the machine STOP button: The block which the TNC is currently executing is not completed. The asterisk in the status display blinks.
- If you do not wish to continue the machining process, you can reset the TNC with the INTERNAL STOP soft key. The asterisk in the status display goes out. In this case, the program must be restarted from the program beginning.

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.



Danger of collision

If you interrupt program run while the working plane is tilted, you can change from a tilted to a non-tilted coordinate system, and vice versa, by pressing the 3-D ON/OFF soft key.

The functions of the axis direction buttons, the electronic handwheel and the positioning logic for returning to the contour are then evaluated by the TNC. When retracting the tool make sure the correct coordinate system is active and the angular values of the tilt axes are entered in the 3-D ROT menu.

Application example: Retracting the spindle after tool breakage

- ▶ Interrupt machining.
- Enable the external direction keys: Press the MANUAL OPERATION soft key.
- ▶ Move the axes with the machine axis direction buttons.



On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to your machine manual.

Resuming program run after an interruption

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.
- An interruption was programmed.

Resuming program run after an error

If the error message is not blinking:

- Remove the cause of the error.
- ▶ To 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 repair 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.



P

Mid-program startup must not begin in a subprogram.

All necessary programs, tables and pallet files 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 also applies to an altered tool length.

If you are working with nested programs, you can use MP7680 to define whether the block scan is to begin at block 0 of the main program or at block 0 of the last interrupted program.

The function M128 is not permitted during a mid-program startup.

If the working plane is tilted, you can use the 3-D ON/OFF soft key to define whether the TNC is to return to the contour in a tilted or in a non-tilted coordinate system.

If you want to use the block scan feature in a pallet table, select the program in which a mid-program startup is to be performed from the pallet table by using the arrow keys. Then press the RESTORE POS AT N soft key.

All touch probe cycles and Cycle 247 are skipped in a midprogram startup. Result parameters that are written to from these cycles might therefore remain empty.

- ▶ To go to the first block of the current program to start a block scan, enter GOTO "0".
- To select mid-program startup, press the RESTORE POS AT N 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, enter the number of repetitions to be calculated in the block scan.
- PLC ON/OFF: To account for tool calls and miscellaneous functions M: Set the PLC to ON (use the ENT key to switch between ON and OFF). If PLC is set to OFF, the TNC considers only the geometry. The tool in the spindle must equal the tool called by the program.
- ▶ To start the block scan, press the machine START button.
- ▶ To return to the contour, see "Returning to the contour," page 468.



11.4 Program Run

Returning to the contour

With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.
- Return to the contour after a block scan 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.
- ► To select a return to contour, press the RESTORE POSITION soft key.
- ▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine START button.
- To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START key.
- ▶ To resume machining, press the machine START key.

Program run, full sequence	est run
N40 T1 G17 S5000* N50 G00 G40 G90 Z+250* N60 X-30 Y+50* N70 G01 Z-5 F200* N80 G01 X+0 Y+50* N90 X+50 Y+100* N100 G42 G25 R20* N100 G42 G25 R20* N120 X+50 Y+0* N130 G25 R15*	1 2 2
X -99.600 M -177.837 2 -318.87 C +0.000 B +0.000 - - - - - - - - - - - - 318.87	
RCTL. + II i Z F 0 H 5/9 RESTORE	
11.5 Automatic Program Start

Function



The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.

In a Program Run operating mode, you can use the soft key AUTOSTART (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:



Show the window for entering the starting time (see figure at lower right).

- Time (h:min:sec): Time of day at which the program is to be started.
- Date (DD.MM.YYYY): Date at which the program is to be started.
- To activate the start, set the AUTOSTART soft key to ON.





11.6 Optional Block Skip

Function

In a test run or program run, the TNC can skip over blocks that begin with a slash $^{\prime\prime}/^{\prime\prime}$:



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 G99 blocks.

After a power interruption the control returns to the most recently selected setting.

Erasing the "/" character

In the Programming and Editing mode you select the block in which the character is to be erased.



Erase the "/" character.



11.7 Optional Program Run Interruption

Function

The TNC optionally interrupts the program or test run at blocks containing M01. If you use M01 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 M01: Set soft key to OFF.



Interrupt Program Run or Test Run at blocks containing M01: Set soft key to ON.







MOD Functions

i

12.1 MOD Functions

The MOD functions provide additional input possibilities and displays. 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.



Press the MOD key. Select the MOD functions for programming/editing and test run. Figures at right, figure on next page: MOD function in a machine operating mode.

Changing the settings

Select the desired MOD function in the displayed menu with the arrow keys.

There are three possibilities for changing a setting, depending on the function selected:

- Enter a numerical value directly, e.g. when determining 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 desired setting directly by pressing the corresponding numerical key (to the left of the colon), or by using the arrow keys and then confirming with ENT. If you don't want to change the setting, close the window again with END.

Exiting the MOD functions

Close the MOD functions with the END key or soft key.

Overview of MOD functions

Depending on the selected mode of operation, you can make the following changes:

Programming and Editing:

- Display software numbers
- Enter code number
- Set data interface
- Machine-specific user parameters (if provided)
- HELP files (if provided)





Test Run:

- Display software numbers
- Enter code number
- Set the data interface
- Show the workpiece in the working space
- Machine-specific user parameters (if provided)
- Display HELP files (if provided)

In all other modes:

- Display software numbers
- Display code digits for installed options
- Select position display
- Unit of measurement (mm/inches)
- Programming language for MDI
- Select the axes for actual position capture
- Set the axis traverse limits
- Display the datums
- Display operating times
- HELP files (if provided)

Manual operation	Programming and editing
Position display 1 <mark>ACTL.</mark> Position display 2 DIST. Change MM/INCH MM Program input HEIDENHAIN Axis selection %00111	2 2
NC : software number 340422 02 PLC: software number BASIS33-03 OPT :%0000111100000111 DSP1: 246261 21	
ICTL1: 246276 20	
POSITION/ TRAVERSE HELP HACHINE	END

12.1 MOD Functions

12.2 Software Numbers and Option Numbers

Function

The following software numbers are displayed on the TNC screen after the MOD functions have been selected:

- **NC:** Number of the NC software (managed by HEIDENHAIN)
- PLC: Number and name of the PLC software (managed by your machine tool builder)
- **SETUP:** Number of the cycle software and the used soft keys (managed by HEIDENHAIN)
- DSP1: Number of the speed controller software (managed by HEIDENHAIN)
- ICTL1: Number of the current controller software (managed by HEIDENHAIN)

In addition, coded numbers for the options available on your control are displayed after the abbreviation **OPT:**

No options active	%00000000000000000000000000000000000000
Bit 0 to bit 7: Additional control loops	%00000000 0000011
Bit 8 to bit 15: Software options	% 00000011 00000011

12.3 Code Numbers

Function

Code numbers allow you to access various functions that are not always required for normal operation of the TNC.

The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Enable special functions for Q-parameter programming	555343
Configuring an Ethernet card	NET123

In addition, you can use the keyword **version** to create a file containing all current software numbers of your control:

- ▶ Enter the keyword **version** and confirm with the ENT key.
- ▶ The TNC displays all current software numbers on the screen.
- ▶ To terminate the version overview, press the END key.

<u></u>
_

If necessary, you can output the file **version.a** saved in the directory TNC:, and send it to your machine manufacturer or HEIDENHAIN for diagnostic purposes.

12.4 Setting the Data Interfaces

Function

To set up the data interfaces, press the RS-232 / RS-422 SETUP soft key to call a menu for setting the data interfaces:

Setting the RS-232 interface

The mode of operation and baud rates for the RS-232 interface are entered in the upper left of the screen.

Setting the RS-422 interface

The mode of operation and baud rates for the RS-422 interface are entered in the upper right of the screen.

Setting the OPERATING MODE of the external device



The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the operating modes FE2 and EXT.

Setting the BAUD RATE

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

External device	Operating mode	Symbol
PC with HEIDENHAIN software TNCremo for remote operation of the TNC	LSV2	
PC with HEIDENHAIN data transfer software TNCremo	FE1	
HEIDENHAIN floppy disk units FE 401 B FE 401 from prog. no. 230 626 03	FE1 FE1	
HEIDENHAIN floppy disk unit FE 401 up to prog. no. 230 626 02	FE2	
Non-HEIDENHAIN devices such as punchers, PC without TNCremo	EXT1, EXT2	₽



Assign

This function sets the destination for the transferred data.

Applications:

Transferring values with Q parameter function D15

The TNC mode of operation determines whether the PRINT or PRINT TEST function is used:

TNC mode of operation	Transfer function
Program Run, Single Block	PRINT
Program Run, Full Sequence	PRINT
Test Run	PRINT TEST

You can set PRINT and PRINT TEST as follows:

Function	Path
Output data via RS-232	RS232:\
Output data via RS-422	RS422:\
Save data to the TNC's hard disk	TNC:\
Save the data in the same directory as the program with D15.	- vacant -

File names

Data	Operating mode	File name
Values with D15	Program Run	%FN15RUN.A
Values with D15	Test Run	%FN15SIM.A



Software for data transfer

For transfer of files to and from the TNC, we recommend using one of the HEIDENHAIN TNCremo data transfer software products for data transfer, such as TNCremo or TNCremoNT. With TNCremo/ TNCremoNT, data transfer is possible with all HEIDENHAIN controls via serial interface.



Please contact your HEIDENHAIN agent if you would like to receive the TNCremo or TNCremoNT data transfer software.

System requirements for TNCremoNT:

- PC with 486 processor or higher
- Operating system Windows 95, Windows 98, Windows NT 4.0, Windows 2000
- 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>, <TNCremoNT>

When you start TNCremoNT for the first time, TNCremoNT automatically tries to set up a connection with the TNC.

Data transfer between the TNC and TNCremoNT

Ensure that:

- The TNC is connected to the correct serial port on your PC or to the network, respectively.
- The operating mode of the interface is set to **LSV2** on the TNC.

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 key PGM MGT (see "Data transfer to or from an external data medium" on page 80) and transfer the desired files.

End 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.

🚋 TNCremoNT				_ 🗆
<u>File ⊻iew Extras H</u> elp				
🖯 🗈 🖻 🗙 🗉) 📰 🖬 🐣	9		
	z:\CYCLE\2	80474XX\NC	.u	Control
Name	Size	Attribute	Date	TNC 430PA
<u> </u>				File status
200.CYC	1858	A	24.08.99 08:00:58	Free: 3367 MByte
🖃 200.H	2278	A	24.08.99 07:41:58	
🗋 201.CYC 🖌	1150	A	24.08.99 08:00:58	Total: [39
₽ 201.H	1410	A	24.08.99 07:41:58	Masked: 39
202.CYC	2532	A	24.08.99 13:18:58	
H) 202.H	3148	A	24.08.99 13:14:58	·
	TNC:\NK	\TSWORK[*.*]	Connection
Name	Size	Attribute	Date	 Protocol:
🚞				LSV-2
3DTASTDEM.H	372		24.08.99 09:27:30	Serial port
🕒 419.H	5772		24.08.99 09:27:24	COM2
H 440.H	4662		24.08.99 09:27:26	JCOM2
🖃 HRUEDI.I 🛛 🎽	92		24.08.99 09:27:34	Baud rate (autodetect):
🖻 U 👘 👘	12		24.08.99 09:27:32	115200
H T419.H	308		24.08.99 09:27:32	
H T440.H	154		24.08.99 09:27:28	-
	0000		00.00.00.00.00.00	
DNC connection establishe	d			



12.5 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 through the Ethernet card in accordance with the Transmission Control Protocol/ Internet Protocol (TCP/IP) family of protocols and with the aid of the Network File System (NFS).

Connection possibilities

You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX). The connection is metallically isolated from the control electronics.

RJ45 connection X26 (100BaseTX or 10BaseT)

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).

If you connect the TNC directly with a PC you must use a transposed cable.



Connecting the iTNC directly with a Windows PC

You don't need any large effort or special networking knowledge to attach the iTNC 530 directly to a PC that has an Ethernet card. You simply have to make some settings on the TNC and the corresponding settings on the PC.

Settings on the iTNC

- Connect the iTNC (connection X26) and the PC with a crossed Ethernet cable (trade names: crossed patch cable or STP cable).
- ▶ In the Programming and Editing mode of operation, press the MOD key. Enter the keyword NET123. The iTNC will then display the main screen for network configuration (see figure at top right).
- Press the DEFINE NET soft key to enter the network setting for a specific device (see figure at center right).
- Enter any network address. Network addresses consist of four numbers separated by periods, e.g. 160.1.180.23
- Press the right arrow key to select the next column, and enter the subnet mask. The subnet mask also consists of four numbers separated by periods, e.g. 255.255.0.0
- ▶ Press the END key to leave the network configuration screen.
- Press the DEFINE MOUNT soft key to enter the network settings for a specific PC (see figure at bottom right).
- Define the PC name and drive that you want to access, beginning with two slashes, e.g. //PC3444/C
- Press the right arrow key to select the next column, and enter the name that the iTNC's file manager uses to display the PC, e.g. PC3444:
- Press the right arrow key to select the next column, and enter the file system type smb
- Press the right arrow key to select the next column and enter the following information (depending on the PC operating system): ip=160.1.180.1,username=abcd,workgroup=SALES,password=uvwx
- ▶ To exit the network configuration, press the END key twice. The iTNC restarts automatically.

The parameters **username**, **workgroup** and **password** do not need to be entered in all Windows operating systems.









Settings on a PC with Windows 2000

12.5 Ethernet Interface

Prerequisite:

The network card must already be installed on the PC and ready for operation.

If the PC that you want to connect the iTNC to is already integrated in your company network, then keep the PC's network address and adapt the iTNC'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.

Internet Protocol (TCP/IP) Propertie	s <u>? ×</u>
General	
You can get IP settings assigned autom this capability. Otherwise, you need to a the appropriate IP settings.	atically if your network supports sk your network administrator for
O Obtain an IP address automaticall	y I
□ Use the following IP address: —	
IP address:	160 . 1 . 180 . 1
S <u>u</u> bnet mask:	255.255.0.0
Default gateway:	· · ·
C Obtain DNS server address autom	natically
	Iresses:
Preferred DNS server:	
Alternate DNS server:	· · ·
	Ad <u>v</u> anced
	OK Cancel

Configuring the TNC



Make sure that the person configuring your TNC is a network specialist.

In the Programming and Editing mode of operation, press the MOD key. Enter the keyword NET123. The TNC will then display the main screen for network configuration.

General network settings

Press the DEFINE NET soft key to enter the general network settings and enter the following information:

Setting	Meaning
ADDRESS	Address that your network specialist must assign to the TNC. Input: four numerical values separated by points, e.g. 160.1.180.20
MASK	The SUBNET MASK serves to differentiate between the network ID and the host ID in the network. Input: four numerical values separated by points. Ask your network specialist for the values, e.g. 255.255.0.0
BROADCAST	The broadcast address of the control is required only if it differs from the standard setting. The standard setting is formed from the network ID and the host ID, for which all bits are set to 1, e.g. 160.1.255.255
ROUTER	Internet address of your default router. Enter the Internet address only if your network consists of several parts. Input: four numerical values separated by points. Ask your network specialist for the values, e.g. 160.1.0.2
HOST	Name under which the TNC identifies itself in the network
DOMAIN	Domain name of the control (is not evaluated until later)
NAMESERVER	Network address of the domain server (is not evaluated until later)

You do not need to indicate the protocol with the iTNC 530. It uses the transmission protocol according to RFC 894.





Network settings specific to the device

Press the soft key DEFINE MOUNT to enter the network settings for a specific device. You can define any number of network settings, but you can manage only seven at one time.

Setting	Meaning
MOUNTDEVICE	Connection via NFS: Name of the directory that is to be logged on. This is formed by the network address of the server, a colon and the name of the directory to be mounted. Input: four numerical values separated by points. Ask your network specialist for the values, e.g. 160.1.13.4. Directory of the NFS server that you wish to connect to the TNC. Be sure to differentiate between small and capital letters when entering the path.
	Connection to individual Windows computer: Enter the network name and the share name of the computer, e.g. //PC1791NT/C
MOUNTPOINT	Name that the TNC shows in the file manager for a connected device. Remember that the name must end with a colon.
FILESYSTEM- TYPE	File system type. nfs: Network File System smb: Windows network
OPTIONS for FILESYSTEM- TYPE=nfs	Data without spaces, separated by commas, and written in sequence. Switch between upper and lower case letters. rsize=: Packet size in bytes for data reception. Input range: 512 to 8192 wsize=: Packet size in bytes for data transmission. Input range: 512 to 8192 time0=: Time, in tenths of a second, after which the TNC repeats a Remote Procedure Call. Input range: 0 to 100 000. If there is no entry, the standard value 7 is used. Use higher values only if the TNC must communicate with the server through several routers. Ask your network specialist for the proper value. soft=: Definition of whether the TNC should repeat the Remote Procedure Call until the NFS server answers. "soft" entered: Do not repeat the Remote Procedure Call. "soft" not entered: Always repeat the Remote Procedure Call.



i

Setting	Meaning
OPTIONS for FILESYSTEM- TYPE=smb for direct connection to Windows networks	Data without spaces, separated by commas, and written in sequence. Switch between upper and lower case letters. ip=: ip address of PC to which the TNC is to be connected username=: User name under which the TNC is to log on workgroup=: Work group under which the TNC is to log on password=: Password that the TNC is to use for logon (up to 80 characters)
AM	Definition of whether the TNC upon switch-on should automatically connect with the network drive. 0: Do not automatically connect 1: Connect automatically



The entries **username**, **workgroup** and **password** in the OPTIONS column may not be necessary in Windows 95 and Windows 98 networks.

With the ENCODE PASSWORD soft key, you can encode the password defined under OPTIONS.

Defining a network identification

Press the soft key DEFINE UID / GID to enter the network identification.

Setting	Meaning
TNC USER ID	Definition of the User Identification under which the end user accesses files in the network. Ask your network specialist for the proper value.
OEM USER ID	Definition of the User Identification under which the machine manufacturer accesses files in the network. Ask your network specialist for the proper value.
TNC GROUP ID	Definition of the group identification with which you access files in the network. Ask your network specialist for the proper value. The group identification is the same for end users and machine manufacturers.
UID for mount	Defines the user identification (UID) for the log-on procedure. USER: The user logs on with the USER identification. R00T: The user logs on with the ID of the ROOT user, value = 0.

Test network connection

- ▶ Press the PING soft key.
- ▶ In the **HOST** line, enter then internet address of the computer for which you want to check the network connection.
- Confirm your entry with the ENT key. The TNC transmits data packets until you exit the test monitor by pressing the END key.

In the **TRY** line the TNC shows the number of data packets that were transmitted to the previously defined addressee. Behind the number of transmitted data packets the TNC shows the status:

Status display	Meaning
HOST RESPOND	Data packet was received again, connection is OK.
TIMEOUT	Data packet was not received, check the connection.
CAN NOT ROUTE	Data packet could not be transmitted. Check the Internet address of the server and of the router to the TNC.

Manual operation	Network	configu	uration		
PING MONITOR					~
HOST : 160.1.	13.6			-	~
TRY 25	5 : TIMEOUT				
					4
					s 🖡
					s I

12.6 Configuring PGM MGT

Function

Use the MOD functions to specify which directories or files are to be displayed by the TNC:

- PGM MGT setting: Simple file management (directories are not displayed) or enhanced file management (directories are displayed).
- **Dependent files** setting: Specify whether dependent files are displayed.



Note: see "Standard File Management," page 77, and see "Advanced File Management," page 84.

Changing the PGM MGT setting

- To select the file manager in the Programming and Editing mode of operation, press the PGM MGT key
- ▶ To select the MOD function, press the MOD key.
- ▶ To select the PGM MGT setting: Using the arrow keys, move the highlight onto the PGM MGT setting and use the ENT key to switch between STANDARD and ENHANCED.

Changing the setting for dependent files

In addition to the file extension .H, dependent files also have the extension .SEC.DEP (SECtion DEPendent) or .T.DEP.

The TNC creates files with the **.SEC.DEP** extension if you work with the structure function. The file contains information needed by the TNC to rapidly jump from one structure point to the next.

The TNC creates files with the extension **.T.DEP** as soon as you have run a program in the **Test Run** mode of operation. In this file the TNC stores all tools (tool numbers, tool radii and tool ages) and program calls that occur in the program.

- To select the file manager in the Programming and Editing mode of operation, press the PGM MGT key
- ▶ To select the MOD function, press the MOD key.
- ▶ To select the Dependent files setting: Using the arrow keys, move the highlight onto the **Dependent files** setting and use the ENT key to switch between **AUTOMATIC** and **MANUAL**.



Dependent files are only visible in the file manager if you selected the MANUAL setting.

If dependent files exist for a file, then the TNC displays a + in the status column of the file manager.

12.7 Machine-Specific User Parameters

Function

To enable you to set machine-specific functions, your machine tool builder can define up to 16 machine parameters as user parameters.



This function is not available on all TNCs. Refer to your machine manual.



12.8 Showing the Workpiece in the Working Space

Function

This MOD function enables you to graphically check the position of the workpiece blank 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 WORK SPACE soft key.

The TNC displays a cuboid for the working space. Its dimensions are shown in the "Traverse range" window. The TNC takes the dimensions for the working space from the machine parameters for the active traverse range. Since the traverse range is defined in the reference system of the machine, the datum of the cuboid is also the machine datum. You can see the position of the machine datum in the cuboid by pressing the soft key M91 in the 2nd soft-key row.

Another cuboid represents the blank form. The TNC takes its dimensions from the workpiece blank definition in the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the cuboid. You can see in the cuboid the position of the datum for input by pressing the corresponding soft key in the 2nd soft-key row.

For a test run it normally does not matter where the workpiece blank is located within the working space. However, if you test programs that contain movements with M91 or M92, you must graphically shift the workpiece blank to prevent contour damage. Use the soft keys shown in the table at right.

You can also activate the working-space monitor for the Test Run mode in order to test the program with the current datum and the active traverse ranges (see table below, last line).

Function	Soft key
Move workpiece blank to the left	← ⊕
Move workpiece blank to the right	→
Move workpiece blank forward	¥ 🕀
Move workpiece blank backward	*



Function	Soft key
Move workpiece blank upward	
Move workpiece blank downward	↓ ⊕
Show workpiece blank referenced to the set datum	
Show the entire traversing range referenced to the displayed workpiece blank	←→
Show the machine datum in the working space	M91 💮
Show a position determined by the machine tool builder (e.g. tool change position) in the working space	M92 🕀
Show the workpiece datum in the working space	
Enable (ON) or disable (OFF) working-space monitoring	OFF ON



12.9 Position Display Types

Function

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
Distance remaining to the programmed position; difference between actual and target positions	DIST.
Servo lag: difference between nominal and actual positions (following error)	LAG
Deflection of the measuring touch probe	DEFL.
Traverses that were carried out with handwheel superpositioning (M118) (only Position display 2)	M118

With the MOD function Position display 1, you can select the position display in the status display.

With Position display 2, you can select the position display in the additional status display.



12.10 Unit of Measurement

Function

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/inches 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/inches function to inches. The value is displayed to 4 decimal places.

If you activate 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.



12.11 Select the Programming Language for \$MDI

Function

The Program input MOD function lets you decide whether to program the \$MDI file in HEIDENHAIN conversational dialog or in ISO format.

- To program the \$MDI.H file in conversational dialog, set the Program input function to HEIDENHAIN
- To program the \$MDI.I file according to ISO, set the Program input function to ISO

i

12.12 Selecting the Axes for Generating L Blocks

Function



This function is only available with conversational dialog programming.

The axis selection input field enables you to define the current tool position coordinates that are transferred to an L block. To generate a separate L block, press the ACTUAL-POSITION-CAPTURE soft key. The axes are selected by bit-oriented definition similar to programming the machine parameters:

Axis selection %11111Transfer the X, Y, Z, IV and V axes

Axis selection %01111Transfer the X, Y, Z and IV axes

Axis selection %00111Transfer the X, Y and Z axes

Axis selection %00011 Transfer the X and Y axes

Axis selection %00001Transfer the X axis



12.13 Enter the Axis Traverse Limits, Datum Display

Function

The AXIS LIMIT MOD function allows you to set limits to axis traverse within the machine's actual working envelope.

Possible application: Protecting an indexing fixture against tool collision.

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the TRAVERSE RANGE MOD function. With this function, you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum. If several traverse ranges are possible on your machine, you can set the limits for each range separately using the soft keys TRAVERSE RANGE (1) to TRAVERSE RANGE (3).

Working without additional traverse limits

To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 99 999 mm) as the TRAVERSE RANGE.



Find and enter the maximum traverse

- Set the Position display MOD function to **REF**.
- Move the spindle to the positive and negative end positions of the X, Y and Z axes.
- ▶ Write down the values, including the algebraic sign.
- ▶ To select the MOD functions, press the MOD key.



Enter the limits for axis traverse: Press the TRAVERSE RANGE soft key and enter the values that you wrote down as limits in the corresponding axes

▶ To exit the MOD functions, press the END soft key.



Active tool radius compensations are not taken into account in the axis traverse limit values.

The traverse range limits and software limit switches become active as soon as the reference points are traversed.

Datum display

The values shown at the top right of the screen define the currently active datum. The datum can have been set manually or can have been activated from the preset table. The datum cannot be changed in the screen menu.

	ų	
5		7

The displayed values depend on your machine configuration. Refer to the notes in Chapter 2 (see "Explanation of values saved in the preset table" on page 55).

Manual opera	tion		Programming and editing
Lisits: X- <u>99999</u> V99999 Z99999	X+ +99999 V+ +99999 Z+ +99999	Datum Points: X -02.3050 Y -114.4027 Z +08.2076 A -09908.0902 C -99998.9952 S +0 S +0 Z +0 - +0 - +0 - +0 - +0 - +0 - +0	
POSITION/ TRAVERSE	HELP MACHINE		END



12.14 Displaying HELP Files

Function

Help files can aid you in situations in which you need clear instructions before you can continue (for example, to retract the tool after an interruption of power). The miscellaneous functions may also be explained in a help file. The figure at right shows the screen display of a help file.



HELP files are not provided on every machine. Your machine tool builder can provide you with further information on this feature.

Selecting HELP files

▶ Press the MOD key to select the MOD function.



To select the last active HELP file, press the HELP soft key.

Call the file manager (PGM MGT key) and select a different help file, if necessary.

Progra	amming	and e	ditin⊆	1		Pros and	ramming editing
	CEG.HLP		ne: 0	Column: 1	INSERT	**	7
23 * * * * *	 only	atten for s	tion ! upervi	!! sor	* * * * * * *	* *	
x	X, Y, +, X-, or	Z can Y+, Y handw	be mov -, Z+, heel	z- ke	⊇у		
		10	4% S-0)VR 14	: 37		4
<mark>X</mark> - ₩A	-191.9 +0.0	96 Y 26 Y 20 * C	3% F-0 +253. +0.	0VR LIN 850 Z 000	1IT 1 -345	5.170	s
ACTL.	∲ ∵ 1	T 5	ZS:	S 2612 F	359.9	338 M 5/9	s 🖡
INSERT	MOVE WORD	MOVE WORD <<	PAGE	PAGE	BEGIN		FIND

12.15 Display Operating Times

Function



The machine tool builder can provide further operating time displays. The machine tool manual provides further information.

The MACHINE TIME soft key enables you to show different operating time displays:

Operating time	Meaning
Control ON	Operating time of the control since commissioning
Machine ON	Operating time of the machine tool since commissioning
Program Run	Duration of controlled operation since commissioning

Manual operation	Programming and editing	Programming and editing	
Control on Machine on Program run Spindle time	 1972:58:06 1073:07:30 12:19:04 29:14:15		
Code number		S .	
		EN	D



12.16 Teleservice

Function

The teleservice functions are enabled and adapted by the machine tool builder. The machine tool manual provides further information.

The TNC provides two soft keys for teleservice, making it possible to configure two different service agencies.

The TNC allows you to carry out teleservice. To be able to use this feature, your TNC should be equipped with an Ethernet card which achieves a higher data transfer rate than the serial RS232-C interface.

With the HEIDENHAIN TeleService software, your machine tool builder can then establish a connection to the TNC via an ISDN modem and carry out diagnostics. The following functions are available:

- On-line screen transfer
- Polling of machine states
- Data transfer
- Remote control of the TNC

Calling/exiting teleservice

- Select any machine mode of operation.
- ▶ To select the MOD function, press the MOD key.



- Establish a connection to the service agency: Set the SERVICE or SUPPORT soft key to ON. The TNC breaks the connection automatically if no new data is transferred for a time set by the machine tool builder (default: 15 min).
- ► To break the connection to the service agency: Set the SERVICE or SUPPORT soft key to OFF. The TNC terminates the connection after approx. one minute.



12.17 External Access

Function



The machine tool builder can configure teleservice settings with the LSV-2 interface. The machine tool manual provides further information.

The soft key SERVICE can be used to grant or restrict access through the LSV-2 interface.

With an entry in the configuration file TNC.SYS you can protect a directory and its subdirectories with a password. The password is requested when data from this directory is accessed from the LSV-2 interface. Enter the path and password for external access in the configuration file TNC.SYS.



The TNC.SYS file must be stored in the root directory TNC: $\!$

If you only supply one entry for the password, then the entire drive TNC: $\$ is protected.

You should use the updated versions of the HEIDENHAIN software TNCremo or TNCremoNT to transfer the data.

Entries in TNC.SYS	Meaning
REMOTE.TNCPASSWORD=	Password for LSV-2 access
REMOTE.TNCPRIVATEPATH=	Path to be protected

Example of TNC.SYS

REMOTE.TNCPASSWORD=KR1402

REMOTE.TNCPRIVATEPATH=TNC:\RK

Permitting/Restricting external access

Select any machine mode of operation.

▶ To select the MOD function, press the MOD key.



- Permit a connection to the TNC: Set the EXTERNAL ACCESS soft key to ON. The TNC will then permit data access through the LSV-2 interface. The password is requested when a directory that was entered in the configuration file TNC.SYS is accessed.
 - Block connections to the TNC: Set the EXTERNAL ACCESS soft key to OFF. The TNC will then block access through the LSV-2 interface.


TNC: DUDG		
CHO: (BHB53	30*.*	
Datei-Na	me	P
DOKU_BOHR	PL .A	byte s
MOVE	. П	1070
125852		1276
DREIECK	• 1	22
	.н	90
UNTUR	.н	472 0
REIS1	.н	76
REIS31XY	.н	70
DEL		76
ADPOT	.н	416
	.н	90
MO	. I	22
SWAHL	. PNT	16
Datei(en)	3716000 μ	<byte frei<="" td=""></byte>

Tables and Overviews



13.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements.

Some examples of user parameters are:

- Dialog language
- Interface behavior
- Traversing speeds
- Sequence of machining
- Effect of overrides

Input possibilities for machine parameters

Machine parameters can be programmed as

- Decimal numbers Enter only the number
- Pure binary numbers Enter a percent sign (%) before the number
- Hexadecimal numbers Enter a dollar sign (\$) before the number

Example:

Instead of the decimal number 27 you can also enter the binary number %11011 or the hexadecimal number \$1B.

The individual machine parameters can be entered in the different number systems.

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign.

Selecting general user parameters

General user parameters are selected with code number 123 in the MOD functions.



The MOD functions also include machine-specific user parameters.

External data transfer	
Integrating TNC interfaces EXT1 (5020.0) and EXT2 (5020.1) to an external device	MP5020.x 7 data bits (ASCII code, 8th bit = parity): +0 8 data bits (ASCII code, 9th bit = parity): +1
	Block Check Character (BCC) any: +0 Block Check Character (BCC) control character not permitted: +2
	Transmission stop through RTS active: +4 Transmission stop through RTS inactive: +0
	Transmission stop through DC3 active: +8 Transmission stop through DC3 inactive: +0
	Character parity even: +0 Character parity odd: +16
	Character parity not desired: +0 Character parity desired: +32
	11/2 stop bits: +0 2 stop bit: +64
	1 stop bit: +128 1 stop bit: +192
	Example:
	Use the following setting to adjust the TNC interface EXT2 (MP 5020.1) to an external non-HEIDENHAIN device:
	8 data bits, any BCC, transmission stop through DC3, even character parity, character parity desired, 2 stop bits
	Input for MP 5020.1: 1+0+8+0+32+64 = 105
Interface type for EXT1 (5030.0) and EXT2 (5030.1)	MP5030.x Standard transmission: 0 Interface for blockwise transfer: 1
2 D touch prohos and disitising	
3-D touch probes and digitizing	
Select signal transmission	MP6010 Touch probe with cable transmission: 0 Touch probe with infrared transmission: 1
Probing feed rate for triggering touch probes	MP6120 1 to 3000 [mm/min]
Maximum traverse to first probe point	MP6130 0.001 to 99 999.9999 [mm]
Safety clearance to probing point during automatic measurement	MP6140 0.001 to 99 999.9999 [mm]

Rapid traverse for triggering touch probes

3-D touch probes and digitizing	
Measure center misalignment of the stylus when calibrating a triggering touch probe	 MP6160 No 180° rotation of the 3-D touch probe during calibration: 0 M function for 180° rotation of the touch probe during calibration: 1 to 999
M function for orienting the infrared sensor before each measuring cycle	MP6161 Function inactive: 0 Orientation directly through the NC: -1 M function for orienting the touch probe: 1 to 999
Angle of orientation for the infrared sensor	MP6162 0 to 359.9999 [°]
Difference between the current angle of orientation and the angle of orientation set in MP 6162; when the entered difference is reached, an oriented spindle stop is to be carried out.	MP6163 O to 3.0000 [°]
Automatically orient the infrared sensor before probing to the programmed probing direction	MP6165 Function inactive: 0 Orient infrared sensor: 1
Multiple measurement for programmable probe function	MP6170 1 to 3
Confidence range for multiple measurement	MP6171 0.001 to 0.999 [mm]
Automatic calibration cycle: Center of the calibration ring in the X axis referenced to the machine datum	MP6180.0 (traverse range 1) to MP6180.2 (traverse range3) 0 to 99 999.9999 [mm]
Automatic calibration cycle: Center of the calibration ring in the Y axis referenced to the machine datum	MP6181.0 (traverse range 1) to MP6181.2 (traverse range 3) 0 to 99 999.9999 [mm]
Automatic calibration cycle: Upper edge of the calibration ring in the Z axis referenced to the machine datum	MP6182.0 (traverse range 1) to MP6182.2 (traverse range 3) 0 to 99 999.9999 [mm]
Automatic calibration cycle: Distance below the upper edge of the ring where the calibration is carried out by the TNC	MP6185.0 (traverse range 1) to MP6185.2 (traverse range 3) 0.1 to 99 999.9999 [mm]
Radius measurement with the TT 130 touch probe: Probing direction	MP6505.0 (traverse range 1) to 6505.2 (traverse range 3) Positive probing direction in the angle reference axis (0° axis): 0 Positive probing direction in the +90° axis: 1 Negative probing direction in the angle reference axis (0° axis): 2 Negative probing direction in the +90° axis: 3
Probing feed rate for second measurement with TT 120, stylus shape, corrections in TOOL.T	MP6507 Calculate feed rate for second measurement with TT 130, with constant tolerance: +0 Calculate feed rate for second measurement with TT 130, with variable tolerance: +1 Constant feed rate for second measurement with TT 130: +2

3-D touch probes and digitizing	
Maximum permissible measuring error with TT 130 during measurement with	MP6510.0 0.001 to 0.999 [mm] (recommended input value: 0.005 mm)
rotating tool	MP6510.1
Required for calculating the probing feed rate in connection with MP6570	0.001 to 0.999 [mm] (recommended input value: 0.01 mm)
Feed rate for probing a stationary tool with the TT 130	MP6520 1 to 3000 [mm/min]
Radius measurement with the TT 130: Distance from lower edge of tool to upper edge of stylus	MP6530.0 (traverse range 1) to MP6530.2 (traverse range 3) 0.001 to 99.9999 [mm]
Set-up clearance in the tool axis above the stylus of the TT 130 for pre-positioning	MP6540.0 0.001 to 30 000.000 [mm]
Clearance zone in the machining plane around the stylus of the TT 130 for pre- positioning	MP6540.1 0.001 to 30 000.000 [mm]
Rapid traverse for TT 130 in the probe cycle	MP6550 10 to 10 000 [mm/min]
M function for spindle orientation when measuring individual teeth	MP6560 0 to 999
Measuring rotating tools: Permissible rotational speed at the circumference of the milling tool	MP6570 1.000 to 120.000 [m/min]
Required for calculating rpm and probe feed rate	
Measuring rotating tools: Permissible	MP6572
rotational rpm	0.000 to 1000.000 [rpm]
	if you enter 0, the speed is limited to 1000 rpm

3-D touch probes and digitizing	
Coordinates of the TT 120 stylus center relative to the machine datum	MP6580.0 (traverse range 1) X axis
	MP6580.1 (traverse range 1) Y axis
	MP6580.2 (traverse range 1) Z axis
	MP6581.0 (traverse range 2) X axis
	MP6581.1 (traverse range 2) Y axis
	MP6581.2 (traverse range 2) Z axis
	MP6582.0 (traverse range 3) X axis
	MP6582.1 (traverse range 3) Y axis
	MP6582.2 (traverse range 3) Z axis
Monitoring the position of rotary axes and parallel axes	MP6585 Function inactive: 0 Function active: 1
Defining the rotary axes and parallel axes to be monitored	MP6586.0 Do not monitor the position of the A axis: 0 Monitor the position of the A axis: 1
	MP6586.1 Do not monitor the position of the B axis: 0 Monitor the position of the B axis: 1
	MP6586.2 Do not monitor the position of the C axis: 0 Monitor the position of the C axis: 1
	MP6586.3 Do not monitor the position of the U axis: 0 Monitor the position of the U axis: 1
	MP6586.4 Do not monitor the position of the V axis: 0 Monitor the position of the V axis: 1
	MP6586.5 Do not monitor the position of the W axis: 0 Monitor the position of the W axis: 1

TNC displays, TNC edito	r
Cycles 17, 18 and 207: Oriented spindle stop at beginning of cycle	MP7160 Oriented spindle stop: 0 No oriented spindle stop: 1
	Bit 1 to bit 3: Function
Programming station	MP7210 TNC with machine: 0 TNC as programming station with active PLC: 1 TNC as programming station with inactive PLC: 2
Acknowledgment of POWER INTERRUPTED after switch-on	MP7212 Acknowledge with key: 0 Acknowledge automatically: 1
ISO programming: Set the block number increment	MP7220 0 to 150
Disabling the selection of file types	MP7224.0 All file types selectable via soft key: +0 Disable selection of HEIDENHAIN programs (soft key SHOW .H): +1 Disable selection of ISO programs (soft key SHOW .I): +2 Disable selection of tool tables (soft key SHOW .T): +4 Disable selection of datum tables (soft key SHOW .D): +8 Disable selection of pallet tables (soft key SHOW .P): +16 Disable selection of text files (soft key SHOW .A):+32 Disable selection of point tables (soft key SHOW .PNT): +64
Disabling the editor for certain file types Note: If a particular file type is inhibited, the TNC will erase all files of this type.	MP7224.1 Do not disable editor: +0 Disable editor for HEIDENHAIN programs: +1 ISO programs: +2 Tool tables: +4 Datum tables: +8 Pallet tables: +16 Text files: +32 Point tables: +64
Configure pallet files	MP7226.0 Pallet table inactive: 0 Number of pallets per pallet table: 1 to 255
Configure datum files	MP7226.1 Datum table inactive: 0 Number of datums per datum table: 1 to 255
Program length for program check	MP7229.0 Blocks 100 to 9999
Program length up to which FK blocks are permitted	MP7229.1 Blocks 100 to 9999

TNC displays, TNC edito	r se
Dialog language	MP7230.0 to MP7230.3 English: 0 German: 1 Czech: 2 French: 3 Italian: 4 Spanish: 5 Portuguese: 6 Swedish: 7 Danish: 8 Finnish: 9 Dutch: 10 Polish: 11 Hungarian: 12 Reserved: 13 Russian: 14
Internal clock of the TNC	MP7235 Universal time (Greenwich time): 0 Central European Time (CET): 1 Central European Summer Time: 2 Time difference to universal time: -23 to +23 [hours]
Configure tool tables	MP7260 Inactive: 0 Number of tools generated by the TNC when a new tool table is opened: 1 to 30 000
Configure pocket tables	MP7261.0 (magazine 1) MP7261.1 (magazine 2) MP7261.2 (magazine 3) MP7261.3 (magazine 4) Inactive: 0 Number of pockets in the tool magazine: 1 to 254 If the value 0 is entered in MP7261.1 to MP7261.3, only one tool magazine will be used.
Index tool numbers in order to be able to assign different compensation data to one tool number	MP7262 Do not index: 0 Number of permissible indices: 1 to 9
POCKET TABLE soft key	MP7263 Show the POCKET TABLE soft key in the tool table: 0 Do not show the POCKET TABLE soft key in the tool table: 1

TNC displays, TNC editor

Configure tool table	MP7266.0
(enter 0 to omit from table): Column number	Tool name – NAME: 0 to 32; column width: 16 characters
in the tool table	Tool length – L: 0 to 32; column width: 11 characters
	Tool radius – R: 0 to 32; column width: 11 characters
	Tool radius 2 – R2: 0 to 32; column width: 11 characters
	MP/266.4 Oversize length – DL: 0 to 32: column width: 8 characters
	MP7266.5
	MP7266.6
	Oversize radius 2 – DR2: 0 to 32; column width: 8 characters MP7266.7
	Tool locked – TL: 0 to 32; column width: 2 characters
	Replacement tool – RT: 0 to 32 ; column width: 3 characters
	Maximum tool life – TIME1: 0 to 32; column width: 5 characters
	MP7266.10
	Maximum toor ine for FOOL CALL – TIME2. 0 to 32, column width. 5 characters MP7266.11
	Current tool life – CUR. TIME: 0 to 32; column width: 8 characters
	Tool comment – DOC: 0 to 32; column width: 16 characters
	Number of teeth – CUT.: 0 to 32; column width: 4 characters
	MP7266.14 Tolerance for wear detection in tool length – LTOL: 0 to 32; column width: 6 characters
	MP7266.15
	MP7266.16
	Cutting direction – DIRECT.: 0 to 32; column width: 7 characters MP7266.17
	PLC status – PLC: 0 to 32; column width: 9 characters
	Offset of the tool in the tool axis in addition to MP6530 – TT:L-OFFS: 0 to 32
	MP7266.19
	Offset of the tool between stylus center and tool center – TT:R-OFFS: 0 to 32 column width: 11 characters
	MP7266.20
	Tolerance for break detection in tool length – LBREAK: 0 to 32; column width: 6 characters
	Tolerance for break detection in tool radius – RBREAK: 0 to 32 ; column width: 6 characters
	MP7266.22 Tooth length (Cycle 22) – L CUTS: 0 to 32 : column width: 11 characters
	MP7266.23
	Maximum plunge angle (Cycle 22) – ANGLE.: 0 to 32 ; column width: 7 characters MP7266.24
	Tool type –TYP: 0 to 32; column width: 5 characters MP7266.25
	Tool material – TMAT: 0 to 32; column width: 16 characters
	Cutting data table – CDT: 0 to 32; column width: 16 characters

TNC displays, TNC edito	r
Configure tool table (enter 0 to omit from table); Column number in the tool table	 MP7266.27 PLC value – PLC-VAL: 0 to 32; column width: 11 characters MP7266.28 Center misalignment in reference axis – CAL-OFF1: 0 to 32; column width: 11 characters MP7266.29 Center misalignment in minor axis – CAL-OFF2: 0 to 32; column width: 11 characters MP7266.30 Spindle angle for calibration – CALL-ANG: 0 to 32; column width: 11 characters MP7266.31 Tool type for the pocket table–PTYP: 0 to 32; column width: 2 characters
Configure pocket table; Column number in the pocket table (enter 0 to omit from table)	MP7267.0 Tool number – T: 0 to 18 MP7267.1 Special tool – ST: 0 to 18 MP7267.2 Fixed pocket – F: 0 to 18 MP7267.3 Pocket locked – L: 0 to 18 MP7267.4 PLC status – PLC: 0 to 18 MP7267.5 Tool name from tool table – TNAME: 0 to 18 MP7267.6 Comment from tool table – DOC: 0 to 18
Configure pocket table; Column number in the pocket table when using a box magazine (enter 0 to omit from table)	MP7267.7 to MP7267.17 Evaluated by the PLC: 0 to 18
Manual Operation mode: Display of feed rate	MP7270 Display feed rate F only if an axis direction button is pressed: 0 Display feed rate F even if no axis direction button is pressed (feed rate defined via soft key F or feed rate of the "slowest" axis): 1
Decimal character	MP7280 The decimal character is a comma: 0 The decimal character is a point: 1
Display mode	MP7281.0 Programming and Editing operating mode
	 MP7281.1 Program Run operating modes Always display multiple line blocks completely: 0 Display multiline blocks completely if the multiline block is the active block: 1 Display multiline blocks completely if the multiline block is being edited: 2
Position display in the tool axis	MP7285 Display is referenced to the tool datum: 0 Display in the tool axis is referenced to the tool face: 1

TNC displays, TNC editor	r	รา
Display step for the spindle position	MP7289 0.1°: 0 0.05°: 1 0.01°: 2 0.005°: 3 0.001°: 4 0.0005°: 5 0.0001°: 6	er Paramete
Display step	MP7290.0 (X axis) to MP7290.8 (9th axis) 0.1 mm: 0 0.05 mm: 1 0.01 mm: 2 0.005 mm: 3 0.001 mm: 4 0.0005 mm: 5 0.0001 mm: 6	General Use
Disable datum setting	MP7295 Do not disable datum setting: +0 Disable datum setting in the X axis: +1 Disable datum setting in the Y axis: +2 Disable datum setting in the Z axis: +4 Disable datum setting in the IVth axis: +8 Disable datum setting in the Vth axis: +16 Disable datum setting in the 6th axis: +32 Disable datum setting in the 7th axis: +64 Disable datum setting in the 8th axis: +128 Disable datum setting in the 9th axis: +256	13.1
Disable datum setting with the orange axis keys	MP7296 Do not disable datum setting: 0 Disable datum setting with the orange axis keys: 1	
Reset status display, Q parameters and tool data	MP7300 Reset all when a program is selected: 0 Reset all when a program is selected and with M02, M30, END PGM: 1 Reset only status display and tool data when a program is selected: 2 Reset only status display and tool data when a program is selected and with M02, M30, END PGM: 3 Reset status display and Q parameters when a program is selected: 4 Reset status display and Q parameters when a program is selected and with M02, M30, END PGM: 5 Reset status display when a program is selected: 6 Reset status display when a program is selected and with M02, M30, END PGM: 7	
Graphic display mode	MP7310 Projection in three planes according to ISO 6433, projection method 1: +1 Projection in three planes according to ISO 6433, projection method 2: +1 Do not rotate coordinate for graphic display: +0 Rotate coordinate system for graphic display by 90°: +2 Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the old datum: +0 Display new BLK FORM in Cycle 7 DATUM SHIFT referenced to the new datum: +4 Do not show cursor position during projection in three planes: +0 Show cursor position during projection in three planes: +8	

TNC displays, TNC edite	or
Graphic simulation without programmed tool axis: Tool radius	MP7315 0 to 99 999.9999 [mm]
Graphic simulation without programmed tool axis: Penetration depth	MP7316 0 to 99 999.9999 [mm]
Graphic simulation without programmed tool axis: M function for start	MP7317.0 0 to 88 (0: Function inactive)
Graphic simulation without programmed spindle axis: M function for end	MP7317.1 0 to 88 (0: Function inactive)
Screen saver	MP7392
Enter the time after which the TNC should start the screen saver	0 to 99 [min] (0: Function inactive)
iviachining and program	i run

Effect of Cycle 11 SCALING FACTOR	MP7410 SCALING FACTOR effective in 3 axes: 0 SCALING FACTOR effective in the working plane only: 1
Manage tool data/calibration data	MP7411 Overwrite current tool data by the calibrated data from the 3-D touch probe system: +0 Current tool data are retained: +1 Manage calibrated data in the calibration menu: +0 Manage calibrated data in the tool table: +2

SL cycles	 MP7420 Mill channel around the contour - clockwise for islands and counterclockwise for pockets: +0 Mill channel around the contour - clockwise for pockets and counterclockwise for islands: +1 First mill the channel, then rough out the contour: +0 First rough out the contour, then mill the channel: +2 Combine compensated contours: +0 Complete one process for all infeeds before switching to the other process: +0 Mill channel and rough-out for each infeed depth before continuing to the next depth: +8 The following note applies to the Cycles G56, G57, G58, G59, G121, G122, G123 and G124: At the end of the cycle, move the tool to the position that was last programmed before the cycle call: +0 At the end of the cycle, retract the tool in the tool axis only: +16
Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING: Overlap factor	MP7430 0.1 to 1.414
Permissible deviation of circle radius between circle end point and circle starting point	MP7431 0.0001 to 0.016 [mm]
Operation of various miscellaneous functions M	MP7440 Program stop with M06: +0
Note:	No program stop with M06: +1 No cycle call with M89: +0
The k _V factors for position loop gain are set by the machine tool builder. Refer to your machine manual.	Cycle call with M89: +2 Program stop with M functions: +0 No program stop with M functions: +4 k _V factors cannot be switched through M105 and M106: +0 k _V factors switchable through M105 and M106: +8 Reduce the feed rate in the tool axis with M103 F Function inactive: +0 Reduce the feed rate in the tool axis with M103 F Function active: +16 Exact stop for positioning with rotary axes Not active: +0 Exact stop for positioning with rotary axes Active: +64
Error message during cycle call	MP7441 Error message when M3/M4 not active: 0 Suppress error message when M3/M4 not active: +1 Reserved: +2 Suppress error message when positive depth programmed: +0 Output error message when negative depth programmed: +4
M function for spindle orientation in the fixed cycles	MP7442 Function inactive: 0 Orientation directly through the NC: -1 M function for orienting the spindle: 1 to 999

Machining and program run

Machining and program run	
Maximum contouring speed at feed rate override setting of 100% in the Program Run modes	MP7470 0 to 99 999 [mm/min]
Feed rate for rotary-axis compensation movements	MP7471 0 to 99 999 [mm/min]
Compatibility machine parameters for datum tables	MP7475 Datum shifts are referenced to the workpiece datum: 0 If the value 1 was entered in older TNC controls or in controls with software 340 420-xx, datum shifts were referenced to the machine datum. This function is no longer available. You must now use the preset table instead of datum tables referenced to REF (see "Datum management with the preset table" on page 54).

13.2 Pin Layout and Connecting Cable 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		Adapter block 310 085-01		Connecting cable 365 725-xx					
Male	Assignment	Female	Color	Female	Male	Female	Male	Color	Female
1	Do not assign	1		1	1	1	1	WH/BN	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		Connecting cable 355 484-xx			Adapter block 363 987-02		Connecting cable 366 964-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	WH/GN	8	8	8	8	WH/GN	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 pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device.

This often depends on the unit and type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363 987-02		Connecting cable 366 964-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	WH/GN	7	
9	9	9	Green	9	
Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.	

RS-422/V.11 interface

Only non-HEIDENHAIN devices are connected to the RS-422 interface.



The interface complies with the requirements of EN 50 178 for "low voltage electrical separation."

The pin layouts on the TNC logic unit (X28) and on the adapter block are identical.

TNC		Conne 355 48	ecting cat 34-xx	Adapter block 363 987-01		
Female	Assignment	Male	Color	Female	Male	Female
1	RTS	1	Red	1	1	1
2	DTR	2	Yellow	2	2	2
3	RXD	3	White	3	3	3
4	TXD	4	Brown	4	4	4
5	Signal GND	5	Black	5	5	5
6	CTS	6	Violet	6	6	6
7	DSR	7	Gray	7	7	7
8	RXD	8	WH/GN	8	8	8
9	TXD	9	Green	9	9	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	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	

13.3 Technical Information

Explanation of symbols

Standard

- Axis option
- ○Software option 1
- □Software option 2

User functions	
Description	 Basic version: 3 axes plus spindle 4th NC axis plus auxiliary axis or 8 additional axes or 7 additional axes plus 2nd spindle Digital current and speed control
Programming	HEIDENHAIN conversational and ISO formats
Position entry	 Nominal positions for line segments and arcs in Cartesian or polar coordinates Absolute or incremental dimensions Display and entry in mm or inches Display of the handwheel path during machining with handwheel superimposition
Tool compensation	 Tool radius in the working plane and tool length Calculating the radius-compensated contour up to 99 blocks in advance (M120) Three-dimensional tool-radius compensation for subsequent changing of tool data without having to recalculate the program
Tool tables	Multiple tool tables with any number of tools
Cutting data tables	Cutting data tables for automatic calculation of spindle speed and feed rate from tool- specific data (cutting speed, feed per tooth)
Constant cutting speed	 With respect to the path of the tool center With respect to the cutting edge
Background programming	Create one program with graphical support while another program is running.
3-D machining (software option 2)	 Motion control with minimum jerk 3-D compensation through surface normal vectors Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = Tool Center Point Management) Keeping the tool normal to the contour Tool radius compensation normal to the direction of traverse and the tool direction Spline interpolation
Rotary table machining (software option 1)	 Programming of cylindrical contours as if in two axes Feed rate in length per minute

Contour elements	 Straight line Chamfer Circular path Circle center Circle radius Tangentially connecting circle Corner rounding
Contour approach and departure	 Via straight line: tangential or perpendicular Via circular arc
FK free contour programming	FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	 Subprograms Program section repeat Program as subprogram
Fixed cycles	 Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping Cycles for milling internal and external threads Milling and finishing rectangular and circular pockets Cycles for multipass milling of flat and twisted surfaces Cycles for milling linear and circular slots Linear and circular hole patterns Contour pockets—also with contour-parallel machining Contour train OEM cycles (special cycles developed by the machine tool builder) can also be integrated
Coordinate transformation	 Datum shift, rotation, mirroring Axis-specific scaling Tilting the working plane (software option 1)
O parameters Programming with variables	 Mathematic functions =, +, -, *, /, sin α, cos α, angle α of sin α and cos α, √a² + b² √a Logical comparisons (=, =/, <, >) Calculating with parentheses tan α, arc sin, arc cos, arc tan, aⁿ, eⁿ, ln, log, absolute value of a number, the constant π, negation, truncation of digits before or after the decimal point Functions for calculating circles
Programming support	 Pocket calculator Context-sensitive help function for error messages Graphical support during programming of cycles Comment blocks in the NC program
Actual position capture	Actual positions can be transferred directly into the NC program

rmation
Info
ical
Techn
13.3

Oser functions	
Test Run graphics	Graphic simulation before a program run, even while another program is being run
Display modes	Plan view / projection in 3 planes / 3-D view
	Magnification of details
Interactive Programming graphics	In the Programming and Editing 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
Program Run graphics Display modes	Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	Calculating the machining time in the Test Run mode of operation
	Display of the current machining time in the Program Run modes
Returning to the contour	Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining
	Program interruption, contour departure and reapproach
Datum tables	Several datum tables
Pallet tables	Pallet tables (with as many entries as desired for the selection of pallets, NC programs and datums) can be machined workpiece by workpiece or tool by tool
Touch probe cycles	 Calibrating a touch probe Compensation of workpiece misalignment, manual or automatic Datum setting, manual or automatic Automatic workpiece measurement Cycles for automatic tool measurement
Onesifications	
Components	= MC 422 main computer
components	CC 422 controller unit
	Keyboard
	TFT 10.4-inch or 15.1-inch flat-panel display with soft keys
Program memory	Hard disk with at least 2 GB for NC programs
Input resolution and display	■ To 0.1 µm for linear axes
step	To 0.0001° for angular axes
Input range	Maximum 99 999.999 mm (3937 in.) or 99 999.999°
Interpolation	Line in 4 axes
	□Line in 5 axes (subject to export permit) (software option 1)
	Arc in 2 axes
	Combination of circular and linear motion
	Spline: Execution of splines (3rd degree polynomials)

Specifications	
Block processing time	■ 3.6 ms
compensation	□0.5 ms (software option 2)
Axis feedback control	Position loop resolution: Signal period of the position encoder/1024
	Cycle time of position controller: 1.8 ms
	Cycle time of speed controller: 600 µs
	Cycle time of current controller: minimum 100 µs
Traverse range	Maximum 100 m (3973 inches)
Spindle speed	Maximum 40 000 rpm (with 2 pole pairs)
Error compensation	 Linear and nonlinear axis error, backlash, reversal spikes during circular movements,
Data interfaces	One each RS-232-C /V.24 and RS-422 / V.11 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 100 Base T
	approx. 2 to 5 megabaud (depending on file type and network load)
Ambient temperature	■ Operation: 0 °C to +45 °C (32 °F to 113 °F)
	■ Storage: –30 °C to +70 °C (–22 °F to 158 °F)
Accessories	
Electronic handwheels	One HB 410 : portable bandwheel or
	One HB 130 : panel-mounted handwheel or
	Up to three HR 150: panel-mounted handwheels via HRA 110 handwheel adapter
Touch probes	TS 220: 3-D touch trigger probe with cable connection, or
	TS 632: 3-D touch trigger probe with infrared transmission

TT 130: 3-D touch trigger probe for workpiece measurement

13 Tables and Overviews

Software option 1	
Rotary table machining	 Programming of cylindrical contours as if in two axes Feed rate in length per minute
Coordinate transformations	○Tilting the working plane
Interpolation	OCircle in 3 axes (with tilted working plane)
Software option 2	
3-D machining	 Motion control with minimum jerk 3-D compensation through surface normal vectors Using the electronic handwheel to change the angle of the swivel head during program run without affecting the position of the tool point (TCPM = Tool Center Point Management) Keeping the tool normal to the contour Tool radius compensation normal to the direction of traverse and the tool direction Spline interpolation
Interpolation	□Line in 5 axes (subject to export permit)
Block processing time	□0.5 ms



Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5.4: places before decimal point, places after decimal point) [mm]
Tool numbers	0 to 32 767.9 (5.1)
Tool names	16 characters, enclosed by quotation marks with TOOL CALL. 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/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) [°]
Angle for polar coordinates, rotation, tilting the working plane	-360.0000 to 360.0000 (3.4) [°]
Polar coordinate angle for helical interpolation (CP)	-5400.0000 to +5400.0000 (4.4) [°]
Datum numbers in Cycle 7	0 to 2999 (4.0)
Scaling factor in Cycles 11 and 26	0.000 001 to 99.999 999 (2.6)
Miscellaneous functions M	0 to 999 (1.0)
Q parameter numbers	0 to 399 (1.0)
Q parameter values	-99 999.9999 to +99 999.9999 (5.4)
Labels (LBL) for program jumps	0 to 254 (3.0)
Number of program section repeats REP	1 to 65 534 (5.0)
Error number with Q parameter function FN14	0 to 1099 (4.0)
Spline parameter K	–9.999 999 99 to +9.999 999 99 (1.8)
Exponent for spline parameter	-255 to 255 (3.0)
Surface-normal vectors N and T with 3-D compensation	–9.999 999 99 to +9.999 999 99 (1.8)

13.4 Exchanging the Buffer Battery

A buffer battery supplies the TNC with current to prevent the data in RAM memory from being lost when the TNC is switched off.

If the TNC displays the error message **Exchange buffer battery**, then you must replace the batteries:



To exchange the buffer battery, first switch off the TNC.

The buffer battery must be exchanged only by trained service personnel.

Battery type:1 Lithium battery, type CR 2450N (Renata) ID Nr. 315 878-01

- 1 The buffer battery is at the back of the MC 422 (see 1, figure at right).
- **2** Exchange the battery. The new battery can only be inserted correctly.



13.5 Addresses (ISO)

G functions

Group	G	Function	Blockwise function	Note
Positioning	00	Straight-line interpolation, Cartesian coordinates, rapid traverse		page 173
	01 02 03	Straight-line interpolation, Cartesian coordinates Circular interpolation, Cartesian coordinates, clockwise Circular interpolation, Cartesian coordinates, counterclockwise	■ (with R) ■ (with R)	page 173 page 177 page 177
	05	Circular interpolation, Cartesian coordinates, without indication of direction		page 177
	06	Circular interpolation, Cartesian coordinates, tangential contour approach		page 180
	07 10 11 12 13 15 16	Paraxial positioning block Straight-line interpolation, polar coordinates, rapid traverse Straight-line interpolation, polar coordinates Circular interpolation, polar coordinates, clockwise Circular interpolation, polar coordinates, counterclockwise Circular interpolation, polar coordinates, without indication of direction Circular interpolation, polar coordinates, tangential contour approach		page 186 page 186 page 186 page 186 page 186 page 187
Machining contours, approaching/departing	24 25 26 27	Chamfer with length R Corner rounding with radius R Tangential approach of a contour with R Tangential departure of a contour with R		page 174 page 175 page 170 page 170
Cycles for drilling, tapping and thread milling	83 84 85 200 201 202 203 204 205 206 207 208 209 262 263 264 265 267	Pecking Tapping with a floating tap holder Rigid tapping Thread cutting Drilling Reaming Boring Universal drilling Back boring Universal pecking Tapping with a floating tap holder Rigid tapping Bore milling Tapping with chip breaking Thread milling/countersinking Thread drilling/milling Helical thread drilling/milling Outside thread milling		page 232 page 249 page 255 page 255 page 233 page 235 page 237 page 239 page 241 page 244 page 250 page 253 page 247 page 256 page 260 page 265 page 265 page 269 page 272

Group	G	Function	Blockwise function	Note
Cycles for milling pockets, studs and slots	74 75 76 77 78 210 211 212 213 214 215 251 252 253 254	Slot milling Rectangular pocket milling in clockwise direction Circular path in counterclockwise direction Circular pocket milling in clockwise direction Circular pocket milling in counterclockwise direction Slot milling with reciprocating plunge Round slot with reciprocating plunge Rectangular pocket finishing Rectangular stud finishing Circular pocket finishing Rectangular pocket Circular stud finishing Rectangular pocket Slot milling Circular slot		page 312 page 300 page 300 page 306 page 306 page 314 page 317 page 302 page 304 page 308 page 310 page 283 page 288 page 291 page 295
Cycles for creating point patterns	220 221	Circular pattern Linear pattern		page 325 page 327
Cycles for creating complex contours	37 56 57 58 59 37 120 121 122 123 124 125 127 128	Definition of pocket contour Pilot drilling of the contour pocket (with G37) SLI Rough-out of the contour pocket (with G37) SLI Contour milling in clockwise direction (with G37) SLI Contour milling in counterclockwise direction (with G37) SLI Definition of pocket contour Contour data Pilot drilling (with G37) SLII Reaming (with G37) SLII Floor finishing (with G37) SLII Side finishing (with G37) SLII Contour train (with G37) Cylinder surface (with G37) Cylindrical surface slot (with G37)		page 333 page 334 page 335 page 336 page 336 page 337 page 342 page 343 page 344 page 345 page 346 page 347 page 349 page 351
Cycles for multipass milling	60 230 231	Run 3-D data Multipass milling of plane surfaces Multipass milling of tilted surfaces		page 371 page 372 page 374
Coordinate transformation cycles	28 53 54 72 73 80	Mirror image Datum shift in a datum table Datum shift in program Scaling factor Rotation of the coordinate system Machining plane		page 385 page 381 page 380 page 388 page 387 page 389
Special cycles	04 36 39 62	Dwell time Oriented spindle stop Cycle for program call, program call via G79 Tolerance deviation for fast contour milling		page 396 page 398 page 397 page 399

Group	G	Function	Blockwise function	Note
Cycles for measurement of workpiece misalignment	400 401 402 403 404 405	Basic rotation from two points Basic rotation from two holes Basic rotation from two studs Compensating misalignment with rotary axis Setting a basic rotation directly Compensating misalignment with the C axis		See User's Manual "Touch Probe Cycles"
Cycles for automatic datum setting	410 411 412 413 414 415 416 417 418 419	Datum in center of rectangular pocket Datum in center of rectangular stud Datum in center of circular pocket/hole Datum in center of circular stud Datum in inside corner Datum in outside corner Datum in center of bolt hole circle Datum in the touch probe axis Datum in intersection of two connecting lines each connecting two holes Datum in single axis		See User's Manual "Touch Probe Cycles"
Cycles for automatic workpiece measurement	55 420 421 422 423 424 425 426 427 430 431	Measuring any coordinate in any axis Measuring angles Measuring position and diameter of a circular pocket/hole Measuring position and diameter of a circular stud Measuring position and diameter of a rectangular pocket Measuring position and diameter of a rectangular stud Measuring the slot width Measuring a ridge Measuring any coordinate in any axis Measuring position and diameter of a bolt hole circle Measuring a plane		See User's Manual "Touch Probe Cycles"
Cycles for automatic tool measurement	480 481 482 483	Calibrating the TT Measuring tool length Measuring tool radius Measuring tool length and radius		See User's Manual "Touch Probe Cycles"
Cycles in general	79	Call the cycle		page 224
Selection of the machining plane	17 18 19 20	Plane selection XY, tool axis Z Plane selection ZX, tool axis Y Plane selection YZ, tool axis X Tool axis IV		page 147
Capture of coordinates	29	Transfer the last nominal position value as a pole		page 176
Define the workpiece blank	30 31	Define workpiece blank for graphics, min. point Define workpiece blank for graphics, max. point		page 98
Influencing the program run	38	Program run STOP		

13 Tables and Overviews

Group	G	Function	Blockwise function	Note
	40 41 42 43 44	No tool compensation (R0) Tool radius compensation, to the left of the contour (RL) Tool radius compensation, to the right of the contour (RR) Paraxial compensation, lengthening (R+) Paraxial compensation, shortening (R–)		page 152
Tools	51	Next tool number (in active central tool memory) Tool definition	-	page 148
	99			page 138
Unit of measure	70 71	Unit of measure: inches (set at start of program) Unit of measure: millimeters (set at start of program)		page 99
Dimensions	90 91	Absolute dimensions Incremental dimensions		page 73 page 73
Subprograms	98	Setting a label number		

Assigned addresses

Address	Function
%	Program start or program call
#	Datum number with Cycle G53
A	Rotation about X axis
B	Rotation about Y axis
C	Rotation about Z axis
D	Definition of parameters (program parameters Q)
DL	Length wear compensation with tool call
DR	Radius wear compensation with tool call
E	Tolerance for M112 and M124
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
F	Factor for feed-rate reduction with M103
G	Preparatory function, cycle definition
H	Polar coordinates angle in incremental value/absolute value
H	Rotation angle with G73
H	Tolerance angle for M112
I	Z coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Setting a label number with G98
L	Jump to a label number
L	Tool length with G99

Address	Function
LA	Number of blocks for block scan with M120
М	Miscellaneous Functions
Ν	Block number
P P	Cycle parameters in machining cycles Parameters in parameter definitions
Q	Program parameters/Cycle parameters
R R R R	Polar coordinate radius Circular radius with G02/G03/G05 Rounding radius with G25/G26/G27 Chamfer section with G24 Tool radius with G99
S S	Spindle speed Oriented spindle stop with G36
T T	Tool definition with G99 Tool call
U V W	Linear movement parallel to X axis Linear movement parallel to Y axis Linear movement parallel to Z axis
X Y Z	X axis Y axis Z axis
*	End of block

Parameter functions

Parameter definition	Function	Note
D00	Assign	page 421
D01 D02 D03 D04	Addition Subtraction Multiplication Division	page 421 page 421 page 421 page 421
D05	Root	page 421
D06 D07	Sine Cosine	page 424 page 424
D08	Root sum of squares	page 424
D09 D10 D11 D12	If equal, go to If not equal, go to If greater than, go to If less than, go to	page 426 page 426 page 426 page 426

Parameter definition	Function	Note
D13	Angle from $c \cdot sin a and c \cdot cos a$)	page 424
D14	Error number	page 430
D15	Print	page 432
D19	Transfer of values to the PLC	page 432







iTNC 530 with Windows 2000 (Option)

14.1 Introduction

G

General information

The special features of the iTNC 530 with Windows 2000 are described in this chapter. For the Windows 2000 system functions, please refer to the Windows documentation.

The TNC controls from HEIDENHAIN have always been user-friendly: Simple programming in HEIDENHAIN conversational format, fieldproven cycles, unambiguous function keys and clearly structured graphic functions make them extremely popular shop-floor programmable controls.

The standard Windows operating system is now also provided as a user interface. The new and highly efficient HEIDENHAIN hardware with two processors is the basis for the iTNC 530 with Windows 2000.

The first processor handles real-time jobs and the HEIDENHAIN operating system, whereas the second processor is available only to the standard Windows operating system and thus provides the user access to the world of information technology.

Again, ease of operation has been given top priority:

- A complete PC keyboard with touchpad is integrated in the operating panel.
- The 15-inch high-resolution color flat-panel monitor displays both the iTNC interface and the Windows applications.
- Standard PC equipment such as a mouse or drives can easily be connected to the control through USB interfaces.

Specifications

Specifications	iTNC 530 with Windows 2000
Version	Dual-processor control with
	HEROS real-time operating system for controlling the machine
	Windows 2000 PC operating system as user interface
Memory	Random access memory (RAM)
	64 MB for control applications
	128 MB for Windows applications
	Hard disk
	2.63 GB for TNC files
	 9 GB for Windows files, of which approx. 7.7 GB are available for applications
Data interfaces	Ethernet 10/100 BaseT (up to 100 Mbps depending on network utilization)
	RS-232-C/V.24 (max. 115 200 bps)
	RS-422/V.11 (max. 115 200 bps)
	■ 2 x USB
	■ 2 x PS/2



14.2 Starting an iTNC 530 Application

Logging on to Windows

After you have switched on the power supply, the iTNC 530 starts booting automatically. When the input dialog for logging on to Windows appears, there are two possibilities for logging in:

Logging on as a TNC user

Logging on as a local administrator

Logging on as a TNC user

- ▶ Enter the user name "TNC" in the **User name** input box. Leave the **Password** input box blank and press the OK button.
- ► The TNC software is automatically started. The status message **STARTING, PLEASE WAIT...** appears in the iTNC Control Panel...



Do not open or use any other Windows programs as long as the iTNC Control Panel is displayed (see figure at right). When the iTNC software has successfully started, the Control Panel minimizes itself to a HEIDENHAIN symbol on the task bar.

This user identification permits very limited access to the Windows operating system. You are neither allowed to change the network settings, nor are you allowed to install new software.


Logging on as a local administrator



Please contact your machine tool builder for the user name and the password.

As a local administrator, you are allowed to install software and change the network settings.



HEIDENHAIN does not assist you in installing Windows applications and offers no guarantee for the function of the applications you installed.

HEIDENHAIN is not liable for faulty hard disk contents caused by installing updates to third-party software or additional application software.

If HEIDENHAIN is required to render service after programs or data have been changed, HEIDENHAIN will charge you for the service costs incurred.

In order to guarantee the trouble-free function of the iTNC application, the Windows 2000 system must at all times have sufficient

- CPU performance
- free hard disk memory on the C drive
- RAM
- bandwidth for the hard drive interface

available.

al

By sufficiently buffering the TNC data, the control can compensate brief interruptions (up to one second at a block cycle time of 0.5 ms) to the data transfer from the Windows PC. However, if the data transfer from the Windows PC is interrupted for a longer time period, problems can occur with the feed rate during program run, resulting in damage to the workpiece.

Keep in mind the following requirements for software installations:

The program to be installed may not force the Windows PC to extreme performances (128 MB RAM, 266 MHz clock frequency).

Programs executed under Windows with the priority levels **above normal, high** or **real time** (e.g. games), may not be installed.



14.3 Switching Off the iTNC 530

Fundamentals

To prevent data from being lost at switch-off, you must shut down the iTNC 530 properly. The following sections describe the various possibilities for doing so.



Inappropriate switch-off of the iTNC 530 can lead to data loss.

Exit the iTNC 530 application before exiting Windows.

Logging a user off

You can log a user off of Windows at any time without adversely influencing the iTNC software, However, the iTNC screen is not visible during the log-off process, and you cannot make any entries during this time.



Note that machine-specific keys (such as NC Start or the axis direction keys) remain active.

After a new user has logged on, the iTNC screen reappears.

1

Exiting the iTNC application



Caution!

Before you exit the iTNC application, you absolutely must press the Emergency Stop key. Otherwise you could lose data or the machine could become damaged.

There are two possibilities for exiting the iTNC application:

- Internal exiting via the Manual operating mode; simultaneously exits Windows
- External exiting via the iTNC Control Panel; only exits the iTNC application

Internal exiting via the Manual operating mode

- Select the Manual operating mode
- Shift the soft-key row until the soft key for shutting down the iTNC application appears.



Select the function for shutting down and confirm the following dialog prompt again with the YES soft key.

When the message It is now safe to turn off your computer. appears on the iTNC screen, you may interrupt the power supply to the iTNC 530.

External exiting via the iTNC Control Panel

- Press the Windows key on the ASCII keyboard to minimize the iTNC application and display the Task Bar.
- Double-click the green HEIDENHAIN symbol to the lower right in the Task Bar for the iTNC Control Panel to appear (see figure at upper right).
- Stop iTNC

Select the function for exiting the iTNC 530 application: Press the Stop iTNC button.

After you have pressed the Emergency Stop button, acknowledge the iTNC message with the **Yes** screen button. The iTNC application is stopped.

▶ The iTNC Control Panel remains active. To restart the iTNC 530, press the **Restart iTNC** button.

To exit Windows, select

- the Start button
- ▶ the menu item **Shut down...**
- > again the menu item **Shut down...**
- ▶ and confirm with OK





Shutting down Windows

If you try to shut down Windows while the iTNC software is still active, the control displays a warning (see figure at top right).



Caution!

Before you confirm with OK, you absolutely must press the Emergency Stop button. Otherwise you could lose data or the machine could become damaged.

If you confirm with OK, the iTNC software is exited and Windows is shut down.



Caution!

After several seconds Windows displays its own warning, covering the iTNC warning (see figure at center right). Never confirm the warning with End Now, since you could lose data or the machine could become damaged.



i

14.4 Network Settings

Prerequisite

ф,

You must log on as a local administrator to be able to change the network settings. Please contact your machine tool builder for the required user name and password.

The network should be configured only by network specialists.

Adjusting the network settings

The iTNC 530 is shipped with two network connections: The **Local Area Connection** and the **iTNC Internal Connection** (see figure at right).

The **Local Area Connection** is the iTNC's connection to its network. You may adjust all familiar Windows 2000 settings to your network (also see the Windows 2000 network description).

The **iTNC Internal Connection** is exactly that: an internal iTNC connection. The configuration of this connection must not be changed. Changes might cause the iTNC to stop functioning.

This internal network address has a default setting of **192.168.254.253** and may not collide with your company network, meaning that the subnet **192.168.254.xxx** cannot already exist in your network.

The option **Obtain IP address automatically** must be inactive.





Controlling access

Administrators have access to the TNC drives D, E and F. Please note that some of the data in these partitions is binary encoded, and write-accesses might lead to undefined behavior of the iTNC.

The partitions D, E and F have access rights for the user groups **SYSTEM** and **Administrators.** The group **SYSTEM** ensures that the Windows service that starts the control has access. The group **Administrators** ensures that the real-time processor of the iTNC receives a network connection via the **iTNC Internal Connection**.



You may not restrict access by these groups, nor may you add groups and prohibit certain accesses by these groups (in Windows, access restrictions have priority over access rights).

1

14.5 Specifics About File Management

The iTNC drive

When you call the iTNC file manager, the left window shows all available drives. For example:

- **C:**\: Windows partition of the built-in hard disk
- **RS232:**\: Serial interface 1
- **RS422:**\: Serial interface 2

al,

TNC:\: Data partition of the iTNC

There might also be other networks available if you have connected them with Windows Explorer.

Please note that the data drive of the iTNC appears in the file manager with the name **TNC:** \. In Windows Explorer, this drive (partition) appears with the letter **D**.

Subdirectories on the TNC drive (e.g. **RECYCLER** and **System Volume Identifier**) are created by Windows 2000 and may not be deleted.

If you connect a new network drive with Windows Explorer, you may have to update the iTNC's display of available drives:

- ▶ To call the file manager, press the PGM MGT key.
- Move the highlight to the drive window at left.
- Switch to the second level of the soft-key row.
- ▶ To update the drive overview, press the UPDATE TREE soft key.

Manual operation	Prog File	rammi name	ng and = <mark>7</mark> 327	edi .H	ting		
;		TNC:\3D-DU	FTER*.*		2		-
∌ 🐙 I:∖ ∌ 📾 RS232:∖ ∌ 📾 RS422:∖	1	3dtorus	.н	Bytes S 518	+ 02-10-	Time 2002 11:08:14	
- C 3D-DUFTER	ogramme 1-68198 27	7327	.н	2602 3860	E + 02-10- + 02-10-	2002 11:08:10 2002 11:08:10	
- C Grosse_Pro		BRERX1_8	.н	481K	+ 0Z-10-	2002 11:08:20	
RECYCLER		BOHRXYZ	.н	1006	02-10-	2002 11:08:20	Profib
- 🗁 Stefan		FISCH	.н	2410K	+ 02-10-	2002 11:08:30	s
L 📩 System Vol	ume Infd	KEGEL	.н	376	+ 02-10-	2002 11:08:38	-
		M114_128	.н	600	02-10-	2002 11:08:38	: 🦽
		M128_0	.н	490	02-10-	2002 11:08:38	
		ROHRKR	.н	3162	+ 02-10-	2002 11:08:40	
		SCHRAUB1	.н	692	02-10-	2002 11:08:40	
		13 file(s) 2669504 kt	oyte vaca	nt		s .
PAGE P	AGE	SELECT	COPY	SELECT	WINDO	J LAST	



Data transfer to the iTNC 530



ф

Before you can initiate a data transfer on the iTNC, the network drive must have been connected with Windows Explorer. Access to UNC network names (e.g. \\PC0815\DIR1) is not possible.

TNC-specific files

After integrating the iTNC 530 into your network, you can access any computer and transfer files to it from the iTNC. However, certain file types may only be transferred if the data transfer was initiated by the iTNC. The reason is that these files must be converted into binary format during the data transfer to the iTNC.



Simply copying the file types listed below to the D drive using Windows Explorer is both prohibited and useless.

File types that may not be copied using Windows Explorer:

- Conversational dialog programs (extension .H)
- ISO programs (extension .I)
- Tool tables (extension .T)
- Pocket tables (extension .TCH)
- Pallet tables (extension .P)
- Datum tables (extension .D)
- Point tables (extension .PNT)
- Cutting data tables (extension .CDT)
- Freely definable tables (extension .TAB)

Procedure for data transfer: (see "Data transfer to or from an external data medium" on page 94).

ASCII files

There are no limitations regarding the direct copying of ASCII files (files with the extension .A) with Windows Explorer.



Please note that all the files you want to use on the TNC must be stored on drive $\mathsf{D}.$

SYMBOLE

3-D compensationPeripheral milling ... 1543-D view ... 453

Α

Accessories ... 43 Actual position capture ... 102, 173 Adding Comments ... 112 Approach contour ... 168 ASCII files ... 113 Automatic cutting data calculation ... 141, 155 Automatic Program Start ... 469 Automatic tool measurement ... 140 Auxiliary axes ... 71

В

Back boring ... 241 Block scan ... 466 Blocks Deleting ... 103 Inserting, editing ... 104 Bolt hole circle ... 325 Bore milling ... 247 Boring ... 237 Buffer battery, exchanging ... 529

С

Calculating with parentheses ... 433 Chamfer ... 174 Changing the block number increment ... 106 Changing the spindle speed ... 51 Circle center ... 176 Circular path ... 177, 178, 180, 186, 187 Circular pocket Finishing ... 308 Roughing ... 306 Roughing+finishing ... 288 Circular slot Roughing+finishing ... 295 Circular slot milling ... 317 Circular stud finishing ... 310 Code numbers ... 477

С

Constant contouring speed: M90 ... 199 Contour train ... 347 Conversational format ... 101 Coordinate transformation ... 379 Copying program sections ... 105 Corner rounding ... 175 Cutting data calculation ... 155 Cutting data table ... 155 Cycle Calling ... 224 Defining ... 222 Groups ... 223 Cycles and point tables ... 228 Cylinder ... 444 Cylinder surface ... 349, 351

D

Data backup ... 76 Data interface Assigning ... 479 Pin layout ... 519 Setting ... 478 Data transfer rate ... 478 Data transfer software ... 480 Datum management ... 54 Datum setting ... 52 Without a 3-D touch probe ... 52 Datum shift With datum tables ... 381 Within the program ... 380 Deepened starting point for drilling ... 246 Define the blank ... 99 Depart contour ... 168 Dependent files ... 490 Dialog ... 101 Directory ... 84, 88 Copying ... 90 Creating ... 88 Deleting ... 91 Drilling ... 233, 239, 244 Deepened starting point ... 246 Drilling Cycles ... 230 Dwell time ... 396

Ε

Ellipse ... 442 Enter the desired spindle speed, ... 147 Error messages ... 118 Help with ... 118 Outputting ... 430 Ethernet Interface Ethernet interface Configuring ... 485 Connecting and disconnecting network drives ... 97 Connection possibilities ... 482 Introduction ... 482 External Access ... 503 External data transfer iTNC 530 with Windows 2000 ... 547

F

Feed rate ... 51 Changing ... 51 For rotary axes, M116 ... 210 Feed rate factor for plunging movements: M103 ... 202 Feed rate in millimeters per spindle revolution: M136 ... 203 File Management File management Advanced ... 84 Overview ... 85 Calling ... 77, 86 Configuring with MOD ... 489 Copying a file ... 79, 89 Copying a table ... 90 Deleting a file ... 78, 91 Dependent files ... 490 Directories ... 84 Copying ... 90 Creating ... 88 External data transfer ... 80, 94 File name ... 75 File protection ... 83, 93 File type ... 75 Overwriting files ... 96 Renaming a file ... 82, 93 Selecting a file ... 78, 87 Standard 77 Tagging files ... 92

Index

File status ... 77, 86 Floor finishing ... 345 FN xx: See Q parameter programming Full circle ... 177 Fundamentals ... 70

G

F

Graphic simulation ... 456 Graphics Display modes ... 450 During programming ... 109 Magnifying a detail ... 110 Magnifying details ... 454

Η

Hard disk ... 75 Helical interpolation ... 187 Helical thread drilling/milling ... 269 Helix ... 187 Help files, displaying ... 500 Help with error messages ... 118 Hole patterns Circular ... 325 Linear ... 327 Overview ... 323

I

Indexed tools ... 143 Information on formats ... 528 Interrupt machining. ... 463 iTNC 530 ... 32 with Windows 2000 ... 538

К

Keyboard ... 35

L

Laser cutting machines, miscellaneous functions ... 218 L-block generation ... 497 Look-ahead ... 205

Μ

M functions: See Miscellaneous functions Machine parameters For 3-D touch probes ... 507 For external data transfer ... 507 For machining and program run ... 516 For TNC displays and TNC editor ... 511 Machine-referenced coordinates: M91. M92 ... 196 Measuring the machining time ... 457 Milling an inside thread ... 260 Mirror image ... 385 **Miscellaneous Functions** entering ... 194 For contouring behavior ... 199 For coordinate data ... 196 For laser cutting machines ... 218 for program run control ... 195 For rotary axes ... 210 For spindle and coolant ... 195 **MOD** Function MOD function Exiting ... 474 Overview ... 474 Select ... 474 Modes of Operation ... 36 Moving the machine axes In increments ... 50 With the electronic handwheel ... 49 With the machine axis direction buttons ... 48

Ν

NC error messages ... 118 Nesting ... 407 Network connection ... 97 Network connection, testing ... 488 Network settings ... 485 iTNC 530 with Windows 2000 ... 545

0

Oblong hole milling ... 314 Open contours: M98 ... 202 Operating time ... 501 Option number ... 476 Oriented spindle stop ... 398

Ρ

Pallet table Entering coordinates ... 119, 124 executing ... 121, 133 Function ... 119, 123 Selecting and leaving ... 121, 128 Parametric programming: See Q parameter programming Part families ... 420 Path ... 84 Path contours Cartesian coordinates Circular arc with tangential connection ... 180 Circular path around circle center CC ... 177 Circular path with defined radius ... 178 Overview ... 172, 185 Straight line ... 173 Polar coordinates Circular arc with tangential connection ... 187 Circular path around pole CC ... 186 Straight line ... 186 Path functions Fundamentals ... 164 Circles and circular arcs ... 166 Pre-position ... 167 Pecking ... 232, 244 Deepened starting point ... 246 Pin layout for data interfaces ... 519 Ping ... 488 Plan view ... 451 Pocket calculator ... 117 Pocket table ... 145 Point tables ... 226 Polar coordinates Fundamentals ... 72 Programming ... 185 Positionina With a tilted working plane ... 198, 217 with manual data input (MDI) ... 64 Preset table ... 54 Principal axes ... 71 Probing Cycles: See "Touch Probe Cycles" User's Manual

Ρ

Program Editing ... 103 Open new ... 99 Structure ... 98 Structuring ... 111 Program call Program as subprogram ... 405 Via cycle ... 397 Program management. See File management Program name: See File Management, File name Program Run Block scan ... 466 Executing ... 462 Interrupting ... 463 Optional block skip ... 470 Overview ... 461 Resuming after an interruption ... 465 Program run Program section repeat ... 404 Program sections, copying ... 105 Programming tool movements ... 101 Projection in 3 planes ... 452

Q

Q parameters Checking ... 428 Preassigned ... 437 Transferring values to the PLC ... 432 Unformatted output ... 432 Q-parameter programming ... 418 Additional functions ... 429 Basic arithmetic (assign, add, subtract, multiply, divide, square root) ... 421 If/then decisions ... 426 Programming notes ... 418 Trigonometric functions ... 424

R

Radius compensation ... 151 Input ... 152 Outside corners, inside corners ... 153 Reaming ... 235 Rectangular pocket Rectangular pockets Finishing process ... 302 Roughing process ... 300 Roughing+finishing ... 283 Rectangular stud finishing ... 304 Reference system ... 71 Replacing texts ... 108 Retraction from the contour ... 207 Returning to the contour ... 468 Rotary axis Reducing display: M94 ... 212 Shorter-path traverse: M126 ... 211 Rotation ... 387 Rough out: See SL Cycles: Rough-out Ruled surface ... 374 Run 3-D data ... 371

S

Scaling factor ... 388 Screen layout ... 34 Search function ... 107 Select the unit of measure ... 99 Setting the BAUD rate ... 478 Setting the datum ... 74 Side finishing ... 346 SL Cycles SL cycles Contour data ... 342 Contour geometry cycle ... 333, 339 Contour train ... 347 Floor finishing ... 345 Fundamentals ... 331, 337, 362 Overlapping contours ... 339, 364 Pilot drilling ... 334, 336, 343 Rough-out ... 335, 344 Side finishing ... 346 SL Cycles with Contour Formula Slot milling ... 312 Reciprocating ... 314 Roughing+finishing ... 291 Software number ... 476 Software options ... 527 Specifications ... 523 iTNC 530 with Windows 2000 ... 539

S

Sphere ... 446 Status display ... 39 Additional ... 40 General ... 39 Straight line ... 173, 186 Structuring programs ... 111 Subprogram ... 403 Superimposing handwheel positioning: M118 ... 206 Switch between upper and lower case letters ... 114 Switch-off ... 47

Т

Tapping With a floating tap holder ... 249, 250 Without a floating tap holder ... 252, 253, 256 Teleservice ... 502 Test Run Executing ... 459 Overview ... 458 Up to a certain block ... 460 Text files Delete functions ... 115 Editing functions ... 114 Finding text sections ... 116 Opening and exiting ... 113 Thread cutting ... 255 Thread drilling/milling ... 265 Thread milling, fundamentals ... 258 Thread milling, outside ... 272 Thread milling/countersinking ... 262 Tilted axes ... 213, 214 Tilting the working plane ... 59, 389 Cycle ... 389 Guide ... 392 Manually ... 59 TNCremo ... 480 TNCremoNT ... 480 Tool change ... 148 **Tool Compensation** Tool compensation Length ... 150 Radius ... 151 Tool Data

Index

Т

Tool data Calling ... 147 Delta values ... 138 Enter them into the program ... 138 Entering into tables ... 139 Indexing ... 143 Tool length ... 137 Tool material ... 141, 157 Tool measurement ... 140 Tool name ... 137 Tool number ... 137 Tool radius ... 138 Tool table Editing functions ... 142 Editing, exiting ... 142 Input possibilities ... 139 Tool type, selecting ... 141 Touch probe monitoring ... 208 Trigonometric functions ... 424 Trigonometry ... 424

U

Universal drilling ... 239, 244 USB interface ... 538 User parameters ... 506 General For 3-D touch probes and digitizing ... 507 For external data transfer ... 507 For machining and program run ... 516 For TNC displays, TNC editor ... 511 Machine-specific ... 491

V

Visual display unit ... 33

W

Windows 2000 ... 538 Windows, logging on ... 540 WMAT.TAB ... 156 Workpiece material, defining ... 156 Workpiece positions Absolute ... 73 Incremental ... 73 Workspace monitoring ... 459, 492

Table of Miscellaneous Functions

Μ	Effect Effective at block	start	end	Page
M00	Stop program/Spindle STOP/Coolant OFF			page 195
M01	Optional program STOP			page 471
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1			page 195
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP			page 195
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP			page 195
M08 M09	Coolant ON Coolant OFF			page 195
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON			page 195
M30	Same function as M02			page 195
M89	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameter)			page 224
M90	Only in lag mode: Constant contouring speed at corners			page 199
M91	Within the positioning block: Coordinates are referenced to machine datum			page 196
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position			page 196
M94	Reduce display of rotary axis to value under 360°			page 212
M97	Machine small contour steps			page 201
M98	Machine open contours completely			page 202
M99	Blockwise cycle call		-	page 224

Μ	Effect Effective at block	start	end	Page
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Reset M101		-	page 148
M103	Reduce feed rate during plunging to factor F (percentage)			page 202
M104	Reactivate the datum as last defined			page 198
M105 M106	Machining with second kv factor Machining with first kv factor			page 517
M107 M108	Suppress error message for replacement tools Reset M107	-		page 148
M109	Constant contouring speed at tool cutting edge			page 204
M110	Constant contouring speed at tool cutting edge			
M111	Reset M109/M110			
M114 M115	Automatic compensation of machine geometry when working with tilted axes Reset M114			page 213
M116 M117	Feed rate for angular axes in mm/min Reset M116			page 210
M118	Superimpose handwheel positioning during program run			page 206
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)			page 205
M124	Do not include points when executing non-compensated line blocks			page 200
M126 M127	Shortest-path traverse of rotary axes Reset M126			page 211
M128 M129	Maintain the position of the tool tip when positioning with tilted axes (TCPM) Reset M128			page 214
M130	Moving to position in an untilted coordinate system with a tilted working plane			page 198
M134 M135	Exact stop at nontangential contour transitions when positioning with rotary axes Reset M134	-		page 216
M136 M137	Feed rate F in millimeters per spindle revolution Reset M136			page 203
M138	Select tilting axes			page 216
M142	Delete modal program information			page 209
M143	Delete basic rotation			page 209

ISO Function Overview

iTNC 530

M functions Stop program/Spindle STOP/Coolant OFF M00 M01 Optional program STOP Stop program run/Spindle STOP/Coolant OFF/Clear M02 status display (depending on machine parameter)/Go to block 1 M03 Spindle ON clockwise M04 Spindle ON counterclockwise M05 Spindle STOP M06 Tool change/Stop program run (depending on machine parameter)/Spindle STOP M08 Coolant ON M09 Coolant OFF M13 Spindle ON clockwise/Coolant ON M14 Spindle ON counterclockwise/Coolant ON M30 Same function as M02 M89 Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameter) M90 Only in lag mode: Constant contouring speed at corners M99 Blockwise cycle call M91 Within the positioning block: Coordinates are referenced to machine datum M92 Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position M94 Reduce display of rotary axis to value under 360° M97 Machine small contour steps M98 Machine open contours completely M101 Automatic tool change with replacement tool if maximum tool life has expired M102 Reset M101 M103 Reduce feed rate during plunging to factor F (percentage) M104 Reactivate the datum as last defined M105 Machining with second kv factor M106 Machining with first kv factor M107 Suppress error message for replacement tools M108 Reset M107 M109 Constant contouring speed at tool cutting edge (increase and decrease feed rate) M110 Constant contouring speed at tool cutting edge (feed rate decrease only)

M111 Reset M109/M110

M functions

- M114 Automatic compensation of machine geometry when working with tilted axes:
- M115 Reset M114
- M116 Feed rate for angular axes in mm/min M117 Reset M116
- M118 Superimpose handwheel positioning during program run
- M120 Pre-calculate radius-compensated contour (LOOK AHEAD)
- M124 Do not include points when executing noncompensated line blocks
- M126 Shortest-path traverse of rotary axes M127 Reset M126
- M128 Maintain the position of the tool tip when positioning with tilted axes (TCPM)
 M129 Reset M128
- M130 Moving to position in an untilted coordinate system with a tilted working plane
- M134 Exact stop at nontangential contour transitions when positioning with rotary axes M135 Reset M134
- M136 Feed rate F in millimeters per spindle revolution M137 Reset M136
- M138 Select tilting axes
- M142 Delete modal program information
- M143 Delete basic rotation
- M144 Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block M145 Reset M144
- M200 Laser cutting: Output programmed voltage directlyM201 Laser cutting: Output voltage as a function of distance
- M202 Laser cutting: Output voltage as a function of speed M203 Laser cutting: Output voltage as a function of time
- (ramp) M204 Laser cutting: Output voltage as a function of time
 - (pulse)

G functions

Tool Movements

- G00 Straight-line interpolation, Cartesian coordinates,
- G01 rapid traverse
- G02 Straight-line interpolation, Cartesian coordinates
- G03 Circular interpolation, Cartesian coordinates, clockwise Circular interpolation, Cartesian coordinates, counterclockwise
- G05 Circular interpolation, Cartesian coordinates, without indication of direction
- G06 Circular interpolation, Cartesian coordinates, tangential contour connection
- G07* Paraxial positioning block
- G10 Straight-line interpolation, polar coordinates, rapid traverse
- G11 Straight-line interpolation, polar coordinates
- G12 Circular interpolation, polar coordinates, clockwise
- G13 Circular interpolation, polar coordinates, counterclockwise
- G15 Circular interpolation, polar coordinates, without indication of direction
- G16 Circular interpolation, polar coordinates, tangential contour connection

Chamfer/Rounding/Approach contour/Depart contour

- G24* Chamfer with length R
- G25* Corner rounding with radius R
- G26* Tangential contour approach with tool radius R
- G27* Tangential contour departure with tool radius R

Define the tool

G99* With tool number T, length L, radius R

Tool radius compensation

- G40 No tool radius compensation
- G41 Tool radius compensation, left of the contour
- G42 Tool radius compensation, right of the contour
- G43 Paraxial compensation for G07, lengthening
- G44 Paraxial compensation for G07, shortening

Blank form definition for graphics

G30 (G17/G18/G19) min. point G31 (G90/G91) max. point

Cycles for drilling, tapping and thread milling

- G83 Pecking
- G84 Tapping with a floating tap holder
- G85 Rigid tapping
- G86 Thread cutting
- G200 Drilling
- G201 Reaming
- G202 Boring
- G203 Universal drilling
- G204 Back boring
- G205 Universal pecking
- G206 Tapping with a floating tap holder
- G207 Rigid tapping G208 Bore milling
- G209 Tapping with chip breaking

G functions

Cycles for drilling, tapping and thread milling

- G262 Thread milling
- G263 Thread milling/countersinking
- G264 Thread drilling/milling
- G265 Helical thread drilling/milling
- G267 External thread milling

Cycles for milling pockets, studs and slots

- G74 Slot milling
- G75 Rectangular pocket milling in clockwise direction
- G76 Rectangular pocket milling in counterclockwise direction
- G77 Circular pocket milling in clockwise direction
- G78 Circular pocket milling in counterclockwise direction
- G210 Slot milling with reciprocating plunge
- G211 Round slot with reciprocating plunge
- G212 Rectangular pocket finishing
- G213 Rectangular stud finishing
- G214 Circular pocket finishing
- G215 Circular stud finishing

Cycles for creating point patterns

- G220 Circular pattern
- G221 Linear pattern

SL Cycles, group 1

- G37 Contour geometry, list of subcontour program numbers
 G56 Pilot drilling
- G57 Rough-out
- G58 Contour milling in clockwise direction (finishing)
- G59 Contour milling in clockwise direction (mishing)

SL Cycles, group 2

- G37 Contour geometry, list of subcontour program numbers
- G120 Contour data (applies to G121 to G124)
- G121 Pilot drilling
- G122 Rough-out
- G123 Floor finishing
- G124 Side finishing
- G125 Contour train (machining open contour)
- G127 Cylinder surface
- G128 Cylindrical surface slot

Coordinate transformations

- G53 Datum shift in datum table
- G54 Datum shift in program
- G28 Mirror image
- G73 Rotation of the coordinate system
- G72 Scaling factor (reduce or enlarge contour)
- G80 Tilting the Working Plane
- G247 Datum setting

Cycles for multipass milling

- G60 Run 3-D data
- G230 Multipass milling of plane surfaces
- G231 Multipass milling of tilted surfaces

*) Non-modal function

Touch probe cycles for measuring workpiece		Unit of measure			
misalignment		G70	Inches (set at start of program)		
G400 G401	Basic rotation from two points Basic rotation from two holes Basic rotation from two studs Compensate a basic rotation via a rotary axis		Millimeters (set at start of program)		
G402			Other G functions		
G403			Transfer the last nominal position value as a pole		
G404 G405	Set basic rotation	G38	(circle center)		
- ·		G51^	Program run STOP Next tool number (with central tool file)		
louch	Touch probe cycles for datum setting		Cycle call		
G410	Datum from inside of rectangle	G98*	Set label number		
G411 G412	Datum from outside of rectangle	*) NI	and all formation		
G413	Datum from outside of circle) Not-modal function			
G414 G415	Datum in outside corner	Addre	esses		
G415 G416	Datum circle center	%	Start of program		
G417	Datum in touch probe axis	%	Program call		
G418	Datum in center of 4 holes		Datum number with G53		
Touch	probe cycles for automatic tool measurement	^	Pototion shout V avia		
G55	Measure any coordinate	B	Rotation about Y axis		
G420	Measure any angle		Rotation about Z axis		
G421 G422	Measure cylindrical stud Measure rectangular pocket	D	Q-parameter definitions		
G423			Learning and the second s		
G424 G425	Measure rectangular stud Measure slot	DL DR	Radius wear compensation with T		
G426	Measure ridge				
G427	Measure any coordinate	E	Tolerance with MTTZ and MTZ4		
G431	Measure any plane	F	Feed rate		
Touch	Probe Cycles for Automatic Tool Measurement	F	Scaling factor with G72		
G480	Calibrating the TT	F	Factor for feed-rate reduction F with M103		
G481	Measure tool length	G	G functions		
G482	Measure tool radius	Ц	Polar coordinato anglo		
6465		H	Rotation angle with G73		
Special	l cycles	Н	Tolerance angle with M112		
G04* G36	Dwell time with F seconds Oriented spindle stop Program call Tolerance deviation for fast contour milling Measure axis shift	I	Z coordinate of the circle center/pole		
G39* G62		J	Y coordinate of the circle center/pole		
G440		K	Z coordinate of the circle center/pole		
Define	machining plane	L	Setting a label number with G98		
G17	Working plane: X/Y; tool axis: Z	L	Jump to a label number Tool length with G99		
G18 G19	Working plane: Z/X; tool axis: Y Working plane: Y/Z; tool axis: X Tool axis IV				
G20			M functions		
Dimens	sions	Ν	Block number		
G90 G91	Absolute dimensions Incremental dimensions	P P	Cycle parameters in machining cycles Value or Q parameter in Q-parameter definition		
			Q parameter		
		R	Polar coordinate radius		
		R	Circular radius with G02/G03/G05		
		К R	Rounding radius with G25/G26/G27 Tool radius with G99		

i

G function

Addresses

S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
T	Next tool with G51
U	Axis parallel to X axis
V	Axis parallel to Y axis
W	Axis parallel to Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

Contour cycles

Sequence of program steps for m with several tools	achining
List of subcontour programs	G37 P01
Define contour data	G120 Q1
Define/Call drill Contour cycle: pilot drilling Cycle call	G121 Q10
Define/Call roughing mill Contour cycle: rough-out Cycle call	G122 Q10
Define/Call finishing mill Contour cycle: floor finishing Cycle call	G123 Q11
Define/Call finishing mill Contour cycle: side finishing Cycle call	G124 Q11
End of main program, return	M02
Contour subprograms	G98 G98 L0

Radius compensation of the contour subprograms

Contour	Programming sequence of the contour elements	Radius compens.
Inside	Clockwise (CW)	G42 (RR)
(pocket)	Counterclockwise (CCW)	G41 (RL)
Outside	Clockwise (CW)	G41 (RL)
(island)	Counterclockwise (CCW)	G42 (RR)

Coordinate transformations

Coordinate transformation	Activate	Cancel
Datum shift	G54 X+20 Y+30 Z+10	G54 X0 Y0 Z0
Mirror image	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F 0.8	G72 F1
Machining plane	G80 A+10 B+10 C+15	G80

Q-parameter definitions

D	Function
00	Assign
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Root
06	Sine
07	Cosine
08	Root sum of squares $c = \sqrt{a^2 + b^2}$
09	If equal, go to label number
10	If not equal, go to label number
11	If greater than, go to label number
12	If less than, go to label number
13	Angle from $c \cdot sin a$ and $c \cdot cos a$)
14	Error number
15	Print
19	Assignment PLC

1

HEIDENHAIN

 DR. JOHANNES HEIDENHAIN GmbH

 Dr.-Johannes-Heidenhain-Straße 5

 83301 Traunreut, Germany

 [®] +49 (8669) 31-0

 ^{EXX} +49 (8669) 5061

 e-mail: info@heidenhain.de

 Technical support

 ^{EXX} +49 (8669) 31-1000

 e-mail: service@heidenhain.de

 Measuring systems

 [#] +49 (8669) 31-3104

 e-mail: service.ms-support@heidenhain.de

 TNC support

 [®] +49 (8669) 31-3101

e-mail: service.nc-support@heidenhain.de NC programming @ +49 (8669) 31-31 03 e-mail: service.nc-pgm@heidenhain.de PLC programming @ +49 (8669) 31-31 02 e-mail: service.plc@heidenhain.de Lathe controls @ +49 (711) 9528 03-0 e-mail: service.hsf@heidenhain.de

www.heidenhain.de

3-D Touch Probe Systems from HEIDENHAIN help you to reduce non-cutting time:

For example in

- workpiece alignment
- datum setting
- workpiece measurement
- digitizing 3-D surfaces

with the workpiece touch probes **TS 220** with cable **TS 640** with infrared transmission

- tool measurement
- wear monitoring
- tool breakage monitoring





with the tool touch probe **TT 130**