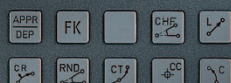
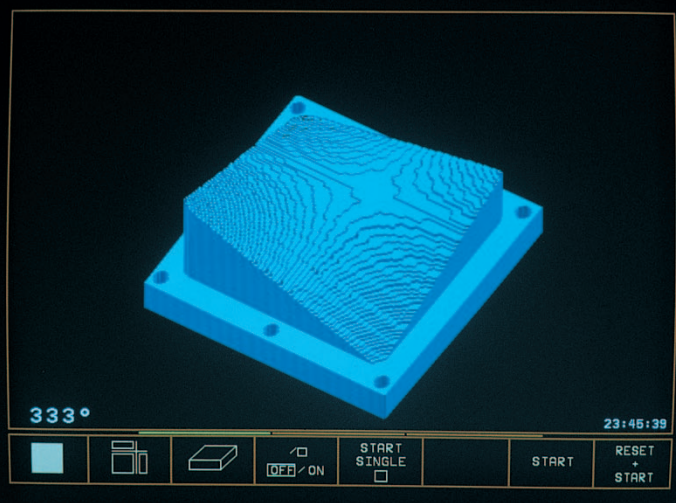


HEIDENHAIN



HEIDENHAIN








TNC 410

NC-Software
286 060-xx
286 080-xx

**User's Manual
Conversational
Programming**

English (en)
6/2001






Controls on the visual display unit

-  Split screen layout
-  Toggle display between machining and programming modes
-  Soft keys for selecting functions in screen
-  Shift soft-key rows for the soft keys
-  Change screen settings (BC 120 only)



Typewriter keyboard for entering letters and symbols

-  File name
Comments
-  ISO programs






Machine operating modes

-  MANUAL OPERATION
-  ELECTRONIC HANDWHEEL
-  POSITIONING WITH MDI
-  PROGRAM RUN, SINGLE BLOCK
-  PROGRAM RUN, FULL SEQUENCE



Programming modes

-  PROGRAMMING AND EDITING
-  TEST RUN

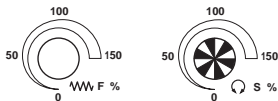
Program/file management, TNC functions

-  Select or delete programs and files
External data transfer
-  Enter program call in a program
-  MOD functions
-  HELP functions
-  Pocket calculator









Moving the cursor, going directly to blocks, cycles and parameter functions

-  Move highlight
-  Go directly to blocks, cycles and parameter functions



Override control knobs for feed rate/spindle speed









Programming path movements

-  Approach/depart contour
-  Free contour programming
-  Straight line
-  Circle center/pole for polar coordinates
-  Circle with center
-  Circle with radius
-  Tangential circle
-  Chamfer
-  Corner rounding














Tool functions

-   Enter or call tool length and radius

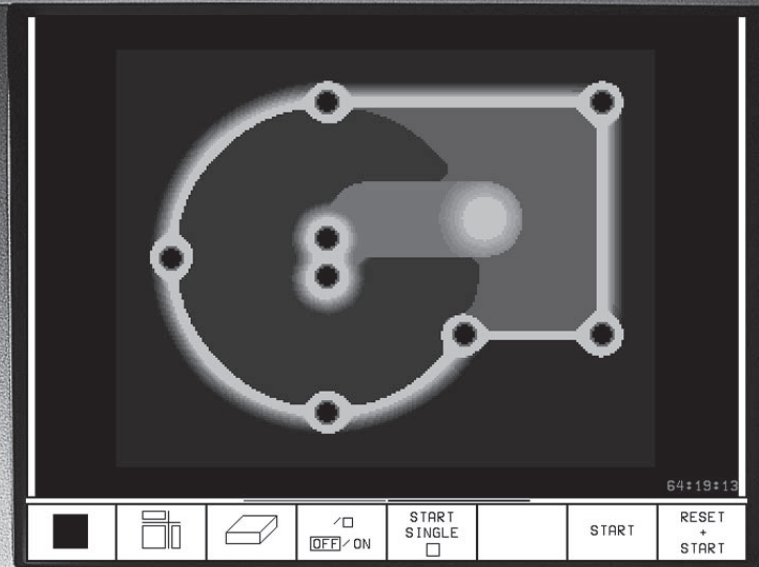
Cycles, subprograms and program section repeats

-   Define and call cycles
-   Enter and call labels for subprogramming and program section repeats
-  Program stop in a program
-  Enter touch probe functions in a program

Coordinate axes and numbers, editing

-  Select coordinate axes or enter them in a program
-  Numbers
-  Decimal point
-  Change arithmetic sign
-  Polar coordinates
-  Incremental dimensions
-  Q parameters
-  Capture actual position
-  Skip dialog questions, delete words
-  Confirm entry and resume dialog
-  End block
-  Clear numerical entry or TNC error message
-  Abort dialog, delete program section

HEIDENHAIN



64:19:13



TNC Models, Software and Features

This manual describes functions and features provided by the TNCs with the following NC software number.

TNC Model	NC Software No.
TNC 410	286 060-xx
TNC 410	286 080-xx

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

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

- Probing function for the 3-D touch probe
- Digitizing option
- Tool measurement with the TT 120
- Rigid tapping

Please contact your machine tool builder to become familiar with the individual implementation of the control on 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.

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.

Contents

Introduction	1
Manual Operation and Setup	2
Positioning with Manual Data Input	3
Programming: Fundamentals of NC, File Management, Programming Aids	4
Programming: Tools	5
Programming: Programming Contours	6
Programming: Miscellaneous Functions	7
Programming: Cycles	8
Programming: Subprograms and Program Section Repeats	9
Programming: Q Parameters	10
Test Run and Program Run	11
3-D Touch Probes	12
Digitizing	13
MOD Functions	14
Tables and Overviews	15

1 INTRODUCTION 1

- 1.1 TheTNC 410 2
- 1.2 Visual Display Unit and Keyboard 3
- 1.3 Modes of Operation 5
- 1.4 Status Displays 9
- 1.5 Accessories: HEIDENHAIN 3-DTouch Probes and Electronic Handwheels 12

2 MANUAL OPERATION AND SETUP 13

- 2.1 Switch-On 14
- 2.2 Moving the MachineAxes 15
- 2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M 18
- 2.4 Setting the Datum (Without a 3-DTouch Probe) 19

3 POSITIONING WITH MANUAL DATA INPUT (MDI) 21

- 3.1 Programming and Executing Simple Positioning Blocks 22

4 PROGRAMMING: FUNDAMENTALS OF NC, FILE MANAGEMENT, PROGRAMMING AIDS 25

- 4.1 Fundamentals of NC 26
- 4.2 File Management 31
- 4.3 Creating andWriting Programs 34
- 4.4 Interactive Programming Graphics 39
- 4.5 Adding Comments 40
- 4.6 HELP Function 41

5 PROGRAMMING: TOOLS 43

- 5.1 EnteringTool-Related Data 44
- 5.2 Tool Data 45
- 5.3 Tool Compensation 52
- 5.4 MeasuringTools with the TT 120 56

6 PROGRAMMING: PROGRAMMING CONTOURS 63

6.1 Overview of Tool Movements	64
6.2 Fundamentals of Path Functions	65
6.3 Contour Approach and Departure	68
Overview: Types of paths for contour approach and departure	68
Important positions for approach and departure	68
Approaching on a straight line with tangential connection: APPR LT	70
Approaching on a straight line perpendicular to the first contour point: APPR LN	70
Approaching on a circular arc with tangential connection: APPR CT	71
Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT	72
Departing tangentially on a straight line: DEP LT	73
Departing on a straight line perpendicular to the last contour point: DEP LN	73
Departing tangentially on a circular arc: DEP CT	74
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT	75
6.4 Path Contours — Cartesian Coordinates	76
Overview of path functions	76
Straight line L	77
Inserting a chamfer CHF between two straight lines	77
Circle center CC	78
Circular path C around circle center CC	79
Circular path CR with defined radius	80
Circular path CT with tangential connection	81
Corner Rounding RND	82
Example: Linear movements and chamfers with Cartesian coordinates	83
Example: Circular movements with Cartesian coordinates	84
Example: Full circle with Cartesian coordinates	85
6.5 Path Contours – Cartesian Coordinates	86
Polar coordinate origin: Pole CC	86
Straight line LP	87
Circular path CP around pole CC	87
Circular path CTP with tangential connection	88
Helical interpolation	88
Example: Linear movement with polar coordinates	90
Example: Helix	91

6.6 Path Contours – FK Free Contour Programming	92
Fundamentals	92
Graphics during FK programming	92
Initiating the FK dialog	93
Free programming of straight lines	94
Free programming of circular arcs	94
Auxiliary points	96
Relative data	97
Closed contours	97
Example: FK programming 1	98
Example: FK programming 2	99
Example: FK programming 3	100

7 PROGRAMMING: MISCELLANEOUS FUNCTIONS 103

7.1 Entering Miscellaneous Functions M and STOP	104
7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant	105
7.3 Miscellaneous Functions for Coordinate Data	105
Programming machine-referenced coordinates: M91/M92	105
7.4 Miscellaneous Functions for Contouring Behavior	107
Smoothing corners: M90	107
Entering contour transitions between contour elements: M112	108
Contour filter: M124	110
Machining small contour steps: M97	112
Machining open contours: M98	113
Feed rate factor for plunging movements: M103	114
Constant feed rate at the tool cutting edge: M109/M110/M111	115
Calculating the radius-compensated path in advance (LOOKAHEAD): M120	115
7.5 Miscellaneous Functions for Rotary Axes	117
Shorter-path traverse of rotary axes: M126	117
Reducing display of a rotary axis to a value less than 360°: M94	117

8 PROGRAMMING: CYCLES 119

8.1 General Overview of Cycles	120
8.2 PointTables	122
Creating a point table	122
Selecting point tables in the program.	122
Calling a cycle in connection with point tables	123
8.3 Drilling Cycles	124
PECKING (Cycle 1)	124
DRILLING (Cycle 200)	126
REAMING (Cycle 201)	127
BORING (Cycle 202)	128
UNIVERSAL DRILLING (Cycle 203)	129
BACK BORING (Cycle 204)	131
TAPPING with a floating tap holder (Cycle 2)	133
RIGIDTAPPING GS (Cycle 17)	134
Example: Drilling cycles	135
Example: Drilling cycles	136
Example: Calling drilling cycles in connection with point tables	137
8.4 Cycles for Milling Pockets, Studs and Slots	139
POCKET MILLING (Cycle 4)	140
POCKET FINISHING (Cycle 212)	141
STUD FINISHING (Cycle 213)	143
CIRCULAR POCKET MILLING (Cycle 5)	144
CIRCULAR POCKET FINISHING (Cycle 214)	146
CIRCULAR STUD FINISHING (Cycle 215)	147
SLOT MILLING (Cycle 3)	149
SLOT (Slot milling) with reciprocating plunge cut (Cycle 210)	150
CIRCULAR SLOT with reciprocating plunge-cut (Cycle 211)	152
Example: Milling pockets, studs and slots	154
Example: Roughing and finishing a rectangular pocket in connection with point tables	156
8.5 Cycles for Machining Hole Patterns	158
CIRCULAR PATTERN (Cycle 220)	159
LINEAR PATTERN (Cycle 221)	160
Example: Circular hole patterns	162

- 8.6 SL cycles 164
 - CONTOUR GEOMETRY (Cycle 14) 165
 - Overlapping contours 166
 - Pilot drilling (Cycle 15) 168
 - ROUGH-OUT (Cycle 6) 169
 - CONTOUR MILLING (Cycle 16) 171
 - Example: Rough-out a pocket 172
 - Example: Pilot drilling, roughing-out and finishing overlapping contours 174
- 8.7 Cycles for multipass milling 176
 - MULTIPASS MILLING (Cycle 230) 176
 - RULED SURFACE (Cycle 231) 178
 - Example: Multipass milling 180
- 8.8 CoordinateTransformation Cycles 181
 - DATUM SHIFT (Cycle 7) 182
 - DATUM SHIFT with datum tables (Cycle 7) 182
 - MIRROR IMAGE (Cycle 8) 184
 - ROTATION (Cycle 10) 185
 - SCALING FACTOR (Cycle 11) 186
 - AXIS-SPECIFIC SCALING (Cycle 26) 187
 - Example: Coordinate transformation cycles 188
- 8.9 Special Cycles 190
 - DWELLTIME (Cycle 9) 190
 - PROGRAM CALL (Cycle 12) 190
 - ORIENTED SPINDLE STOP (Cycle 13) 191

9 PROGRAMMING: SUBPROGRAMS AND PROGRAM SECTION REPEATS 193

- 9.1 Marking Subprograms and Program Section Repeats 194
- 9.2 Subprograms 194
- 9.3 Program section repeats 195
- 9.4 Program as Subprogram 196
- 9.5 Nesting 197
 - Subprogram within a subprogram 197
 - Repeating program section repeats 198
 - Repeating a subprogram 199
- 9.6 Programming Examples 200
 - Example: Milling a contour in several infeeds 200
 - Example: Groups of holes 201
 - Example: Groups of holes with several tools 202

10 PROGRAMMING: Q PARAMETERS 205

- 10.1 Principle and Overview 206
- 10.2 Part Families — Q Parameters in Place of Numerical Values 207
- 10.3 Describing Contours Through Mathematical Functions 208
- 10.4 Trigonometric Functions 210
- 10.5 If-Then Decisions with Q Parameters 211
- 10.6 Checking and Changing Q Parameters 212
- 10.7 Additional Functions 213
- 10.8 Entering Formulas Directly 219
- 10.9 Preassigned Q Parameters 222
- 10.10 Programming Examples 224
 - Example: Ellipse 224
 - Example: Concave cylinder machined with spherical cutter 2267
 - Example: Convex sphere machined with end mill 228

11 TEST RUN ND PROGRAM RUN 231

- 11.1 Graphics 232
- 11.2 Test run 236
- 11.3 Program run 238
- 11.4 Blockwise Transfer: Running Longer Programs 245
- 11.5 Optional Block Skip 246
- 11.6 Optional Program Run Interruption 246

12 3-D TOUCH PROBES 247

- 12.1 Touch Probe Cycles in the Manual and Electronic Handwheel modes. 248
- 12.2 Setting the Datum with a 3-D Touch Probe 251
- 12.3 Measuring Workpieces with a 3-D Touch Probe 254

13 DIGITIZING 259

- 13.1 Digitizing with a Triggering Touch Probe (Optional) 260
- 13.2 Programming Digitizing Cycles 261
- 13.3 Meander Digitizing 262
- 13.4 Contour Line Digitizing 263
- 13.5 Using Digitized Data in a Part Program 265

14 MOD FUNCTIONS 267

- 14.1 Selecting, Changing and Exiting the MOD Functions 268
- 14.2 System Information 268
- 14.3 Code Number 269
- 14.4 Setting the Data Interface 269
- 14.5 Machine-Specific User Parameters 271
- 14.6 Position DisplayTypes 272
- 14.7 Unit of Measurement 272
- 14.8 Select the Programming Language 273
- 14.9 Enter AxisTraverse Limits 274
- 14.10The HELP Function 275

15 TABLES AND OVERVIEWS 277

- 15.1 General User Parameters 278
 - Input possibilities for machine parameters 278
 - Selecting user parameters 278
 - External data transfer 279
 - 3-D touch probes and digitizing 280
 - TNC displays,TNC editor 282
 - Machining and program run 287
 - Electronic handwheels 289
- 15.2 Pin Layout and Connecting Cable for the Data Interface 290
- 15.3 Technical Information 292
 - TNC features 292
 - Programmable functions 293
 - TNC Specifications 294
- 15.4TNC Error Messages 295
 - TNC error messages during programming 295
 - TNC error messages during test run and program run 296
 - TNC error messages during digitizing 299
- 15.5 Changing the Buffer Battery 300



1

Introduction

1.1 The TNC 410

HEIDENHAIN TNC controls are shop-floor programmable contouring controls for milling, drilling and boring machines, as well as machining centers with up to four axes. You can program conventional milling, drilling and boring operations right at the machine with the easily understandable interactive conversational guidance. You can also change the angular position of the spindle under program control.

Keyboard and screen layout are clearly arranged in a such 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 carries out the necessary calculations automatically. Workpiece machining can be graphically simulated during test run. It is also possible to program in ISO format or DNC mode.

You can enter a program while the TNC is running another.

Compatibility

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



1.2 Visual Display Unit and Keyboard

Visual display unit

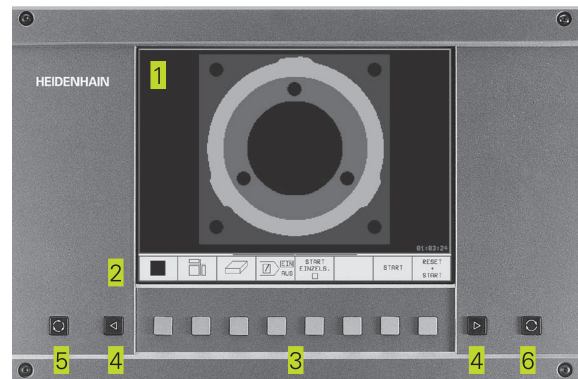
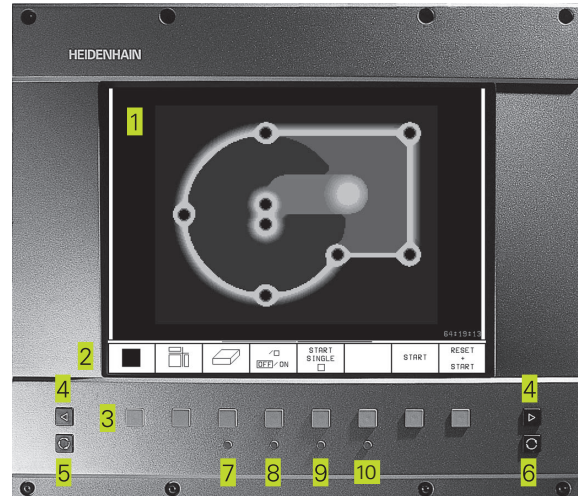
The TNC is available with either a color CRT screen (BC 120) or a TFT flat panel display (BF 120). The figures at right show the keys and controls on the BC 120 (upper right) and the BF 120 (middle right).

- 1 Header
When the TNC is on, the selected operating modes are shown in the screen header.
- 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 3. The lines immediately above the soft-key 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 selector keys
- 4 Switching the soft-key rows
- 5 Setting the screen layout
- 6 Shift key for switchover between machining and programming modes

Keys on BC 120 only

- 7 Screen demagnetization;
Exit main menu for screen settings
- 8 Select main menu for screen settings;
In the main menu: Move highlight downward
In the submenu: Reduce value
Move picture to the left or downward
- 9 In the main menu: Move highlight upward
In the submenu: Increase value
Move picture to the right or upward
- 10 In the main menu: Select submenu
In the submenu: Exit submenu

See next page for the screen settings.




Main menu dialog	Function
BRIGHTNESS	Adjust brightness
CONTRAST	Adjust contrast
H-POSITION	Adjust horizontal position
H-SIZE	Adjust picture width
V-POSITION	Adjust vertical position
V-SIZE	Adjust picture height
SIDE-PIN	Correct barrel-shaped distortion
TRAPEZOID	Correct trapezoidal distortion
ROTATION	Correct tilting
COLORTEMP	Adjust color temperature
R-GAIN	Adjust strength of red color
B-GAIN	Adjust strength of blue color
RECALL	No function

The BC 120 is sensitive to magnetic and electromagnetic noise, which can distort the position and geometry of the picture. Alternating fields can cause the picture to shift periodically or to become distorted.


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 help graphics for cycle definition 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.



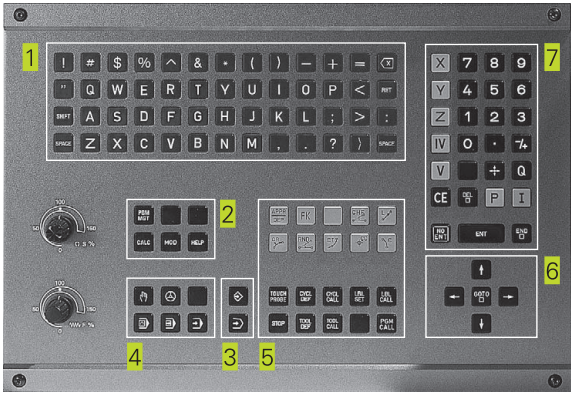
Select the desired screen layout.

Keyboard

The figure at right shows the keys of the keyboard grouped according to their functions:

- 1 Alphanumeric keyboard
for entering texts and file names, as well as for programming in ISO format
- 2 File management,
MOD functions,
HELP functions
- 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

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



1.3 Modes of Operation

The TNC offers the following modes of operation for the various functions and working steps that you need to machine a workpiece:

Manual Operation and Electronic

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

The 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

The same selection is available as in the Positioning with MDI mode. The TNC always shows the positions at left in the divided screen.

Manual operation							
ACTL. X -219.210							
Y +0.795							
Z +212.795							
DIST. X +29999.210							
Y +29999.205							
Z +29999.205							
				T	0		
				F			
				S	M5 / 9		
M	S	TOUCH PROBE		INCRE- MENT DEF ON	DATUM SET		TOOL TABLE

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 blocks	PGM
Left: program, right: general program information	PGM + PGM STATUS
Left: program, right: positions and coordinates	PGM + POS. STATUS
Left: program, right: information on tools	PGM + TOOL STATUS
Left: Program, right: coordinate transformations	PGM + C. TRANS. STATUS

Programming and Editing

In this mode of operation you can write your part programs. The FK free programming feature, the various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the individual steps.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program blocks	PGM
Left: program, right: help graphics for cycle programming	PGM + FIGURE
Left: program blocks, right: programming graphics	PGM + GRAPHICS
Interactive Programming Graphics	GRAPHICS

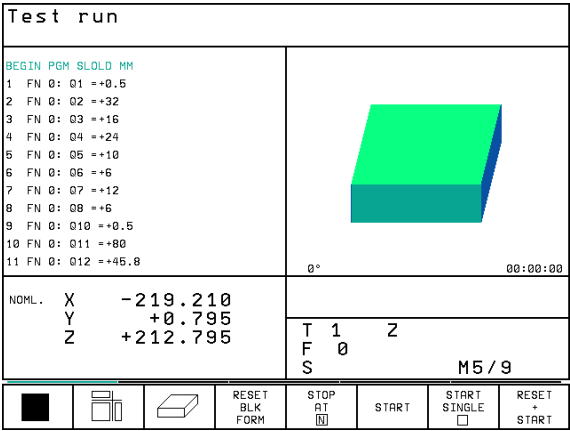
Programming and editing							
1	BLK	FORM	0.1	Z	X-20	Y-20	Z-20
2	BLK	FORM	0.2	X+20	Y+20	Z+0	
3	TOOL	DEF	1	L+0	R+3		
4	TOOL	CALL	1	Z	S1000		
5	L	Z+50	R0	FMAX	M3		
6	L	X+50	Y+50	R0	FMAX	M8	
7	L	Z-5	R0	FMAX	M1		
8	CC	X+0	Y+0				
9	LP	PR+14	PA+45	RR	F500		
10	RND	R1					
11	FC	DR+	R2.5	CLSD+			
12	FLT	AN+180.925	M1				
ACTL.				X	-219.715		
				Y	+0.285		
				Z	+212.680		
				T	2	Z	
				F	0		
				S			M5/9
PAGE	PAGE	BEGIN	END	FIND			
↑	↓	↑	↓				

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

Screen windows	Soft key
Program blocks	PGM
Test run graphics	GRAPHICS
Left: program, right: test run graphics	PGM + GRAPHICS
Left: program, right: general program information	PGM + PGM STATUS
Left: program, right: positions and coordinates	PGM + POS. STATUS
Left: program, right: information on tools	PGM + TOOL STATUS
Left: Program, right: coordinate transformations	PGM + C. TRANS. STATUS



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 blocks	PGM
Left: program, right: general program information	PGM + PGM STATUS
Left: program, right: positions and coordinates	PGM + POS. STATUS
Left: program, right: information on tools	PGM + TOOL STATUS
Left: Program, right: coordinate transformations	PGM + C. TRANS. STATUS
Left: program, right: tool measurement	PGM + T. PROBE STATUS

Program run, full sequence

BEGIN PGM 3507 MM

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

2 BLK FORM 0.2 X+20 Y+20 Z+0

3 TOOL DEF 1 L+0 R+3

4 TOOL CALL 1 Z S1000

5 L Z+50 R0 FMAX M3

6 L X+50 Y+50 R0 FMAX M8

7 L Z-5 R0 FMAX M1

8 CC X+0 Y+0

9 LP PR+14 PA+45 RR F500

10 RND R1

11 FC DR+ R2.5 CLSD+

Programs 3507 / 0

ACTL. X -219.210

Y +0.795

Z +212.795

Basic rotati

+12.357

NOML. X -219.210

Y +0.795

Z +212.795

T F 0

S ROT M5/9

BLOCKWISE TRANSFER

RESTORE POS. AT

ON OFF

ON OFF

TOOL TABLE


1.4 Status Displays

“General” status displays

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

In the Manual mode, Electronic Handwheel mode, and Positioning with MDI mode the position display appears in the large window.

Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
X Y Z	Machine axes
S F M	Spindle speed S, feed rate F and active M functions
*	Program run started
+	Axis locked
	Axes are moving under a basic rotation.

Additional status displays

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

To switch on the additional status display:

- Call the soft-key row for screen layout.
- Select the layout option for the additional status display, e.g. positions and coordinates.

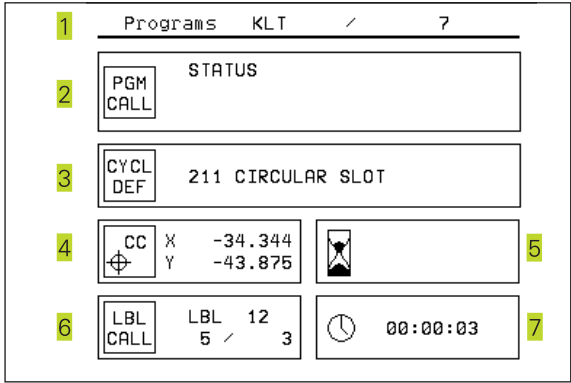
Program run, full sequence											
BEGIN PGM STATUS MM											
1 BLK FORM 0.1 Z X+0 Y+0 Z-40											
2 BLK FORM 0.2 X+100 Y+100 Z+0											
3 TOOL CALL 1 Z S4000 DL+0.05 DR+0.04											
4 L Z+100 R0 FMAX											
5 L X-20 Y+50 R0 FMAX											
6 L Z-2 R0 FMAX M3											
7 LBL 12											
8 CYCL DEF 7.0 DATUM SHIFT											
9 CYCL DEF 7.1 X+25.5											
10 CYCL DEF 7.2 Y+10											
11 CYCL DEF 7.3 Z+12											
NOML. X -244.710						T 2 Z F 0 S <div>ROT M5/9</div>					
Y -9.205											
Z +213.270											
BLOCKWISE TRANSFER				RESTORE POS. AT [N]		<div><div><div></div></div>ON OFF</div>		<div><div><div></div></div>ON OFF</div>		TOOL TABLE	

You can also choose between the following additional status displays:

PGM +
PGM
STATUS

General program information

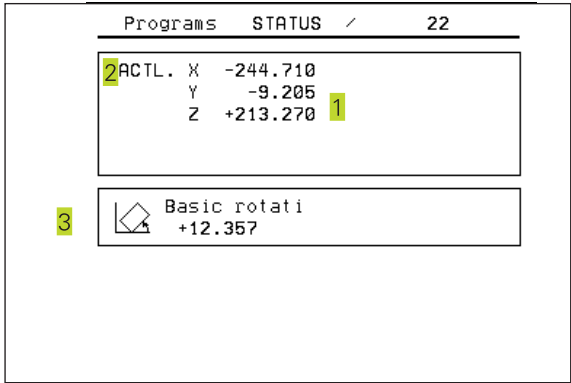
- 1 Name of main program
- 2 Active programs
- 3 Active machining cycle
- 4 Circle center CC (pole)
- 5 Dwell time counter
- 6 Active program section repeats/Counter for current program section repeat (5/3: 5 repetitions programmed, 3 remaining to be run)
- 7 Operating time



PGM +
POS.
STATUS

Positions and coordinates





- 1 Position display
- 2 Type of position display, e.g. actual positions
- 3 Angle of a basic rotation



PGM +
TOOL
STATUS

Information on tools

- 1 T: Tool number and name
RT: Number and name of a replacement tool
- 2 Tool axis
- 3 Tool length 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 T 2 SCHRUPPER											
2		3	L -12.500 R +3.000									
4	<table><tr><td></td><td>DL</td><td>DR</td></tr><tr><td>TAB</td><td>+0.025</td><td>+0.050</td></tr><tr><td>PGM</td><td>+0.050</td><td>+0.040</td></tr></table>				DL	DR	TAB	+0.025	+0.050	PGM	+0.050	+0.040
	DL	DR										
TAB	+0.025	+0.050										
PGM	+0.050	+0.040										
5	<table><tr><td></td><td>CUR. TIME</td><td>TIME1</td><td>TIME2</td></tr><tr><td></td><td></td><td>1:40</td><td></td></tr></table>				CUR. TIME	TIME1	TIME2			1:40		
	CUR. TIME	TIME1	TIME2									
		1:40										
6	<table><tr><td>TOOL CALL</td><td>2 SCHRUPPER</td></tr><tr><td>RT</td><td>↔ 12</td></tr></table>			TOOL CALL	2 SCHRUPPER	RT	↔ 12					
TOOL CALL	2 SCHRUPPER											
RT	↔ 12											

PGM +
C. TRANS.
STATUS

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 (Cycle 11 or Cycle 26)

For further information, refer to section 8.8 "Coordinate Transformation Cycles."

1	Programs STATUS / 22		
2	<div style="text-align: center; margin-bottom: 10px;">DATUM SHIFT</div> <div style="display: flex; align-items: center;"> <div> <p>X +25.500</p> <p>Y +10.000</p> <p>Z +12.000</p> </div> </div>	<div style="text-align: center; margin-bottom: 10px;">ROTATION</div> <div style="display: flex; align-items: center;"> <div> <p>+12.500</p> </div> </div>	3
	<div style="text-align: center; margin-bottom: 10px;">MIRROR IMAGE</div> <div style="display: flex; align-items: center;"> <div> <p>X Y</p> </div> </div>	4	
5	<div style="text-align: center; margin-bottom: 10px;">SCALING</div> <div style="display: flex; align-items: center;"> <div style="flex: 1;"> <div> <p>X +10.000</p> <p>Y +25.000</p> <p>Z -1.000</p> </div> </div> <div style="flex: 1; text-align: right;"> <p>0.995000</p> <p>1.005360</p> <p>0.992000</p> </div> </div>		

PGM +
T. PROBE
STATUS

Tool measurement

- 1 Number of the tool to be measured
- 2 Display whether the tool radius or the tool length is being measured
- 3 MIN and MAX values of the individual cutting edges and the 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	Tool data T 2 SCHRUPPER		
		L 2	3 MIN 2 +1.9664 MAX 3 +2.0035 DYN
4	1 +1.9909 2 +1.9664 * 3 +2.0035 4 +1.9986		

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
- Digitize 3-D surfaces (option), and
- Measure and inspect tools

TS 220 and TS 630 touch trigger probes

These touch probes are particularly effective for automatic workpiece alignment, datum setting, workpiece measurement and for digitizing. 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 features infrared transmission of the triggering signal to the TNC. This makes it 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.

During digitizing the TNC generates a program containing straight line blocks in HEIDENHAIN format from a series of measured position data. You can then output the program to a PC for further processing with the SUSA evaluation software. This evaluation software enables you to calculate male/female transformations or correct the program to account for special tool shapes and radii that differ from the shape of the stylus tip. If the tool has the same radius as the stylus tip you can run these programs immediately.

TT 120 tool touch probe for tool measurement

The TT 120 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 120 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.





2

Manual Operation and Setup

2.1 Switch-On



Switch-on and traversing the reference points can vary depending on the individual 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.

Power Interrupted



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

PLC-Translate PLC program

The PLC program of the TNC is automatically compiled.

Relay Ext. DC Voltage Missing



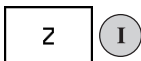
Switch on the control voltage.
The TNC checks the functioning of the EMERGENCY STOP circuit.

Manual Operation

Traverse Reference Points



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, or



Cross the reference points with several axes at the same time: Use soft keys to select the axes (axes are then shown highlighted on the screen), and then press the machine START button.

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

2.2 Moving the Machine Axes



Traversing with the machine axis direction buttons is a machine-dependent function. Your machine manual provides more detailed information.

To traverse with the machine axis direction buttons:

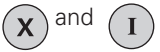


Select the Manual Operation mode.



Press the machine axis direction button and hold it as long as you wish the axis to move.

...or move the axis continuously:



Press and hold the machine axis direction button, then press the machine START button: The axis continues to move after you release the keys.



To stop the axis, press the machine STOP button.

You can move several axes at a time with these two methods.

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 an 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 indicators 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:



Select the Electronic Handwheel mode of operation



Press and hold the permissive button



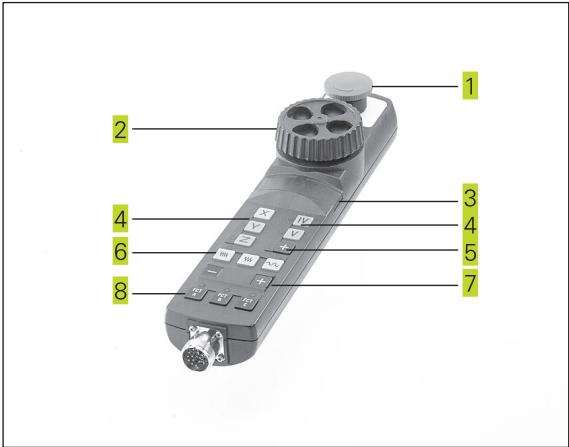
Select the axis.



Select the feed rate.




Move the active axis in the positive or negative direction.




Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance each time you press the corresponding machine axis direction button.



Select the Electronic Handwheel or Manual mode of operation.



Select incremental jog positioning, set the soft key to On.

JOG INCREMENT =

8

ENT

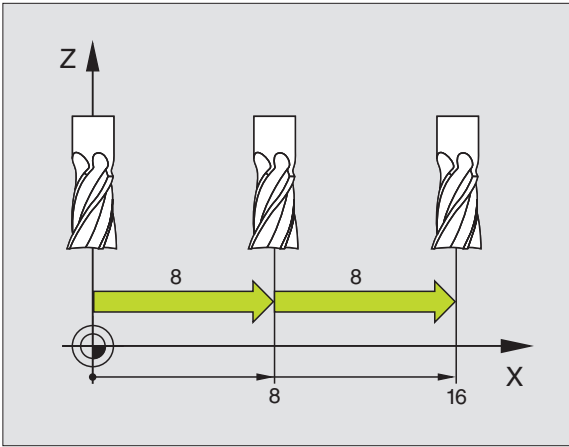
Enter the jog increment in mm, e.g. 8 mm, or

2.5

Enter the jog increment via soft key (preset soft-key values).

X

Press the machine axis direction button as often as desired.



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

In the operating modes Manual and Electronic Handwheel, enter the spindle speed S and the miscellaneous function M using soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous Functions." The feed rate is defined in a machine parameter and can be changed only with the override knobs (see next page).

Entering values

Example: Enter the spindle speed S

S

To enter the spindle speed, press the S soft key.

Spindle speed S=

1000

ENT

Enter the desired spindle speed,

I

and confirm your entry with the machine START button.

The spindle speed S with the entered rpm is started with a miscellaneous function.

Proceed in the same way to enter the miscellaneous functions M.

Changing the spindle speed S and feed rate F

With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value.



The knob for spindle speed override is effective only on machines with an infinitely variable spindle drive.

The machine tool builder determines which miscellaneous functions M are available on your TNC and what effects they have.



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

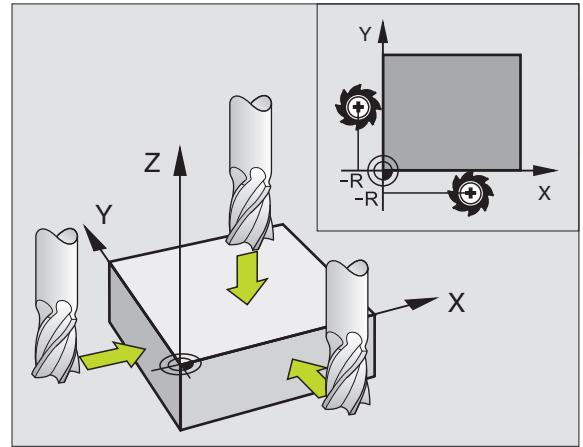
You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

To prepare the TNC:

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

Set the datum

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 .



Select the Manual Operation mode.



Move the tool slowly until it touches the workpiece surface.



Select the axis.

Datum Set



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 $Z=L+d$.



3

**Positioning with Manual Data
Input (MDI)**

3.1 Programming and Executing Simple Positioning Blocks

The operating mode Positioning with Manual Data Input is particularly convenient for simple machining operations or pre-positioning 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 operating mode Positioning with MDI, the additional status displays can also be activated.



Select the Positioning with MDI mode of operation. Program the file \$MDI as you wish.



To start program run, press the machine START button.



Limitations:

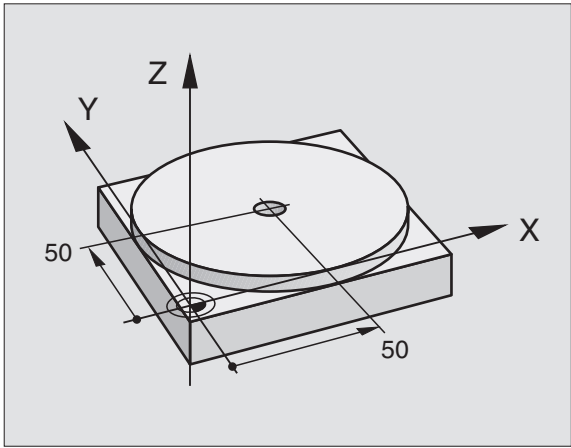
The following functions are not available:

- Tool radius compensation
- FK free contour programming
- Programming graphics and program-run graphics
- The programmable probing functions
- Subprograms, Program section repetitions
- The path functions CT, CR, RND and CHF
- PGM CALL

Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool in L blocks (straight-line blocks) to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle 1 PECKING.



0 BEGIN PGM \$MDI MM	
1 TOOL DEF 1 L+0 R+5	Define tool: zero tool, radius 5
2 TOOL CALL 1 Z S2000	Call tool: tool axis Z
	Spindle speed 2000 rpm
3 L Z+200 R0 FMAX	Retract tool (FMAX = rapid traverse)
4 L X+50 Y+50 R0 FMAX M3	Move the tool at FMAX to a position above the bore-
5 L Z+5 F2000	hole, spindle on Position tool to 5 mm above hole


6 CYCL DEF 1.0 PECKING	Define PECKING cycle:
7 CYCL DEF 1.1 SET UP 5	Setup clearance of the tool above the hole
8 CYCL DEF 1.2 DEPTH -20	Total hole depth (Algebraic sign=working direction)
9 CYCL DEF 1.3 PECKG 10	Depth of each infeed before retraction
10 CYCL DEF 1.4 DWELL 0.5	Dwell time in seconds at the hole bottom
11 CYCL DEF 1.5 F250	Feed rate for pecking
12 CYCL CALL	Call PECKING cycle
13 L Z+200 RO FMAX M2	Retract tool
14 END PGM \$MDI MM	End of program


The straight-line function is described in section 6.4 “Path Contours — Cartesian Coordinates,” the PECKING cycle in section 8.3 “Drilling Cycles.”

Example 2
Correcting workpiece misalignment on machines with rotary tables

Use the 3-D touch probe to rotate the coordinate system.
See “12.1 Touch probe cycles in the Manual and Electronic Handwheel Mode” section “Compensating workpiece misalignment.”

Write down the Rotation Angle and cancel the Basic Rotation.

 Select operating mode: Positioning with MDI.


 **IV** Select the axis of the rotary table, enter the rotation angle you wrote down previously and set the feed rate.
For example: L C+2.561 F50

 Conclude entry.


I Press the machine START button: The rotation of the table corrects the misalignment. After the NC start the highlight is moved onto the next block.

Protecting and erasing programs in \$MDI


The \$MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:




Select operating mode: Programming and Editing



To call the file manager, press the PGM MGT key (program management).



Move the highlight to the \$MDI file.




Select „Copy file“: Press the COPY soft key


Target file =

BOREHOLE

Enter the name under which you want to save the current contents of the \$MDI file.




Copy the file.



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 switch between conversational and ISO programming with the MOD function, you need to delete the current file \$MDI.*, and then select again the Positioning with MDI operating mode.

For further information, refer to section 4.2 “File Management.”



4

Programming:

**Fundamentals of NC,
File Management,
Programming Aids**

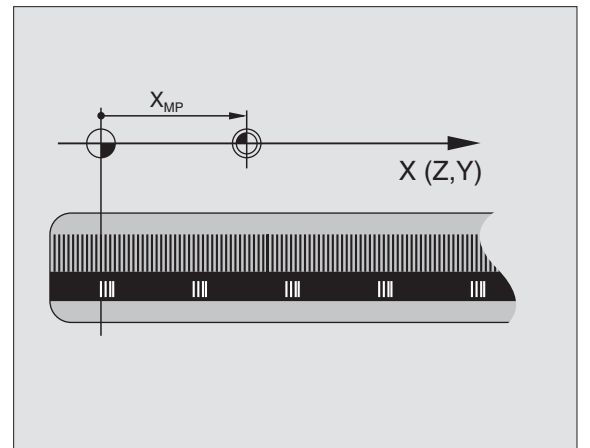
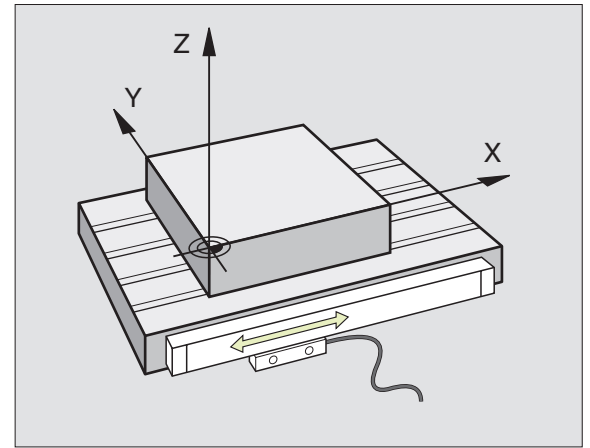
4.1 Fundamentals of NC

Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. 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 an interruption of power, the calculated position will no longer correspond to the actual position of the machine slide. The TNC can re-establish this relationship with the aid of reference marks when power is returned. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From the signal the TNC identifies that position as the machine-axis reference point and can re-establish the assignment of displayed positions to machine axis positions.

Linear encoders are generally used for linear axes. Rotary tables and tilt axes have angle encoders. If the position encoders feature distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders, to re-establish the assignment of the displayed positions to machine axis positions.

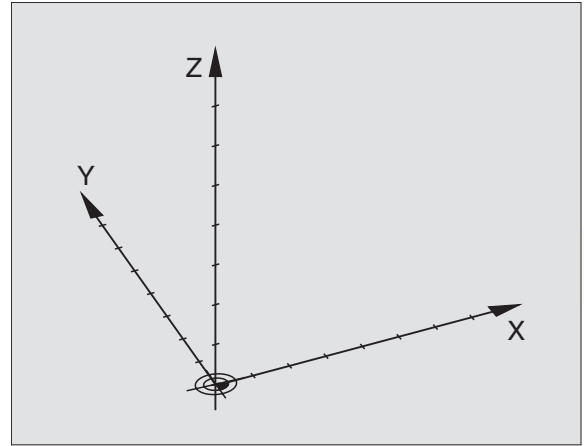


Reference system

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 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.

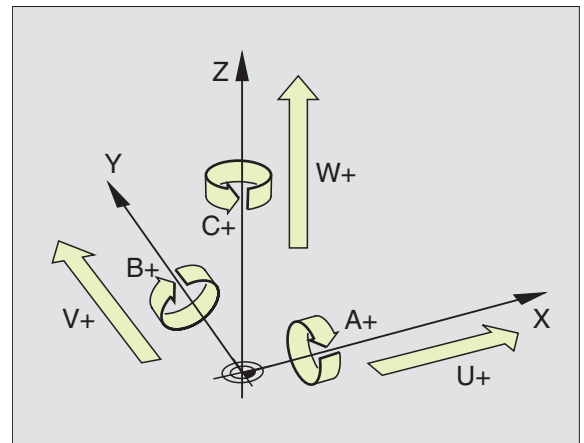
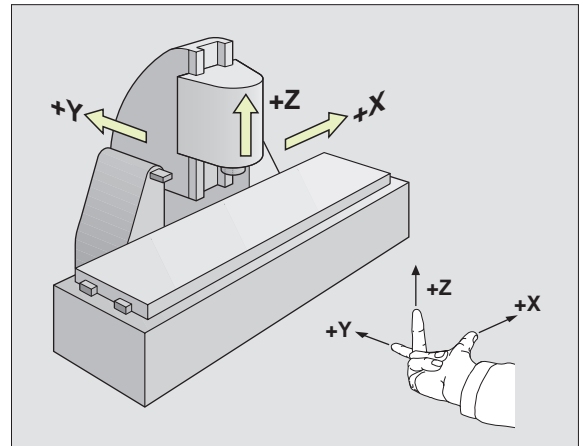
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.



Reference systems on milling machines

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

The TNC 410 can control up to 4 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 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 a circle center (CC), or pole. A position in a plane can be clearly defined by the

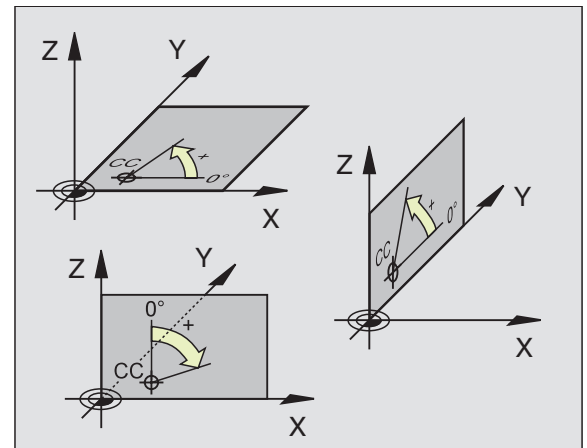
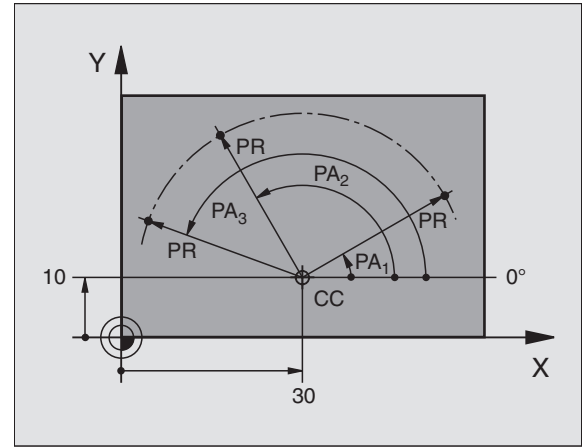
- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the size of the angle between the reference axis and the line that connects the circle center CC with the position.

See figure to the lower right.

Setting the pole and the angle reference axis

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

Coordinates of the pole (plane)	Reference axis of the angle
XY	+X
YZ	+Y
ZX	+Z



Absolute and relative 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 Y=10 mm	X=30 mm Y=20 mm	X=50 mm Y=30 mm

Relative workpiece positions

Relative 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 prefix "I" before the axis.

Example 2: Holes dimensioned in relative coordinates

Absolute coordinates of hole 4:

X= 10 mm
Y= 10 mm

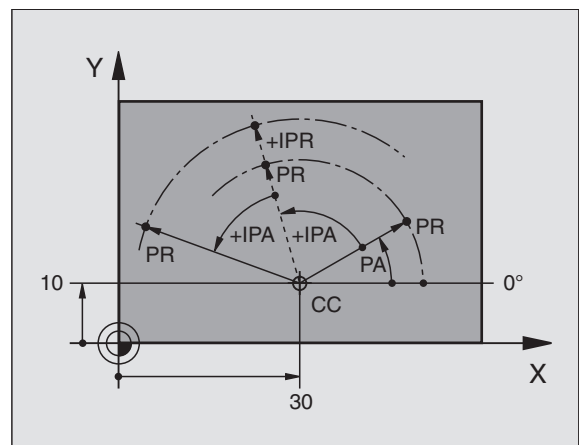
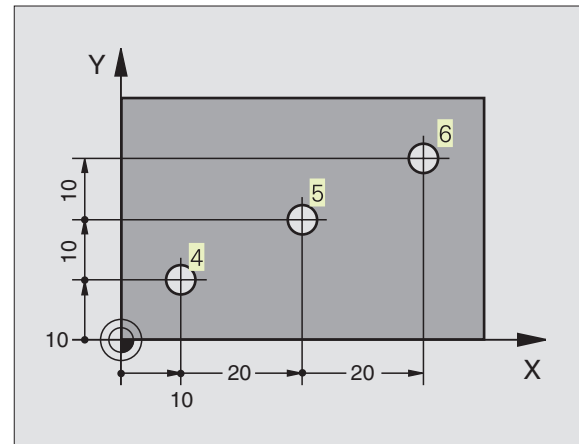
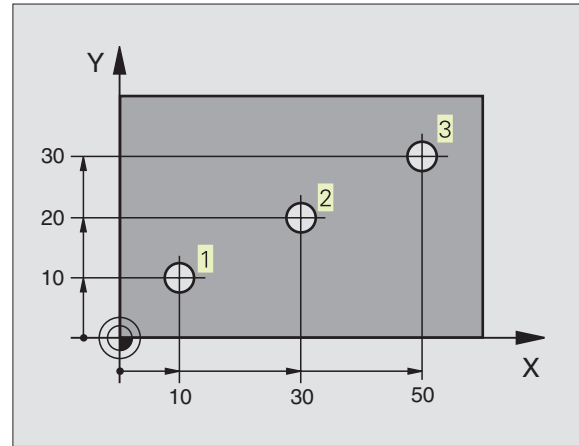
Hole 5 referenced to hole 4 Hole 6 referenced to hole 5

IX= 20 mm IX= 20 mm
IY= 10 mm IY= 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.



Selecting the datum

A production drawing identifies a certain point on 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 to either zero or a predetermined position value. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.

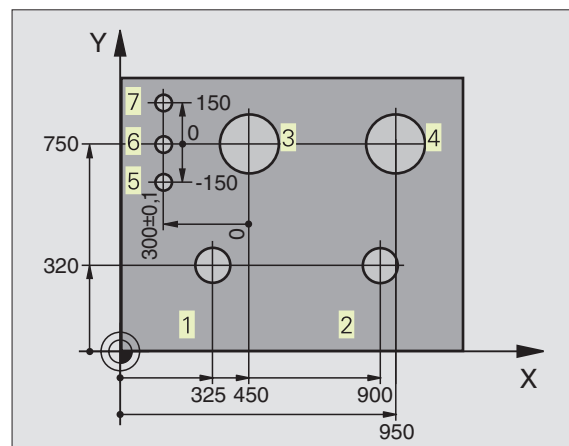
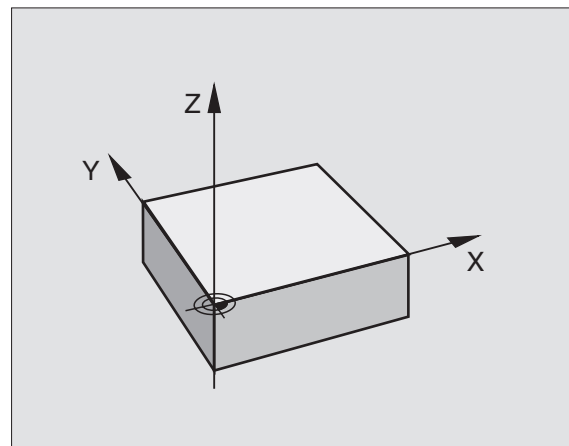
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles. For further information, refer to section 8.8 “Coordinate Transformation Cycles.”

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece, which is the most 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. For further information, refer to section 12.2 “Setting the Datum with a 3-D Touch Probe.”

Example

The workpiece drawing at right illustrates the holes 1 to 4, which are dimensioned to an absolute datum with the coordinates $X=0$ $Y=0$. The holes 5 to 7 are referenced to a relative datum with the absolute coordinates $X=450$ $Y=750$. By using the DATUM SHIFT cycle you can shift the datum temporarily to the position $X=450$, $Y=750$ and program the holes 5 to 7 without any further calculations.



4.2 File Management

Files and file management

When you write a part program on the TNC, you must first enter a file name. The TNC then stores the program as a file with the same name. You can also store tables as files.

File names

The name of a file can have up to 8 characters. The special characters @, \$, _, %, # and & are permitted. When you store programs and tables as files, the TNC adds an extension to the file name, separated by a point. This extension identifies the file type (see table at right).

PROG20	.H
File name	File type



The TNC permits only one file type per file name. You cannot assign several file types to the same name.

The TNC can manage up to 64 files. Their total size, however, must not exceed 256 KB.

Working with the file manager

This section informs you about the meaning of the individual screen information, and describes how to select files and directories. If you are not yet familiar with the TNC file manager, we recommend that you read this section completely and test the individual functions on your TNC.

To call the file manager



Press the PGM MGT key:
the TNC displays the file management window

The window 1 shows you all of the files that are stored in the TNC. Each file is shown with additional information that is illustrated in the table on the next page.

Files in the TNC

Type

Programs

in HEIDENHAIN conversational format	.H
in ISO format	.I

Tables for

Tools	.T
Tool pockets	.TCH
Datums	.D
Points	.PNT

Program selection			
File name =			
CYC210	.H	214	
CYCLS	.H	1362	
DATUM	.D	28	
FK	.H	588	1
FK3	.H	304	
HE	.H	272	
HE2	.H	338	
HE3	.H	228	
HEI	.I	100	
HETEST	.H	278	
HGF	.H	106	
IJP45	.H	330	
ACTL.	X	+0.195	
	Y	-11.000	
	Z	+136.000	
			T 0
			F 1000
			M5/9
PAGE	PAGE	PROTECT / UNPROTECT	RENAME [ABC] = [XYZ]
↑	↓	DELETE	COPY [ABC] = [XYZ]
			EXT
			END

Display

Meaning

File name	Name with up to 8 characters and file type
M	Properties of the file: Program is selected in a Program Run operating mode
P	Protect a file against editing and erasure (Protected)

Display of long file directories

Soft key

Move pagewise up through the file directory	PAGE ↑
Move pagewise down through the file directory	PAGE ↓

Select a file



Call the file manager.

Use the arrow keys to move the highlight to the desired file:



Move the highlight up or down

Enter the first or more letters of the file you wish to select and then press the GOTO key: The highlight moves to the first file that matches these letters.



The selected file is opened in the operating mode from which you have the called file manager: Press ENT.

Copy a file

► Move the highlight to the file you wish to copy.



► Press the COPY soft key to select the copying function.

► Enter the name of the destination file and confirm your entry with the ENT key: The TNC copies the file. The original file is retained.

Rename a file

► Move the highlight to the file you wish to rename.



► Select the renaming function.

► Enter the new file name; the file type cannot be changed.

► To execute renaming, press the ENT key.

Delete a file

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



► To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to erase the file.

► To confirm erasure press the YES soft key. Abort with the NO soft key if you do not wish to erase the file.

Protecting a file/Canceling file protection

► Move the highlight to the file you want to protect.



► To enable file protection, press the PROTECT/UNPROTECT soft key. The file now has status P.

To cancel file protection, proceed in the same way using the PROTECT/UNPROTECT soft key. You also need to enter the code number 86357.

Converting an FK program into HEIDENHAIN conversational format

► Move the highlight to the file you want to convert.



► To select the converting function, press the CONVERT FK->H soft key (2nd soft-key row).

► Enter the name of the destination file.

► To execute conversion, press the ENT key.

Read in/read out files



- To read in or read out files: Press the ENT soft key. The TNC provides the functions described below.



If a file to be read in already exists in the memory of the TNC, the TNC displays the message "File xxx already exists. Read in file? In this case, answer the dialog question with YES (file is the read in) or NO (file is not read in).

Likewise, if a file to be read out already exists on the external device, the TNC asks whether you wish to overwrite the external file.

Read in all files (file types: .H, .I, .T, .TCH, .D, .PNT)



- Read in all of the files that are stored on the external data medium.

Read in offered file



- List all files of a certain file type.



- For example: list all HEIDENHAIN conversational programs. To read-in the listed program, press the YES soft key. If you do not wish the read-in the program, press NO.

Read in a specific file



- Enter the file name. Confirm with the ENT key.



- Select the file type, e.g. HEIDENHAIN dialog program.

If you wish to read-in the tool table TOOL.T, press the TOOL TABLE soft key. If you wish to read-in the tool-pocket table TOOLP.TCH, press the POCKET TABLE soft key.

Read out a specific file



- Select the function for reading out a single file.



- Move the highlight to the file that you wish to read out. Press ENT or TRANSFER soft key to start the transfer.



- To terminate the function for reading out specific files: press the END key.

Read out all files (file types: .H, .I, .T, .TCH, .D, .PNT)



- Output all files stored in the TNC to an external device.

Display a file directory of the external device (File types: .H, .I, .T, .TCH, .D, .PNT)



- Display a list of files stored in the external device. The files are displayed pagewise. To show the next page: press the YES soft key. To return to the main menu: press the NO soft key.

4.3 Creating and Writing Programs

Organization of an NC program in HEIDENHAIN conversational format

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

The TNC numbers the blocks in ascending sequence.

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

The subsequent blocks contain information on:

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

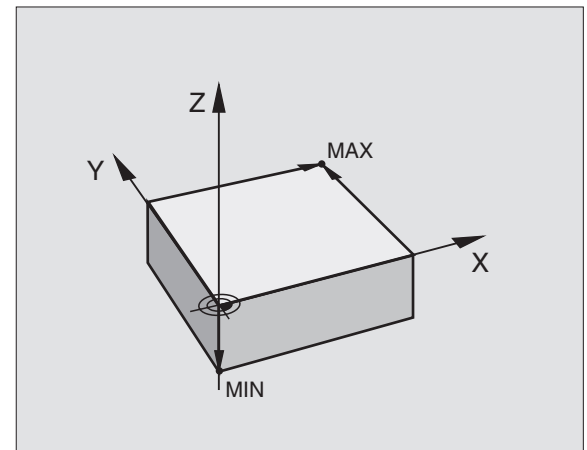
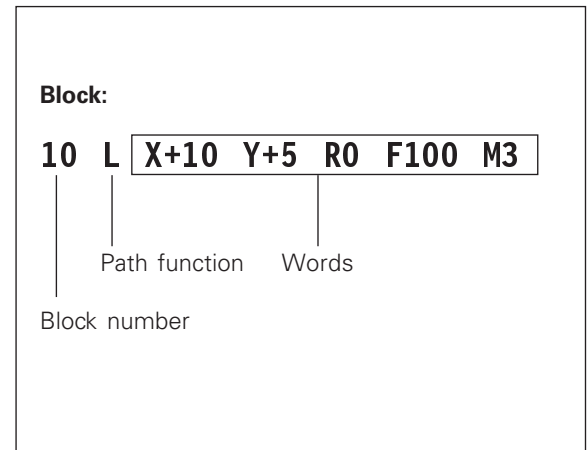
The last block of a program is identified by "END PGM," the program name and the active unit of measure.

Defining the blank form — BLK FORM

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 30 000 mm long. The blank form is defined by two of its corner points:

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

The TNC can display the graphic only if the short side of the BLK FORM is longer than 1/64 of the long side.



Creating a new part program

You always enter a part program in the Programming and Editing mode of operation.

Program initiation in an example:



Select the Programming and Editing mode of operation.



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

File name =

NEW



Entering new program names



Select the file type, e.g. HEIDENHAIN conversational: Press the .H soft key.



If necessary, switch to inches as unit of measure: Press the MM/INCH soft key.



Confirm your entry with the ENT key.

Program selection									
File name = TG.H									
ACTL.		X	+0.195						
		Y	-11.000						
		Z	+136.000						
		T	0						
		F	0						
		S	1000		M5 / 9				

Define the blank



Open the dialog for blank definition: Press the BLK FORM soft key

Working spindle axis X/Y/Z ?



Enter the spindle axis.

Def BLK FORM: Min corner?



Enter in sequence the X, Y and Z coordinates of the MIN point.



End the dialog MIN-corner entry.

Def BLK FORM: Max-corner?



Enter in sequence the X, Y and Z coordinates of the MAX point.



The program blocks window shows the following BLK FORM definition

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


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

Programming and editing																
Def BLK FORM: max-corner?																
BEGIN PGM 1568T MM																
1	BLK FORM	0.1	Z	X+0	Y+0	Z-40										
2	BLK FORM	0.2	X+100	Y+100	Z+0											
END PGM 1568T MM																
ACTL.		X	-219.715				T 2 Z F 0 S M5/9									
		Y	+0.285													
		Z	+212.680													

Programming tool movements in HEIDENHAIN conversational format

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

Dialog initiation in an example

Initiate the dialog.

Coordinates ?

X 10

ENT

Enter the target coordinate for the X axis.

Y 5

ENT

2x

Enter the target coordinate for the Y axis, and go to the next question with ENT.

Radius comp.: RL/RR/no comp. ?

ENT

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

Feed rate ? F=

100

ENT

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

Miscellaneous function M ?

3

END

Enter the miscellaneous function directly, e.g M3 for "spindle on," or

M120

enter miscellaneous functions that require additional input values, e.g. M120: Press M120 and enter the values.

END

Press the END key to end the dialog and save the entered block.

The program blocks window will display the following line:

```
3 L X+10 Y+5 R0 F100 M3
```

Programming and editing
Miscellaneous function M?

1 BLK FORM 0.1 Z X+0 Y+0 Z-40

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

3 TOOL DEF 100 L+10 R+5

4 TOOL CALL 100 Z S4000

5 L X+10 Y+5 R0 F100 M3

END PGM 1568T MM

ACTL. X -219.715

Y +0.285

Z +212.680




T 2 Z

F 0

S

M5 / 9




M	M103	M112	M120	M124			
---	------	------	------	------	--	--	--

Functions during the dialog	Key
Ignore the dialog question	
End the dialog immediately, save the block	
Abort the dialog and erase the block	

Editing program lines

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 (see table at right). While you are entering a new block the TNC identifies the block with a * as long as the block has not been saved.

Looking for the same words in different blocks

	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.

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.

Inserting blocks at any desired location

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

Inserting the previously edited (deleted) block at any desired location

- ▶ Select the block after which you want to insert the block you have just edited (deleted) and press the INSERT NC BLOCK soft key.










Editing and inserting words








- ▶ Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.
- ▶ To save your changes, press the END key.
- ▶ To reject the change, press the DEL 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.

Block display

If a block is so long that the TNC cannot display it in one line (for example in a fixed cycle), this will be indicated with ">>" at the right edge of the screen.

Function	Soft keys/keys
Go to the previous page	
Go to the next page	
Jump to end of program	
Jump to end of program	
Move from one block to the next	 
Select individual words in a block	 
Search for a sequence of characters	

Erasing blocks and words	Key
Set the selected word to zero	
Erase an incorrect number	
Clear a (non-blinking) error message	
Delete the selected word	
In a block: Restore previously saved version	
Delete the selected block (cycle)	
Delete the program sections: First select the last block of the program section to be erased, then erase with the DEL key.	

4.4 Interactive Programming Graphics

While you are writing the part program, you can have the TNC generate a graphic illustration of the programmed contour. The TNC represents movements in a negative spindle axis direction with a white circle (circle diameter = machine tool diameter).

To generate/not generate graphics during programming:

- ▶ 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 is switched ON, graphics are not generated for program section repeats.

To generate 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 are listed in the table at right.

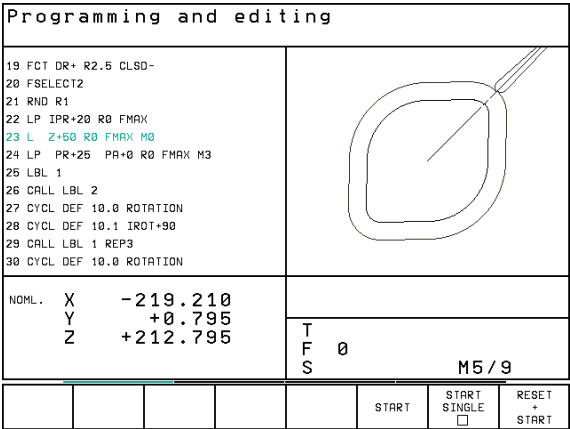
To erase the graphic:



- ▶ Shift the soft-key row (see figure at right)



- ▶ Delete graphic: Press CLEAR GRAPHIC soft key



Functions	Soft key
Generate interactive graphic blockwise	<div>START SINGLE</div> <div>□</div>
Generate a complete graphic or complete it after RESET + START	<div>START</div>
Stop the programming graphics This soft key only appears while the TNC generates the interactive graphics	<div>STOP</div>

Magnifying or reducing a detail

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

- Select the soft-key row for detail magnification/reduction (second row, see figure at right)
The following functions are available:

Principle	Soft key
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	>>
Shift the frame	<div>↓↑</div> <div>←→</div>

WINDOW
DETAIL

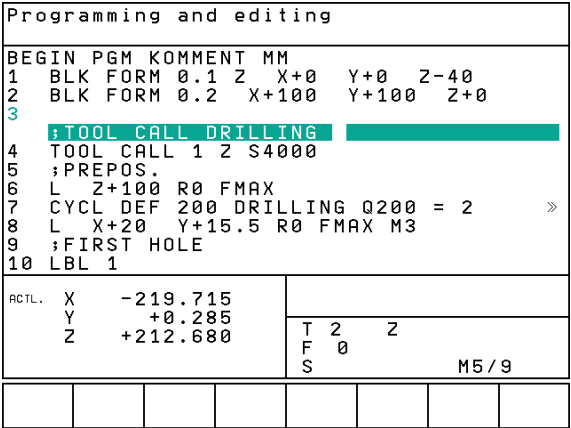
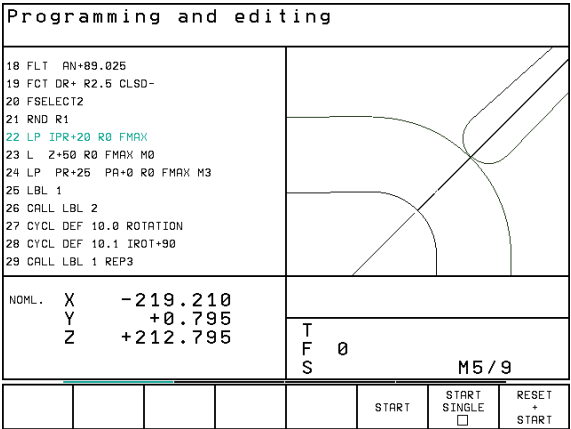
 ► With the WINDOW DETAIL soft key, Confirm the selected area.

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

4.5 Adding Comments

To explain or identify program steps you can enter comment blocks.

- 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.



4.6 HELP Function

Certain TNC programming functions are explained in more detail in the HELP function. You can select a HELP topic using the soft keys

Select the HELP function



- ▶ Press the HELP key
- ▶ Select a topic: Press one of the available soft keys

Help topics / Functions	Soft key
ISO programming: G functions	G
ISO programming: D functions	D
ISO programming: M functions	M
ISO programming: Address letters	ADDR LETTER
Cycle parameters	Q
HELP that is entered by the machine manufacturers (optional, not executable)	PLC
Go to next page	PAGE ↓
Go to previous page	PAGE ↑
Go to beginning of file	BEGIN ↑
Go to end of file	END ↓
Select search functions; Enter text, Begin search with ENT key	FIND

End the HELP function

Press the END soft key twice.

Programming and editing							
G	D	M	ADDR LETTER	Q	PLC		END

Programming and editing							
22' 0 to machine datum M92 - Within the positioning block: Coordinates are referenced to position defined by machine builder, such as tool change position M93 - Within the positioning block: Coordinates are referenced to the current tool position M94 - Reduce display of rotary axis to value under 360° M97 - Machine small contour steps M98 - Completely machine open contours M99 - Blockwise cycle call M101 - Automatic tool change with replacement tool if maximum service life has expired M102 - Reset M101 M103 - Reduce feed rate during plunging to factor F (percentage) M109 - Constant contouring speed at tool cutting edge on arcs (increase and decrease feed rate) M110 - Constant contouring speed at tool cutting edge on arcs (feed rate decrease only) M111 - Reset M109/M110 M112 - Insert contour transitions between contour elements M113 - Reset M112/M124 M120 - Radius-compensated contour look-ahead M124 - Ignore points for calculating contour transition M126 - Move rotary axes by shortest path M127 - Reset M126 [END]							
PAGE ↑	PAGE ↓		BEGIN ↑	END ↓		FIND	END



5

Programming:
Tools



5.1 Entering Tool-Related Data

Feed rate F

The feed rate 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 the individual axes and are set in machine parameters.

Input

You can enter the feed rate in every positioning block. For further information refer to section 6.2 "Fundamentals of Path Contours."

Rapid traverse

If you wish to program rapid traverse, enter FMAX. To enter FMAX, press the ENT key or the FMAX soft key as soon as the dialog question "Feed rate F = ?" appears on the TNC screen.

Duration of effect

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

Changing the spindle speed 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 a TOOL CALL block.

Programmed change

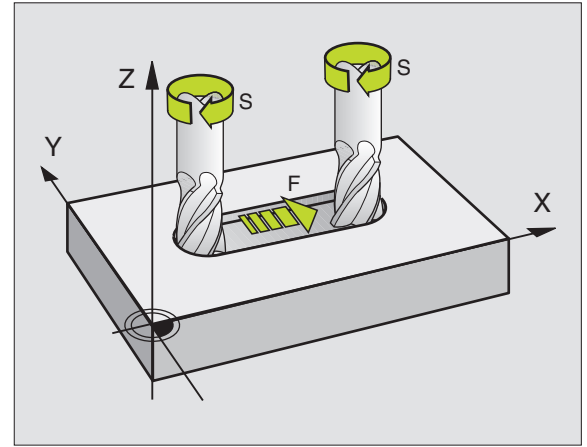
In the part program, you can change the spindle speed in a TOOL CALL block by entering the spindle speed only:



- ▶ To program a tool call, press the TOOL CALL key.
- ▶ Ignore the dialog question for "Tool number ?" with the NO ENT key.
- ▶ Ignore the dialog question for "Working spindle axis X/ Y/Z ?" with the NO ENT key.
- ▶ Enter the new spindle speed for the dialog question "Spindle speed S= ?" and confirm with END.

Changing the spindle speed during program run

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



5.2 Tool Data

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

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

Tool number

Each tool is identified by a number between 0 and 254.

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:

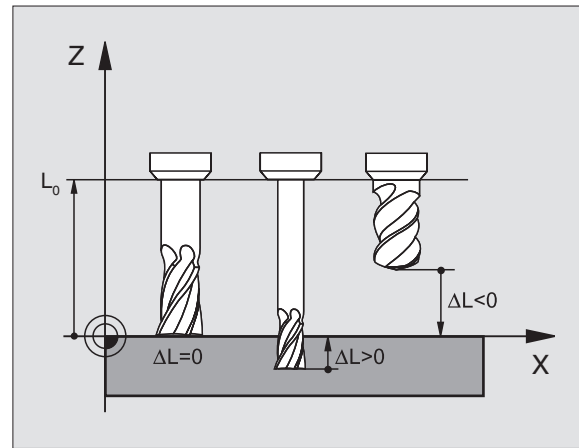
- 1 The length L is the difference between the length of the tool and that of a zero tool L_0 .

For the algebraic sign:

- The tool is longer than the zero tool $L > L_0$
- The tool is shorter than the zero tool: $L < L_0$

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 TOOL DEF block or in the tool table by pressing the actual-position-capture soft key
- 2 If you determine the length L with a tool presetter, this value can be entered directly in the TOOL DEF block 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 ($DR > 0$). If you are programming the machining data with an allowance, enter the oversize value in the TOOL CALL block of the part program.

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

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

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

Entering tool data into the program

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



- ▶ To select tool definition, press the TOOL DEF key.
- ▶ Enter the Tool number: Each tool is uniquely identified by its number.
- ▶ Enter the tool length: Enter the compensation value for the tool length.
- ▶ Enter the tool radius: Enter the compensation value for the tool radius.

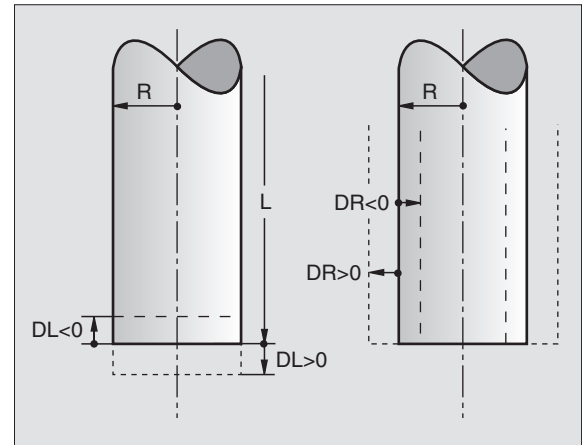


During the dialog, you can take the values for length and radius directly from the position display with the soft keys „CUR.POS X, CUR.POS Y or CUR.POS Z“.

If you use the black key for actual-position capture, the TNC saves the value of the active tool axis for the tool length. If no tool axis is active, the TNC saves the value of the axis that has been defined in the probing function calibration menu as the touch probe axis.

Resulting NC block:

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



Entering tool data in tables

You can define and store up to 254 tools and their tool data in a tool table. (The maximum number of tools in the table can be set in machine parameter 7260). See also the Editing Functions at a later stage in this Chapter.

You must use the tool table if:

- your machine tool has an automatic tool changer
- you want to measure tools automatically with the TT 120 touch probe (see section 5.4 “Measuring Tools”)

Tool table: Available input data

Abbr.	Input	Dialog
T	Number by which the tool is called in the program	–
NAME	Name by which the tool is called in the program	Tool name?
L	Value for tool length compensation	Tool length?
R	Compensation value for the tool radius R	Tool radius?
DL	Delta value for tool length	Tool length oversize?
DR	Delta value for tool radius R	Tool radius oversize?
TL	Set tool lock (TL :Tool Lock)	Tool locked ?
RT	Number of sister tool,if available, as Replacement Tool (RT) (see also 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 ?
TIME2	Maximum tool life in minutes during 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 life 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 sent to the PLC	PLC status?

Tool table: Tool data required for automatic tool measurement

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	Tool length measurement: tool offset between stylus center and tool center. Preset value: R = Tool radius R	Tool offset: radius ?
TT:L-OFFS	Tool radius measurement: tool offset in addition to MP6530 (see section 15.1 +General User Parameters+) between upper edge of stylus and lower edge of tool. Preset value: 0	Tool offset: length ?
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: length ?
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool(status L). Input range: 0 to 0.9999 mm	Breakage tolerance: radius ?

Editing tool tables

The tool table that is active during execution of the part program is designated TOOL.T. Tool.T is automatically active in a Program Run operating mode. In the Programming and Editing mode, tool tables can also have other names.

To open the tool table TOOL.T:

- ▶ Select any machine operating mode.



- ▶ To select the tool table, press the TOOL TABLE soft 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.
- ▶ Select an existing file with the extension .T and press the COPY soft key. Enter a new file name and confirm with the ENT 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 (see figure at upper right). You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table at right.

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 <<.

To leave the tool table:

- ▶ Finish editing the tool table: Press the "End" soft key or the END key.
- ▶ Call the file manager and select a file of a different type, e.g. a part program.



If you edit the tool table parallel to tool change the TNC does not interrupt the program run. However, the changed data does not become effective until the next tool call.

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

Program run, full sequence												
Tool length oversize?												
←	TOOL	. T	MM									→
0	+0	+0	+0	+0								
1	-12.5	+4	+0.025	+0.05	2	100	90	95				
2	-12.5	+3	+0.025	+0.05	12	100	0	0				
3	+0	+1.5	+0	+0			0	0	0			
4	+0	+2.5	+0	+0			0	0	0			
5	+0	+3	+0	+0			0	0	0			
6	-12	+25	+0.01	+0			0	0	0			
7	-25.35	+5	+0.5	+0			0	0	0			
8	+0	+0	+0	+0			0	0	0			
9	+0	+0	+0	+0			0	0	0			
10	-17.356	+2.5	+0.01	+0			0	0	0			
11	+0	+6	+0.05	+0			0	0	0			
12	-17.2	+3	+0	+0			0	0	0			
13	-45	+7.5	+0	+0			0	0	0			
ACTL. X -219.210												
Y +0.795												
Z +212.795												
					T F 0					M5 / 9		
					S							
PAGE	PAGE	WORD	WORD	EDIT							POCKET	END
↑	↓	←	→	OFF / ON							TABLE	

Editing functions for tool tables

Soft key

Go to the previous page of the table



Go to the next page of the table



Move highlight to the left



Move highlight to the right



Lock tool in TL column



Do not lock tool in TL column



Confirm actual positions, e.g. for Z axis



Confirm entered value



Select the next column in the table.
If the highlight is at the end of the line, then jump to the first column of the next line.

Delete incorrect numerical value, re-establish preset value



Re-establish the last value stored



Pocket table for tool changer

The TOOLPTCH (**TOOL** Pocket)table must be programmed to enable automatic tool change.

To select the pocket table:

- ▶ In the Programming and Editing mode,
 - ▶ call the file manager.
 - ▶ Move the highlight to TOOLPTCH. Confirm with the ENT key.

- ▶ In a machine operating mode

TOOL
TABLE

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

POCKET
TABLE

- ▶ To select the pocket table, press the POCKET TABLE soft key

EDIT
OFF / [ON]

- ▶ Set the EDIT soft key to ON

When you have opened the pocket table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys (see figure at upper right). You can overwrite the stored values, or enter new values at any position.

You may not use a tool number twice in the pocket table. If you do so the TNC will output an error message when you exit the table.

You can enter the following information on a tool into a pocket table

Program run, full sequence									
Pocket locked?									
TOOLP		.TCH		MM					
0	1	2	3	4	5	6	7	8	9
0									
1	2								111
2									0
3	11	S	F						101
4									0
5	15								0
6	1								0
7	3		F						0
8									0
9	7		F						0
10									0
11									0
12									0
13									0
ACTL.				X	-219.210				
				Y	+0.795				
				Z	+212.795				
				T	0				
				F					
				S			M5 / 9		
PAGE	PAGE	WORD	WORD				YES	NO	
↑	↓	←	→						

Editing functions for pocket tables Soft key

Select previous page in table



Select next page in table



Move the highlight one column to the left



Move the highlight one column to the right



Reset pocket table



Abbr.	Input	Dialog
P	Pocket number of the tool in the tool magazine	–
T	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 (status L).	Special tool ?
F	Fixed tool number. The tool is always returned to the same pocket.	Fixed pocket?
L	Locked pocket	Locked pocket?
PLC	Information on this tool pocket that is to be sent to the PLC	PLC status?

Calling tool data

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



- ▶ Select the tool call function with the TOOL CALL key.
- ▶ Tool number: Enter the number of the tool. The tool must already be defined in a TOOL DEF block or in the tool table.
- ▶ Working spindle axis X/Y/Z: Enter the tool axis. Parallel axes U, V and W are permitted.
- ▶ Spindle speed S
- ▶ Tool length oversize: Enter the delta value for the tool length.
- ▶ Tool radius oversize: Enter the delta value for the tool radius.

Example:

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.

```
20 TOOL CALL 5 Z S2500 DL+0.2 DR-1
```

The character D preceding L and R designates delta values.

Tool preselection with tool tables

When you use a tool table, enter a TOOL DEF block to preselect the next tool to be selected. Simply enter the tool number or a corresponding Q parameter, or type a tool name in quotation marks.

Tool change



The tool change function can vary depending on the individual machine tool. Your machine manual provides more information on M101.

Tool change position

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

Manual tool change

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

- ▶ Move to the tool change position under program control.
- ▶ Interrupt program run (see section 11.3 "Program Run").
- ▶ Change the tool.
- ▶ Resume the program run (see section 11.3 "Program Run").

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, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101



This function can vary depending on the individual machine tool. Your machine manual provides more information on M101.

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

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 TOOL CALL 0, the distance between tool and workpiece will be reduced.

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

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

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

L is the tool length L from the TOOL DEF block or tool table

$DL_{\text{TOOL CALL}}$ is the oversize for length DL in the TOOL CALL block (not taken into account by the position display)

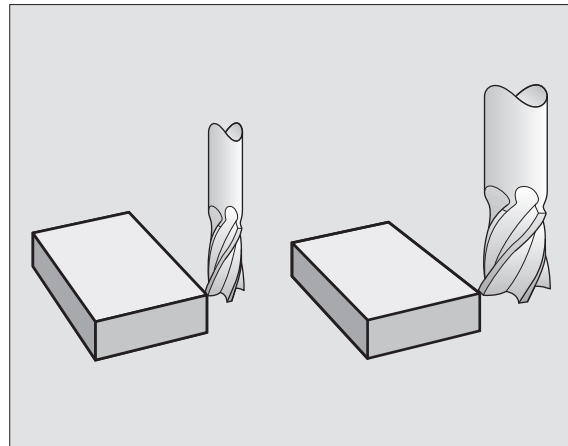
DL_{TAB} is the oversize for length DL in the tool table

Tool radius compensation

The NC block for programming a tool movement contains:

- RL or RR for compensation in the tool radius
- R+ or R- for radius compensation in single-axis movements
- R0 if no radius compensation is required

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with RL or RR. To cancel radius compensation, program a positioning block with R0.



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

Compensation value = $R + DR_{TOOL CALL} + DR_{TAB}$, where

R is the tool radius R from the TOOL DEF block or tool table

DR_{TOOL CALL} is the oversize for radius DR in the TOOL CALL block (not taken into account by the position display)

DR_{TAB} is the oversize for radius DR in the tool table

Tool movements without radius compensation: R0

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

Applications: Drilling and boring, pre-positioning (see figure at center right)

Tool movements with radius compensation: RR and RL

RR The tool moves to the right of the programmed contour

RL 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 illustrations on the next page).

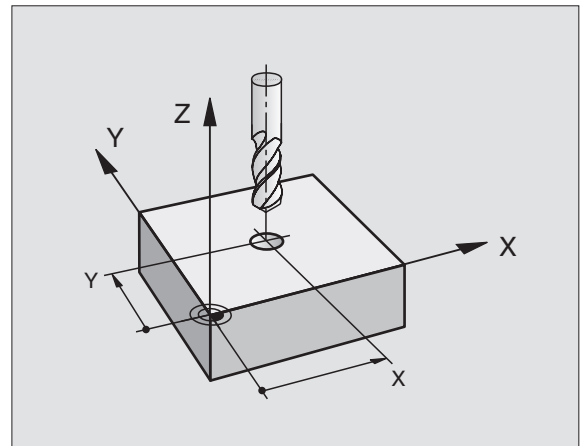
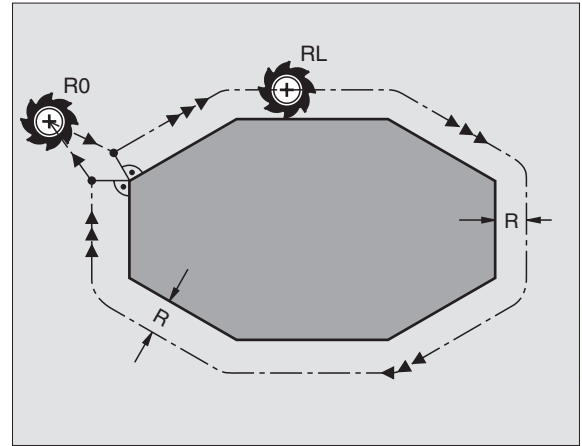


Between two program blocks with different radius compensations (RR and RL) you must program at least one block without radius compensation (that is, with R0).

Radius compensation does not come into 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 too in each following block, since otherwise the TNC will execute the radius compensation in the principal axis again.

Whenever radius compensation is activated with RR/RL or canceled with R0, 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

When you program a path contour, the following dialog question is displayed after entry of the coordinates:

Radius comp.: RL/RR/no comp. ?

RL

To select tool movement to the left of the contour, press the RL soft key, or

RR

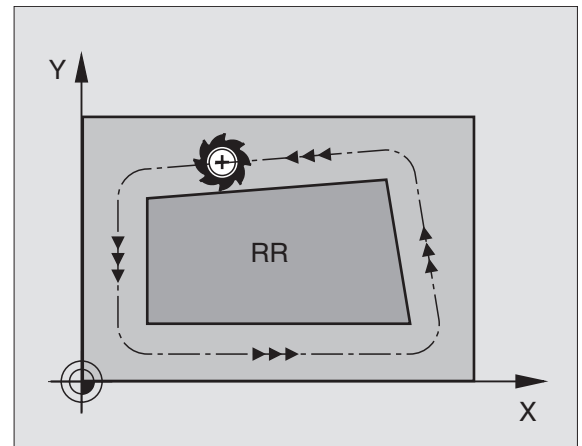
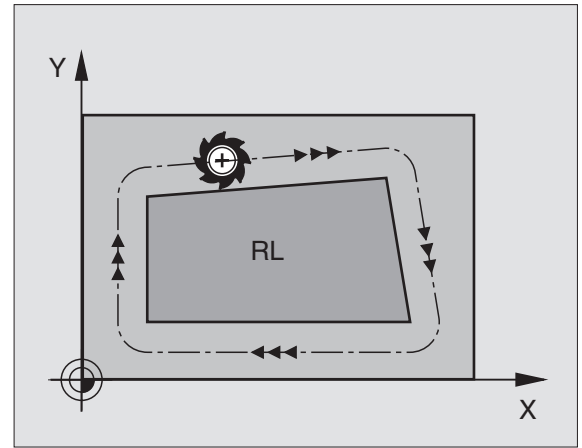
To select tool movement to the right of the contour, press the RR soft key, or

ENT

To select tool movement without radius compensation or to cancel radius compensation, press the ENT key or the R0 soft key.

END

To terminate the dialog, press the END key.



Radius compensation: Machining corners

Outside corners

If you program radius compensation, the TNC moves the tool in a transitional arc around corners. The tool “rolls around” the corner point. 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

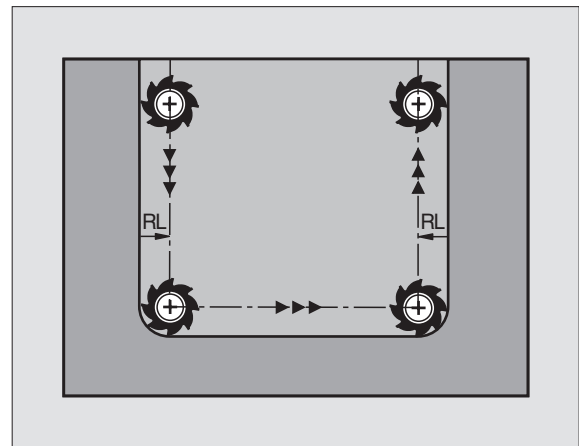
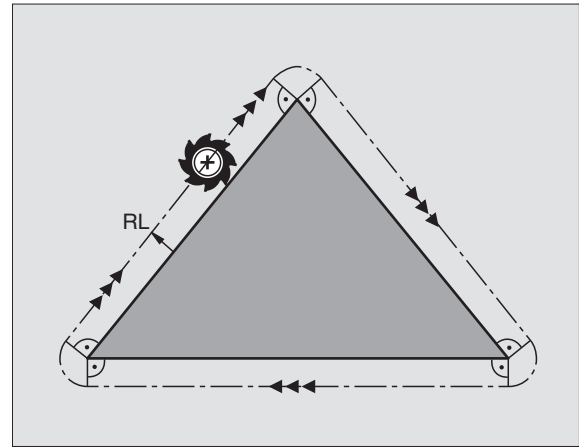
The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.



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 functions M90 and M112. See “7.4 Miscellaneous Functions for Contouring Behavior.”



5.4 Measuring Tools with the TT 120



The TNC and the machine tool must be set up by the machine tool builder for use of the TT 120 touch probe.

Some cycles and functions may not be provided on your machine tool. Refer to your machine manual.

In conjunction with the TNC's tool measurement cycles, the TT 120 enables you to measure tools automatically. The compensation values for tool length and radius can be stored in the central tool file TOOL.T and are accounted for in the next tool call. The following types of tool measurement are provided:

- Tool measurement while the tool is at standstill
- Tool measurement while the tool is rotating
- Measurement of individual teeth

You can define the tool measurement cycles in the PROGRAMMING AND EDITING mode of operation. The following cycles are available:

- TCH PROBE 30.0 TT CALIBRATE
- TCH PROBE 31.0 TOOL LENGTH
- TCH PROBE 32.0 TOOL RADIUS



The measuring cycles can be used only when the central tool file TOOL.T is active.

Before working with the measuring cycles, you must first enter all the required data into the central tool file and call the tool to be measured with TOOL CALL.

Setting the machine parameters



The TNC uses the feed rate for probing defined in MP6520 when measuring a tool at standstill.

When measuring a rotating tool, the TNC automatically calculates the spindle speed and feed rate for probing.

The spindle speed is calculated as follows

$$n = \frac{MP6570}{r \cdot 0.0063}$$

where

- n = spindle speed in rpm
- $MP6570$ = maximum permissible cutting speed in m/min
- r = active tool radius in mm

The feed rate for probing is calculated from:

$$v = \text{meas. tolerance} \cdot n$$

where

- v = feed rate for probing in mm/min
- meas. tolerance = meas. tolerance in mm, depending on MP6507
- n = speed in rpm

MP6507 determines the calculation of the probing feed rate:

MP6507=0:

The measuring tolerance remains constant — regardless of the tool radius. With very large tools, however, the feed rate for probing is reduced to zero. The smaller you set the maximum permissible rotational speed (MP6570) and the permissible tolerance (MP6510), the sooner you will encounter this effect.

MP6507=1:

The measuring tolerance is adjusted relative to the size of the tool radius. This ensures a sufficient feed rate for probing even with large tool radii. The TNC adjusts the measuring tolerance according to the following table:

Tool radius	Meas. tolerance
up to 30 mm	MP6510
30 to 60 mm	2 • MP6510
60 to 90 mm	3 • MP6510
90 to 120 mm	4 • MP6510

MP6507=2:

The feed rate for probing remains constant, the error of measurement, however, rises linearly with the increase in tool radius:

$$\text{Measuring tolerance} = \frac{r \cdot MP6510}{5 \text{ mm}}$$

where

- r = tool radius in mm
- MP6510= maximum permissible error of measurement

Displaying the measuring results

You can display the results of tool measurement in the additional status display when your screen layout includes PGM and T PROBE STATUS (in the machine operating modes). The TNC then shows the program blocks in the left and the measuring results in the right screen window. The measuring results that exceed the permissible wear tolerance are marked in the status display with an asterisk “*”; the results that exceed the permissible breakage tolerance are marked with the character B.

Calibrating the TT 120 touch probe



Before calibrating the touch probe, you must enter the exact length and radius of the calibrating tool into the tool table TOOL.T.




The position of the TT 120 within the machine working space must be defined by setting the machine parameters 6580.0 to 6580.2.

If you change the setting of any of the machine parameters 6580.0 to 6580.2, you must recalibrate the TT 120.

The TT 120 is calibrated automatically with the measuring cycle TCH PROBE 30. The TNC also measures the center misalignment of the calibrating tool automatically by rotating the spindle by 180° after the first half of the calibration cycle.
The calibrating tool must be a precisely cylindrical part, for example a cylinder pin. The resulting calibration values are stored in the TNC memory and are accounted for during subsequent tool measurement.



- ▶ To program the calibration cycle, press the TOUCH PROBE key in the Programming and Editing mode of operation.
- ▶ Select measuring cycle 30 TT CALIBRATE: Press TT CALIBR. soft key
- ▶ Clearance height: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540)

Program run, full sequence									
BEGIN PGM STATUS MM 1 BLK FORM 0.1 Z X+0 Y+0 Z-40 2 BLK FORM 0.2 X+100 Y+100 Z+0 3 TOOL CALL 1 Z S4000 DL+0.05 DR+0.04 4 L Z+100 R0 FMAX 5 L X-20 Y+50 R0 FMAX 6 L Z-2 R0 FMAX M3 7 LBL 12 8 CYCL DEF 7.0 DATUM SHIFT 9 CYCL DEF 7.1 X+25.5 10 CYCL DEF 7.2 Y+10 11 CYCL DEF 7.3 Z+12					Tool data T 2 SCHRUPPER				
					 L MIN 2 +1.9664 MAX 3 +2.0035 DYN				
					1 +1.9909 2 +1.9664 * 3 +2.0035 4 +1.9986				
NOML. X -244.710 Y -9.205 Z +213.270					T 2 Z F 0 S ROT M5/9				
BLOCKWISE TRANSFER					RESTORE POS. AT N	 ON OFF	 ON OFF	TOOL TABLE	

NC sample blocks

6	TOOL CALL 1 Z
7	TCH PROBE 30.0 TT CALIBRATE
8	TCH PROBE 30.1 HEIGHT: +90

Measuring the tool length

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The tool length is measured with the cycle TCH PROBE 31 TOOL LENGTH. Depending on the input parameters you can measure the length of a tool by one of the following methods:

- If the tool diameter is larger than the diameter of the measuring surface of the TT 120, you can measure the tool while it is rotating (Enter TT:R-OFFS = R in the TOOL.T).
- If the tool diameter is smaller than the diameter of the measuring surface of the TT 120, or if you are measuring the length of a drill or spherical cutter, you can measure the tool while it is at standstill (Enter TT:R-OFFS = 0 in the TOOL.T).
- If the tool diameter is larger than the diameter of the measuring surface of the TT 120, you can measure the individual teeth of the tool while it is at standstill.

Measuring cycle for measuring a tool during rotation

The TNC determines the longest tooth of a rotating tool by positioning the tool to be measured at an offset to the center of the touch probe system and then moving it toward the measuring surface until it contacts the surface. The offset is programmed in the tool table under Tool offset: Radius (TT: R-OFFS; preset value: R = tool radius).

Measuring cycle for measuring a tool during standstill (e.g. for drills)

The TNC positions the tool to be measured over the center of the measuring surface. It then moves the tool toward the measuring surface of the TT 120 (without rotation of the spindle) until the tool contacts the surface. To activate this function, enter zero for the Tool offset: Radius (TT: R-OFFS) in the tool table.

Measuring cycle for measuring individual teeth

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the tool to the upper edge of the touch probe head is defined in MP6530. You can enter an additional offset with Tool offset: Length (TT: L-OFFS) in the tool table. The TNC probes the tool radially during rotation to determine the starting angle for measuring the individual teeth. It then measures the length of each tooth by changing the corresponding angle of spindle orientation. To activate this function, program TCH PROBE 31 = 1 for cutter measurement.



- ▶ To program the calibration cycle, press the TOUCH PROBE key in the Programming and Editing mode of operation.
- ▶ Select measuring cycle 31 TT TOOL LENGTH: Press TOOL LENGTH soft key
- ▶ Measure tool=0 / Check tool=1: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool length L in the central tool file TOOL.T by the delta value DL = 0. If you wish to inspect a tool, the TNC compares the measured length with the tool length L that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DL. The deviation can also be used for Q parameter Q115. If the delta value is greater than the permissible tool length tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).
- ▶ Parameter number for result?: Parameter number in which the TNC stores the status of the measurement:
 0.0: Tool is within tolerance
 1.0: Tool is worn out (LTOL exceeded)
 2.0: Tool is broken (LBREAK exceeded)
 If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- ▶ Clearance height: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540)
- ▶ Probe the teeth? 0=No / 1=Yes:
 Choose whether or not the TNC is to measure the individual teeth.

Resulting NC blocks for measuring a rotating tool for the first time and storing the status in Q1

```

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. tool length
8 TCH PROBE 31.1 CHECK: 0 Q1
9 TCH PROBE 31.2 height: +120
10 TCH PROBE 31.3 PROBING THE TEETH:0
  
```

Resulting NC blocks for inspecting a tool and measuring the individual teeth, but not saving the status

```

6 TOOL CALL 12 Z
7 TCH PROBE 31.0 CAL. tool length
8 TCH PROBE 31.1 check: 1
9 TCH PROBE 31.2 height: +120
10 TCH PROBE 31.3 PROBING THE TEETH:1
  
```

Measuring the tool radius

Before measuring a tool for the first time, enter the following data on the tool into the tool table TOOL.T: the approximate radius, the approximate length, the number of teeth, and the cutting direction.

The tool radius is measured with the cycle TCH PROBE 32 TOOL RADIUS. Depending on the input parameters you can measure the radius of a tool by:

- Measuring the tool while it is rotating
- Measuring the tool while it is rotating and subsequently measuring the individual teeth.

Measuring cycle

The TNC pre-positions the tool to be measured to a position at the side of the touch probe head. The distance from the tip of the milling tool to the upper edge of the touch probe head is defined in MP6530. The TNC probes the tool radially while it is rotating. If you have programmed a subsequent measurement of individual teeth, the TNC measures the radius of each tooth with the aid of oriented spindle stops.



- ▶ To program the measuring cycle, press the TOUCH PROBE key in the Programming and Editing mode of operation.
- ▶ Select measuring cycle 32 TT TOOL RADIUS: Press TOOL RADIUS soft key
- ▶ Measure tool=0 / Check tool=1: Select whether the tool is to be measured for the first time or whether a tool that has already been measured is to be inspected. If the tool is being measured for the first time, the TNC overwrites the tool radius R in the central tool file TOOL.T by the delta value DR = 0. If you wish to inspect a tool, the TNC compares the measured radius with the tool radius R that is stored in TOOL.T. It then calculates the positive or negative deviation from the stored value and enters it into TOOL.T as the delta value DR. The deviation can also be used for Q parameter Q116. If the delta value is greater than the permissible tool radius tolerance for wear or break detection, the TNC will lock the tool (status L in TOOL.T).

Resulting NC blocks for measuring a rotating tool for the first time and storing the status in Q1

```
7 TOOL CALL 12 Z
8 TCH PROBE 32.0 CAL. tool radius
9 TCH PROBE 32.1 CHECK: 0 Q1
10 TCH PROBE 32.2 height: +120
11 TCH PROBE 32.3 PROBING THE TEETH:0
```

Resulting NC blocks for inspecting a tool and measuring the individual teeth, but not saving the status

```
7 TOOL CALL 12 Z
8 TCH PROBE 32.0 CAL. tool radius
9 TCH PROBE 32.1 check: 1
10 TCH PROBE 32.2 height: +120
11 TCH PROBE 32.3 PROBING THE TEETH:1
```


- ▶ Parameter number for result?: Parameter number in which the TNC stores the status of the measurement:
 0.0: Tool is within tolerance
 1.0: Tool is worn (RTOL exceeded)
 2.0: Tool is broken (RBREAK exceeded)
 If you do not wish to use the result of measurement within the program, answer the dialog prompt with NO ENT.
- ▶ Clearance height: Enter the position in the spindle axis at which there is no danger of collision with the workpiece or fixtures. The clearance height is referenced to the active workpiece datum. If you enter such a small clearance height that the tool tip would lie below the level of the probe contact, the TNC automatically positions the tool above the level of the probe contact (safety zone from MP6540)
- ▶ Cutter measurement 0=No / 1=Yes:
 Choose whether or not the TNC is to measure the individual teeth.



6

Programming:
Programming Contours

6.1 Overview of 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**.

Free contour (FK) programming

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

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

Miscellaneous functions M

With the TNC's miscellaneous functions you can affect

- program run, such as 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.

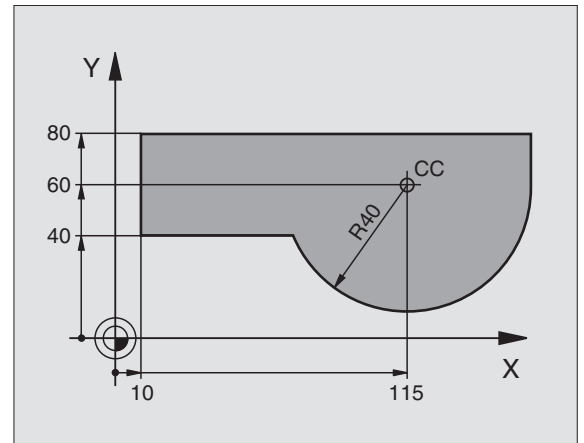
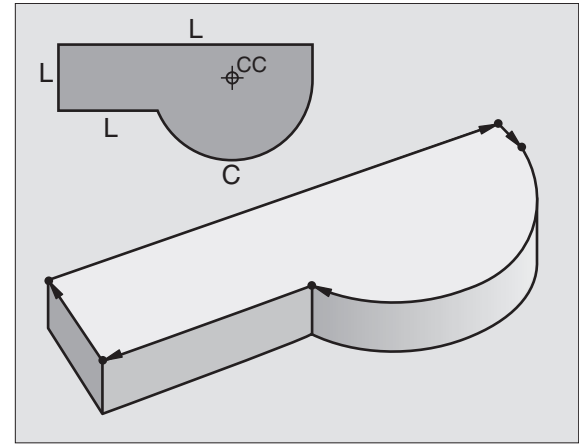
How subprograms and program section repeats are used in programming is described in Chapter 9.

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 parameter 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, and from 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 individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

L X+100

L Path function for "straight line"

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:

L X+70 Y+50

The tool retains the Z coordinate and moves in the X/Y plane to the position X=70, Y=50. See figure at center right.

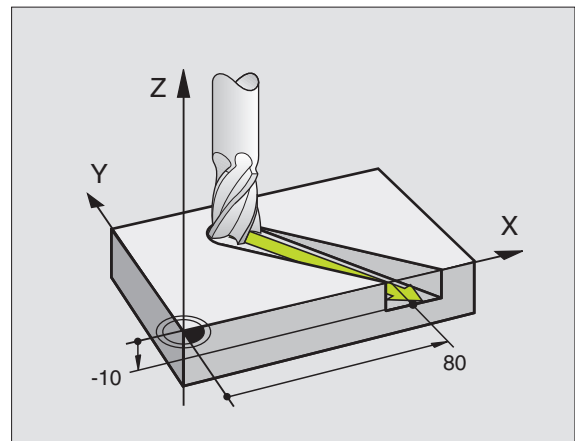
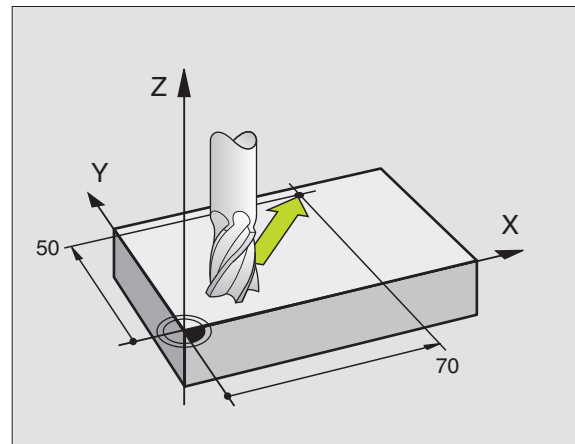
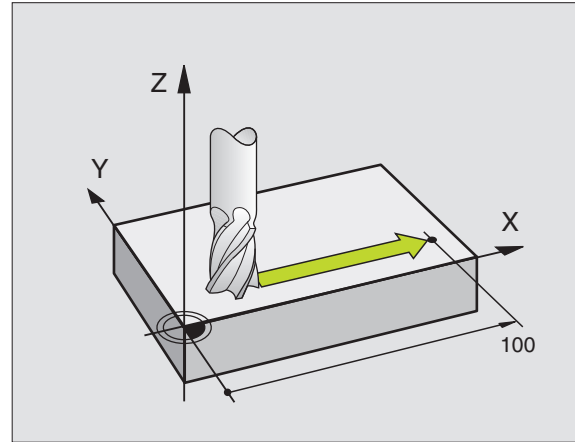
Three-dimensional movement

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

Example:

L X+80 Y+0 Z-10

This example is illustrated in the figure at lower right.



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 the circle center CC.

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:

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



You can program circles that do not lie parallel to a main plane by using Q parameters (see Chapter 10).

Direction of rotation DR for circular movements

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

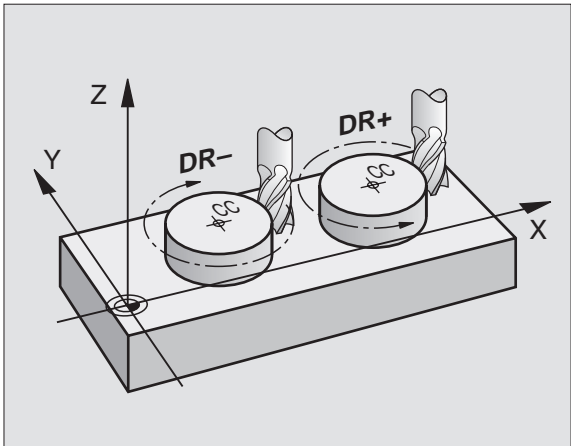
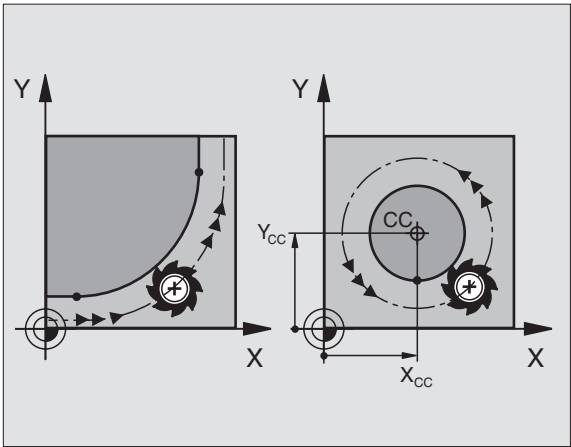
Clockwise direction of rotation: DR-
Counterclockwise direction of rotation: DR+

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 or approach block (APPR block).

Pre-positioning

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



Creating the program blocks with the path function keys

The gray path function keys initiate the plain language dialog. The TNC asks you successively for all the necessary information and inserts the program block into the part program.

Example — programming a straight line:



Initiate the programming dialog (here, for a straight line).

Coordinates ?



10

Enter the coordinates of the straight-line end point.



5



2x

Radius comp.: RL/RR/no comp. ?



Select the radius compensation (here, press the RL soft key — the tool moves to the left of the programmed contour).

Feed rate ? F=

100



Enter the feed rate (here, 100 mm/min), and confirm your entry with ENT.

Miscellaneous function M ?

3



Enter a miscellaneous function (here, M3), and terminate the dialog with END.



Enter a miscellaneous function with parameter: e.g. press the soft key M120 and enter the requested parameter.

The part program now contains the following line:









L X+10 Y+5 RL F100 M3

Programming and editing							
Miscellaneous function M?							
1	BLK	FORM	0.1	Z	X+0	Y+0	Z-40
2	BLK	FORM	0.2		X+100	Y+100	Z+0
3	TOOL	DEF	100	L+10	R+5		
4	TOOL	CALL	100	Z	S4000		
5	L	X+10	Y+5	R0	F100	M3	
END PGM 1568T MM							
ACTL.		X	-219.715				
		Y	+0.285				
		Z	+212.680				
		T	2	Z			
		F	0				
		S				M5 / 9	
M	M103	M112	M120	M124			

6.3 Contour Approach and Departure

Overview: Types of paths for contour approach and departure

The functions for contour approach and departure are activated with the APPR/DEP key. You can then select the desired path function with the corresponding soft key:

Function	Soft keys:	Approach	Departure
Straight line with tangential connection			
Straight line perpendicular to a contour point			
Circular arc with tangential connection			
Circular arc with tangential connection to the contour. Approach and departure to an auxiliary point outside of the contour on a tangentially connecting line.			

Approaching and departing a helix

The tool approaches and departs a helix on its extension by moving in a circular arc that connects tangentially to the contour. You program helix approach and departure with the APPR CT and DEP CT functions.

Important positions for approach and departure









- Starting point P_S
You program this position in the block before the APPR block. P_S lies outside the contour and is approached without radius compensation (R0).
- Auxiliary point P_H
Some of the paths for approach and departure go through an auxiliary point P_H that the TNC calculates from your input in the APPR or DEP block.
- First contour point P_A and last contour point P_E
You program the first contour point P_A in the APPR block. The last contour point P_E can be programmed with any path function.
- If the APPR block also contains a Z axis coordinate, the TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.
- End point P_N
The position P_N lies outside of the contour and results from your input in the DEP block. If the DEP block also contains a Z axis coordinate, the TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.

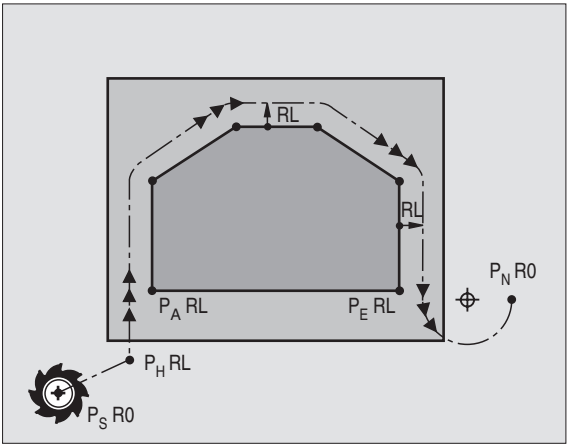
Programming and editing

```
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL DEF 100 L+10 R+5
4 TOOL CALL 100 Z S4000
5 L Z+50 R0 FMAX
END PGM 1568T MM
```

ACTL.	X	-219.715			
	Y	+0.285			
	Z	+212.680			

T	2	Z
F	0	
S		M5/9

							
---	--	---	---	---	---	---	---



You can enter the position data in absolute or incremental coordinates and in Cartesian or polar coordinates.

The TNC does not check whether the programmed contour will be damaged when moving from the actual position to the auxiliary point P_H . Use the test graphics to simulate approach and departure before executing the part program.

When approaching the contour, allow sufficient distance between the starting point P_S and the first contour point P_A to assure that the TNC will reach the programmed feed rate for machining.

The TNC moves the tool from the actual position to the auxiliary point P_H at the feed rate that was last programmed.

Radius compensation

For the TNC to interpret an APPR block as an approach block you must program a change in compensation from R0 to RL/RR. The TNC automatically cancels the radius compensation in a DEP block. If you wish to program a contour element with the DEP block (no change in compensation), then you need to program the active radius compensation again (2nd soft-key row, if the F element is highlighted).

If no change in compensation is programmed in an APPR or in a DEP block, the TNC makes the contour connection as follows:


Function	Contour connection
APPR LT	Tangential connection to the following contour element
APPR LN	Perpendicular connection to the following contour element
APPR CT	without angle of traverse/without radius: Tangentially connecting circular arc between the preceding and the following contour element. without angle of traverse/with radius: Tangentially connecting circular arc with programmed radius to the following contour element with angle of traverse/without radius: Tangentially connecting circular arc with angle of traverse to the following contour element with angle of traverse/with radius: Tangentially connecting circular arc with connecting line and angle of traverse to the following contour element
APPR LCT	Tangent with connecting tangential circular arc to the following contour element

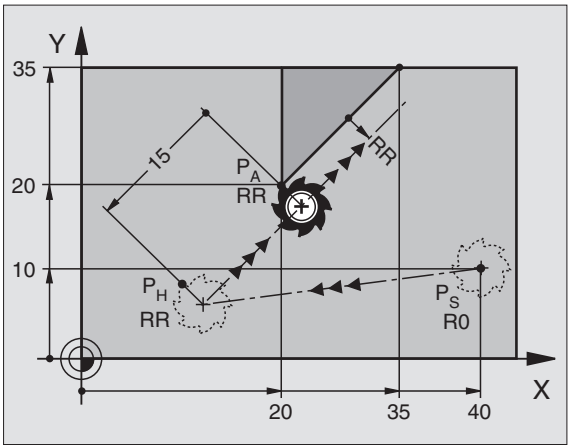
Abbreviation	Meaning
APPR	Approach
DEP	Departure
L	Line
C	Circle
T	Tangential (smooth connection)
N	Normal (perpendicular)

Function	Contour connection
DEP LT	Tangential connection to the preceding contour element
DEP LN	Perpendicular connection to the preceding contour element
DEP CT	without angle of traverse/without radius: Tangentially connecting circular arc between the preceding and the following contour element without angle of traverse/with radius: Tangentially connecting circular arc with programmed radius to the preceding contour element with angle of traverse/without radius: Tangentially connecting circular arc with angle of traverse to the preceding contour element with angle of traverse/with radius: Tangentially connecting circular arc with connecting line and angle of traverse to the preceding contour element
DEP LCT	Tangent with tangentially connecting circular arc to the preceding contour element

Approaching on a straight line with tangential connection: APPR LT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H . It then moves from P_H to the first contour point P_A on a straight line that connects tangentially to the contour. The auxiliary point P_H is separated from the first contour point P_A by the distance LEN.

- Use any path function to approach the starting point P_S .
- 
 - Initiate the dialog with the APPR/DEP key and APPR LT soft key:
 - Coordinates of the first contour point P_A
 - LEN: Distance from the auxiliary point P_H to the first contour point P_A
 - Radius compensation for machining




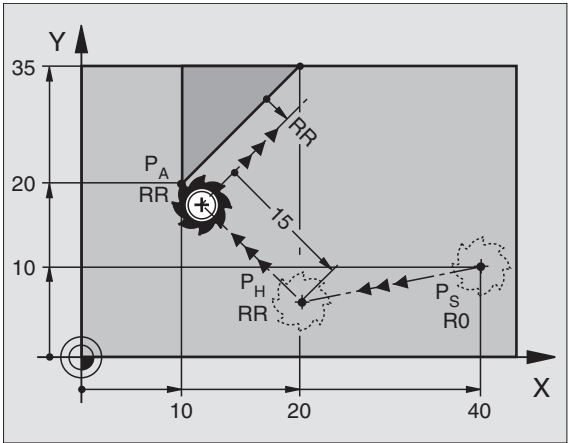
Example NC blocks

7 L X+40 Y+10 R0 FMAX M3	Approach P_S without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	P_A with radius comp. RR
9 L X+35 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a straight line perpendicular to the first contour point: APPR LN

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H . It then moves from P_H to the first contour point P_A on a straight line perpendicular to the first contour element. The auxiliary point P_H is separated from the first contour point P_A by the distance LEN plus the tool radius.

- Use any path function to approach the starting point P_S .
- Initiate the dialog with the APPR/DEP key and APPR LN soft key:
- 
 - Coordinates of the first contour point P_A
 - Length: Distance from the auxiliary point P_H to the first contour point P_A
Always enter LEN as a positive value!
 - Radius compensation RR/RL for machining



Example NC blocks

7 L X+40 Y+10 R0 FMAX M3	Approach P_S without radius compensation
8 APPR LN X+10 Y+20 Z-10 LEN+15 RR F100	P_A with radius comp. RR, distance P_H to P_A : LEN=15
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a circular arc with tangential connection: APPR CT

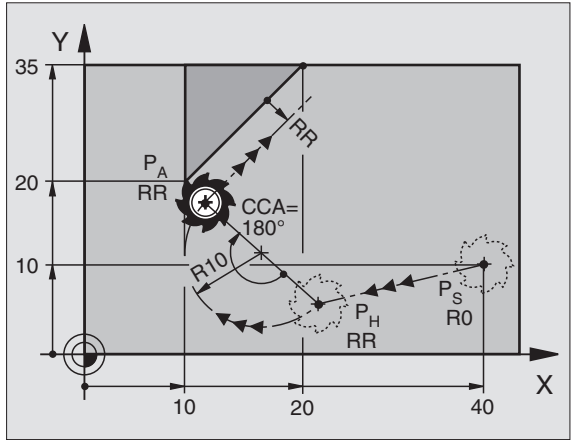
The tool moves on a straight line from the starting point P_S to an auxiliary point P_H . It then moves from P_H to the first contour point P_A following a circular arc that is tangential to the first contour element.

The arc from P_H to P_A is determined through the radius R and the center angle CCA . The direction of rotation of the circular arc is automatically derived from the tool path for the first contour element.

- Use any path function to approach the starting point P_S .
- Initiate the dialog with the APPR/DEP key and APPR CT soft key:



- Coordinates of the first contour point P_A
- Center angle CCA of the arc
 - CCA can be entered only as a positive value.
 - Maximum input value 360°
- Radius R of the circular arc
 - If the tool should approach the workpiece in the direction defined by the radius compensation: Enter R as a positive value.
 - If the tool should approach the workpiece opposite to the radius compensation: Enter R as a negative value.
- Radius compensation RR/RL for machining



Example NC blocks

7 L X+40 Y+10 R0 FMAX M3	Approach P_S without radius compensation
8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100	P_A with radius comp. RR , Radius $R=10$
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT

The tool moves on a straight line from the starting point P_S to an auxiliary point P_H . It then moves from P_H to the first contour point P_A on a circular arc.

The arc is connected tangentially both to the line $P_S - P_H$ as well as to the first contour element. Once these lines are known, the radius then suffices to completely define the tool path.

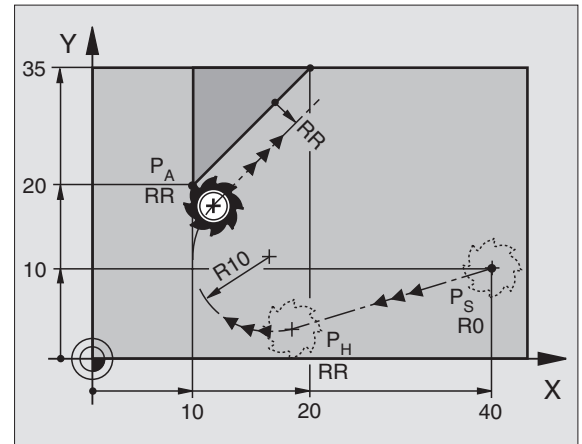
- Use any path function to approach the starting point P_S .
- Initiate the dialog with the APPR/DEP key and APPR LCT soft key:



- Coordinates of the first contour point P_A
- Radius R of the arc
Always enter R as a positive value.
- Radius compensation for machining

Example NC blocks

7 L X+40 Y+10 R0 FMAX M3	Approach P_S without radius compensation
8 APPR LCT X+10 Y+20 Z-10 R10 RR F100	P_A with radius compensation RR , radius $R=10$
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element



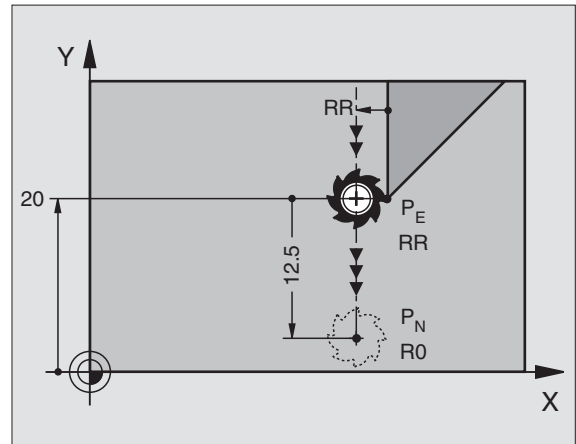
Departing tangentially on a straight line: DEP LT

The tool moves on a straight line from the last contour point P_E to the end point P_N . The line lies in the extension of the last contour element. P_N is separated from P_E by the distance LEN.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and DEP LT soft key:



- LEN: Enter the distance from the last contour element P_E to the end point P_N .



Example NC blocks

23 L Y+20 RR F100

Last contour element: P_E with radius compensation

24 DEP LT LEN12.5 F100

Depart contour by LEN = 12.5 mm

25 L Z+100 FMAX M2

Retract in Z, return to block 1, end program

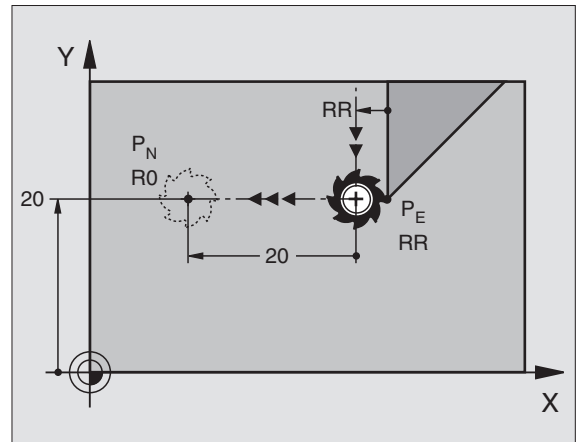
Departing on a straight line perpendicular to the last contour point: DEP LN

The tool moves on a straight line from the last contour point P_E to the end point P_N . The line departs on a perpendicular path from the last contour point P_E . P_N is separated from P_E by the distance LEN plus the tool radius.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and DEP LN soft key:



- LEN: Enter the distance from the last contour element P_E to the end point P_N . Important: Always enter LEN as a positive value!



Example NC blocks

23 L Y+20 RR F100

Last contour element: P_E with radius compensation

24 DEP LN LEN+20 F100

Depart perpendicular to contour by LEN = 20 mm

25 L Z+100 FMAX M2

Retract in Z, return to block 1, end program

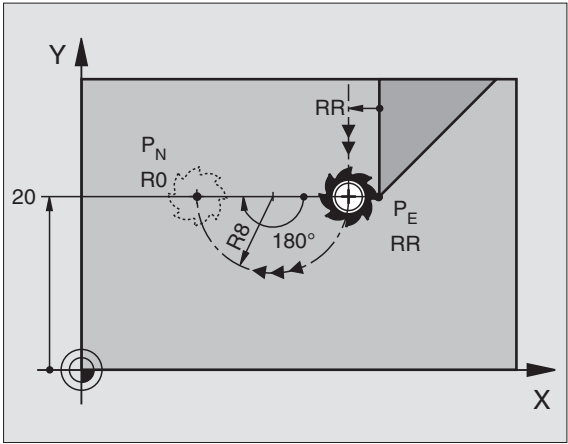
Departing tangentially on a circular arc: DEP CT

The tool moves on a circular arc from the last contour point P_E to the end point P_N . The arc is tangentially connected to the last contour element.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and DEP CT soft key:



- Center angle CCA of the arc
- Radius R of the circular arc
- If the tool should depart the workpiece in the direction of the radius compensation (i.e. to the right with RR or to the left with RL):
Enter R as a positive value.
- If the tool should depart the workpiece on the direction **opposite** to the radius compensation:
Enter R as a negative value.



Example NC blocks

23 L Y+20 RR F100	Last contour element: P_E with radius compensation
24 DEP CT CCA 180 R+8 F100	Center angle=180°, arc radius= 8 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

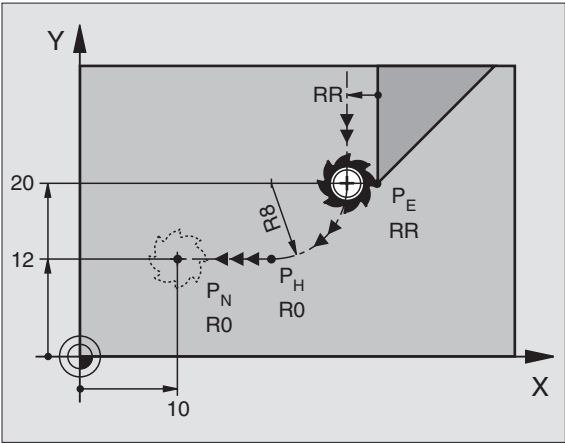
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT

The tool moves on a circular arc from the last contour point P_E to an auxiliary point P_H . It then moves from P_H to the end point P_N on a straight line. The arc is tangentially connected both to the last contour element and to the line from P_H to P_N . Once these lines are known, the radius R then suffices to completely define the tool path.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and DEP LCT soft key:



- Enter the coordinates of the end point P_N .
- Radius R of the arc
Always enter R as a positive value











Example NC blocks

23 L Y+20 RR F100	Last contour element: P_E with radius compensation
24 DEP LCT X+10 Y+12 R8 F100	Coordinates P_N , arc radius = 8 mm
25 L Z+100 FMAX M2	Retract in Z, return to block 1, end program

6.4 Path Contours — Cartesian Coordinates

Overview of path functions

Function	Path function key	Tool movement	Required input
Line L		Straight line	Coordinates of the straight-line end point
CHamFer		Chamfer between two straight lines	Chamfer side length
Circle C enter		No tool movement	Coordinates of the circle center or pole
Circle		Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation
Circle by R adius		Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation
Circle T angential		Circular arc with tangential connection to the preceding contour element	Coordinates of the arc end point
Corner RouNDing		Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding-off radius R
FK free contour programming		Straight line or circular path with any connection to the preceding contour element	See section 6.6

Straight line L

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



► Enter the coordinates of the end point for the line.

Further entries, if necessary:

► Radius compensation RL/RR/R0

► Feed rate F

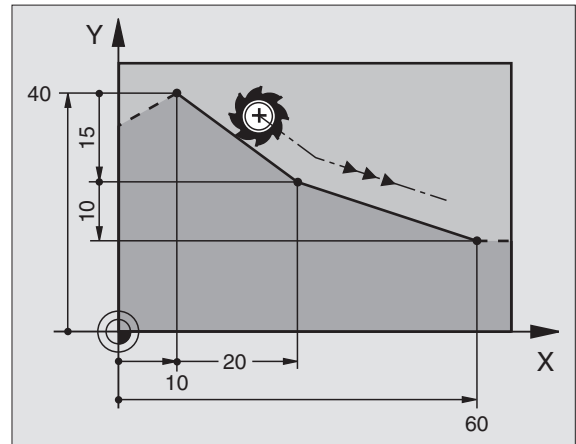
► Miscellaneous function M

Example NC blocks

```
7 L X+10 Y+40 RL F200 M3
```

```
8 L IX+20 IY-15
```

```
9 L X+60 IY-10
```



Actual position capture

In a positioning block you can capture the coordinates of the current tool position:

- Select the Programming and Editing mode of operation.
- Open a new block or move the highlight to a coordinate within an existing block.



► Press the actual-position-capture key: The TNC stores the coordinates of the highlighted axis.

Inserting a chamfer CHF between two straight lines

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

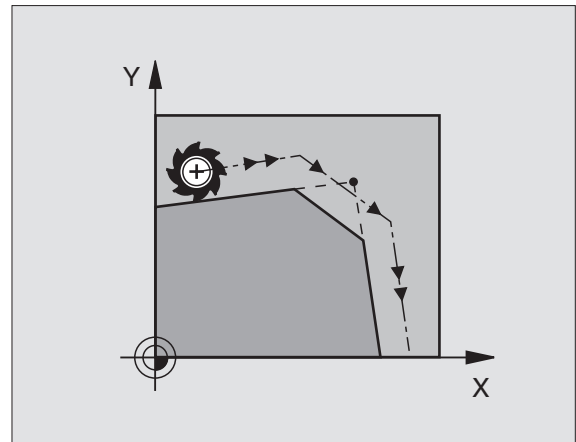
- The blocks before and after the CHF block must be in the same working plane.
- The radius compensation before and after the chamfer block must be the same.
- An inside chamfer must be large enough to accommodate the current tool.



► Chamfer side length: Enter the length of the chamfer

Further entries, if necessary:

► Feed rate F (only effective in CHF block)



Please observe the notes on the next page!

Example NC blocks

```
7 L X+0 Y+30 RL F300 M3
```

```
8 L X+40 IY+5
```

```
9 CHF 12
```

```
10 L IX+5 Y+0
```



You cannot start a contour with a CHF block

A chamfer is possible only in the working plane.

The feed rate for chamfering is taken from the preceding block.

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

Circle center CC

You can define a circle center CC for circles that are programmed with the C key (circular path C). This is done in the following ways:

- Entering the Cartesian coordinates of the circle center
- Using the circle center defined in an earlier block
- Capturing the coordinates with the actual-position-capture key



► Coordinates CC: Enter the circle center coordinates

If you want to use the last programmed position, do not enter any coordinates.

Example NC blocks

```
5 CC X+25 Y+25
```

or

```
10 L X+25 Y+25
```

```
11 CC
```

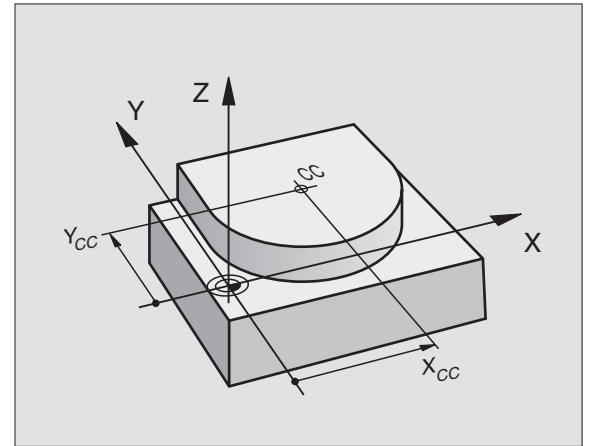
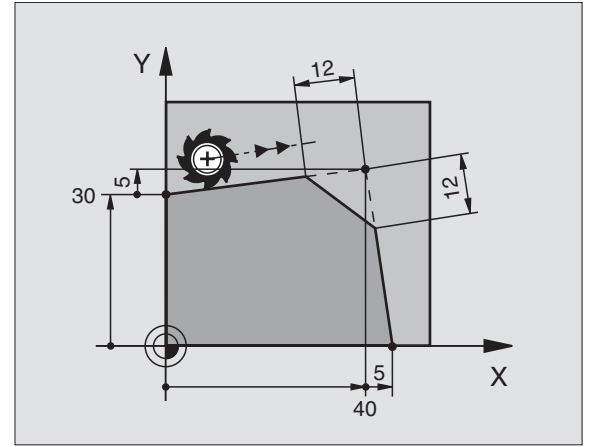
The program blocks 10 and 11 do not refer to the illustration.

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 the circle center CC incrementally

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





The only effect of CC is to define a position as a circle center — the tool does not move to the position.

The circle center also serves as the pole for polar coordinates.

Circular path C around circle center CC

Before programming a circular path C, you must first enter the circle center CC. The last programmed tool position before the C block is used as the circle starting point.

► Move the tool to the circle starting point.



► Enter the coordinates of the circle center.



► Enter the coordinates of the arc end point

► Direction of rotation DR

Further entries, if necessary:

► Feed rate F

► Miscellaneous function M

Example NC blocks

```
5 CC X+25 Y+25
```

```
6 L X+45 Y+25 RR F200 M3
```

```
7 C X+45 Y+25 DR+
```

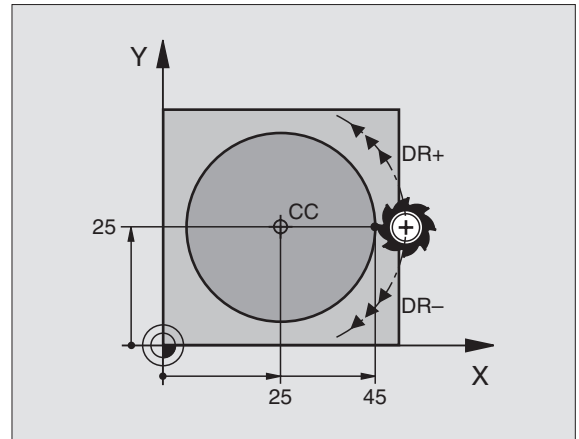
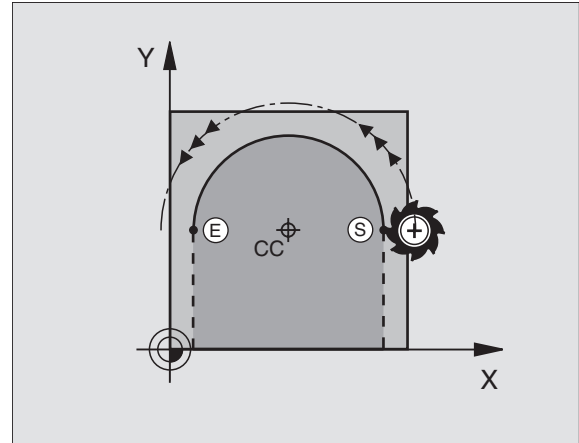
Full circle

Enter the same point you used as the starting point for the end point in a C block.



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

Input tolerance: up to 0.016 mm.



Circular path CR with defined radius

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



- ▶ Enter the coordinates of the arc end point.
 - ▶ Radius R
Note: The algebraic sign determines the size of the arc.
 - ▶ Direction of rotation DR
Note: The algebraic sign determines whether the arc is concave or convex.
- 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. See figure at upper right.

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: $CCA < 180^\circ$

Enter the radius with a positive sign $R > 0$

Larger arc: $CCA > 180^\circ$

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 DR- (with radius compensation RL)

Concave: Direction of rotation DR+ (with radius compensation RL)

Example NC blocks

See figures at middle and lower right.

10 L X+40 Y+40 RL F200 M3

11 CR X+70 Y+40 R+20 DR- (arc 1)

or

11 CR X+70 Y+40 R+20 DR+ (arc 2)

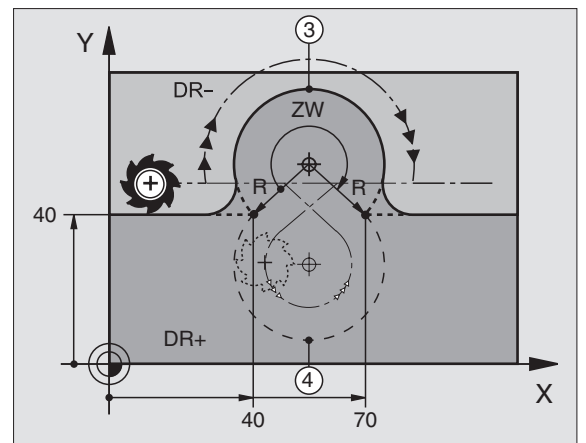
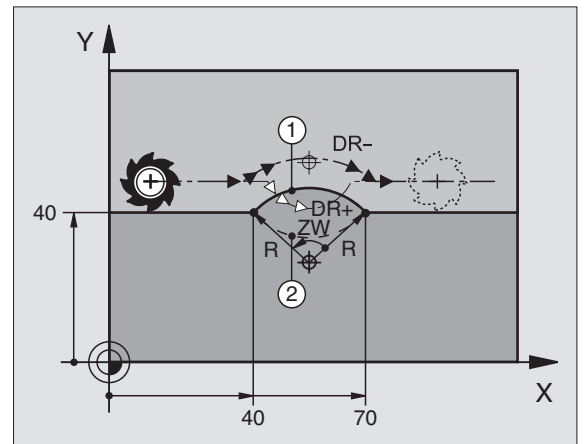
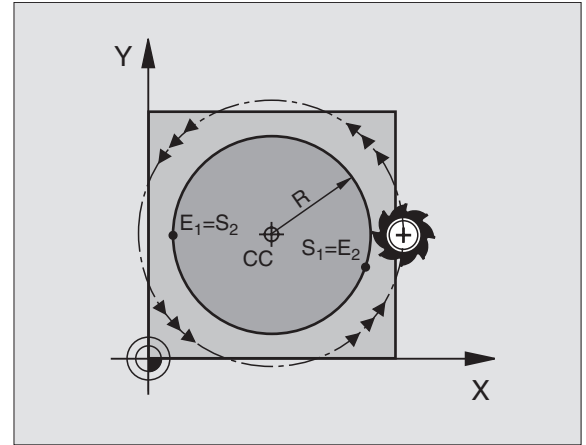
or

11 CR X+70 Y+40 R-20 DR- (arc 3)

or

11 CR X+70 Y+40 R-20 DR+ (arc 4)

Please observe the notes on the next page!





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 9999.999 mm.

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

Circular path CT with tangential connection

The tool moves on an arc that starts at a tangent with 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 CT block. This requires at least two positioning blocks.



► Enter the coordinates of the arc end point.

Further entries, if necessary:

► Feed rate F

► Miscellaneous function M

Example NC blocks

```
7 L X+0 Y+25 RL F300 M3
```

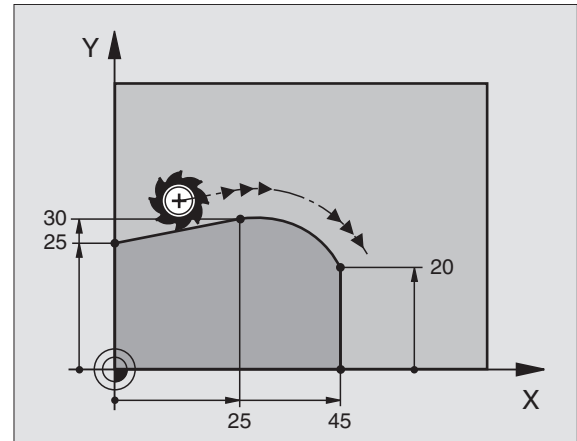
```
8 L X+25 Y+30
```

```
9 CT X+45 Y+20
```

```
10 L Y+0
```



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




Corner Rounding RND

The RND 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.




▶ Rounding-off radius: Enter the radius of the arc.

▶ Feed rate for rounding the corner.

Example NC blocks

```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5
```

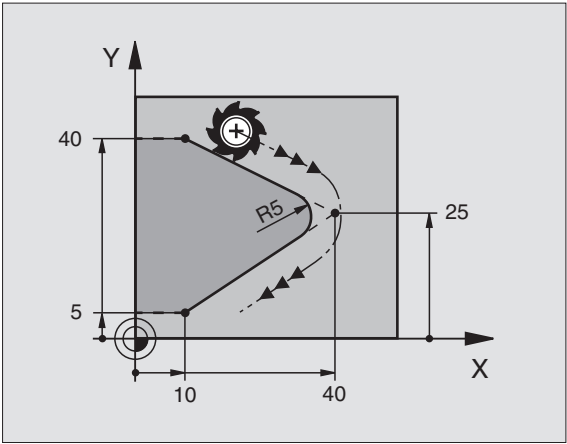


In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc.

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

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

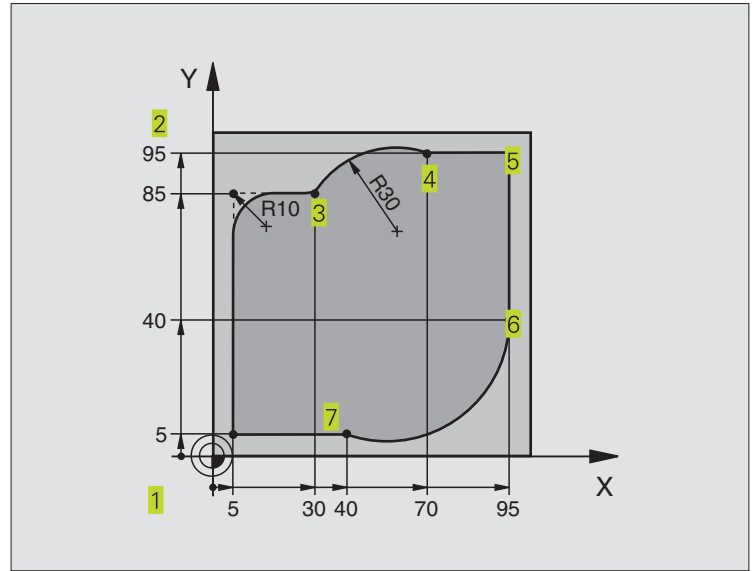
You can also use an RND block for a tangential contour approach if you do not want to use an APPR function.



6.4 Path Contours – Cartesian Coordinates

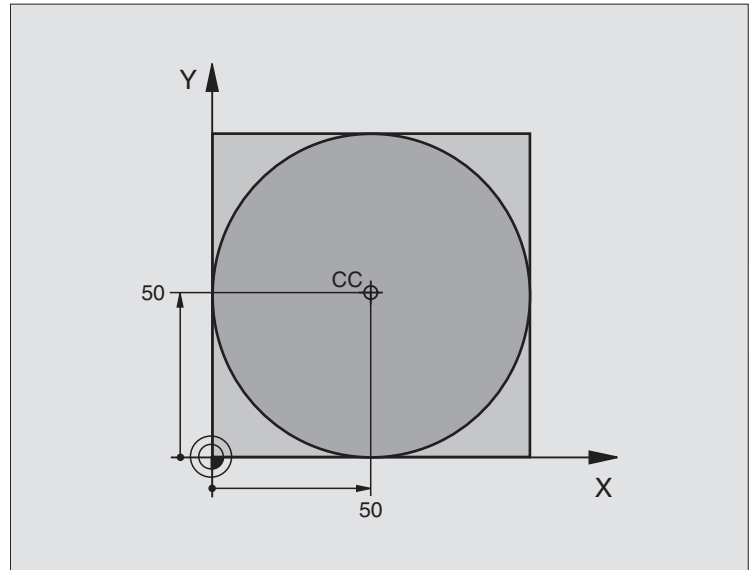
01

Example: Circular movements with Cartesian coordinates



0	BEGIN PGM CIRCULAR MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	T00L DEF 1 L+0 R+10	Define tool in the program
4	T00L CALL 1 Z S4000	Call tool in the spindle axis and with the spindle speed S
5	L Z+250 R0 FMAX	Retract tool in the spindle axis at rapid traverse FMAX
6	L X-10 Y-10 R0 F MAX	Pre-position the tool
7	L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
8	APPR LCT X+5 Y+5 R5 RL F300	Approach the contour at point 1 on a circular arc with tangential connection
9	L X+5 Y+85	Point 2: first straight line for corner 2
10	RND R10 F150	Insert radius with R = 10 mm, feed rate: 150 mm/min
11	L X+30 Y+85	Move to point 3: Starting point of the arc with CR
12	CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
13	L X+95	Move to point 5
14	L X+95 Y+40	Move to point 6
15	CT 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
16	L X+5	Move to last contour point 1
17	DEP LCT X-20 Y-20 R5 R0 F1000	Depart the contour on a circular arc with tangential connection
18	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
19	END PGM CIRCULAR MM	

Example: Full circle with Cartesian coordinates



0	BEGIN PGM C-CC MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+12.5	Define the tool
4	TOOL CALL 1 Z S3150	Call the tool
5	CC X+50 Y+50	Define the circle center
6	L Z+250 R0 F MAX	Retract the touch probe
7	L X-40 Y+50 R0 F MAX	Pre-position the tool
8	L Z-5 R0 F1000 M3	Move to working depth
9	APPR LCT X+0 Y+50 R5 RL F300	Approach the starting point of the circle on a circular arc with tangential connection
10	C X+0 DR-	Move to the circle end point (= circle starting point)
11	DEP LCT X-40 Y+50 R5 R0 F1000	Depart the contour on a circular arc with tangential connection
12	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
13	END PGM CCC MM	





6.5 Path Contours – Cartesian Coordinates

With polar coordinates you can define a position in terms of its angle PA and its distance PR relative to a previously defined pole CC. See section "4.1 Fundamentals of NC."

Polar coordinates are useful with:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

Function	Path function keys	Tool movement	Required input
Line LP	 + P	Straight line	Polar radius, polar angle of the straight-line end point
Circular arc CP	 + P	Circular path around circle center/ pole CC to arc end point	Polar angle of the arc end point, direction of rotation
Circular arc CTP	 + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helical interpolation	 + P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis

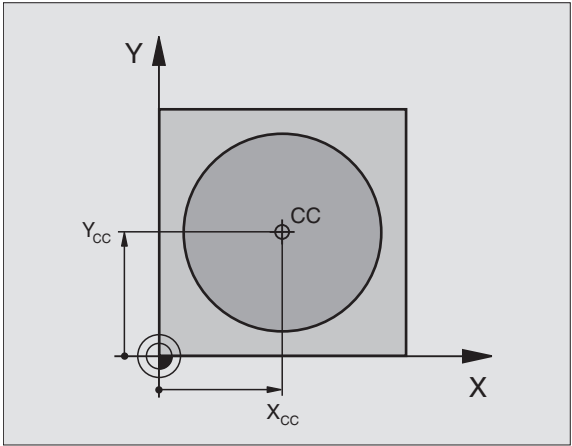
Polar coordinate origin: Pole CC

You can define the pole CC anywhere in the part program before blocks containing polar coordinates. Enter the pole in Cartesian coordinates as a circle center in a CC block.



- Coordinates CC: Enter Cartesian coordinates for the pole, or:

If you want to use the last programmed position, do not enter any coordinates.



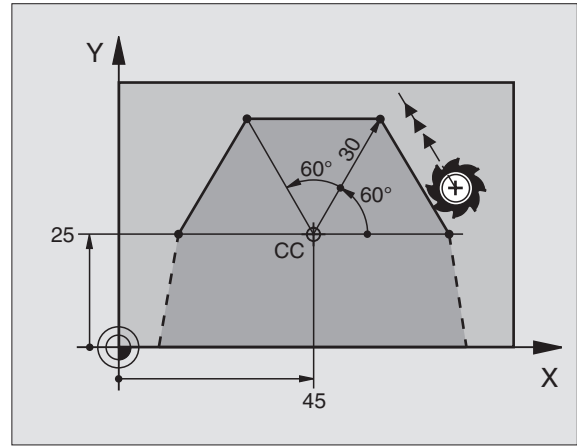
Straight line LP

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


P

- Polar-coordinates radius PR: Enter the distance from the pole CC to the straight-line end point.
- Polar-coordinates angle PA: Angular position of the straight-line end point between -360° and $+360^\circ$

The sign of PA depends on the angle reference axis:
Angle from angle reference axis to PR is counterclockwise: $PA > 0$
Angle from angle reference axis to PR is clockwise: $PA < 0$



Example NC blocks

```
12 CC X+45 Y+25
```

```
13 LP PR+30 PA+0 RR F300 M3
```

```
14 LP PA+60
```

```
15 LP IPA+60
```

```
16 LP PA+180
```

Circular path CP around pole CC

The polar coordinate radius PR is also the radius of the arc. It is defined by the distance from the starting point to the pole CC. The last programmed tool position before the CP block is the starting point of the arc.


P

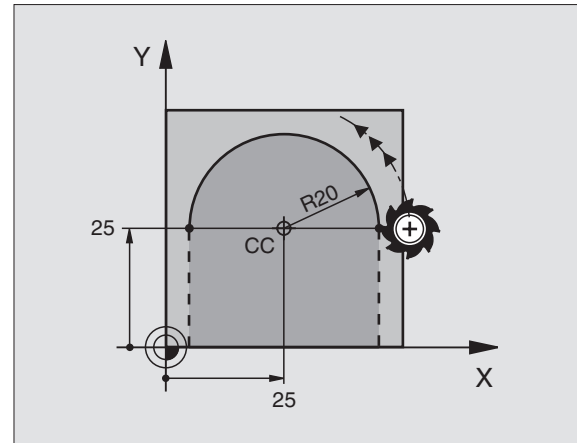
- Polar coordinates angle PA: Angular position of the arc end point between -5400° and $+5400^\circ$
- Direction of rotation DR

Example NC blocks

```
18 CC X+25 Y+25
```

```
19 LP PR+20 PA+0 RR F250 M3
```



```
20 CP PA+180 DR+
```



For incremental coordinates, enter the same sign for DR and PA.

Circular path CTP with tangential connection


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

- 
- Polar coordinates radius PR: Distance from the arc end point to the pole CC.

► Polar coordinates angle PA: Angular position of the arc end point.

Example NC blocks

```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```



The pole CC 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.

Applications

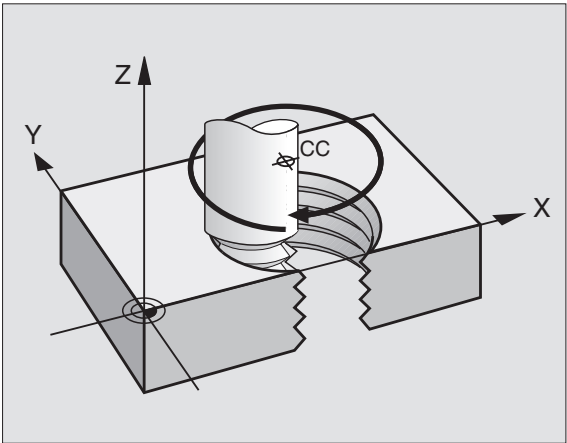
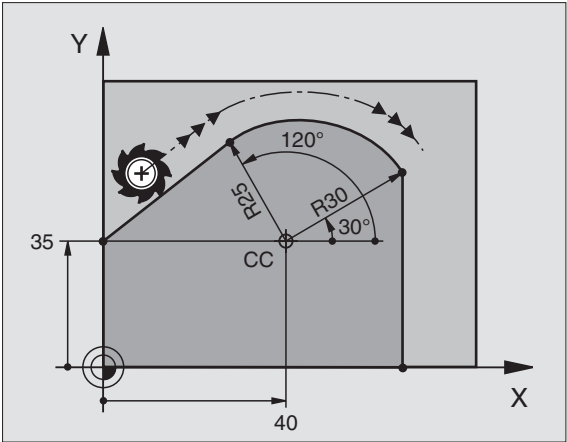
- Large-diameter internal and external threads
- Lubrication grooves

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 a upward direction, you need the following data:

Thread revolutions n	Thread revolutions + thread overrun at the start and end of the thread
Total height h	Thread pitch P x thread revolutions n
Incremental total angle IPA	Thread revolutions x 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Thread pitch P x (thread revolutions + thread overrun at the start of the thread)



Shape of the helix

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

Internal thread	Working dir.	Dir. of rotation	Radius comp.
Right-handed	Z+	DR+	RL
Left-handed	Z+	DR–	RR
Right-handed	Z–	DR–	RR
Left-handed	Z–	DR+	RL
External thread			
Right-handed	Z+	DR+	RR
Left-handed	Z+	DR–	RL
Right-handed	Z–	DR–	RL
Left-handed	Z–	DR+	RR

Programming a helix



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

For the total angle IPA, you can enter a value from -5400° to $+5400^\circ$. If the thread has of more than 15 revolutions, program the helix in a program section repeat (see section 9.3 "Program Section Repeats" and "Example: HELIX" later in this chapter).



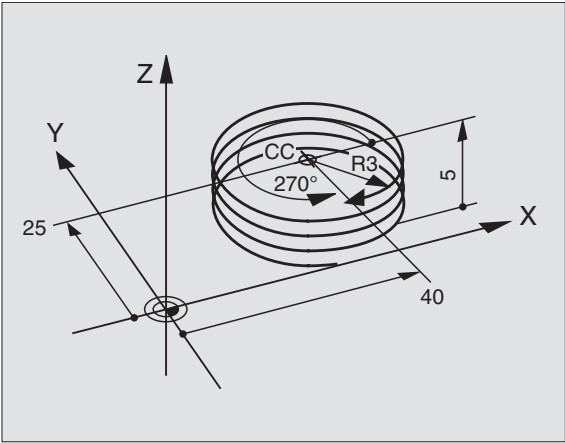
- Polar coordinates angle: Enter the total angle of tool traverse along the helix in incremental dimensions.
After entering the angle, identify the tool axis with an axis selection key.
- Enter the coordinate for the height of the helix in incremental dimensions.
- Direction of rotation DR
Clockwise helix: DR–
Counterclockwise helix: DR+
- Radius compensation RL/RR/R0
Enter the radius compensation according to the table above.

Example NC blocks

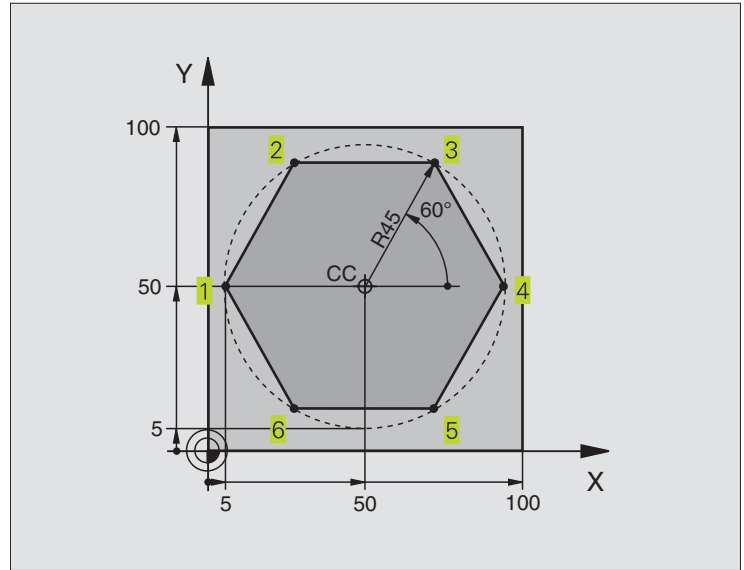
```

12 CC X+40 Y+25
13 Z+0 F100 M3
14 LP PR+3 PA+270 RL
15 CP IPA–1800 IZ+5 DR– RL F50

```

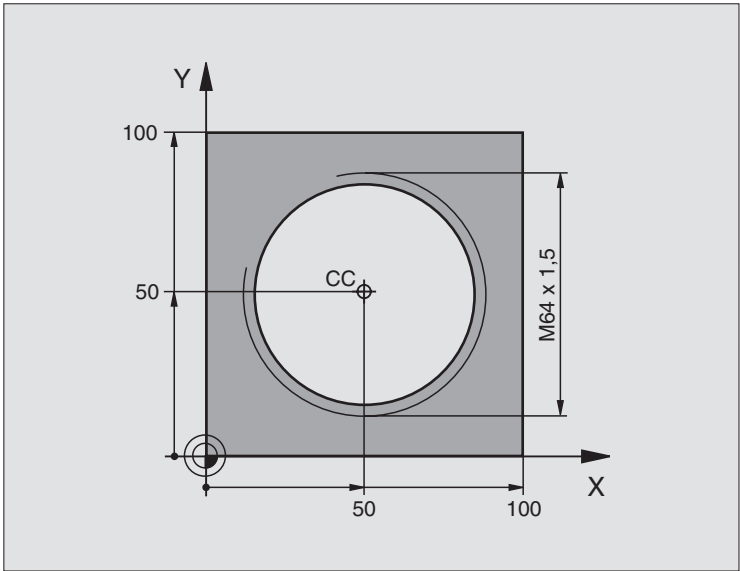


Example: Linear movement with polar coordinates



0	BEGIN PGM LINEARPO MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+7.5	Define the tool
4	TOOL CALL 1 Z S4000	Call the tool
5	CC X+50 Y+50	Define the datum for polar coordinates
6	L Z+250 R0 F MAX	Retract the touch probe
7	LP PR+60 PA+180 R0 F MAX	Pre-position the tool
8	L Z-5 R0 F1000 M3	Move to working depth
9	APPR LCT X+5 Y+50 R5 RL F250	Approach the contour at point 1 on a circular arc with tangential connection
10	LP PA+120	Move to point 2
11	LP PA+60	Move to point 3
12	LP PA+0	Move to point 4
13	LP PA-60	Move to point 5
14	LP PA-120	Move to point 6
15	LP PA+180	Move to point 1
16	DEP LCT X-15 Y+50 R5 R0 F1000	Depart the contour on a circular arc with tangential connection
17	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
18	END PGM LINEARPO MM	

Example: Helix



0	BEGIN PGM HELIX MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+5	Define the tool
4	TOOL CALL 1 Z S1400	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	L X+50 Y+50 R0 F MAX	Pre-position the tool
7	CC	Transfer the last programmed position as the pole
8	L Z-12.75 R0 F1000 M3	Move to working depth
9	APPR CT X+18 Y+50 CCA180 R+2	Approach the contour on a circular arc with tangential
	RL F100	connection
10	CP IPA+3240 IZ+13.5 DR+ F200	Helical interpolation
11	DEP CT CCA180 R+2 R0	Depart the contour on a circular arc with tangential connection
12	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
13	END PGM HELIX MM	

To cut a thread with more than 16 revolutions

...	
8	L Z-12.75 R0 F1000
9	APPR CT X+18 Y+50 CCA180 R+2 RL F100
10	LBL 1
	Identify beginning of program section repeat
11	CP IPA+360 IZ+1.5 DR+ F200
	Enter the thread pitch as an incremental IZ dimension
12	CALL LBL 1 REP 24
	Program the number of repeats (thread revolutions)
13	DEP CT CCA180 R+2 R0

6.6 Path Contours – FK Free Contour Programming

Fundamentals

Workpiece drawings that are not dimensioned for NC often contain unconventional coordinate data that cannot be entered with the gray path function keys. You may, for example, have only the following data on a specific contour element:

- Known coordinates on the contour element or in its proximity
- Coordinate data that are referenced to another contour element
- Directional data and data regarding the course of the contour

You can enter such dimensional data directly by using the FK free contour programming function. The TNC derives the contour from the known coordinate data and supports the programming dialog with the interactive programming graphics. The figure to the upper right shows a workpiece drawing for which FK programming is the most convenient programming method.



If you wish to run FK programs on old TNC models, use the conversion function (see „4.2 File Management; Converting an FK program into HEIDENHAIN conversational format“).

Graphics during FK programming

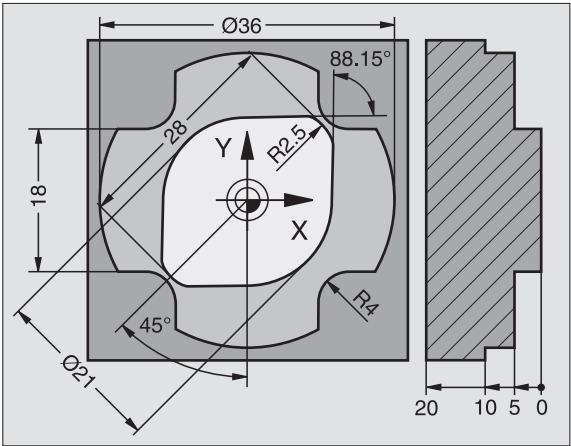
Incomplete coordinate data often are not sufficient to fully define a workpiece contour. In this case, the TNC indicates the possible solutions in the FK graphic. You can then select the contour that matches the drawing. The FK graphic displays the elements of the workpiece contour in different colors:

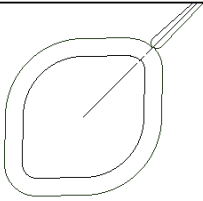
- White** The contour element is fully defined.
- Green** The entered data describe a limited number of possible solutions: select the correct one.
- Red** The entered data are not sufficient to determine the contour element: enter further data.

If the entered data permit a limited number of possible solutions and the contour element is displayed in green, select the correct contour element as follows:

-  ▶ Press the SHOW soft key repeatedly until the correct contour element is displayed.
-  ▶ If the displayed contour element matches the drawing, select the contour element with FSELECT.

Select the green contour elements as soon as possible with the FSELECT soft key. In this way, you can reduce the ambiguity of subsequent elements.



Programming and editing			
19 FCT DR+ R2.5 CLSD- 20 FSELECT2 21 RND R1 22 LP IPR+20 R0 FMAX 23 L Z+50 R0 FMAX M0 24 LP PR+25 PA+0 R0 FMAX M3 25 LBL 1 26 CALL LBL 2 27 CYCL DEF 10.0 ROTATION 28 CYCL DEF 10.1 IROT+90 29 CALL LBL 1 REP3 30 CYCL DEF 10.0 ROTATION			
NOML. X -219.210 Y +0.795 Z +212.795		T 0 F S M5/9	
		START	START SINGLE RESET + START

If you do not yet wish to select a green contour element, press the EDIT soft key to continue the FK dialog.



The machine tool builder may use other colors for the FK graphics.

NC blocks from a program that you called with PGM CALL are displayed in another color.

The TNC represents movements in a negative machine tool axis direction with a white circle (circle diameter = machine tool diameter).

Initiating the FK dialog

If you press the gray FK button, the TNC displays the soft keys you can use to initiate an FK dialog: See table on the right. Press the FK button a second time to deselect the soft keys.

If you initiate the FK dialog with one of these soft keys, the TNC shows additional soft-key rows that you can use for entering known coordinates, directional data and data regarding the course of the contour.



The following prerequisites for FK programming must be observed:

The FK free contour programming feature can only be used for programming contour elements that lie in the working plane. The working plane is defined in the first BLK FORM block of the part program.

You must enter all available data for every contour element. Even the data that does not change must be entered in every block — otherwise it will not be recognized.

Q parameters are permissible in all FK elements, but may not be changed however during program run.

If both FK blocks and conventional blocks are entered in a program, the FK contour must be fully defined before you can return to conventional programming.


The TNC needs a fixed point from which it can calculate the contour elements. Use the gray path function keys to program a position that contains both coordinates of the working plane immediately before programming the FK contour. Do not enter any Q parameters in this block.


If the first block of an FK contour is an FCT or FLT block, you must program at least two NC block with the gray path function keys to fully define the direction of contour approach.

Do not program an FK contour immediately after an LBL label.

contour element	Soft key
Straight line with tangential connection	
Straight line without tangential connection	
Circular arc with tangential connection	
Circular arc without tangential connection	

Free programming of straight lines

- 

▶ To display the soft keys for free contour programming, press the FK key
- 


▶ To initiate the dialog for free programming of straight lines, press the FL soft key. The TNC displays additional soft keys — see table at right.


▶ Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data are entered. If the entered data describe several solutions, the graphic will display the contour element in green. See “Graphics during FK programming.”

See next page for the resulting NC blocks.

Straight line with tangential connection

If the straight line connects tangentially to another contour element, initiate the dialog with the FLT soft key:


- 


▶ To display the soft keys for free contour programming, press the FK key
- 

▶ To initiate the dialog, press the FLT soft key.

▶ Use the soft keys (listed in the table at above right) to enter all known data in the block.

Free programming of circular arcs

- 


▶ To display the soft keys for free contour programming, press the FK key
- 


▶ To initiate the dialog for free programming of circular arcs, press the FC soft key. The TNC displays soft keys with which you can enter direct data on the circular arc or data on the circle center — see table at right.

▶ Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data are entered. If the entered data describe several solutions, the graphic will display the contour element in green (see “Graphics during FK programming”).

Circular arc with tangential connection








If the circular arc connects tangentially to another contour element, initiate the dialog with the FCT soft key:

- 








▶ To display the soft keys for free contour programming, press the FK key
- 

▶ To initiate the dialog, press the FCT soft key.

▶ Use the soft keys (listed in the table at right) to enter all known data in the block.

Known data	Soft key
X coordinate of the straight-line end point	
Y coordinate of the straight-line end point	
Polar coordinate radius	
Polar coordinate angle	
Length of a straight line	
Gradient angle of a straight line	
Beginning or end of a closed contour	

For data that are based on other blocks see “Relative data,” for auxiliary points see “Auxiliary points” in this section.

Direct data on a circular arc	Soft key
X coordinate of the arc end point	
Y coordinate of the arc end point	
Polar coordinate radius	
Polar coordinate angle	
Rotational direction of the arc	
Radius of the arc	
Angle of the leading axis to the arc end point	

Centers of free-programmed circles

The TNC calculates a circle center for free-programmed arcs from the data you enter. This makes it possible to program full circles in an FK program block.

If you wish to define circle centers in polar coordinates you must use FPOL, not CC, to define the pole. FPOL is entered in Cartesian coordinates and remains in effect until the TNC encounters a block in which another FPOL is defined.

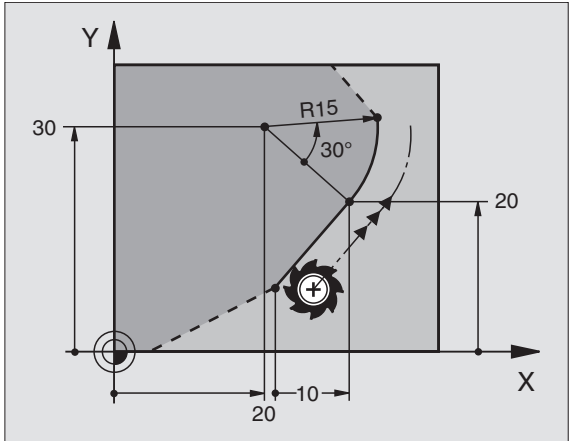
A circle center that was calculated or programmed conventionally is then no longer valid as a pole or circle center for the new FK contour: If you enter conventional polar coordinates that refer to a pole from a CC block you have defined previously, then you must enter the pole again in a CC block after the FK contour.

Resulting NC blocks for FL, FPOL and FCT

```
7 FPOL X+20 Y+30
8 FL IX+10 Y+20 RR F100
9 FCT PR+15 IPA+30 DR+ R15
```

See figure at center right.

Data on the circle center	Soft key
X coordinate of the circle center	<div>CCK</div>
Y coordinate of the circle center	<div>CCV</div>
Polar coordinate radius of the circle center (relative to FPOL)	<div>CCP</div>
Polar coordinate angle of the circle center	<div>CCP</div>



Auxiliary points

You can enter the coordinates of auxiliary points that are located on the contour or in its proximity for both free-programmed straight lines and free-programmed circular arcs. The soft keys are available as soon as you initiate the FK dialog with the FL, FLT, FC or FCT soft key.

Auxiliary points for a straight line

If the auxiliary points are located on a straight line or on the extension of a straight line, use the soft keys listed in the table at upper right.

If the auxiliary points are located at a distance D from a straight line, use the soft keys listed in the second table at right.



Auxiliary points for a circular arc




For circular arcs, you can enter 1 auxiliary point on the contour. The available soft keys are listed in the table at lower right.






Example NC blocks

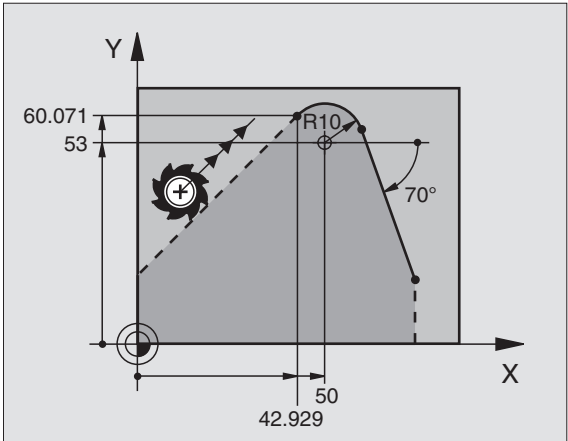
```
13 FC DR- R10 P1X+42.929 P1Y+60.071
14 FLT AN-70 PDX+50 PDY+53 D10
```

This example is illustrated in the figure at lower right.

Auxiliary points on a straight line	Soft key
X coordinate auxiliary point P1	
Y coordinate auxiliary point P1	

Auxiliary points near a straight line	Soft key
X coordinate of an auxiliary point	
Y coordinate of an auxiliary point	
Distance auxiliary point/straight line	

Auxiliary points on/beside the circular arc	Soft key
X coordinate of a P1 auxiliary point	
Y coordinate of a P1 auxiliary point	
Coordinates of an auxiliary point beside the circular arc	 
Distance from the auxiliary point to the circular arc	



Relative data

Data whose values are based on another contour element are called relative data. The soft keys are available as soon as you initiate the FK dialog with the FL or FLT soft key.



The block number of the contour element on which the relative data are based can only be located up to 64 positioning blocks before the block in which you program the reference.

If you delete a block on which relative data are based, the TNC will display an error message. Change the program first before you clear this message.

Relative data for a free-programmed straight line Soft key

Straight line parallel to another contour element



Distance from a straight line to a parallel contour element



Example NC blocks

The known direction and the known distance from the contour element are relative to block N (see figure at top right):

17 FL LEN20 AN+15

18 FL AN+105 LEN12.5

19 FL PAR17 DP12.5

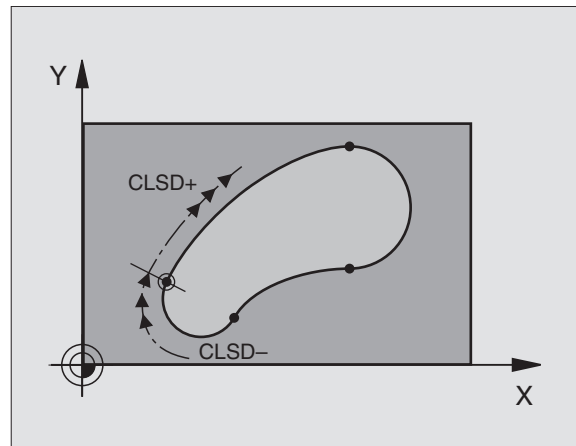
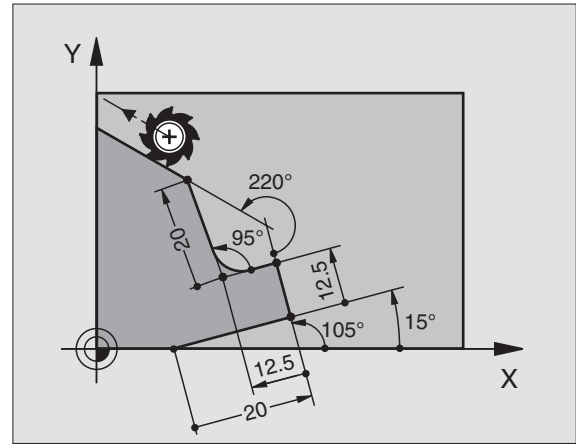
20 FSELECT 2

21 FL LEN20 IAN+95

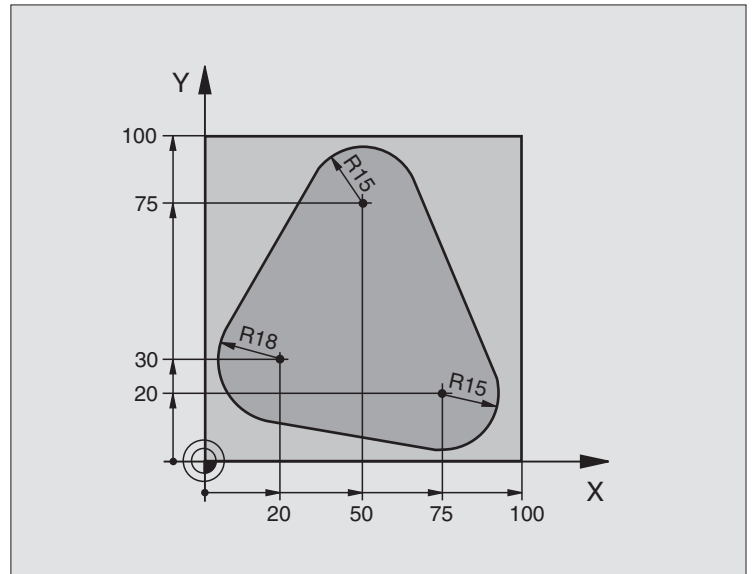
Closed contours

You can identify the beginning and end of a closed contour with the CLSD soft key. This reduces the number of possible solutions for the last contour element.

Enter CLSD as an addition to another contour data entry in the first and last block of an FK section.

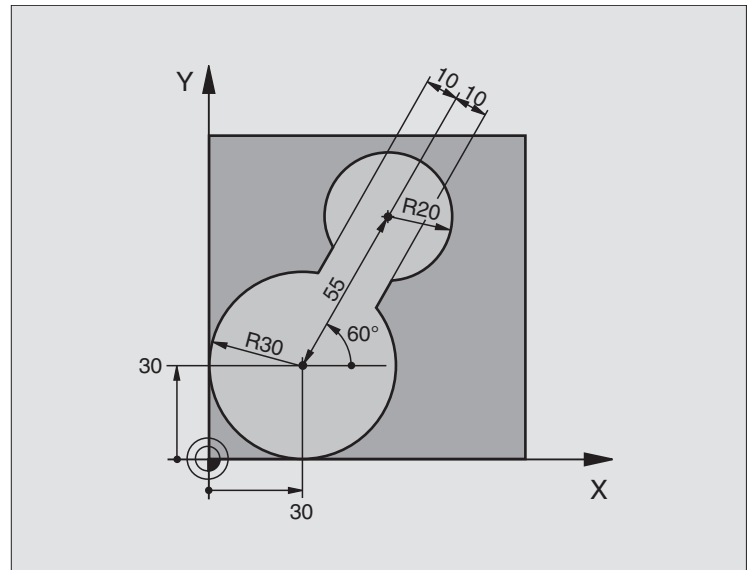


Example: FK programming 1



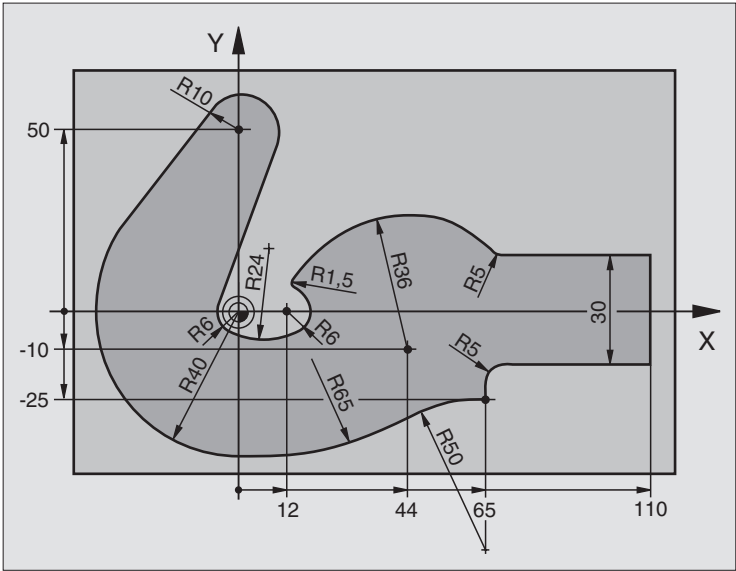
0	BEGIN PGM FK1 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define the tool
4	TOOL CALL 1 Z S500	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	L X-20 Y+30 R0 F MAX	Pre-position the tool
7	L Z-10 R0 F1000 M3	Move to working depth
8	APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
9	FC DR- R18 CLSD+ CCX+20 CCY+30	FK contour:
10	FLT	Program all known data for each contour element
11	FCT DR- R15 CCX+50 CCY+75	
12	FLT	
13	FCT DR- R15 CCX+75 CCY+20	
14	FLT	
15	FCT DR- R18 CLSD- CCX+20 CCY+30	
16	DEP CT CCA90 R+5 R0 F1000	Depart the contour on a circular arc with tangential connection
17	L X-30 Y+0 R0 F MAX	
18	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
19	END PGM FK1 MM	

Example: FK programming 2



0	BEGIN PGM FK2 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	T00L DEF 1 L+0 R+2	Define the tool
4	T00L CALL 1 Z S4000	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	L X+30 Y+30 R0 F MAX	Pre-position the tool
7	L Z+5 R0 FMAX M3	Pre-position the tool in the tool axis
8	L Z-5 R0 F100	Move to working depth
9	APPR LCT X+0 Y+30 R5 RR F350	Approach the contour on a circular arc with tangential connection
10	FPOL X+30 Y+30	FK contour:
11	FC DR- R30 CCX+30 CCY+30	Program all known data for each contour element
12	FL AN+60 PDX+30 PDY+30 D10	
13	FSELECT 3	
14	FC DR- R20 CCPR+55 CCPA+60	
15	FSELECT 2	
16	FL AN-120 PDX+30 PDY+30 D10	
17	FSELECT 3	
18	FC X+0 DR- R30 CCX+30 CCY+30	
19	FSELECT 2	
20	DEP LCT X+30 Y+30 R5 R0	Depart the contour on a circular arc with tangential connection
21	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
22	END PGM FK2 MM	

Example: FK programming 3



0	BEGIN PGM FK3 MM	
1	BLK FORM 0.1 Z X-45 Y-45 Z-20	Define the blank form
2	BLK FORM 0.2 X+120 Y+70 Z+0	
3	TOOL DEF 1 L+0 R+3	Define the tool
4	TOOL CALL 1 Z S4500	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	L X-70 Y+0 R0 F MAX	Pre-position the tool
7	L Z-5 R0 F1000 M3	Move to working depth
8	APPR CT X-40 Y+0 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
9	FC DR- R40 CCX+0 CCY+0	FK contour:
10	FLT	Program all known data for each contour element
11	FCT DR- R10 CCX+0 CCY+50	
12	FLT	
13	FCT DR+ R6 CCX+0 CCY+0	
14	FCT DR+ R24	
15	FCT DR+ R6 CCX+12 CCY+0	
16	FSELECT 2	
17	FCT DR- R1.5	
18	FCT DR- R36 CCX+44 CCY-10	
19	FSELECT 2	
20	FCT DR+ R5	
21	FLT X+110 Y+15 AN+0	
22	FL AN-90	

23	FL X+65 AN+180 PAR21 DP30	
24	RND R5	
25	FL X+65 Y-25 AN-90	
26	FC DR+ R50 CCX+65 CCY-75	
27	FCT DR- R65	
28	FSELECT 1	
29	FCT Y+0 DR- R40 CCX+0 CCY+0	
30	FSELECT 4	
31	DEP CT CCA90 R+5 R0 F1000	Depart the contour on a circular arc with tangential connection
32	L X-70 R0 F MAX	
33	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
34	END PGM FK3 MM	



7

Programming:
Miscellaneous Functions

7.1 Entering Miscellaneous Functions M and STOP

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

- Program run, such as 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 for more information.

M functions are always entered at the end of a positioning block or with the soft key M. The TNC then displays the following dialog question:

Miscellaneous function M ?

You usually enter only the number of the M function in the programming dialog. For M functions that you can select directly via soft key, the dialog is continued so that you can enter parameters for the function.

In the operating modes Manual and Electronic Handwheel, you enter the miscellaneous functions with the soft key M. With the NC start key the TNC executes the entered M function immediately.

Please note that some F functions become effective at the start of a positioning block, and other 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 program. Some M functions are effective only in the block in which they are called.

Entering an M function in a STOP block

If you program a STOP block, the program run or test run is interrupted at the block, for example for tool inspection. You can also enter an M function in a STOP block:



- ▶ To program an interruption of program run, press the STOP key.
- ▶ Enter miscellaneous function M

Resulting NC block

87 STOP M5

Programming and editing									
Miscellaneous function M?									
1	BLK	FORM	0.1	Z	X+0	Y+0	Z-40		
2	BLK	FORM	0.2		X+100	Y+100	Z+0		
3	TOOL	DEF	100	L+10	R+5				
4	TOOL	CALL	100	Z	S4000				
5	L	X+10	Y+5	R0	F100	M3			
END PGM 1568T MM									
ACTL.		X	-219.715						
		Y	+0.285						
		Z	+212.680						
		T	2		Z				
		F	0						
		S						M5/9	
M	M103	M112	M120	M124					

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

M	Effect	Effective at
M00	Stop program run Spindle STOP Coolant OFF	Block end
M01	Stop program run	Block end
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (dependent on machine parameter 7300)	Block end
M03	Spindle ON clockwise	Block start
M04	Spindle ON counterclockwise	Block start
M05	Spindle STOP	Block end
M06	Tool change Spindle STOP Program run stop (dependent on machine parameter 7440)	Block end
M08	Coolant ON	Block start
M09	Coolant OFF	Block end
M13	Spindle ON clockwise Coolant ON	Block start
M14	Spindle ON counterclockwise Coolant ON	Block start
M30	Same as M02	Block end

7.3 Miscellaneous Functions for Coordinate Data

Programming machine-referenced coordinates: M91/M92

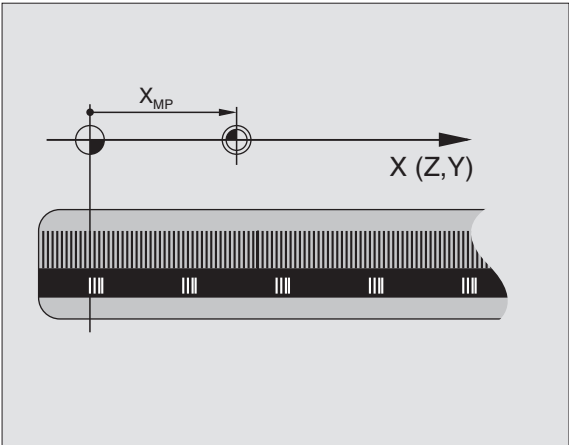
Scale reference point

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").

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 referenced to the machine datum. Switch the display of coordinates in the status display to REF (see section 1.4 "Status Displays").

Behavior with M92 — Additional machine datum



In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.



Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

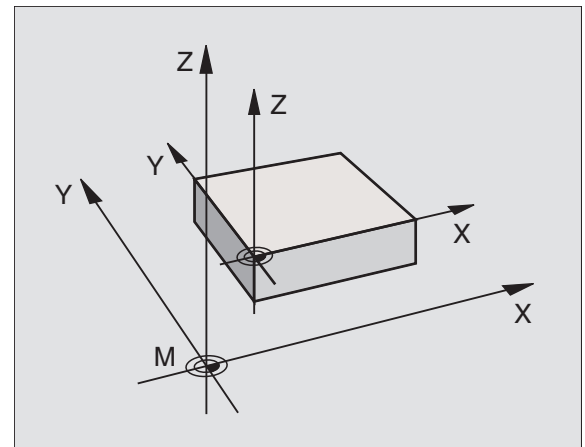
Effect

M91 and M92 are effective only in the blocks in which they are programmed with M91 or M92.

M91 and M92 become effective at the start of block.

Workpiece datum

The figure at right shows coordinate systems with the machine datum and workpiece datum.



7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90



Instead of M90, you should use the function M112 (see further on in this chapter). Old programs can however be run with M112 and M90 combined.

Standard behavior

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

In program blocks with radius compensation (RR/RL), 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.

Example application: Surface consisting of a series of straight line segments.

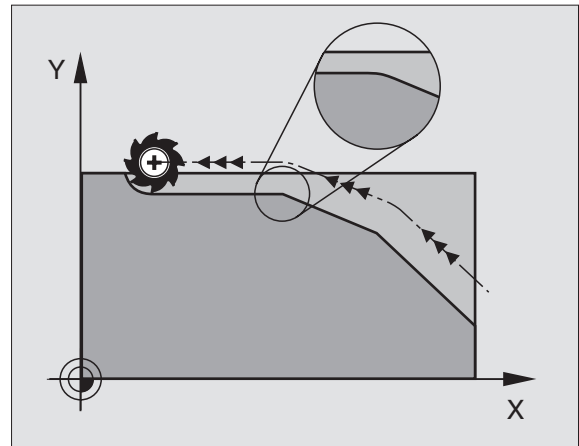
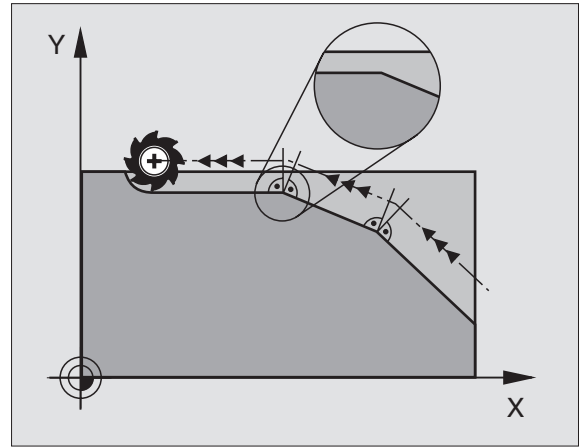
Effect

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.



Independently of M90, you can define a limit value via machine parameter MP7460 up to which the tool moves at constant path speed (effective with servo lag and feedforward control).



Entering contour transitions between contour elements: M112

Standard behavior

The TNC stops briefly for all changes in direction that are greater than the limit angle defined in MP7460(exact stop).

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

Behavior with M112



You can adjust the effect of M112 by redefining machine parameters.

M112 is effective in operation with servo lag as well as operation with feedforward control.

The TNC inserts a selectable contour transition between **any contour elements (compensated and uncompensated)**, in the plane or in three dimensions :

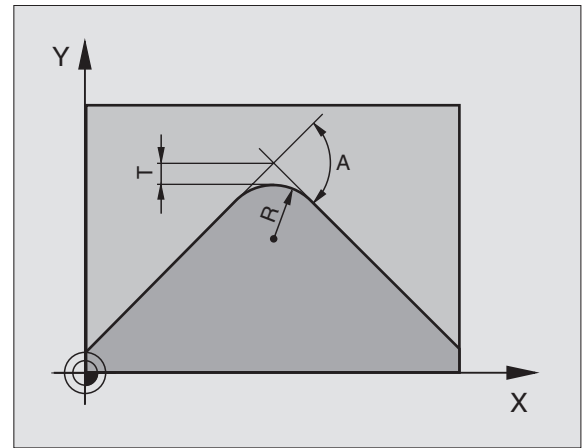
- Tangential circle: MP7415.0 = 0
an acceleration jump results from the change in the curvature at the connection points.
- Third-degree polynomial (cubic spline): MP7415.0 = 1
There is no velocity jump at the connection points.
- Fifth-degree polynomial: MP7415.0 = 2
There is no acceleration jump at the connection points.
- Seventh-degree polynomial: MP7415.0 = 3 (standard setting)
There is no jump in the rate of acceleration change.

Permissible contour deviationT

With the tolerance value T you define the distance by which the milled contour can deviate from the programmed contour. If you do not enter a tolerance value, the TNC calculates the most exact contour transition possible at the programmed feed rate.

Tolerance angle A

If you enter a tolerance angle A, the TNC smoothens only those contour transitions whose angle of directional change is greater than the programmed tolerance angle. If you enter a limit angle = 0, the TNC moves the tool at a constant acceleration also over tangential transitions. Input range: 0° to 90°



Entering M112 in a positioning block

If you press the soft key M112 in a positioning block (in answer to the "Miscellaneous function?" prompt), the TNC then continues the dialog by asking you for the permissible tolerance T and the tolerance angle A.

You can also define T and A through Q parameters. See Chapter „10 Programming: Q Parameters“

Effect

M112 is effective during operation with velocity feedforward as well as with servo lag.

M112 becomes effective at the start of block.

To cancel M112, enter M113.

Resulting NC block

```
L X+123.723 Y+25.491 R0 F800 M112 T0.01 A10
```


Contour filter: M124**Standard behavior**

The TNC includes all available points in its calculation of a contour transition between contour elements.

Behavior with M124

You can adjust the effect of M124 by redefining machine parameters.

The TNC filters contour elements with small point spacing and inserts a transitional contour.

Shape of contour transition

- Tangential circle: $MP7415.0 = 0$
an acceleration jump results from the change in the curvature at the connection points.
- Third-degree polynomial (cubic spline): $MP7415.0 = 1$
There is no velocity jump at the connection points.
- Fifth-degree polynomial: $MP7415.0 = 2$
There is no acceleration jump at the connection points.
- Seventh-degree polynomial: $MP7415.0 = 3$ (standard setting)
There is no jump in the rate of acceleration change.

Rounding of contour transitions

- Do not round the contour transition: $MP7415.1 = 0$
execute the contour transition as defined in $MP7415.0$ (standard contour transition: 7th-degree polynomial)
- Round the contour transition: $MP7415.1 = 1$
Execute the contour transition so that the straight line segments remaining between the contour transitions are also rounded.

Maximum length T of a contour element

With parameter T you define the length up to which the TNC will filter out contour elements. If you have defined a permissible contour deviation in M112 the TNC will respect it. If you do not enter a maximum contour deviation, the TNC calculates the most exact contour transition possible without reducing the programmed feed rate.

Programming M124

If in a positioning block (with the dialog "Miscellaneous function") you press the soft key M124, the TNC then continues the dialog for this block and asks for the tolerance value T.

You can also define T through Q parameters. See Chapter „10 Programming: Q Parameters“

Effect

M124 becomes effective at the start of block. Like M112, M124 is reset with M113.

Resulting NC block

```
L X+123.723 Y+25.491 R0 F800 M124 T0.01
```

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. See figure at upper right.

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. See figure at center right.

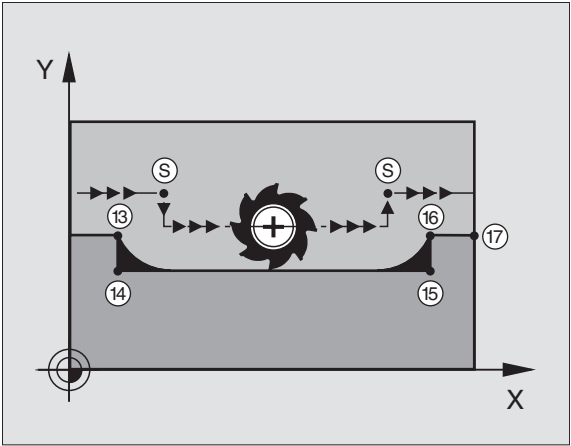
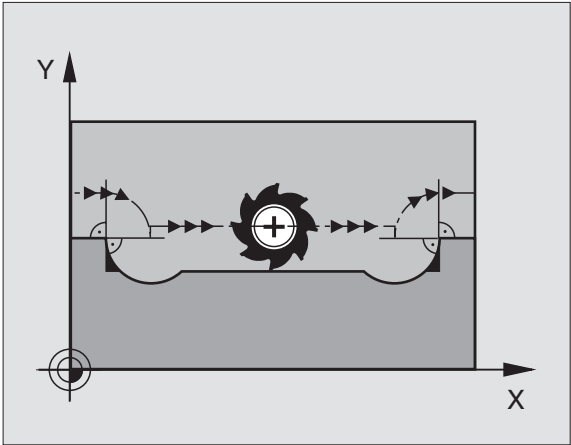
Program M97 in the same block as the outside corner.

Effect

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



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



Example NC blocks

5	T00L DEF L ... R+20	Large tool radius
...		
13	L X ... Y ... R.. F .. M97	Move to contour point 13
14	L IY-0.5 R .. F..	Machine small contour step 13 to 14
15	L IX+100 ...	Move to contour point 15
16	L IY+0.5 ... R .. F.. M97	Machine small contour step 15 to 16
17	L 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 (see figure at upper right).

Behavior with M98

With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined (see figure at lower right).

Effect

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

M98 becomes effective at the end of block.

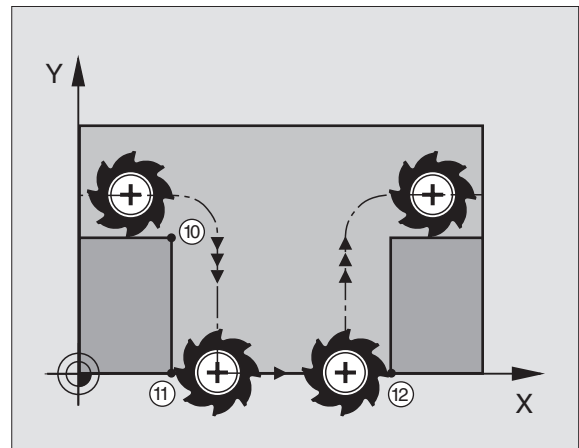
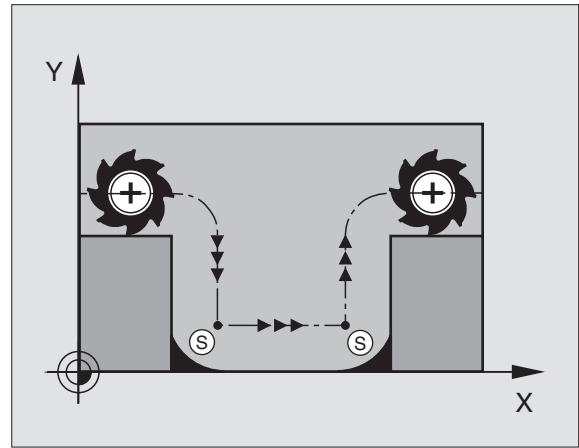
Example NC blocks

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

```
10 L X ... Y... RL F
```

```
11 L X... IY... M98
```

```
12 L IX+ ...
```



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 (defined in machine parameter 7440). 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 press the soft key M103 in a positioning block (with the dialog "Miscellaneous function?"), the TNC then continues the dialog by asking you for the factor F.

Effect

M103 becomes effective at the start of block. To cancel M103, program M103 once again **without a factor**.

Example NC blocks

The feed rate for plunging is to be 20% of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
17 L X+20 Y+20 RL F500 M103 F20	500
18 L Y+50	500
19 L IZ-2.5	100
20 L IY+5 IZ-5	141
21 L IX+50	500
22 L Z+5	500

Constant feed rate at the tool cutting edge: 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 at inside and outside contours such that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110

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

Effect

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

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

Standard behavior

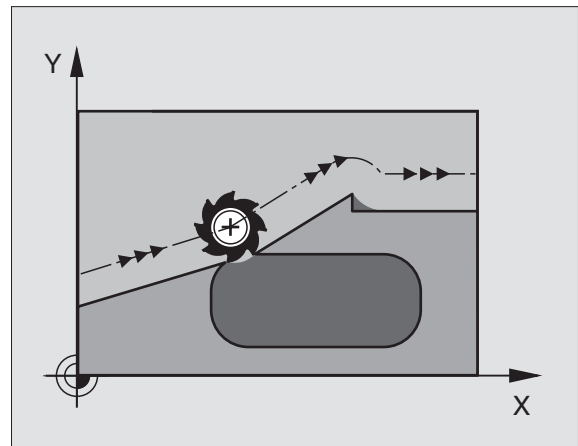
If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. Although you can use M97 to inhibit the error message (see "Machining small contour steps: M97"), this will result in dwell marks and will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

If the programmed contour contains undercut features, the tool may damage the contour. —See figure at right.

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) behind 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 press the soft key M120 in a positioning block (with the dialog "Miscellaneous function?"), the TNC then continues the dialog by asking you for the number of "look ahead" blocks (LA)to be precalculated).

Effect

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

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA.
- another program is called with PGM CALL

M120 becomes effective at the start of block.

7.5 Miscellaneous Functions for Rotary Axes

Shorter-path traverse of rotary axes: M126

Standard behavior

If you reduce display of a rotary axis to a value less than 360°, the TNC will move the axis by the difference between the nominal and actual positions. The table at upper right shows examples.

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°. The table at lower right shows examples.

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 path 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.

Example NC blocks

To reduce display of all active rotary axes:

```
L M94
```

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

```
L C+180 FMAX M94
```

Effect

M94 is effective only in the block in which M94 is programmed.
M94 becomes effective at the start of block.

Standard behavior of the TNC

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

Behavior with M126

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



8

**Programming:
Cycles**

8.1 General Overview of 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. The table at right shows the various cycle groups.

Fixed cycles with number starting with 200 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 setup clearance, Q202 the plunging depth, etc.

Defining a cycle



► The soft-key row shows the available groups of cycles



► Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles



► Select the desired cycle, for example PECKING. 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. Select the screen layout PROGRAM + HELP GRAPHIC.

► Enter all parameters asked by the TNC and conclude each entry with the ENT key

► The TNC terminates the dialog when all required data have been entered

Resulting NC blocks

CYCL DEF 1.0	PECKING
CYCL DEF 1.1	DIST2
CYCL DEF 1.2	DEPTH-30
CYCL DEF 1.3	PLNGNG5
CYCL DEF 1.4	DWELL1
CYCL DEF 1.5	F150

Group of cycles	Soft Key
Cycles for pecking, reaming, boring and tapping	DRILLING
Cycles for milling pockets, studs and slots	POCKETS/STUDS/SLOTS
Cycles for producing regular point patterns, such as circular hole patterns, hole surfaces or irregular point patterns via point tables	PATTERN
Subcontour List (SL) cycles, which allow the machining of more complex contours comprising several overlapping subcontours.	SL CYCLES
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
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 and oriented spindle stop	SPECIAL CYCLES

Programming and editing Pitch?

```

3 TOOL DEF 1 L=0 R=3
4 TOOL CALL 1 Z S4000
5 L Z+100 R0 FMAX M103 F1 M124 T5
6 L X=0 Y=0 Z-20 R0 FMAX M8
7 CYCL DEF 17 RIGID TAPPING
  SET UP+2 DEPTH-15
  M103 F1
END PGM CYC210 MM

```

ACTL.	X	-219.715							
	Y	+0.285							
	Z	+212.680							
	T	2	Z						
	F	0							
	S								

Calling the cycle



Prerequisites

The following data must always be programmed before a cycle call

- BLK FORM for graphic display (needed only for test graphics)
- Call the tool
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF).

For some cycles, additional prerequisites must be observed. They are described with the individual 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:

- Cycles for circular and linear hole patterns
- SL cycle CONTOUR GEOMETRY
- Coordinate transformation cycles
- DWELL TIME cycle

All other cycles are called as described below.

If the TNC is to execute the cycle once after the last programmed block, program the cycle call with the miscellaneous function M99 or with CYCL CALL::



- ▶ To program the cycle call, press the CYCL CALL key.
- ▶ Press the CYCL CALL soft key to enter a cycle call.
- ▶ Enter a miscellaneous function M or press END to end the dialog.

If the TNC is to execute the cycle automatically after every positioning block, program the cycle call with M89 (depending on machine parameter 7440).

To cancel M89, enter

- M99 or
- CYCL CALL or
- CYCL DEF

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 principle axes X, Y or Z. Exceptions:

- you program secondary axes for the side lengths in cycles 3 SLOT MILLING and 4 POCKET MILLING
- you program secondary axes in the contour geometry subprogram of an SL cycle.

8.2 Point Tables

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.

Programming and editing									
MUSTPKT .PNT MM									
NR	X	Y	Z						
0	+35	+30	+0						
1	+65	+30	+0						
2	+80	+50	+0						
3	+50	+50	+0						
4	+20	+50	+0						
5	+35	+70	+0						
6	+65	+70	+0						
[END]									
ACTL. X +0.195									
Y -11.000									
Z +136.000									
				T					
				F 0					
				S 1000		M5/9			
PAGE		PAGE		WORD		WORD			
↑		↓		←		→			
				INSERT				DELETE	
				N LINES				LINE	
								INSERT	
								LINE	



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

File name =

NEW

ENT

Enter the name of the point table and confirm your entry with the ENT key.



If necessary, switch to inches as unit of measure: Press the MM/INCH soft key.



Press the .PNT soft key to select the point table file type.

Selecting point tables in the program.

Select the Programming and Editing mode of operation.



Press the PGM CALL key to call the function for selecting the point table.



Press the POINT TABLE soft key.

Enter the name of the point table and confirm your entry with the END key.

Calling a cycle in connection with point tables



Before programming, note the following:

With CYCL CALL PAT the TNC runs the points table that you last defined (even if you have defined the point table in a program that was nested with CALL PGM).

The TNC uses the coordinate in the spindle axis as the clearance height for the cycle call.

If you want the TNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with CYCLE CALL PAT:



- ▶ To program the cycle call, press the CYCL CALL key.
- ▶ Press the CYCL CALL PAT soft key to call a point table.
- ▶ Enter the feed rate at which the TNC is to move from point to point (if you make no entry the TNC will move at the last programmed feed rate).
- ▶ If required, enter miscellaneous function M, then confirm with the END key.

The TNC moves the tool back to the safe height over each successive starting point (safe height = the spindle axis coordinate for cycle call). To use this procedure also for the cycles number 200 and greater, you must define the 2nd set-up clearance (Q204) as 0.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103 (see section "7.4 Miscellaneous Functions for Contouring Behavior").

Effect of the point tables with Cycles 1 to 5 and 17

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 12

The TNC interprets the points as an additional datum shift.

Effect of the point tables with Cycles 200 to 204









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 (see the example in section "8.3 Drilling Cycles").

Effect of the point tables with Cycles 210 to 215

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 (see the example in section "8.4 Cycles for Milling Pockets, Studs and Slots").

8.3 Drilling Cycles

The TNC offers 8 cycles for all types of drilling operations:

Cycle	Soft Key
1 PECKING Without automatic pre-positioning	
200 DRILLING With automatic pre-positioning and 2nd set-up clearance	
201 REAMING With automatic pre-positioning and 2nd set-up clearance	
202 BORING With automatic pre-positioning and 2nd set-up clearance	
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrement	
204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	
2 TAPPING With a floating tap holder	
17 RIGID TAPPING Without a floating tap holder	

PECKING (Cycle 1)

- 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 in rapid traverse FMAX 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: $t = 0.6 \text{ mm}$
 - At a total hole depth exceeding 30 mm: $t = \text{hole depth} / 50$

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 total hole depth is reached.
- 6 After a dwell time at the hole bottom, the tool is returned to the starting position in rapid traverse FMAX 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 R0.

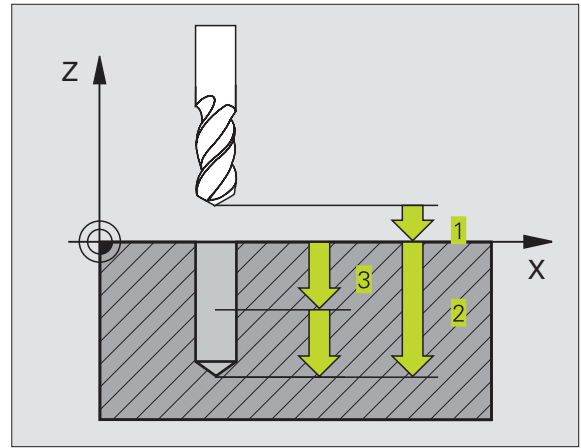
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the cycle parameter TOTAL HOLE DEPTH determines the working direction.



- ▶ Setup 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 TNC will go to depth in one movement if:
 - the plunging depth equals the total hole depth
 - the plunging depth is greater than the total hole depth

The total hole depth does not have to be a multiple of the plunging 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 blocks:

```

1 CYCL DEF 1.0 PECKING
2 CYCL DEF 1.1 SET UP 2
3 CYCL DEF 1.2 DEPTH -20
4 CYCL DEF 1.3 PECKG 5
5 CYCL DEF 1.4 DWELL 0
6 CYCL DEF 1.5 F500

```


DRILLING (Cycle 200)

- 1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the set-up clearance above the workpiece surface.
- 2 The tool drills to the first plunging depth at the programmed feed rate F.
- 3 The TNC returns the tool at FMAX to the setup clearance, dwells there (if a dwell time was entered), and then moves at FMAX 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 total hole depth is reached.
- 6 At the hole bottom, the tool is retracted to set-up clearance or — if programmed — to the 2nd set-up clearance in rapid traverse FMAX.



Before programming, note the following:

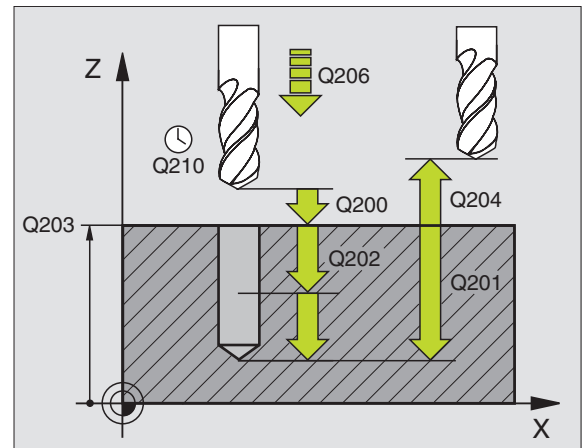
Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

The algebraic sign for the depth parameter determines the working direction.



- ▶ 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 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

The depth does not have to be a multiple of the plunging 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.



Example NC blocks:

7 CYCL DEF 200 DRILLING	
Q200=2	; SET-UP CLEARANCE
Q201=-20	; DEPTH
Q206=150	; FEED RATE FOR PLUNGING
Q202=5	; PLUNGING DEPTH
Q210=0	; DWELL TIME AT TOP
Q203=+0	; SURFACE COORDINATE
Q204=50	; 2. SET-UP CLEARANCE

REAMING (Cycle 201)

- 1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.
- 2 The tool reams to the entered depth at the programmed feed rate F.
- 3 If programmed, the tool remains at the hole bottom for the entered dwell time.
- 4 The tool then retracts to set-up clearance at the feed rate F, and from there — if programmed — to the 2nd set-up clearance in FMAX.



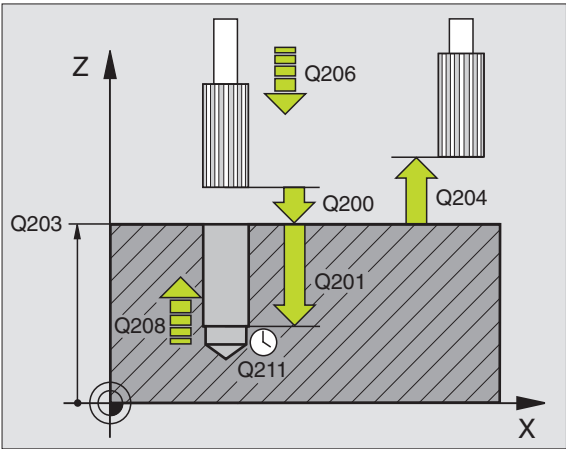
Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

The algebraic sign for the depth parameter determines the working direction.



- 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:

8	CYCL DEF 201	REAMING
Q200=2	; SET-UP CLEARANCE	
Q201=-20	; DEPTH	
Q206=150	; FEED RATE FOR PLUNGING	
Q211=0.25	; DWELL TIME AT BOTTOM	
Q208=500	; RETRACTION FEED TIME	
Q203=+0	; SURFACE COORDINATE	
Q204=50	; 2. SET-UP CLEARANCE	

BORING (Cycle 202)



Machine and control must be specially prepared by the machine tool builder to enable Cycle 202.

- 1 The TNC positions the tool in the tool axis at rapid traverse FMAX to set-up 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 with M19 to the 0° position with an oriented spindle stop.
- 5 If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
- 6 The tool then retracts to set-up clearance at the retraction feed rate, and from there — if programmed — to the 2nd set-up clearance in FMAX.



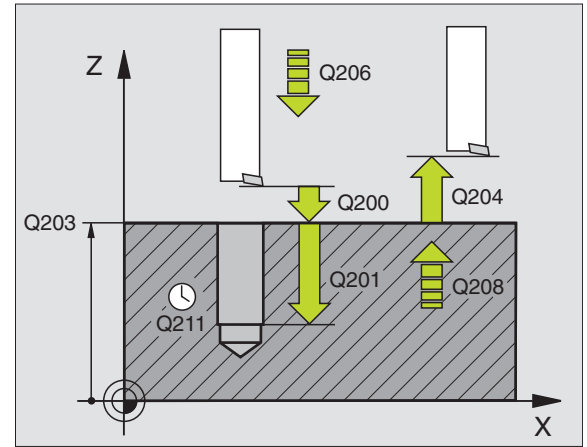
Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

The algebraic sign for the cycle parameter TOTAL HOLE DEPTH determines the working direction.



- ▶ 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).



Example NC blocks:

9 CYCL DEF 202 BORING	
Q200=2	;SET-UP CLEARANCE
Q201=-20	;DEPTH
Q206=150	;FEED RATE FOR PLUNGING
Q211=0.5	;DWELL TIME AT BOTTOM
Q208=500	;RETRACTION FEED TIME
Q203=+0	;SURFACE COORDINATE
Q204=50	;2. SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN

- 0:** Do not retract tool
- 1:** Retract tool in the negative main axis direction
- 2:** Retract tool in the negative secondary axis direction
- 3:** Retract tool in the positive main axis direction
- 4:** Retract tool in the positive secondary axis direction



Danger of collision

Check the position of the tool tip when you use M19 to program a spindle orientation (for example, in the Positioning with Manual Data Input mode of operation). Align the tool tip so that it is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

UNIVERSAL DRILLING (Cycle 203)

- 1** The TNC positions the tool in the tool axis at rapid traverse FMAX 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 setup clearance. If you are working without chip breaking, the tool retracts at the RETRACTION FEED RATE to setup clearance, remains there — if programmed — for the entered dwell time, and advances again in FMAX to the setup 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 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 FMAX.



Before programming, note the following:

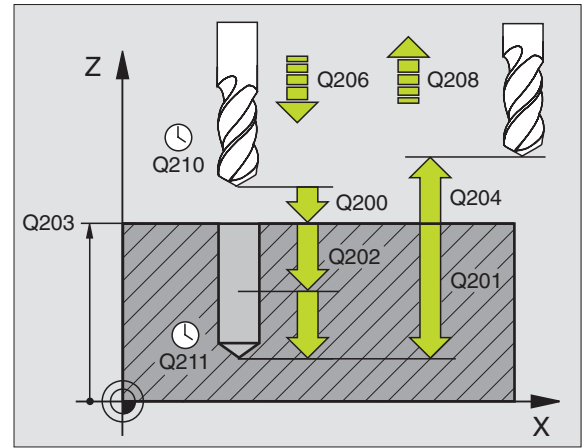
Program a positioning block for the starting point (hole center) in the working plane with **RADIUS COMPENSATION R0**.

The algebraic sign for the cycle parameter **TOTAL HOLE DEPTH** determines the working direction.



- ▶ 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 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

The depth does not have to be a multiple of the plunging 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 after each infeed.
- ▶ Nr 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 setup clearance Q200.
- ▶ 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 tool retracts at the feed rate in Q206.



Example NC blocks:

10	CYCL DEF 203 UNIVERSAL DRILLING
Q200=2	; SET-UP CLEARANCE
Q201=-20	; DEPTH
Q206=150	; FEED RATE FOR PLUNGING
Q202=5	; PLUNGING DEPTH
Q210=0	; DWELL TIME AT TOP
Q203=+0	; SURFACE COORDINATE
Q204=50	; 2. SET-UP CLEARANCE
Q212=0.2	; DECREMENT
Q213=3	; BREAKS
Q205=3	; MIN. PLUNGING DEPTH
Q211=0.25	; DWELL TIME AT BOTTOM
Q208=500	; RETRACTION FEED TIME

BACK BORING (Cycle 204)



Machine and control must be specially prepared by the machine tool builder to enable back boring.

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 FMAX to set-up clearance above the workpiece surface.
- 2 The TNC orients the spindle with M19 to the 0° position and moves the tool by its 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 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. The TNC carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.
- 6 The TNC moves the tool at the pre-positioning feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance with FMAX.



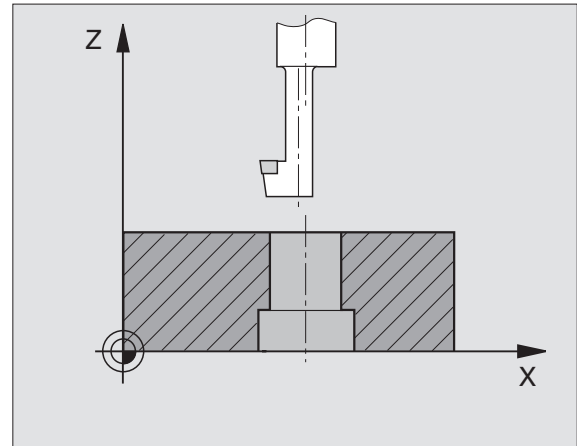
Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

When calculating the starting point for the boring, the TNC takes into consideration the tooth length of the boring bar and the thickness of the material.





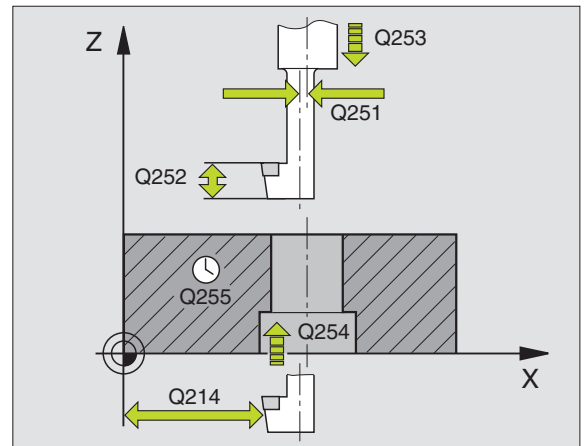
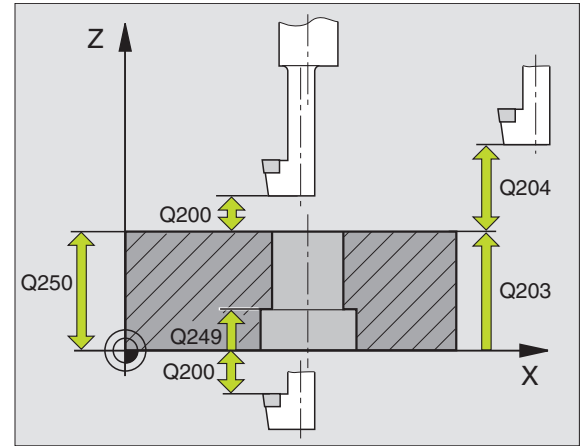
- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Depth of counterbore Q249 (incremental value): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction.
- ▶ Material thickness Q250 (incremental value): Thickness of the workpiece
- ▶ Off-center distance Q251 (incremental value): Off-center 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 moves the tool by its off-center distance (after spindle orientation).

- 0: Entry is not possible in this cycle
- 1: Displace tool in the negative main axis direction
- 2: Displace tool in the negative secondary axis direction
- 3: Displace tool in the positive main axis direction
- 4: Displace tool in the positive secondary axis direction



Danger of collision

Check the position of the tool tip when you use M19 to program a spindle orientation to 0° (for example, in the Positioning with Manual Data Input mode of operation). Align the tool tip so that it is parallel to a coordinate axis. Select a disengaging direction in which the tool can plunge into the hole without danger of collision.



Example NC blocks:

11	CYCL DEF 204 BACK BORING
Q200=2	;SET-UP CLEARANCE
Q249=+5	;DEPTH OF COUNTERBORE
Q250=20	;MATERIAL THICKNESS
Q251=3.5	;OFF-CENTER DISTANCE
Q252=15	;TOOL EDGE HEIGHT
Q253=750	;F PRE-POSITIONING
Q254=200	;F COUNTERBORING
Q255=0	;DWELL TIME
Q203=+0	;SURFACE COORDINATE
Q204=50	;2. SET-UP CLEARANCE
Q214=1	;DISENGAGING DIRECTN

TAPPING with a floating tap holder (Cycle 2)

- 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.



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

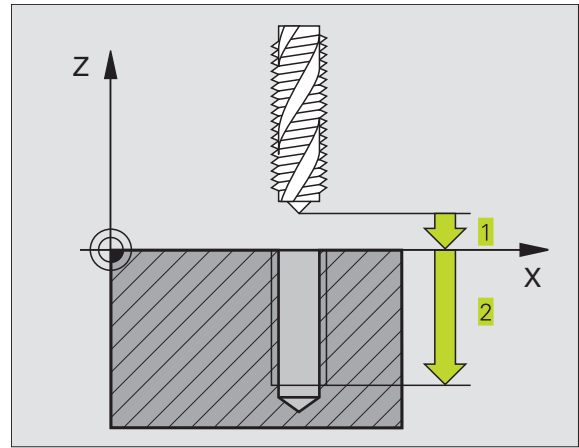
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the depth parameter determines the working direction.

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.



Example NC blocks:

```
13 CYCL DEF 2.0 TAPPING
```

```
14 CYCL DEF 2.1 SET UP 2
```

```
15 CYCL DEF 2.2 DEPTH -20
```

```
16 CYCL DEF 2.3 DWELL 0
```

```
17 CYCL DEF 2.4 F100
```



- ▶ Setup 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: $F = S \times p$, where

F is the feed rate in mm/min),

S is the spindle speed in rpm,

and p is the thread pitch in mm

RIGID TAPPING GS (Cycle 17)



Machine and control must be specially prepared by the machine tool builder to enable rigid tapping.

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 machine parameter 7160).
- Increased traverse range of the spindle axis due to absence of a floating tap holder.



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The algebraic sign for the parameter total hole depth determines the working direction.

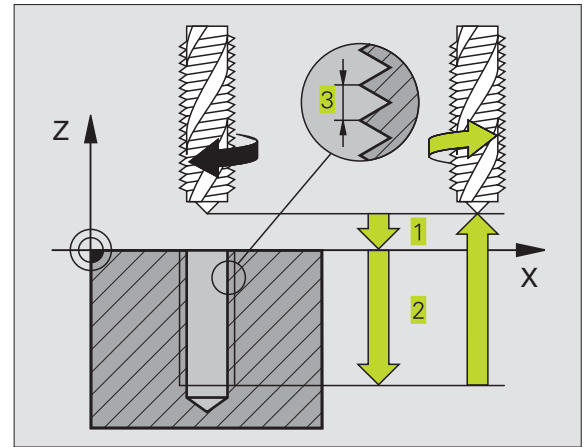
The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

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).



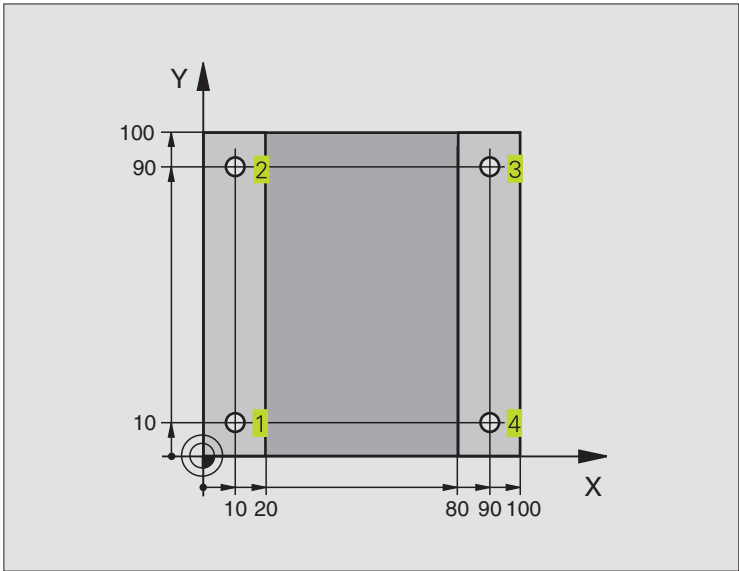
- ▶ Setup clearance **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- ▶ Total hole depth **2** (incremental value): Distance between workpiece surface (beginning of thread) and end of thread
- ▶ PITCH **3**:
Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
+ = right-hand thread
- = left-hand thread



Example NC blocks:

```
18 CYCL DEF 17.0 RIGID TAPPING GS
19 CYCL DEF 17.1 SET UP 2
20 CYCL DEF 17.2 DEPTH -20
21 CYCL DEF 17.3 PITCH +1
```

Example: Drilling cycles

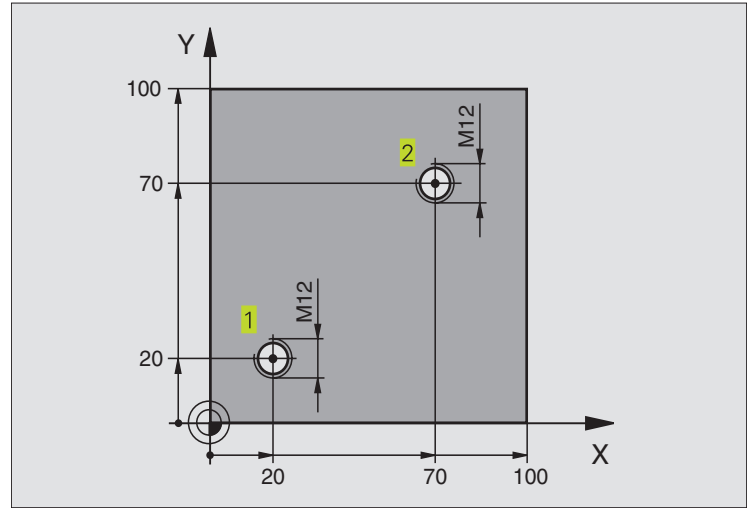


0	BEGIN PGM C200 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	T00L DEF 1 L+0 R+3	Tool definition
4	T00L CALL 1 Z S4500	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	CYCL DEF 200 DRILLING	Define cycle
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-15 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q202=5 ;PLUNGING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=-10 ;SURFACE COORDINATE	
	Q204=20 ;2ND SET-UP CLEARANCE	
7	L X+10 Y+10 R0 F MAX M3	Approach hole 1, spindle ON
8	CYCL CALL	Cycle call
9	L Y+90 R0 FMAX M99	Approach hole 2, call cycle
10	L X+90 R0 FMAX M99	Approach hole 3, call cycle
11	L Y+10 R0 FMAX M99	Approach hole 4, call cycle
12	L Z+250 R0 FMAX M2	Retract the tool, end of program
13	END PGM C200 MM	

Example: Drilling cycles

Program sequence

- Plate has already been pilot drilled for M12, depth of the plate: 20 mm
- Program tapping cycle
- For safety reasons, pre-positioning should be done first of all in the main plane and then in the spindle axis



0 BEGIN PGM C2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+4.5	Tool definition
4 TOOL CALL 1 Z S100	Call the tool
5 L Z+250 R0 FMAX	Retract the touch probe
6 CYCL DEF 2 .0 TAPPING	Cycle definition for tapping
7 CYCL DEF 2 .1 SET UP 2	
8 CYCL DEF 2 .2 DEPTH -25	
9 CYCL DEF 2 .3 DWELL 0	
10 CYCL DEF 2 .4 F175	
11 L X+20 Y+20 R0 FMAX M3	Approach hole 1 in the machining plane
12 L Z+2 R0 FMAX M99	Pre-position in the tool axis
13 L X+70 Y+70 R0 FMAX M99	Approach hole 2 in the machining plane
14 L Z+250 R0 FMAX M2	Retract the tool, end of program
15 END PGM C2 MM	

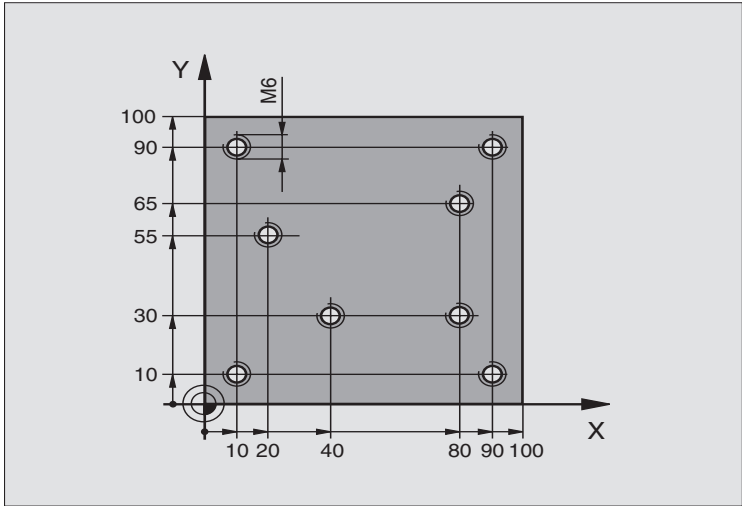
Example: Calling drilling cycles in connection with point tables

Program sequence

- Centering
- Drilling
- Tapping M6

The drill hole coordinates are stored in the point table TAB1.PNT (see next page) and are called by the TNC with CYCL CALL PAT.

The tool radii are selected so that all work steps can be seen in the test graphics.












0 BEGIN PGM 1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+4	Tool definition of center drill
4 TOOL DEF 2 L+0 R+2.4	Define tool: drill
5 TOOL DEF 3 L+0 R+3	Tool definition of tap
6 TOOL CALL 1 Z S5000	Tool call of centering drill
7 L Z+10 R0 F5000	Move tool to clearance height (Enter a value for F.
	The TNC positions to the clearance height after every cycle)
8 SEL PATTERN "TAB1"	Defining point tables
9 CYCL DEF 200 DRILLING	Cycle definition: Centering
Q200=2 ;SET-UP CLEARANCE	
Q201=-2 ;DEPTH	
Q206=150 ;FEED RATE FOR PLUNGING	
Q202=2 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	Surface coordinate (0 must be entered here)
Q204=0 ;2. SET-UP CLEARANCE	2nd set-up clearance (0 must be entered here)
10 CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT.
	Feed rate between points: 5000 mm/min
11 L Z+100 R0 FMAX M6	Retract the tool, change the tool

12	TOOL CALL 2 Z S5000	Call the drilling tool
13	L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
14	CYCL DEF 200 DRILLING	Cycle definition: drilling
	Q200=2 ;SET-UP CLEARANCE	Setup clearance
	Q201=-25 ;DEPTH	Depth
	Q206=150 ;FEED RATE FOR PLUNGING	Feed rate for plunging
	Q202=5 ;PLUNGING DEPTH	Plunging depth
	Q210=0 ;DWELL TIME AT TOP	Dwell time
	Q203=+0 ;SURFACE COORDINATE	Surface coordinate (0 must be entered here)
	Q204=0 ;2. SET-UP CLEARANCE	2nd set-up clearance (0 must be entered here)
15	CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
16	L Z+100 R0 FMAX M6	Retract the tool, change the tool
17	TOOL CALL 3 Z S200	Tool call for tap
18	L Z+50 R0 FMAX	Move tool to clearance height
19	CYCL DEF 2 .0 TAPPING	Cycle definition for tapping
20	CYCL DEF 2 .1 SET UP+2	Setup clearance
21	CYCL DEF 2 .2 DEPTH-15	Depth
22	CYCL DEF 2 .3 DWELLO	Dwell time
23	CYCL DEF 2 .4 F150	Feed rate
24	CYCL CALL PAT F5000 M3	Cycle call in connection with point table TAB1.PNT
25	L Z+100 R0 FMAX M2	Retract the tool, end of program
26	END PGM 1 MM	

Point tableTAB1.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

Cycle	Soft Key
4 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning	
212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
5 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning	
214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
3 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical downfeed	
210 SLOT WITH RECIPROCATING PLUNGE-CUT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	
211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	

POCKET MILLING (Cycle 4)

- 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 3) 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:

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

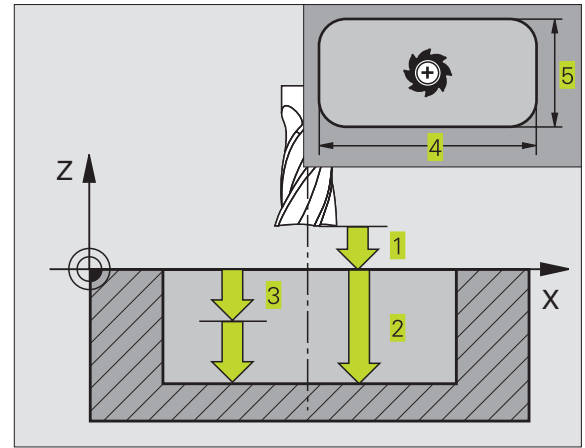
The algebraic sign for the depth parameter determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

The length and the width must each be greater than 2 times the rounding radius.



- ▶ Setup 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 advance to the depth in one movement if:
 - the plunging depth equals the depth
 - the plunging depth is greater than the depth
- ▶ Feed rate for plunging: Traversing speed of the tool during penetration
- ▶ 1st side length **4**: Pocket length, parallel to the main axis of the working plane
- ▶ 2nd side length **5**: Pocket width
- ▶ Feed rate F: Traversing speed of the tool in the working plane



Example NC blocks:

```

27 CYCL DEF 4.0 POCKET MILLING
28 CYCL DEF 4.1 SET UP 2
29 CYCL DEF 4.2 DEPTH -20
30 CYCL DEF 4.3 PLNGNG 5 F100
31 CYCL DEF 4.4 X80
32 CYCL DEF 4.5 Y60
33 CYCL DEF 4.6 F275 DR+ RADIUS 5
    
```

- ▶ Direction of the milling path
 DR + : climb milling with M3
 DR - : up-cut milling with M3
- ▶ Rounding radius: Radius of the pocket corners. If radius = 0 is entered, the pocket corners will be rounded with the radius of the cutter.

Calculations:

Stepover factor $k = K \times R$

where

K: is the overlap factor, preset in machine parameter 7430, and

R: is the cutter radius

POCKET FINISHING (Cycle 212)

- 1 The TNC automatically moves the tool in the tool axis to 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 plunge-cuts into the pocket center.
- 3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX 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 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).

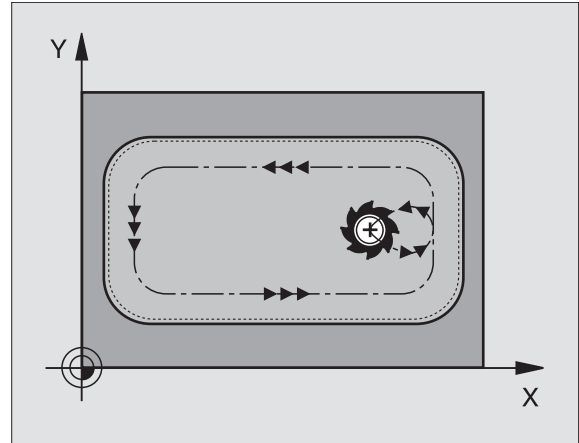


Before programming, note the following:

The algebraic sign for the depth parameter determines the working direction.

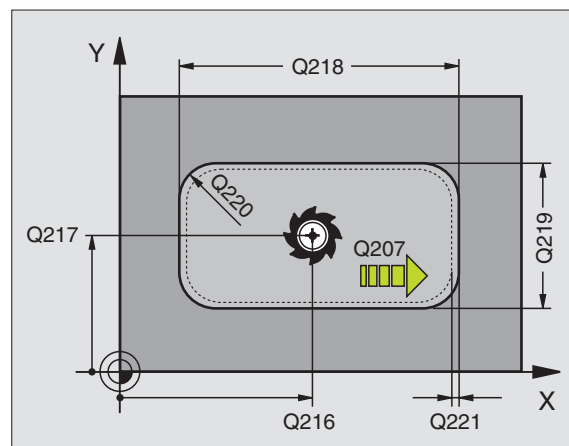
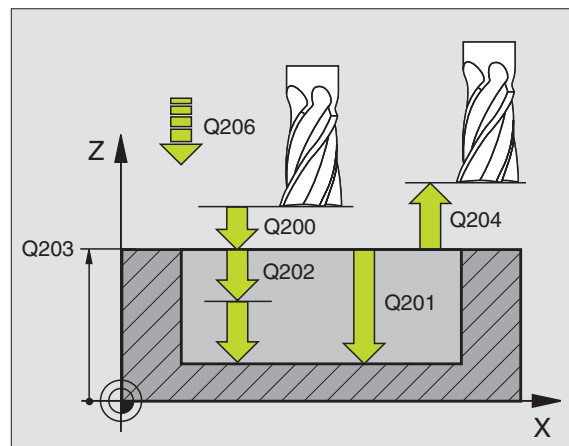
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Minimum size of the pocket: 3 times the tool radius.





- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ 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 lower value 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 main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the pocket in the secondary axis of the working plane
- ▶ First side length Q218 (incremental value): Pocket length, parallel to the main axis of the working plane
- ▶ Second side length Q219 (incremental value): Pocket length, parallel to the secondary 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): Allowance in the main axis of the working plane referenced to the length of the pocket. This value is only required by the TNC for calculating the preparatory position.



Example NC blocks:

34 CYCL DEF 212 POCKET FINISHING	
Q200=2	; SET-UP CLEARANCE
Q201=-20	; DEPTH
Q206=150	; FEED RATE FOR PLUNGING
Q202=5	; PLUNGING DEPTH
Q207=500	; FEED RATE FOR MILLING
Q203=+0	; SURFACE COORDINATE
Q204=50	; 2. SET-UP CLEARANCE
Q216=+50	; CENTER IN 1ST AXIS
Q217=+50	; CENTER IN 2ND AXIS
Q218=80	; 1ST SIDE LENGTH
Q219=60	; 2ND SIDE LENGTH
Q220=5	; CORNER RADIUS
Q221=0	; ALLOWANCE

STUD FINISHING (Cycle 213)

- 1** The TNC moves the tool in the tool axis to 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. 3.5 times the tool radius.
- 3** If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX 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 in FMAX to set-up clearance, or — if programmed — to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).



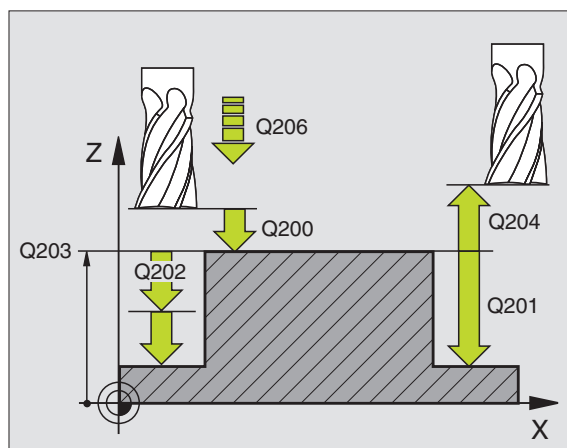
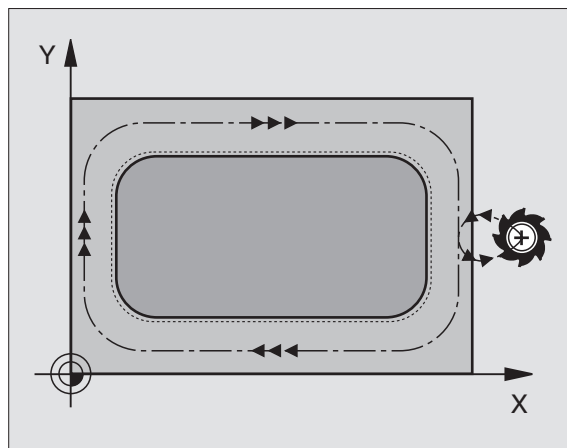
Before programming, note the following:

The algebraic sign for the depth parameter determines the working direction.

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.



- ▶ 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.



Example NC blocks:

35 CYCL DEF 213 STUD FINISHING

Q200=2 ; SET-UP CLEARANCE

```
Q201=-20      ;DEPTH
```

Q206=150 ; FEED RATE FOR PLUNGING

Q202=5 ; PLUNGING DEPTH

Q207=500 ; FEED RATE FOR MILLING

Q203=+0 ; SURFACE COORDINATE

Q204=50 ; 2. SET-UP CLEARANCE

Q216=+50 ; CENTER IN 1ST AXIS

Q217=+50 ;CENTER IN 2ND AXIS

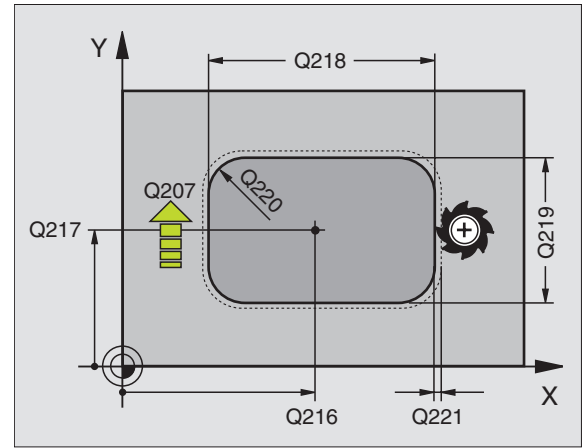
Q218=80 ; 1ST SIDE LENGTH

Q219=60 ; 2ND SIDE LENGTH

Q220=5 ; CORNER RADIUS

Q221=0 ; ALLOWANCE

- ▶ 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 main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the stud in the secondary axis of the working plane
- ▶ First side length Q218 (incremental value): Stud length, parallel to the main axis of the working plane
- ▶ Second side length Q219 (incremental value): Stud length, 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 in the main axis of the working plane referenced to the length of the stud. This value is only required by the TNC for calculating the preparatory position.



CIRCULAR POCKET MILLING (Cycle 5)

- 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 Cycle 4 POCKET MILLING.
- 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.



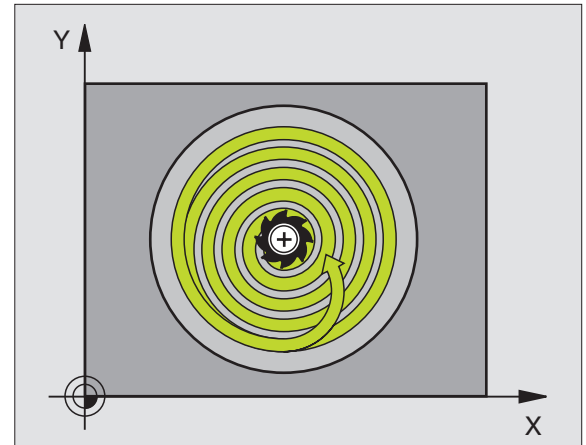
Before programming, note the following:

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

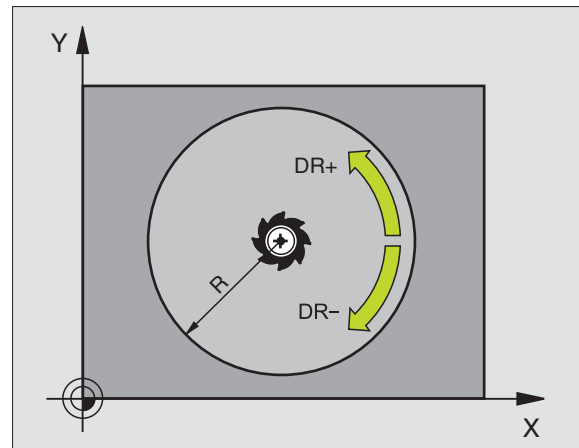
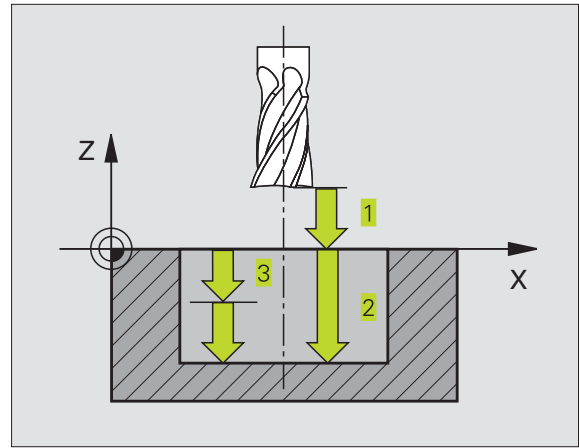
The algebraic sign for the depth parameter determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.





- ▶ Setup 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 advance to the depth in one movement if:
 - the plunging depth equals the depth
 - the plunging depth is greater than the depth
- ▶ Feed rate for plunging: Traversing speed of the tool during penetration
- ▶ CIRCLE RADIUS: Radius of the circular pocket
- ▶ Feed rate F: Traversing speed of the tool in the working plane
- ▶ Direction of the milling path
 - DR + : climb milling with M3
 - DR - : up-cut milling with M3



Example NC blocks:

```

36 CYCL DEF 5.0 CIRCULAR POCKET
37 CYCL DEF 5.1 SET UP 2
38 CYCL DEF 5.2 DEPTH -20
39 CYCL DEF 5.3 PLNGNG 5 F100
40 CYCL DEF 5.4 RADIUS 40
41 CYCL DEF 5.5 F250 DR+
  
```

CIRCULAR POCKET FINISHING (Cycle 214)

- 1 The TNC automatically moves the tool in the tool axis to 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 FMAX 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 (4 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, the TNC retracts the tool in FMAX 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).



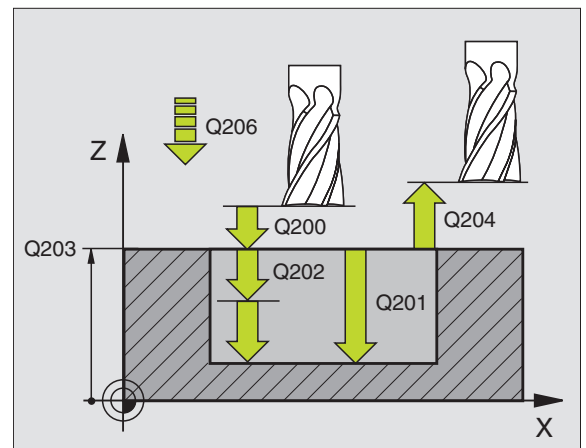
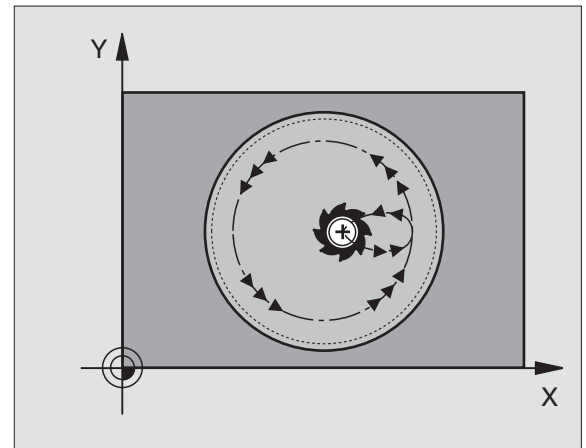
Before programming, note the following:

The algebraic sign for the depth parameter determines the working direction.

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.



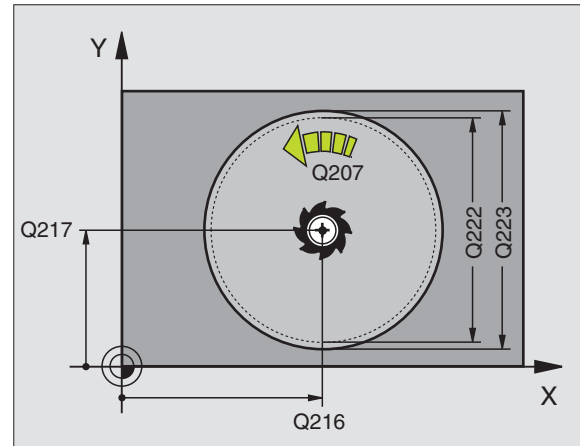
- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Depth Q201 (incremental value): Distance between workpiece surface and bottom of pocket
- ▶ Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a lower value 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



Example NC blocks:

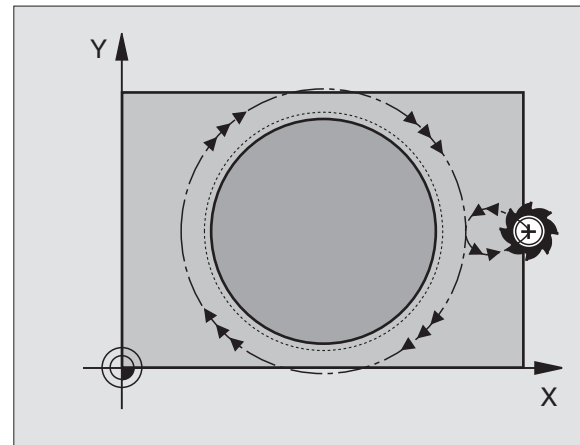
42 CYCL DEF 214 CIRCULAR POCKET FINISHING	
Q200=2	; SET-UP CLEARANCE
Q201=-20	; DEPTH
Q206=150	; FEED RATE FOR PLUNGING
Q202=5	; PLUNGING DEPTH
Q207=500	; FEED RATE FOR MILLING
Q203=+0	; SURFACE COORDINATE
Q204=50	; 2. SET-UP CLEARANCE
Q216=+50	; CENTER IN 1ST AXIS
Q217=+50	; CENTER IN 2ND AXIS
Q222=79	; WORKPIECE BLANK DIA.
Q223=80	; FINISHED PART DIA.

- ▶ 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 main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the pocket in the secondary axis of the working plane
- ▶ Workpiece blank dia. Q222: Diameter of the premachined pocket. Enter a workpiece blank diameter less than the diameter of the finished part. If you enter Q222 = 0, then the TNC plunge-cuts into the pocket center.
- ▶ Finished part dia. Q223: Diameter of the finished pocket. Enter the diameter of the finished part to be greater than the workpiece blank diameter.



CIRCULAR STUD FINISHING (Cycle 215)

- 1 The TNC automatically moves the tool in the tool axis to 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. 3.5 times the tool radius.
- 3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX 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 (4 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, the TNC retracts the tool in FMAX 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).





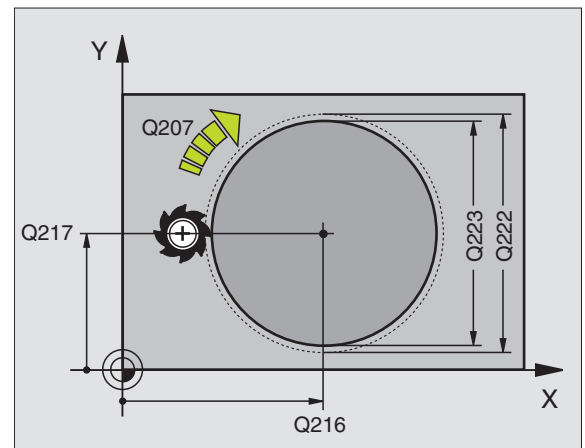
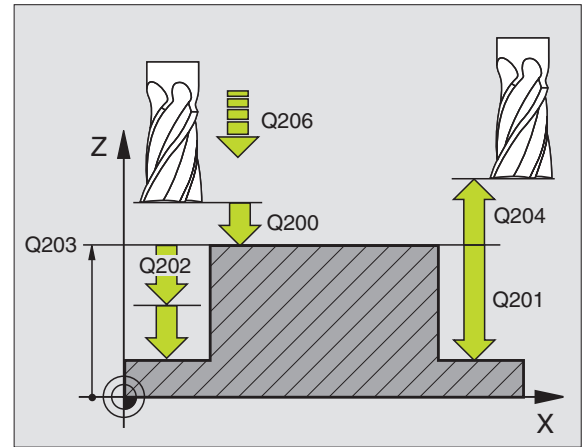
Before programming, note the following:

The algebraic sign for the depth parameter determines the working direction.

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.



- ▶ 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 main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the stud in the secondary axis of the working plane
- ▶ Workpiece blank diameter Q222: Diameter of the premachined stud. 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.



Example NC blocks:

43 CYCL DEF 215 C. STUD FINISHING	
Q200=2	; SET-UP CLEARANCE
Q201=-20	; DEPTH
Q206=150	; FEED RATE FOR PLUNGING
Q202=5	; PLUNGING DEPTH
Q207=500	; FEED RATE FOR MILLING
Q203=+0	; SURFACE COORDINATE
Q204=50	; 2. 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 3)

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 in rapid traverse FMAX to 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:

Program a positioning block for the starting point in the working plane — to the center of the slot (second side length) and, within the slot, offset by the tool radius — with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

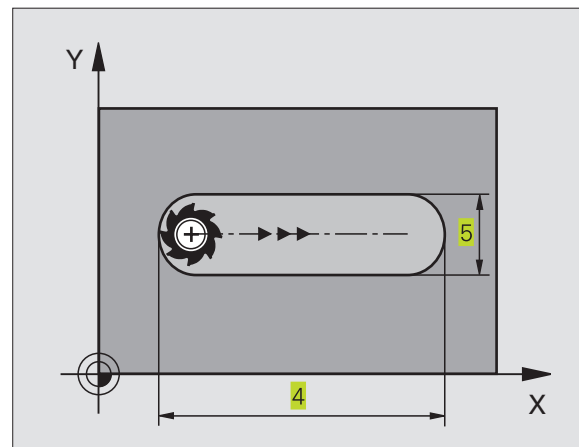
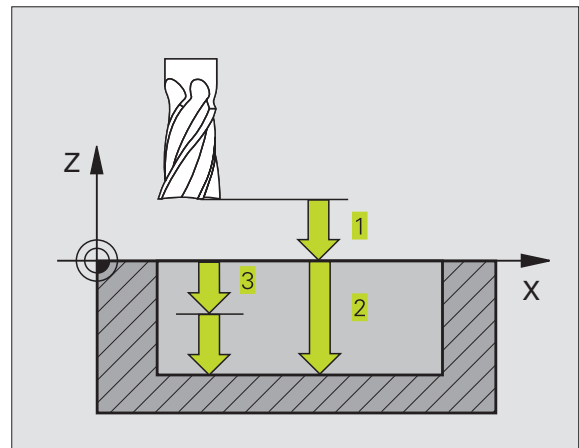
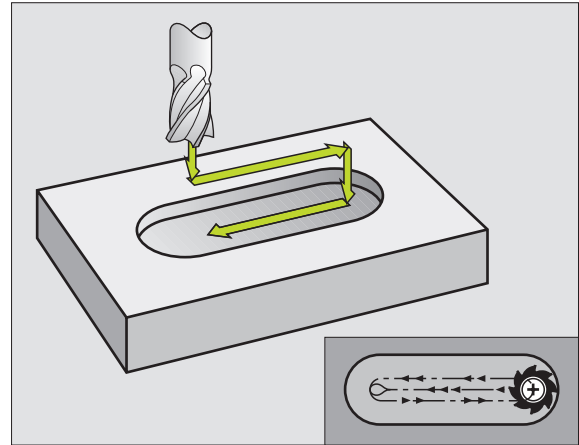
The algebraic sign for the depth parameter determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point.

The cutter diameter must be not be larger than the slot width and not smaller than half the SLOT WIDTH.



- ▶ Setup 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 advance to the depth in one movement if:
 - the plunging depth equals the depth
 - the plunging depth is greater than the depth



- ▶ Feed rate for plunging: Traversing speed of the tool 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

SLOT (Slot milling) with reciprocating plunge cut (Cycle 210)



Before programming, note the following:

The algebraic sign for the depth parameter determines the working direction.

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.

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 set-up clearance above the workpiece surface.
- 2 The tool moves at reduced feed rate to the workpiece surface. From there, the cutter advances at the feed rate for milling 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 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.

Finishing process

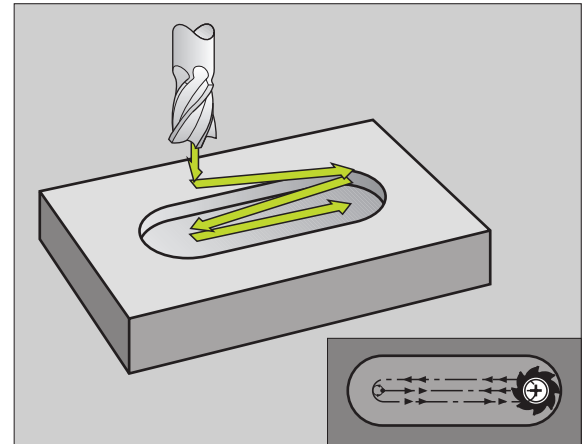
- 5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3).
- 6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.
- 7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to set-up clearance and — if programmed — to the 2nd set-up clearance.

Example NC blocks:

```

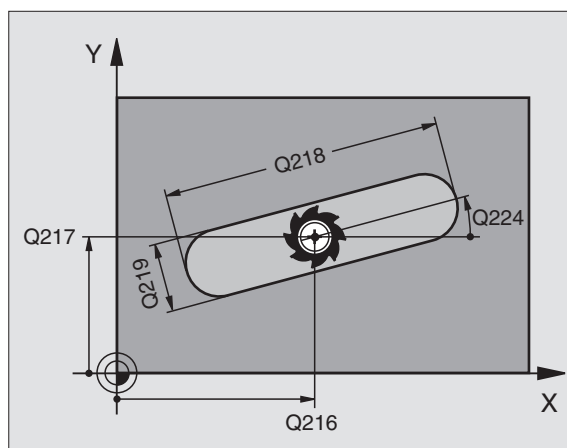
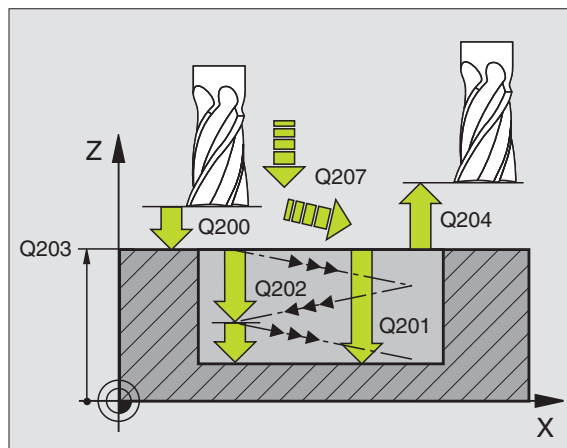
44 CYCL DEF 3.0 SLOT MILLING
45 CYCL DEF 3.1 SET UP 2
46 CYCL DEF 3.2 DEPTH -20
47 CYCL DEF 3.3 PLNGNG 5 F100
48 CYCL DEF 3.4 X+80
49 CYCL DEF 3.5 Y12
50 CYCL DEF 3.6 F275

```





- ▶ 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 extent of machining:
0: Roughing and finishing
1: Roughing only
2: Finishing only
- ▶ 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 main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the slot in the secondary axis of the working plane
- ▶ First side length Q218 (value parallel to the main 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).
- ▶ 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.



Example NC blocks:

51	CYCL DEF 210 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=+0	;SURFACE COORDINATE
Q204=50	;2. SET-UP CLEARANCE
Q216=+50	;CENTER IN 1ST AXIS
Q217=+50	;CENTER IN 2ND AXIS
Q218=80	;1ST SIDE LENGTH
Q219=12	;2ND SIDE LENGTH
Q224=+15	;ANGLE OF ROTATION

CIRCULAR SLOT with reciprocating plunge-cut (Cycle 211)

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 reduced feed rate to the workpiece surface. From there, the cutter advances at the feed rate for milling — plunge-cutting obliquely into the material — until it reaches the center of 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 (2 to 3) is repeated until the programmed milling depth is reached.
- 4 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot.

Finishing process

- 5 For finishing the slot, the TNC advances the tool tangentially to the contour of the finished part. The tool subsequently climb-mills the contour (with M3). 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 in rapid traverse FMAX to set-up clearance and — if programmed — to the 2nd set-up clearance.

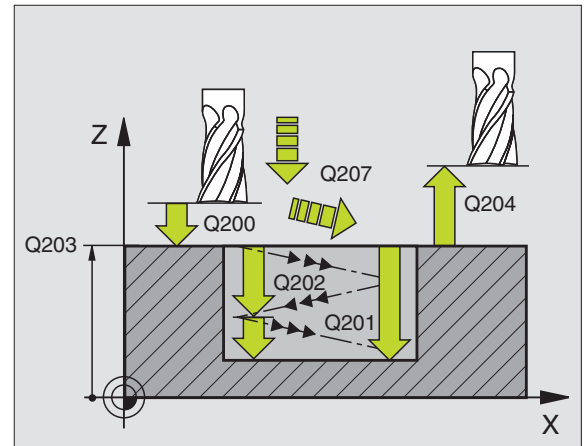
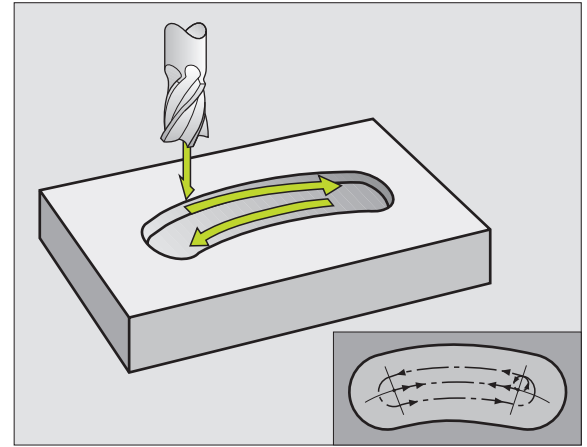


Before programming, note the following:

The algebraic sign for the depth parameter determines the working direction.

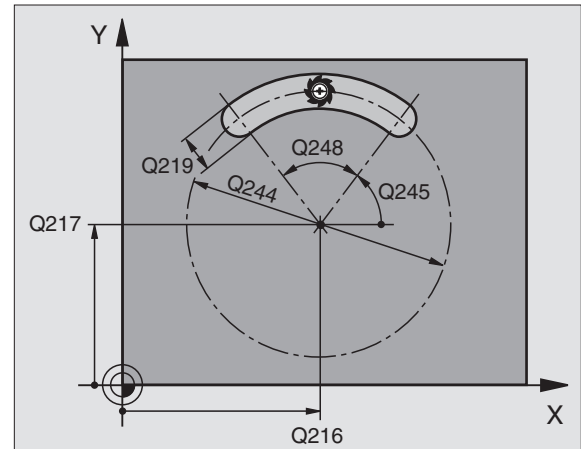
The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.





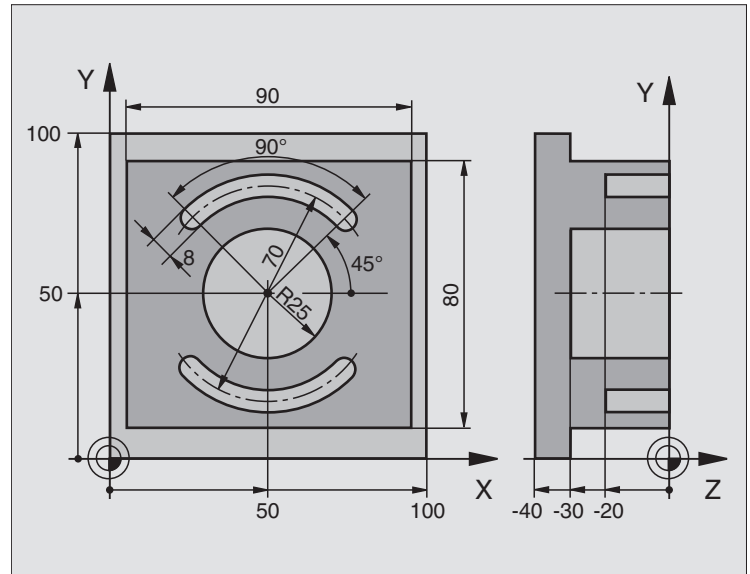
- ▶ Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- ▶ Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot
- ▶ Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- ▶ Plunging depth Q202 (incremental 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 extent of machining:
0: Roughing and finishing
1: Roughing only
2: Finishing only
- ▶ 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 main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the slot in the secondary 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).
- ▶ 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



Example NC blocks:

52	CYCL DEF 211	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=+0	; SURFACE COORDINATE
	Q204=50	; 2. SET-UP CLEARANCE
	Q216=+50	; CENTER IN 1ST AXIS
	Q217=+50	; CENTER IN 2ND AXIS
	Q244=80	; PITCH CIRCLE DIAMETER
	Q219=12	; 2ND SIDE LENGTH
	Q245=+45	; STARTING ANGLE
	Q248=90	; ANGULAR LENGTH

Example: Milling pockets, studs and slots



0	BEGIN PGM C210 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+6	Define the tool for roughing/finishing
4	TOOL DEF 2 L+0 R+3	Define slotting mill
5	TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
6	L Z+250 R0 F MAX	Retract the touch probe
7	CYCL DEF 213 STUD FINISHING	Define cycle for machining the contour outside
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-30 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q202=5 ;PLUNGING DEPTH	
	Q207=250 ;FEED RATE FOR MILLING	
	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 ;ALLOWANCE IN 1ST AXIS 1ST AXIS	
8	CYCL CALL M3	Define CIRCULAR POCKET MILLING cycle

9	CYCL DEF 5.0 CIRCULAR POCKET	
10	CYCL DEF 5.1 SET UP 2	
11	CYCL DEF 5.2 DEPTH -30	
12	CYCL DEF 5.3 PLNGNG 5 F250	
13	CYCL DEF 5.4 RADIUS 25	
14	CYCL DEF 5.5 F400 DR+	Call CIRCULAR POCKET MILLING cycle
15	L Z+2 R0 FMAX M99	Tool change/
16	L Z+250 R0 FMAX M6	Call slotting mill
17	TOOL CALL 2 Z S5000	Define cycle for slot 1
18	CYCL DEF 211 CIRCULAR SLOT	
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-20 ;DEPTH	
	Q207=250 ;FEED RATE FOR MILLING	
	Q202=5 ;PLUNGING DEPTH	
	Q215=0 ;MACHINING OPERATION	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=100 ;2ND SET-UP CLEARANCE	
	Q216=+50 ;CENTER IN 1ST AXIS	
	Q217=+50 ;CENTER IN 2ND AXIS	
	Q244=70 ;PITCH CIRCLE DIAMETR	
	Q219=8 ;SECOND SIDE LENGTH	
	Q245=+45 ;STARTING ANGLE	
	Q248=90 ;ANGULAR LENGTH	
19	CYCL CALL M3	Call cycle for slot 1
20	FN 0: Q245 = +225	New starting angle for slot 2
21	CYCL CALL	Call cycle for slot 2
22	L Z+250 R0 FMAX M2	Retract the tool, end of program
23	END PGM C210 MM	

Example: Roughing and finishing a rectangular pocket in connection with point tables

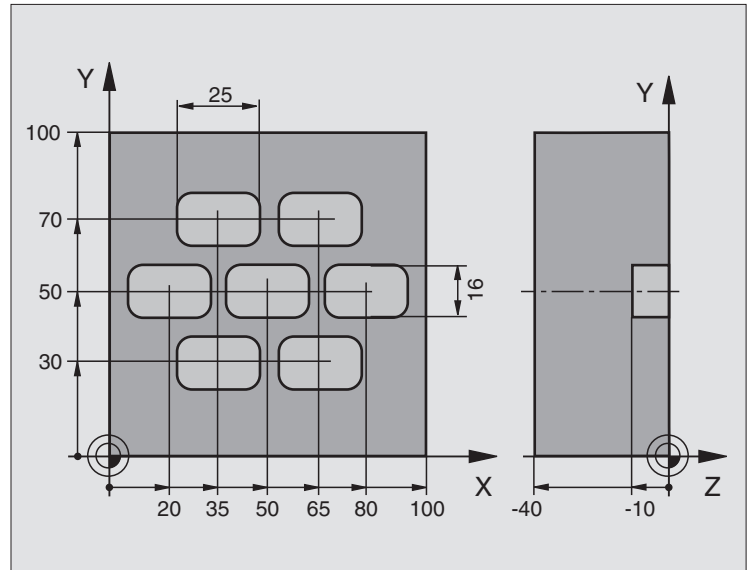
Program sequence

- Rectangular pocket roughing with Cycle 4
- Rectangular pocket finishing with Cycle 212

The center point coordinates are stored in the point table MUSTPKT.PNT (see next page) and are called by the TNC with CYCL CALL PAT.

Make sure that in the cycle definition 212 both the coordinates for the pocket center (Q212 and Q213), and the coordinate of the workpiece surface is defined as 0.

To mill the pockets to different levels of depth, change the coordinates in the point table MUSTPKT.PNT.



0	BEGIN PGM TAKOM MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+3	Tool definition for roughing
4	TOOL DEF 2 L+0 R+3	Tool definition for finishing
5	TOOL CALL 1 Z S5000	Tool call for roughing
6	L Z+10 R0 F5000	Move tool to clearance height (enter a value for F)
		The TNC positions to the clearance height after every cycle)
7	SEL PATTERN "MUSTPKT"	Defining point tables
8	CYCL DEF 4 .0 POCKET MILLING	Define cycle for roughing a pocket
9	CYCL DEF 4 .1 SET UP+2	
10	CYCL DEF 4 .2 DEPTH-10	
11	CYCL DEF 4 .3 PLNGNG+3 F150	
12	CYCL DEF 4 .4 X+25	
13	CYCL DEF 4 .5 Y+15	
14	CYCL DEF 4 .6 F350 DR+ RADIUS4	
15	CYCL CALL PAT F5000 M3	Cycle call in connection with point table MUSTPKT.PNT



16	L	Z+100	R0	FMAX	M6	Retract the tool, change the tool
17	T00L	CALL	2	Z	S5000	
18	L	Z+10	R0	F5000		Move tool to clearance height (enter a value for F)
19	CYCL	DEF	212	POCKET	FINISHING	Cycle definition for pocket finishing
	Q200=2				;SET-UP CLEARANCE	
	Q201=-10				;DEPTH	
	Q206=150				;FEED RATE FOR PLUNGING	
	Q202=5				;PLUNGING DEPTH	
	Q207=500				;FEED RATE FOR MILLING	
	Q203=+0				;SURFACE COORDINATE	Surface coordinate (0 must be entered here)
	Q204=0				;2. SET-UP CLEARANCE	2nd set-up clearance (0 must be entered here)
	Q216=+0				;CENTER IN 1ST AXIS	Center of X axis (0 must be entered here)
	Q217=+0				;CENTER IN 2ND AXIS	Center of Y axis (0 must be entered here)
	Q218=25				;1ST SIDE LENGTH	
	Q219=16				;2ND SIDE LENGTH	
	Q220=4				;CORNER RADIUS	
	Q221=0.5				;ALLOWANCE IN 1ST AXIS	
20	CYCL	CALL	PAT	F5000	M3	Cycle call in connection with point table MUSTPKT.PNT
21	L	Z+100	R0	FMAX	M2	Retract the tool, end of program
22	END	PGM	TAKOM	MM		


Point table MUSTPKT.PNT

	MUSTPKT	.PNT	MM
NR	X	Y	Z
0	+35	+30	+0
1	+65	+30	+0
2	+80	+50	+0
3	+50	+50	+0
4	+20	+50	+0
5	+35	+70	+0
6	+65	+70	+0
[END]			

8.5 Cycles for Machining Hole Patterns

The TNC offers two cycles for machining surfaces with the regular point patterns:

Cycle	Soft Key
220 CIRCULAR PATTERN	
221 LINEAR PATTERN	

 To create irregular point patterns, use point tables (see section "8.2 Point Tables").

You can combine Cycle 220 and Cycle 221 with the following fixed cycles:

Cycle 1	PECKING
Cycle 2	TAPPING with a floating tap holder
Cycle 3	SLOT MILLING
Cycle 4	POCKET MILLING
Cycle 5	CIRCULAR POCKET MILLING
Cycle 17	RIGID TAPPING
Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL MILLING CYCLE
Cycle 204	BACK BORING
Cycle 212	POCKET FINISHING
Cycle 213	STUD FINISHING
Cycle 214	CIRCULAR POCKET FINISHING
Cycle 215	CIRCULAR STUD FINISHING

CIRCULAR PATTERN (Cycle 220)

- 1 At rapid traverse, the TNC moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:

- Move to 2nd set-up clearance (tool axis)
- Approach starting point in the working plane
- Move to set-up clearance above the workpiece surface (tool 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.



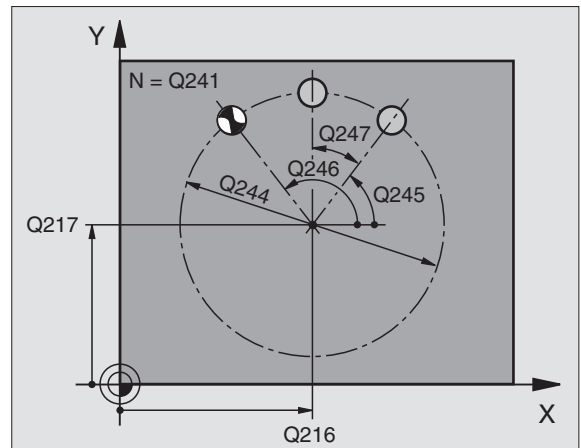
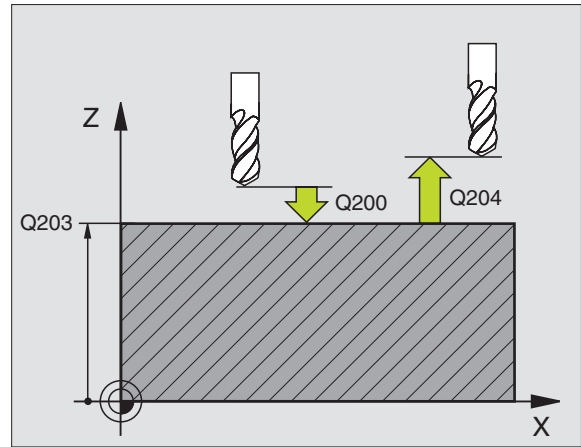
Before programming, note the following:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

If you combine Cycle 220 with one of the fixed cycles 200 to 204 and 212 to 215, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 220 will be effective for the selected fixed cycle.



- ▶ Center in 1st axis Q216 (absolute value): Center of the pitch circle in the main axis of the working plane
- ▶ Center in 2nd axis Q217 (absolute value): Center of the pitch circle in the secondary axis of the working plane
- ▶ Pitch circle diameter Q244: Diameter of the pitch circle
- ▶ Starting angle Q245 (absolute value): Angle between the main 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 main axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise.
- ▶ ANGLE STEP Q247 (incremental): Angle between two machining operations on a pitch circle. If you enter an ANGLE STEP of 0, the TNC will calculate the ANGLE STEP from the STARTING and STOPPING ANGLES and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the STOPPING ANGLE into account. The sign for the ANGLE STEP determines the working direction (- = clockwise).



Example NC blocks:

53 CYCL DEF 220 POLAR PATTERN

Q216=+50 ;CENTER IN 1ST AXIS

Q217=+50 ;CENTER IN 2ND AXIS

Q244=80 ;PITCH CIRCLE DIAMETER

Q245=+0 ;STARTING ANGLE

Q246=+360 ;STOPPING ANGLE

Q247=+0 ;STEPPING ANGLE

Q241=8 ;NR OF REPETITIONS

Q200=2 ;SET-UP CLEARANCE

Q203=+0 ;SURFACE COORDINATE

Q204=50 ;2. SET-UP CLEARANCE

- ▶ Number of repetitions Q241: Number of machining operations on a pitch circle
- ▶ Set-up clearance Q200 (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.

LINEAR PATTERN (Cycle 221)



Before programming, note the following:

Cycle 221 is DEF active, which means that Cycle 221 calls the last defined fixed cycle.

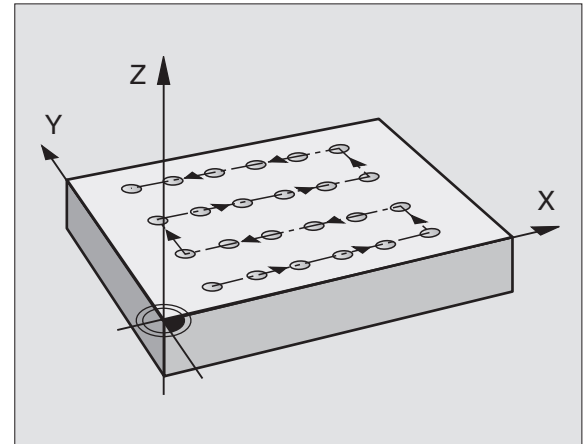
If you combine Cycle 221 with one of the fixed cycles 200 to 204 and 212 to 215, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 221 will be effective for the selected fixed cycle.

- 1 The TNC automatically moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:

- Move to 2nd set-up clearance (tool axis)
- Approach starting point in the working plane
- Move to set-up clearance above the workpiece surface (tool axis)

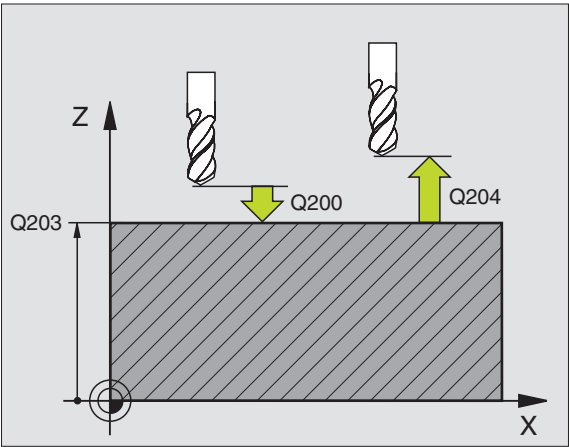
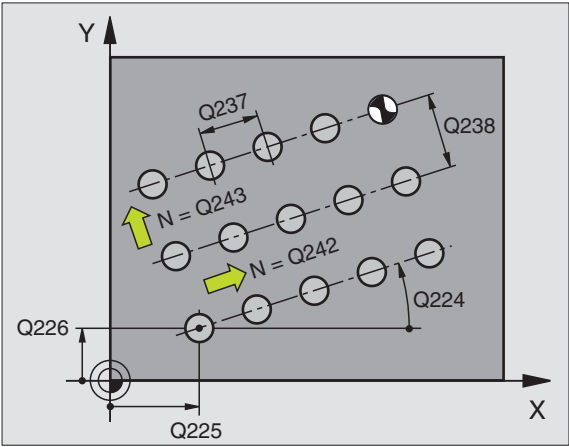
- 2 From this position, the TNC executes the last defined fixed cycle.
- 3 The tool then approaches the starting point for the next machining operation in the positive main axis direction at set-up clearance (or 2nd set-up clearance).
- 4 This process (1 to 3) is repeated until all machining operations 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 The tool then approaches the starting point for the next machining operation in the negative main axis direction and executes the operation there.



- 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.



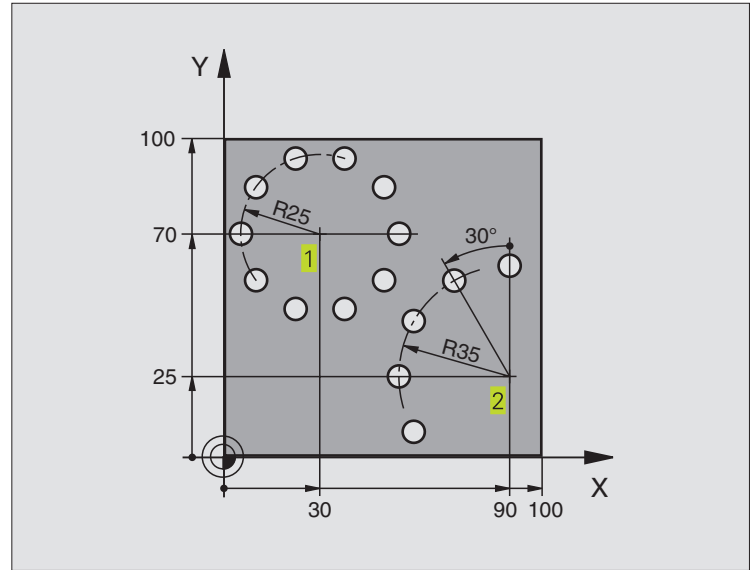
- ▶ Starting point 1st axis Q225 (absolute value): Coordinate of the starting point in the main axis of the working plane
- ▶ Starting point 2nd axis Q226 (absolute value): Coordinate of the starting point in the secondary axis of the working plane
- ▶ Spacing in 1st axis Q237 (incremental value): Spacing between the individual points on a line
- ▶ Spacing in 2nd axis Q238 (incremental): Spacing between the individual lines
- ▶ 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 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.



Example NC blocks:

54	CYCL DEF 221	CARTESIAN PATTRN
Q225=+15	; STARTNG PNT 1ST AXIS	
Q226=+15	; STARTNG PNT 2ND AXIS	
Q237=+10	; SPACING IN 1ST AXIS	
Q238=+8	; SPACING IN 2ND AXIS	
Q242=6	; NUMBER OF COLUMNS	
Q243=4	; NUMBER OF LINES	
Q224=+15	; ANGLE OF ROTATION	
Q200=2	; SET-UP CLEARANCE	
Q203=+0	; SURFACE COORDINATE	
Q204=50	; 2. SET-UP CLEARANCE	

Example: Circular hole patterns



0	BEGIN PGM HOLEPAT MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+3	Tool definition
4	TOOL CALL 1 Z S3500	Call the tool
5	L Z+250 R0 FMAX M3	Retract the touch probe
6	CYCL DEF 200 DRILLING	Cycle definition: drilling
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-15 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q202=4 ;PLUNGING DEPTH	
	Q210=0 ;DWELL TIME	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=0 ;2. SET-UP CLEARANCE	

7	CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 1, CYCL 200 is called automatically,
		Q200, Q203 and Q204 are effective as defined in Cycle 220
	Q216=+30 ;CENTER IN 1ST AXIS	
	Q217=+70 ;CENTER IN 2ND AXIS	
	Q244=50 ;PITCH CIRCLE DIAMETR	
	Q245=+0 ;STARTING ANGLE	
	Q246=+360 ;STOPPING ANGLE	
	Q247=+0 ;STEPPING ANGLE	
	Q241=10 ;NR OF REPETITIONS	
	Q200=2 ;SET-UP CLEARANCE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=100 ;2ND SET-UP CLEARANCE	
8	CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 2, CYCL 200 is called automatically,
		Q200, Q203 and Q204 are effective as defined in Cycle 220
	Q216=+90 ;CENTER IN 1ST AXIS	
	Q217=+25 ;CENTER IN 2ND AXIS	
	Q244=70 ;PITCH CIRCLE DIAMETR	
	Q245=+90 ;STARTING ANGLE	
	Q246=+360 ;STOPPING ANGLE	
	Q247=+30 ;STEPPING ANGLE	
	Q241=5 ;NR OF REPETITIONS	
	Q200=2 ;SET-UP CLEARANCE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=100 ;2ND SET-UP CLEARANCE	
9	L Z+250 R0 FMAX M2	Retract the tool, end of program
10	END PGM HOLEPAT MM	

8.6 SL cycles

SL cycles allow the contour-oriented machining of complex contours.

Characteristics of the contour

- A contour can be composed of several overlapping subcontours (up to 12 subcontours are possible). Islands and pockets can form a subcontour.
- The subcontour list (subprogram numbers) is entered in Cycle 14 CONTOUR GEOMETRY. The TNC calculates the contour from the subcontours.
- The individual subcontours are defined in subprograms.
- The memory capacity for programming an SL cycle is limited. All subprograms together can contain, for example, up to 128 straight-line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed. When programmed within subcontours, they are also effective for the following subprograms, but do not need to be reset after the cycle has been called.
- 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 RR.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL.
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. Parallel axes are allowed.

Characteristics of the fixed cycles



With MP7420.0 and MP7420.1 you define how the TNC should move the tool during area clearance (see "15.1 General User Parameters").

- 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 6). 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 6) into consideration

Overview of SL cycles

Cycle	Soft Key
14 CONTOUR GEOMETRY (essential)	
15 PILOT DRILLING (optional)	
6 ROUGH-OUT (essential)	
16 CONTOUR MILLING (optional)	

CONTOUR GEOMETRY (Cycle 14)

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY (see figure bottom right).



Before programming, note the following:

Cycle 14 is DEF active which means that it becomes effective as soon as it is defined in the part program.

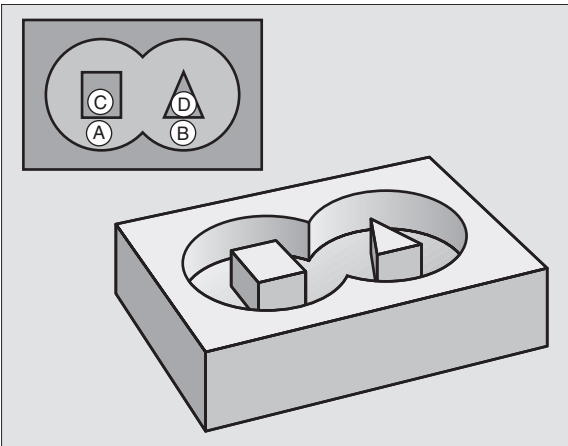
You can list up to 12 subprograms (subcontours) in Cycle 14.



► 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.

Program structure: Working with SL cycles

```
0 BEGIN PGM SL MM
...
12 CYCL DEF 14.0 contour geometry ...
...
16 CYCL DEF 15.0 PILOT DRILLING
17 CYCL CALL
...
18 CYCL DEF 6.0 ROUGH-OUT
19 CYCL CALL
...
26 CYCL DEF 16.0 CONTOUR MILLING
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL MM
```



Example NC blocks:

```
3 CYCL DEF 14.0 CONTOUR GEOMETRY
4 CYCL DEF 14.1 CONTOUR LABEL 1 /2 /3
```


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 14 CONTOUR GEOMETRY in a main program.

Pockets *A* and *B* overlap.

The TNC calculates the points of intersection S_1 and S_2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Left pocket

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

Subprogram 2: Right pocket

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```

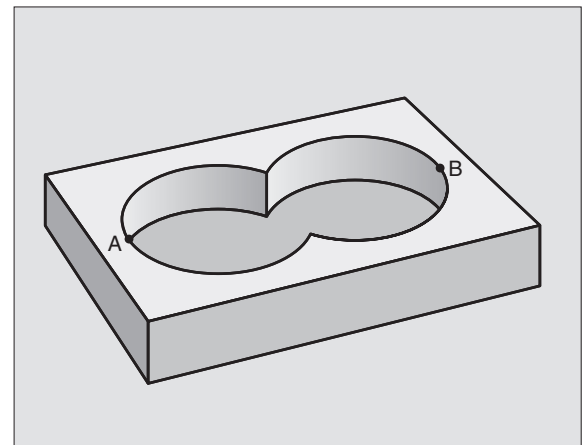
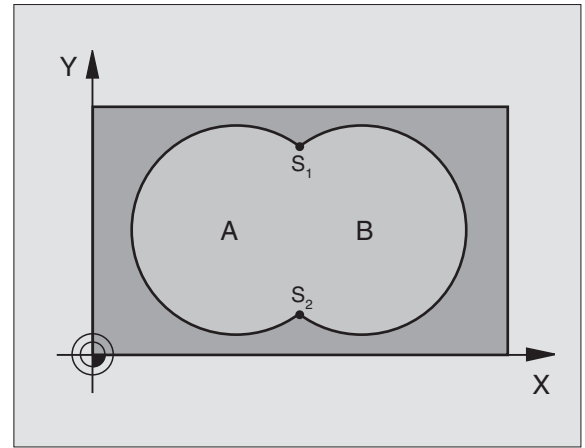
Area of inclusion

Both surfaces *A* and *B* are to be machined, including the mutually overlapped area:

- The surfaces *A* and *B* must be pockets.
- The first pocket (in Cycle 14) must start outside the second pocket.

Surface *A*

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```



Surface *B*

56	LBL	2
57	L	X+90 Y+50 RR
58	CC	X+65 Y+50
59	C	X+90 Y+50 DR-
60	LBL	0

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*

51	LBL	1
52	L	X+10 Y+50 RR
53	CC	X+35 Y+50
54	C	X+10 Y+50 DR-
55	LBL	0

Surface *B*

56	LBL	2
57	L	X+90 Y+50 RL
58	CC	X+65 Y+50
59	C	X+90 Y+50 DR-
60	LBL	0

Area of intersection

Only the area overlapped by both *A* and *B* is to be machined. (The areas covered by *A* or *B* alone are to be left unmachined.)

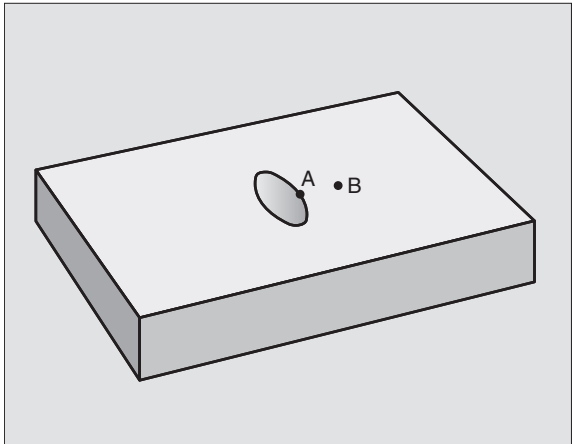
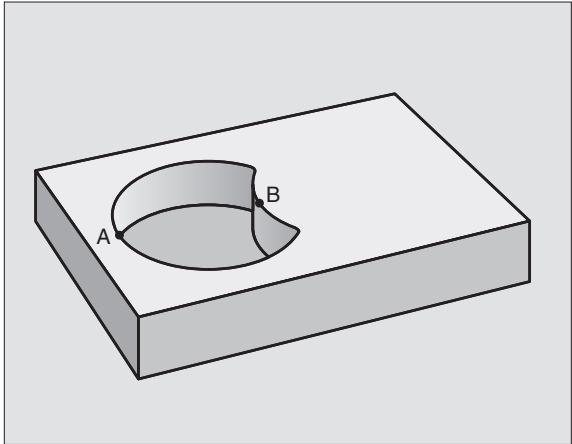
- *A* and *B* must be pockets.
- *A* must start inside of *B*.

Surface *A*

51	LBL	1
52	L	X+60 Y+50 RR
53	CC	X+35 Y+50
54	C	X+60 Y+50 DR-
55	LBL	0

Surface *B*

56	LBL	2
57	L	X+90 Y+50 RR
58	CC	X+65 Y+50
59	C	X+90 Y+50 DR-
60	LBL	0



Pilot drilling (Cycle 15)

Process

Same as Cycle 1 Pecking (see "8.3 Drilling Cycles").

Application

Cycle 15 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.



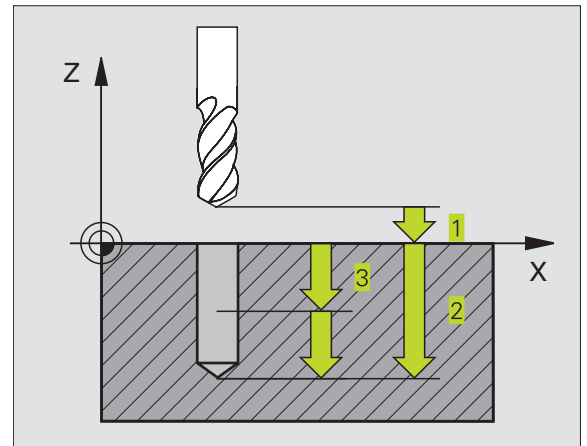
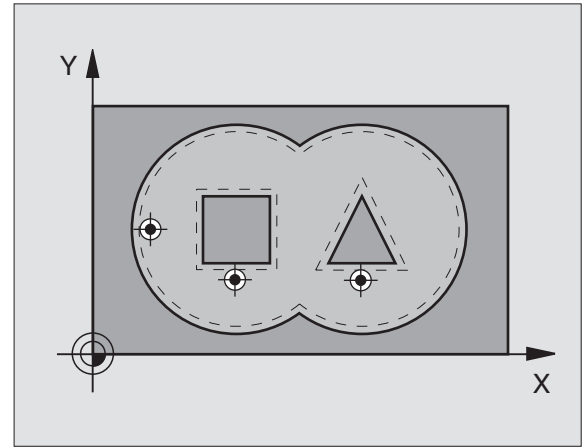
Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).



- ▶ Setup 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 TNC will go to depth in one movement if:
 - the plunging depth equals the total hole depth
 - the plunging depth is greater than the total hole depth

The total hole depth does not have to be a multiple of the plunging depth.
- ▶ Feed rate for plunging: Traversing speed in mm/min for drilling
- ▶ Finishing allowance: Allowance in the machining plane



Example NC blocks:

```
5 CYCL DEF 15.0 PILOT DRILLING
6 CYCL DEF 15.1 DIST+2 DEPTH-25
7 CYCL DEF 15.2 PLNGNG+3 F250 ALLOW+0.1
```

ROUGH-OUT (Cycle 6)

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 entered milling depth is reached.

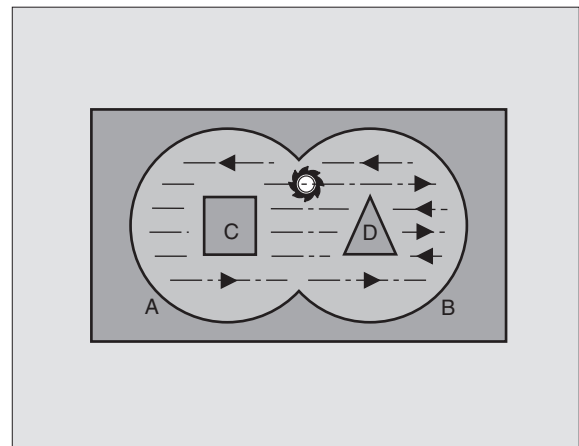
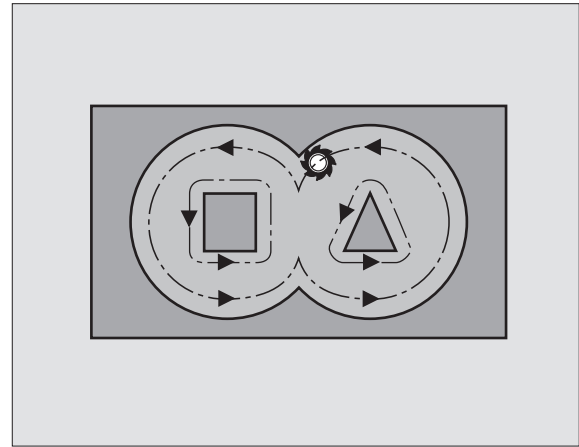


Before programming, note the following:

With MP7420.0 and MP7420.1 you define how the TNC should machine the contour (see section "15.1 General User Parameters").

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

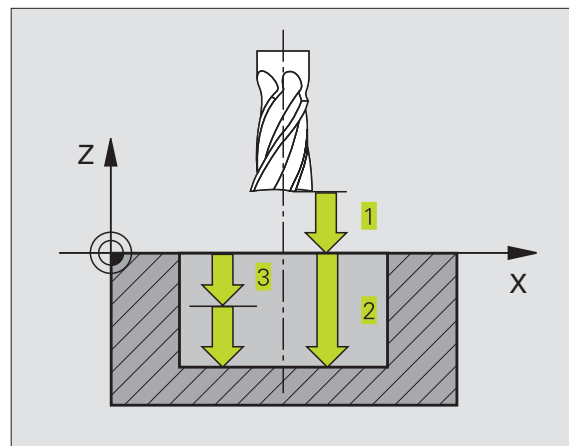
This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 15.





- ▶ Setup clearance **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- ▶ Milling depth **2** (incremental value): Distance between workpiece surface and pocket floor
- ▶ Plunging depth **3** (incremental value): Infeed per cut. The TNC will go to depth in one movement if:
 - The plunging depth and the milling depth are equal
 - The plunging depth is greater than the milling depth

The milling depth does not have to be a multiple of the plunging 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 main 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



Example NC blocks:

```

8 CYCL DEF 6.0 ROUGH-OUT
9 CYCL DEF 6.1 DIST+2 DEPTH-25
10 CYCL DEF 6.2 PLNGNG+3 F150 ALLOW+0.1
11 CYCL DEF 6.3 ANGLE+0 F350
    
```

CONTOUR MILLING (Cycle 16)

Application

Cycle 16 CONTOUR MILLING serves for finishing the contour pocket.



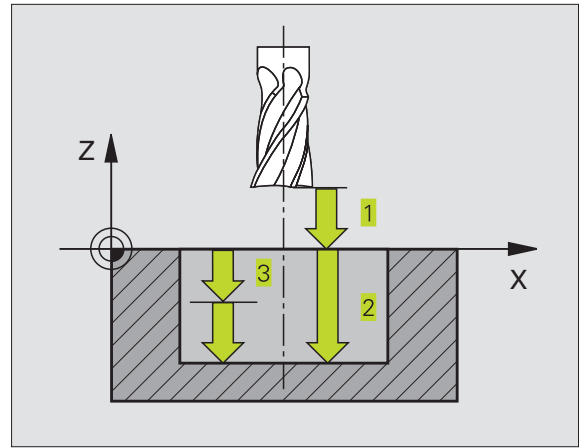
Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

The TNC finishes each subcontour separately, even at several infeed depths.



- ▶ Setup clearance **1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
 - ▶ Milling depth **2** (incremental value): Distance between workpiece surface and pocket floor
 - ▶ Plunging depth **3** (incremental value): Infeed per cut. The TNC will go to depth in one movement if:
 - The plunging depth and the milling depth are equal
 - The plunging depth is greater than the milling depth
- The milling depth does not have to be a multiple of the plunging depth.
- ▶ Feed rate for plunging: Traversing speed of the tool in mm/min during penetration
 - ▶ Clockwise rotation:
 - DR + : Climb milling with M3
 - DR - : Up-cut milling with M3
 - ▶ Feed rate: Feed rate for milling in mm/min



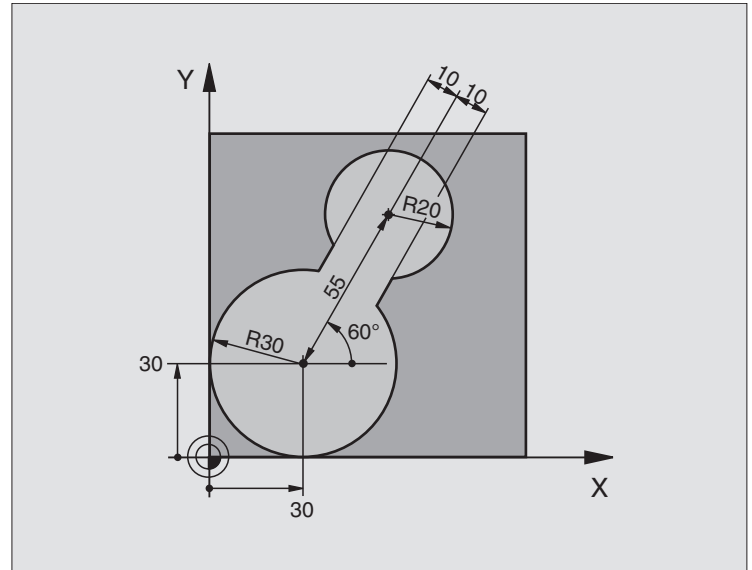
Example NC blocks:

```
12 CYCL DEF 16.0 CONTOUR MILLING
```

```
13 CYCL DEF 16.1 DIST+2 DEPTH-25
```

```
14 CYCL DEF 16.2 PLNGNG+5 F150 DR+ F500
```

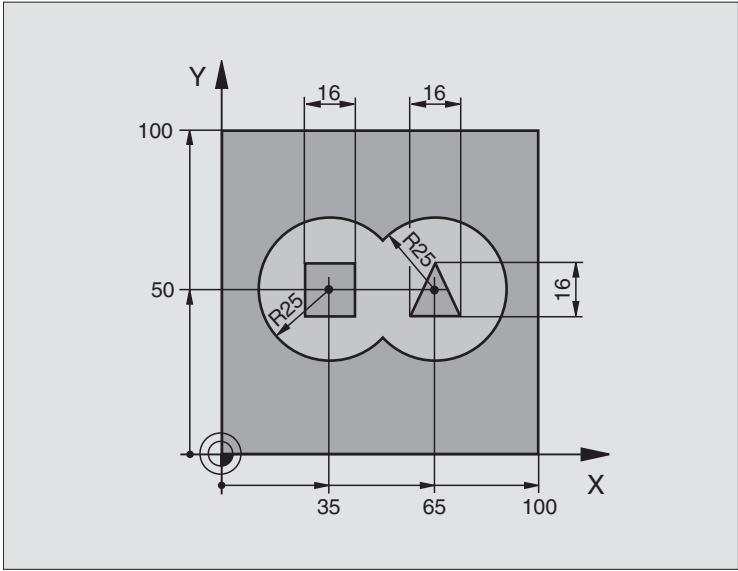
Example: Rough-out a pocket



0 BEGIN PGM C20 MM	
1 BLK FORM 0.1 Z X-10 Y-10 Z-40	Define the blank form
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+5	Tool definition
4 TOOL CALL 1 Z S4500	Call the tool
5 L Z+250 R0 FMAX	Retract the touch probe
6 CYCL DEF 14 .0 CONTOUR	Define contour subprogram
7 CYCL DEF 14 .1 CONTOUR LABEL 1	
8 CYCL DEF 6 .0 ROUGH-OUT	Cycle definition for rough-out
9 CYCL DEF 6 .1 SET UP 2 DEPTH -20	
10 CYCL DEF 6 .2 PLNGNG 5 F150 ALLOW +0	
11 CYCL DEF 6 .3 ANGLE +60 F250	
12 L X+30 Y+30 R0 FMAX M3	Pre-positioning in the working plane
13 L Z+2 R0 F1000 M99	Pre-position in the spindle axis, cycle call
14 L Z+250 R0 FMAX M2	Retract the tool, end of program

15	LBL 1	Contour subprogram
16	L X+0 Y+30 RR	(see second FK programming example on page 99)
17	FC DR- R30 CCX+30 CCY+30	
18	FL AN+60 PDX+30 PDY+30 D+10	
19	FSELECT 03	
20	FPOL X+30 Y+30	
21	FC DR- R20 CCPR+55 CCPA+60	
22	FSELECT 02	
23	FL AN-120 PDX+30 PDY+30 D+10	
24	FSELECT 03	
25	FC X+0 DR- R30 CCX+30 CCY+30	
26	FSELECT 02	
27	LBL 0	
28	END PGM C20 MM	

Example: Pilot drilling, roughing-out and finishing overlapping contours





0	BEGIN PGM C21 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+3	Define tool: drill
4	TOOL DEF 2 L-12,53 R+3	Define the tool for roughing/finishing
5	TOOL CALL 1 Z S4500	Call the drilling tool
6	L Z+250 R0 F MAX	Retract the touch probe
7	CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
8	CYCL DEF 14.1 CONTOUR LABEL 1 /2 /3 /4	
9	CYCL DEF 15.0 PILOT DRILLING	Cycle definition: PILOT DRILLING
10	CYCL DEF 15.1 SET UP 2 DEPTH -20	
11	CYCL DEF 15.2 PLNGNG 5 F200 ALLOW +1	
12	L X+50 Y+50 R0 F MAX M3	Pre-positioning in the working plane
13	L Z+2 R0 FMAX M99	Pre-positioning in the spindle axis, cycle call pilot drilling
14	L Z+250 R0 FMAX M6	Tool change/
15	TOOL CALL 2 Z S4000	Call the tool for roughing/finishing
16	CYCL DEF 6.0 ROUGH-OUT	Cycle definition for rough-out
17	CYCL DEF 6.1 SET UP 2 DEPTH -20	
18	CYCL DEF 6.2 PLNGNG 5 F150 ALLOW +1	
19	CYCL DEF 6.3 ANGLE +0 F250	
20	L Z+2 R0 F1000 M3	Pre-position in the tool axis
21	CYCL CALL	Cycle call for rough-out

22	CYCL DEF 16.0 CONTOUR MILLING	Cycle definition for finishing
23	CYCL DEF 16.1 SET UP 2 DEPTH -20	
24	CYCL DEF 16.2 PLNGNG 5 F100 DR+ F300	
25	L Z+2 R0 FMAX M99	Cycle call for finishing
26	L Z+250 R0 FMAX M2	Retract the tool, end of program
27	LBL 1	Contour subprogram 1: left pocket
28	CC X+35 Y+50	
29	L X+10 Y+50 RR	
30	C X+10 DR-	
31	LBL 0	
32	LBL 2	Contour subprogram 2: right pocket
33	CC X+65 Y+50	
34	L X+90 Y+50 RR	
35	C X+90 DR-	
36	LBL 0	
37	LBL 3	Contour subprogram 3: square left island
38	L X+27 Y+50 RL	
39	L Y+58	
40	L X+43	
41	L Y+42	
42	L X+27	
43	LBL 0	
44	LBL 4	Contour subprogram 4: triangular right island
45	L X+65 Y+42 RL	
46	L X+57	
47	L X+65 Y+58	
48	L X+73 Y+42	
49	LBL 0	
50	END PGM C21 MM	

8.7 Cycles for multipass milling

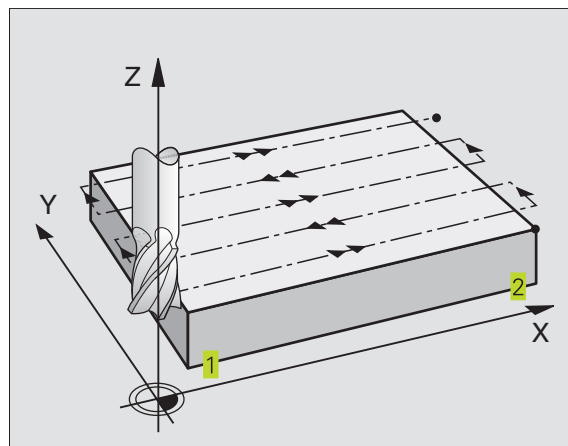
The TNC offers two cycles for machining surfaces with the following characteristics:

- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

Cycle	Soft Key
230 MULTIPASS MILLING For flat rectangular surfaces	
231 RULED SURFACE For oblique, inclined or twisted surfaces	

MULTIPASS MILLING (Cycle 230)

- 1 From the current position, the TNC positions the tool in rapid traverse in the working plane to the starting position **1**. During this movement, the TNC also offsets the tool by its radius to the left and upward.
- 2 The tool then moves in FMAX 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 subsequently advances to the stopping point **2** at the feed rate for milling. The stopping point is calculated from the programmed starting point, the programmed 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 X direction
- 6 Multipass milling is repeated until the programmed surface has been completed.
- 7 At the end of the cycle, the tool is retracted in FMAX to set-up clearance.





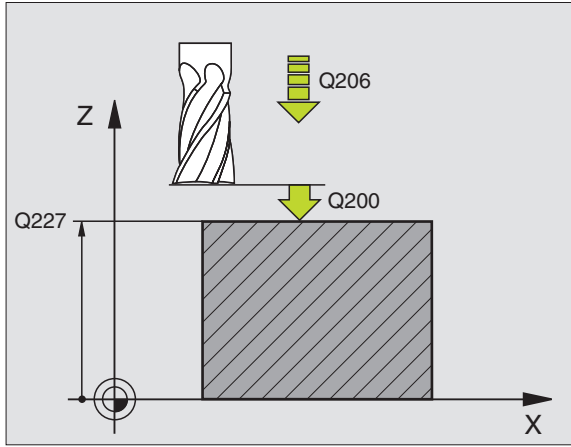
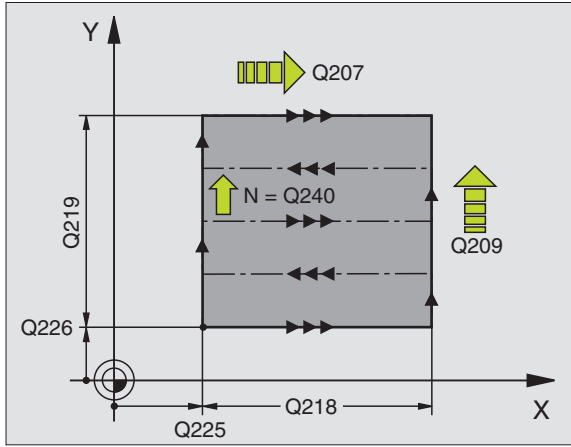
Before programming, note the following:

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): Min. point coordinate of the surface to be multipass-milled in the main axis of the working plane
- ▶ Starting point in 2nd axis Q226 (absolute value): Min. point coordinate of the surface to be multipass-milled in the secondary 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 main axis of the working plane, referenced to the starting point in 1st axis
- ▶ Second side length Q219 (incremental value): Length of the surface to be multipass-milled in the secondary axis of the working plane, referenced to the starting point in 2nd axis
- ▶ Number of cuts Q240: Number of passes to be made over the width
- ▶ Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving from 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 blocks:

71 CYCL DEF 230 MULTIPASS MILLNG		
Q225=+10	; STARTNG PNT 1ST AXIS	
Q226=+12	; STARTNG PNT 2ND AXIS	
Q227=+2.5	; STARTING PNT 3RD AXIS	
Q218=150	; 1ST SIDE LENGTH	
Q219=75	; 2ND SIDE LENGTH	
Q240=25	; NUMBER OF CUTS	
Q206=150	; FEED RATE FOR PLUNGING	
Q207=500	; FEED RATE FOR MILLING	
Q209=200	; STEPOVER FEED RATE	
Q200=2	; SET-UP CLEARANCE	

RULED SURFACE (Cycle 231)

- 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 in rapid traverse FMAX by the tool diameter in the positive tool axis direction, and then back to starting point **1**.
- 4 At the starting position **1** the TNC moves the tool back to the 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 stopping point using point **2** and an offset 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

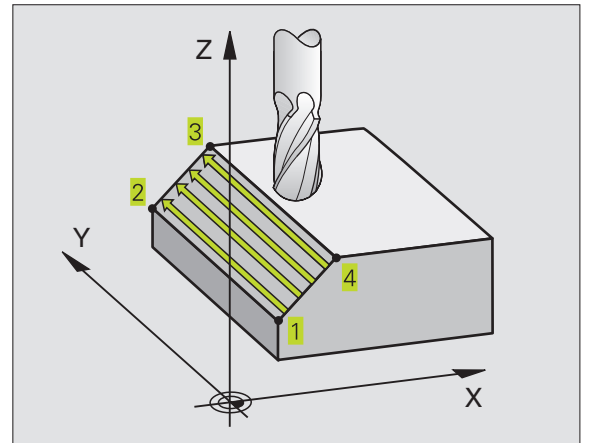
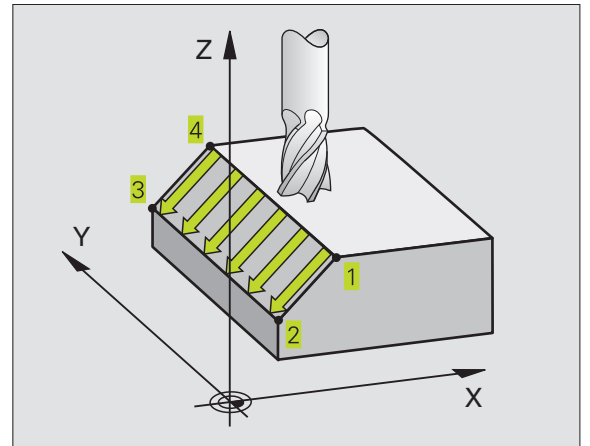
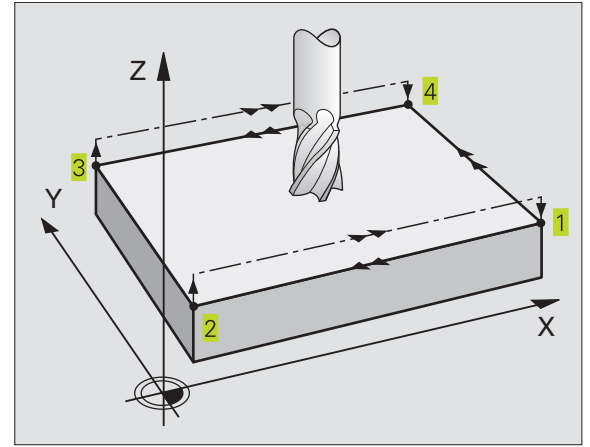
You can freely choose the starting point and thus the milling direction since the TNC always performs the individual cuts from point **1** to point **2** and the process sequence is executed from points **1** / **2** to point **3** / **4**. You can position point **1** in any corner of the surface to 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 (tool axis coordinate of point **1** greater than tool axis coordinate of point **2**) for slightly inclined surfaces.
- a drawing cut (tool axis coordinate of point **1** less than tool 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. See figure at center right.

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 steeper inclination. See figure at lower right.





Before programming, note the following:

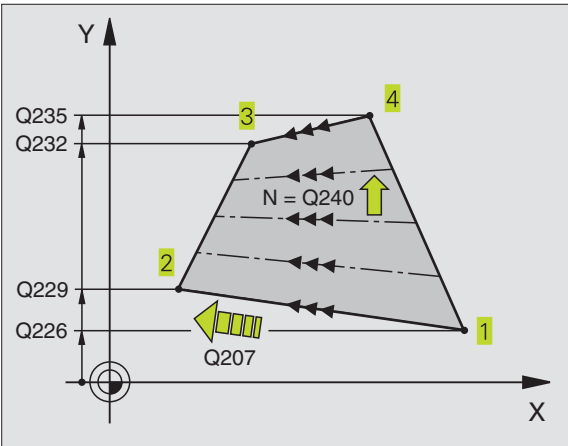
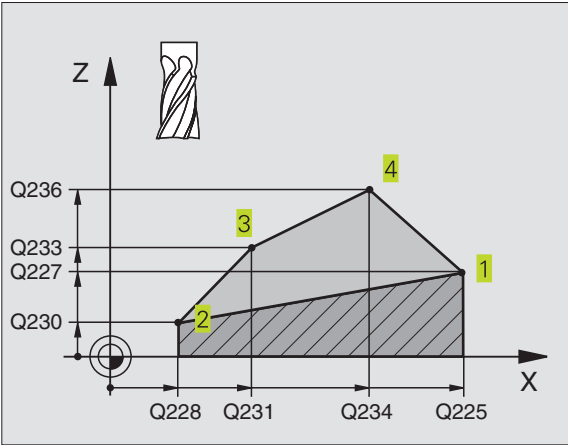
From the current position, the TNC positions the tool in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

The TNC moves the tool with radius compensation R0 to the programmed positions.

If required, use a center-cut end mill (ISO 1641).



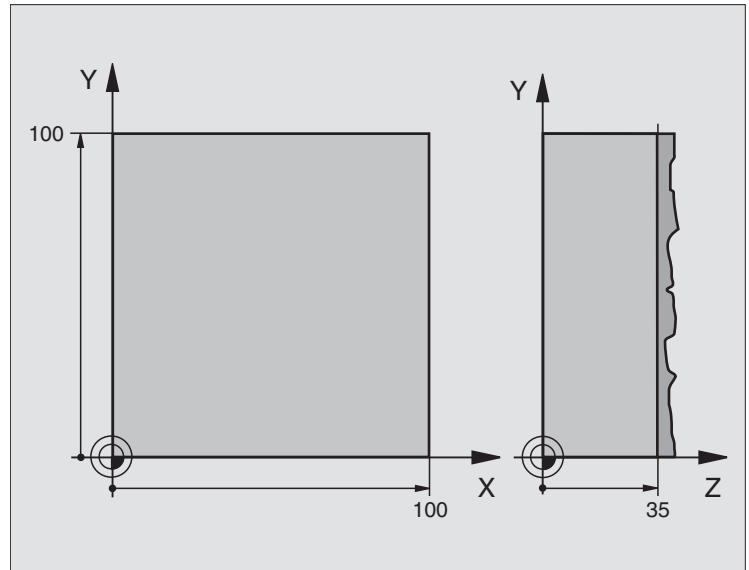
- ▶ Starting point in 1st axis Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the main axis of the working plane
- ▶ Starting point in 2nd axis Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the secondary axis of the 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 main 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 secondary 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 main axis of the working plane
- ▶ 3rd point in 2nd axis Q232 (absolute value): Coordinate of point **3** in the subordinate 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 main axis of the working plane
- ▶ 4th point in 2nd axis Q235 (absolute value): Coordinate of point **4** in the subordinate 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**, and between points **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:

72 CYCL DEF 231 RULED SURFACE	
Q225=+0	; STARTNG PNT 1ST AXIS
Q226=+5	; STARTNG PNT 2ND AXIS
Q227=-2	; STARTING PNT 3RD AXIS
Q228=+100	; 2ND POINT 1ST AXIS
Q229=+15	; 2ND POINT 2ND AXIS
Q230=+5	; 2ND PNT IN 3RD AXIS
Q231=+15	; 3RD PNT IN 1ST AXIS
Q232=+125	; 3RD PNT IN 2ND AXIS
Q233=+25	; 3RD PNT IN 3RD AXIS
Q234=+85	; 4TH PNT IN 1ST AXIS
Q235=+95	; 4TH PNT IN 2ND AXIS
Q236=+35	; 4TH PNT IN 3RD AXIS
Q240=40	; NUMBER OF CUTS
Q207=500	; FEED RATE FOR MILLING






Example: Multipass milling



0	BEGIN PGM C230 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z+0	Define the blank form
2	BLK FORM 0.2 X+100 Y+100 Z+40	
3	TOOL DEF 1 L+0 R+5	Tool definition
4	TOOL CALL 1 Z S3500	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	CYCL DEF 230 MULTIPASS MILLNG	Cycle definition: MULTIPASS MILLING
	Q225=+0 ;STARTNG PNT 1ST AXIS	
	Q226=+0 ;STARTNG PNT 2ND AXIS	
	Q227=+35 ;STARTNG PNT 3RD AXIS	
	Q218=100 ;FIRST SIDE LENGTH	
	Q219=100 ;SECOND SIDE LENGTH	
	Q240=25 ;NUMBER OF CUTS	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q207=400 ;FEED RATE FOR MILLNG	
	Q209=150 ;STEPOVER FEED RATE	
	Q200=2 ;SET-UP CLEARANCE	
7	L X-25 Y+0 R0 F MAX M3	Pre-position near the starting point
8	CYCL CALL	Cycle call
9	L Z+250 R0 FMAX M2	Retract the tool, end of program
10	END PGM C230 MM	

8.8 Coordinate Transformation Cycles

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
7 DATUM SHIFT For shifting contours directly within the program or from a datum table	
8 MIRROR IMAGE For mirroring contours	
10 ROTATION For rotating contours in the working plane	
11 SCALING FACTOR For increasing or reducing the size of contours	
26 AXIS-SPECIFIC SCALING For increasing or reducing the size of contours with axis-specific scaling factors	

Effect of coordinate transformations

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 END PGM block (depending on machine parameter 7300)
- Select a new program

DATUM SHIFT (Cycle 7)

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.



► 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.



► REF: Press the REF soft key to reference the programmed datum to the machine datum. In this case the TNC indicates the first cycle block with REF

Example NC blocks:

```
73 CYCL DEF 7.0 DATUM SHIFT
```

```
74 CYCL DEF 7.1 X+10
```

```
75 CYCL DEF 7.2 Y+10
```

```
76 CYCL DEF 7.3 Z-5
```

Cancellation

A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Status Displays

- The actual position values are referenced to the active (shifted) datum.
- The actual position values shown in the additional status display are referenced to the manually set datum.

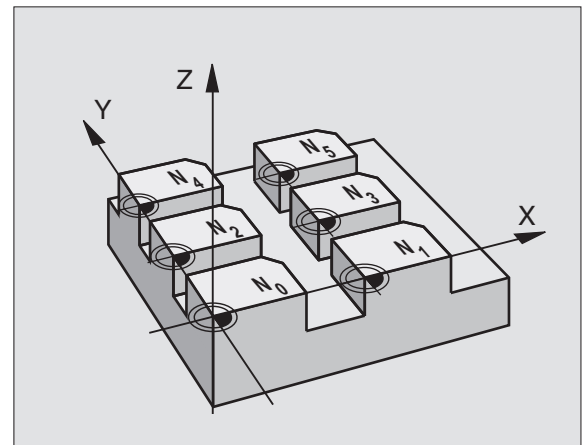
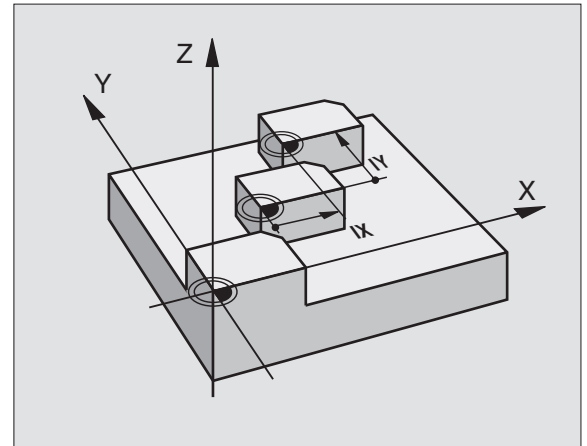
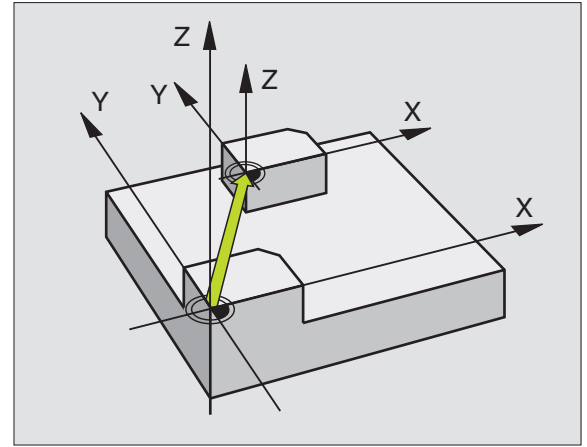
DATUM SHIFT with datum tables (Cycle 7)



Datums from a datum table can be referenced either to the current datum or to the machine datum (depending on machine parameter 7475).

The datum points from datum tables are only effective with absolute coordinate values.

Remember that the datum numbers shift whenever you insert lines in an existing datum table (edit part program if necessary).



Application

Datum tables are applied 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.



► Define Cycle 7



► Press the soft key for entering the datum number. Enter the datum number and confirm it with the END key.

Example NC blocks:

77 CYCL DEF 7.0 DATUM SHIFT

78 CYCL DEF 7.1 #12

Cancellation

- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
- Execute a datum shift to the coordinates X=0; Y=0 etc. directly via cycle definition.

Selecting a datum table in the part program

With the SEL TABLE function you select the table from which the TNC takes the datums:



- To select the functions for program call, press the PGM CALL key.
- Press the DATUM TABLE soft key.
- Enter the name of the datum table, then confirm with the END key.

Editing a datum table















Select the datum table in the PROGRAMMING AND EDITING mode of operation.



- To call the file manager, press the PGM MGT key — see section 4.2 “File Management” for more information.
- Move the highlight to any datum table. Confirm with the ENT key.
- File editing: See the “Editing functions” table.

To leave a datum table

- Call the file manager and select a file of a different type, e.g. a part program.

Editing functions	Soft key
Select the axis.	 / 
Scroll downwards line by line	
Scroll upwards line by line	
Go to the previous page	
Go to the previous page	
Move one word to the right	
Move one word to the left	
Confirm current position, e.g. for the Z-axis	
Enter the number of lines to be inserted	
Delete and temporarily store a line	
Insert a new line or the line last deleted	
Go to the beginning of the table	
Go to the end of the table	

MIRROR IMAGE (Cycle 8)

The TNC can machine the mirror image of a contour in the working plane. See figure at upper right.

Effect

The MIRROR IMAGE cycle becomes effective as soon as it is defined in the program. Cycle 10 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 — see figure at lower right.
- If the datum lies outside the contour to be mirrored, the element also “jumps” to another location — see figure at lower right.



► Mirror image axis ?: Enter the axis to be mirrored. You can mirror all axes, including rotary axes, except for the spindle axis and its auxiliary axes.

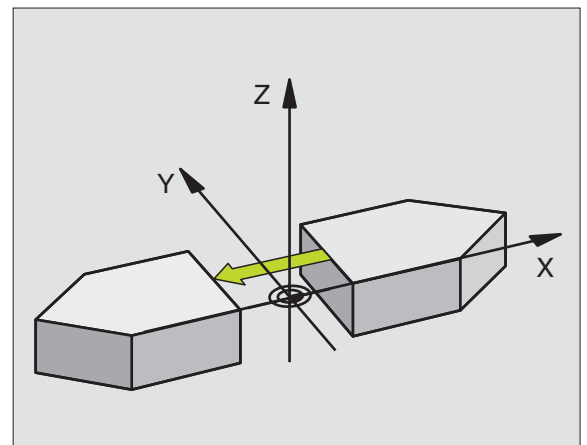
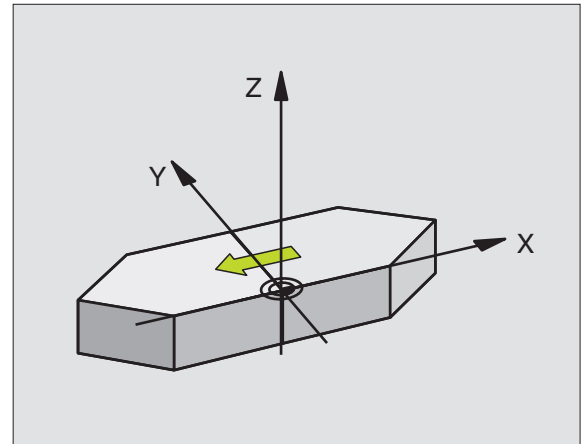
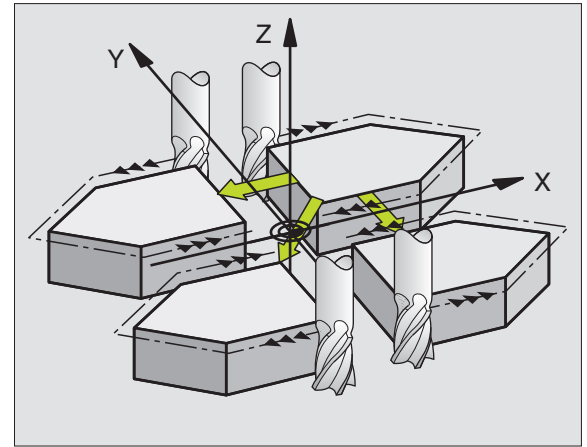
Example NC blocks:

```
79 CYCL DEF 8.0 MIRROR IMAGE
```

```
80 CYCL DEF 8.1 X Y
```

Cancellation

Program the MIRROR IMAGE cycle once again with NO ENT.



ROTATION (Cycle 10)

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. This cycle 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 Spindle axis



Before programming, note the following:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

After defining Cycle 10, 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 $+360^\circ$ (absolute or incremental).

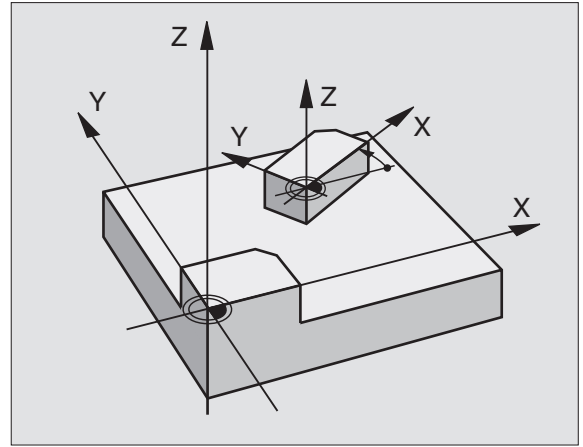
Example NC blocks:

```
81 CYCL DEF 10.0 ROTATION
```

```
82 CYCL DEF 10.1 ROT+12.357
```

Cancellation

Program the ROTATION cycle once again with a rotation angle of 0° .



SCALING FACTOR (Cycle 11)

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 can be applied

- in the working plane, or on all three coordinate axes at the same time (depending on machine parameter 7410)
- to the dimensions in cycles
- to 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 SCL. The TNC multiplies the coordinates and radii by the SCL factor (as described under "Activation" above)

Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction SCL less than 1 (down to 0.000 001)

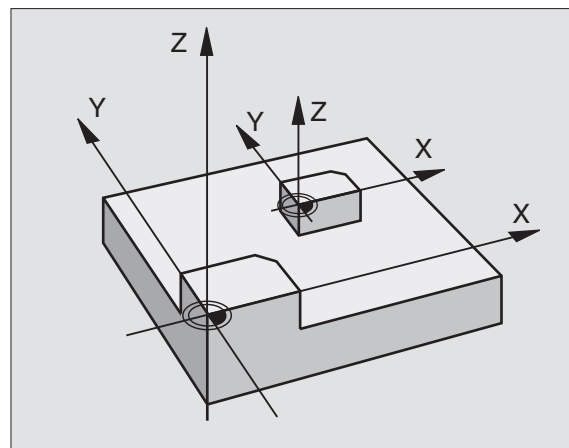
Example NC blocks:

```
83 CYCL DEF 11.0 SCALING
```

```
84 CYCL DEF 11.1 SCL0.99537
```

Cancellation

Program the SCALING FACTOR cycle once again with a scaling factor of 1.



AXIS-SPECIFIC SCALING (Cycle 26)



Before programming, note the following:

You cannot use different factors to expand or compress coordinate axes with positions for circular arcs.

Otherwise you can program each coordinate axis with its own axis-specific scaling factor.

In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING FACTOR) with reference to the active datum.

Effect

The SCALING FACTOR becomes effective as soon as it is defined in the program. It also works in the Positioning with MDI mode. The TNC shows the active scaling factor in the additional status display.

The SCALING FACTOR is also effective in the Positioning with Manual Data Input mode. The active scaling factor is shown in the additional status display.



- ▶ Axis and scaling factor: Enter the coordinate axis/axes as well as the factor(s) involved in enlarging or reducing. Enter a positive value up to 99.999 999.
- ▶ Center coordinates: Enter the center of the axis-specific enlargement or reduction.

The coordinate axes are selected with soft keys.

Cancellation

Program the AXIS-SPECIFIC SCALING cycle once again with a scaling factor of 1 for the same axis.

Example

Axis-specific scaling factors in the working plane

Preset: Quadrangle (see figure at lower right)

Corner 1: X = 20.0 mm Y = 2.5 mm

Corner 2: X = 32.5 mm Y = 15.0 mm

Corner 3: X = 20.0 mm Y = 27.5 mm

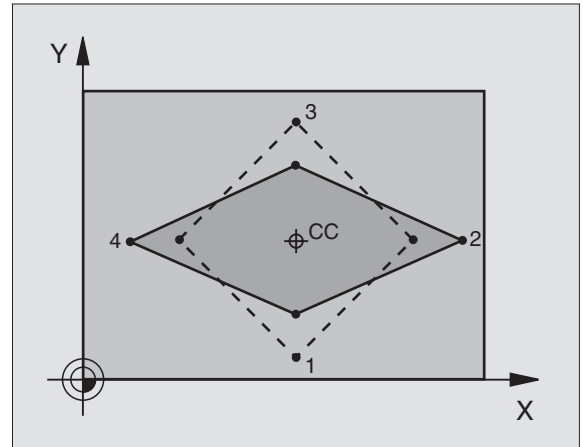
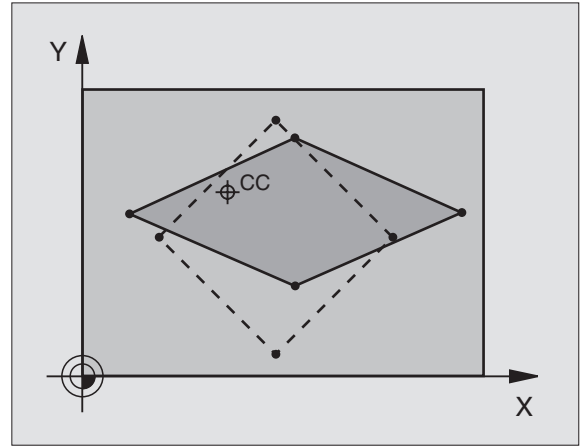
Corner 4: X = 7.5 mm Y = 15.0 mm

- Enlarge X axis by factor 1.4
- Reduce Y axis by factor 0.6
- Center at CCX = 15 mm CCY = 20 mm

Resulting NC blocks

CYCL DEF 26.0 axis-spec. scaling

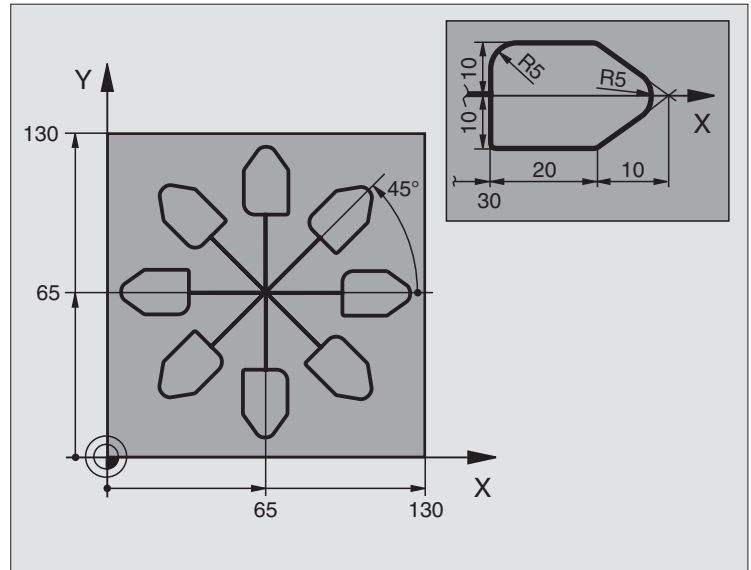
CYCL DEF 26.1 X1.4 Y0.6 CCX+15 CCY+20



Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- Program the machining operation in subprogram 1 (see section 9 "Programming: Subprograms and Program Section Repeats")



0	BEGIN PGM KOU MR MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
2	BLK FORM 0.2 X+130 Y+130 Z+0	
3	TOOL DEF 1 L+0 R+1	Tool definition
4	TOOL CALL 1 Z S4500	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
7	CYCL DEF 7.1 X+65	
8	CYCL DEF 7.2 Y+65	
9	CALL LBL 1	Call milling operation
10	LBL 10	Set label for program section repeat
11	CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
12	CYCL DEF 10.1 IROT+45	
13	CALL LBL 1	Call milling operation
14	CALL LBL 10 REP 6	Return jump to LBL 10; execute the milling operation six times
15	CYCL DEF 10.0 ROTATION	Reset the rotation
16	CYCL DEF 10.1 ROT+0	
17	CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
18	CYCL DEF 7.1 X+0	
19	CYCL DEF 7.2 Y+0	
20	L Z+250 R0 FMAX M2	Retract the tool, end of program

21	LBL 1	Subprogram 1:
22	L X+0 Y+0 R0 F MAX	Define milling operation
23	L Z+2 R0 FMAX M3	
24	L Z-5 R0 F200	
25	L X+30 RL	
26	L IY+10	
27	RND R5	
28	L IX+20	
29	L IX+10 IY-10	
30	RND R5	
31	L IX-10 IY-10	
32	L IX-20	
33	L IY+10	
34	L X+0 Y+0 R0 F500	
35	L Z+20 R0 F MAX	
36	LBL 0	
37	END PGM KOUMR MM	

8.9 Special Cycles

DWELL TIME (Cycle 9)

This cycle 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 30 000 seconds (approx. 8.3 hours) in increments of 0.001 seconds

Resulting NC blocks

89 CYCL DEF 9.0 DWELL TIME

90 CYCL DEF 9.1 DWELL 1.5

PROGRAM CALL (Cycle 12)

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.



- Program name: Name of the program to be called

The program is called with

- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (modally)

Example: Program call

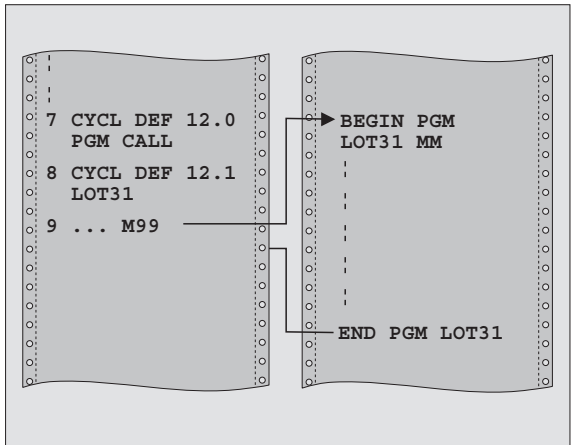
A callable program 50 is to be called into a program via a cycle call.

Resulting NC blocks

55 CYCL DEF 12.0 PGM CALL

56 CYCL DEF 12.1 PGM 50.H

57 L X+20 Y+50 FMAX M99



Definition:

Program 50 is a cycle"

Call program 50

ORIENTED SPINDLE STOP (Cycle 13)



The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle 13.

The control can address the machine tool spindle as a 6th axis 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.

If you program M19 without having defined Cycle 13, 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.

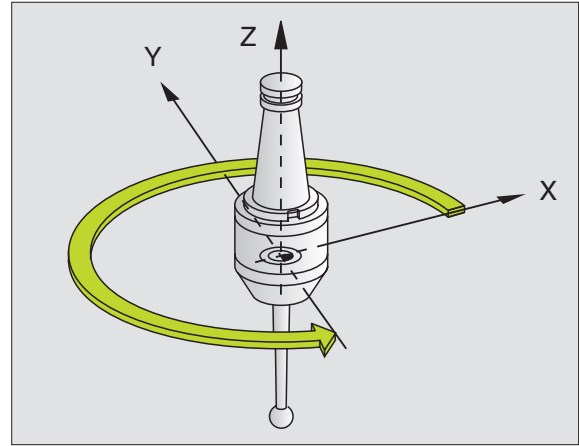
Input range 0 to 360°

Input resolution 0.001°

Resulting NC blocks

93 CYCL DEF 13.0 ORIENTATION

94 CYCL DEF 13.1 ANGLE 180





9

Programming:

**Subprograms and
Program Section Repeats**

9.1 Marking 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

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

A label is identified by a number between 1 and 254. Each label can be set only once with LABEL SET in a program.

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

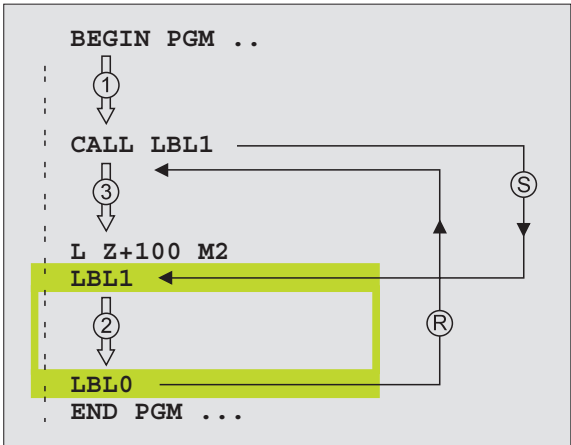
9.2 Subprograms

Operating sequence

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

Operating constraints

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

Calling a subprogram



- ▶ To call a subprogram, press the LBL CALL key.
- ▶ Label number: Enter the label number of the subprogram to be called, then confirm with the END key.



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

9.3 Program section repeats

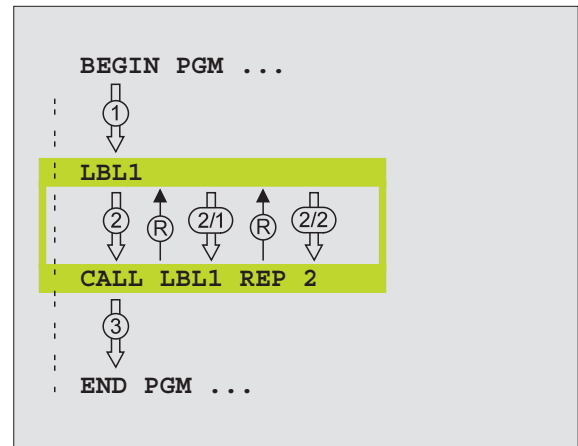
The beginning of a program section repeat is marked by the label LBL. The end of a program section repeat is identified by CALL LBL REP.

Operating sequence

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

Operating constraints

- You can repeat a program section up to 65 534 times in succession.
- In the additional status display the TNC shows how many repetitions remain (see section "1.4 Status Displays").
- The total number of times the program section is executed is always one more than the programmed number of repeats.



Programming a program section repeat



- ▶ To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat.
- ▶ Enter the program section.

Calling a program section repeat



- ▶ Press the LBL CALL key and enter the label number of the program section you want to repeat as well as the number of repeats (with Repeat REP).

9.4 Program as Subprogram

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

Operating constraints

- No labels are needed to call any program as a subprogram.
- The called program may not contain the miscellaneous functions M2 or M30.
- The called program must not contain a program call into the calling program.

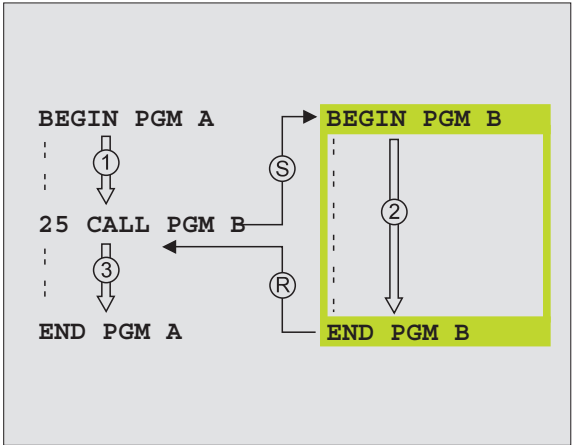
Calling any program as a subprogram



- ▶ To select the functions for program call, press the PGM CALL key.
- ▶ Press the PROGRAM soft key.
- ▶ Enter the name of the program to be called. Using soft keys you can also indicate the type of program you wish to call and where the program is stored (see table at right).



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



Function	Soft key
Call externally stored programs	EXT
Call conversational programs	.H
Call ISO programs	.I
Convert a CALL PGM EXT block to CALL PGM INT (call internally stored program)	INT
Call program type defined in the "Program input" MOD function.	DEFAULT

9.5 Nesting

You can nest subprograms and program section repeats in the following ways:

- 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
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example NC blocks

0	BEGIN PGM UPGMS MM	
...		
17	CALL LBL 1	Call the subprogram marked with LBL1
...		
35	L Z+100 R0 FMAX M2	Last program block of the main program (with M2)
36	LBL 1	Beginning of subprogram 1
...		
39	CALL LBL 2	Call the subprogram marked with LBL2
...		
45	LBL 0	End of subprogram 1
46	LBL 2	Beginning of subprogram 2
...		
62	LBL 0	End of subprogram 2
63	END PGM UPGMS MM	

Program execution

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

Repeating program section repeats

Example NC blocks

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

Program execution

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

Repeating a subprogram

Example NC blocks

0	BEGIN PGM UPGREP MM	
...		
10	LBL 1	Beginning of the program section repeat
11	CALL LBL 2	Subprogram call
12	CALL LBL 1 REP 2	The program section between this block and LBL1
...		(block 10) is repeated twice
19	L Z+100 R0 FMAX M2	Last program block of the main program with M2
20	LBL 2	Beginning of subprogram
...		
28	LBL 0	End of subprogram
29	END PGM UPGREP MM	

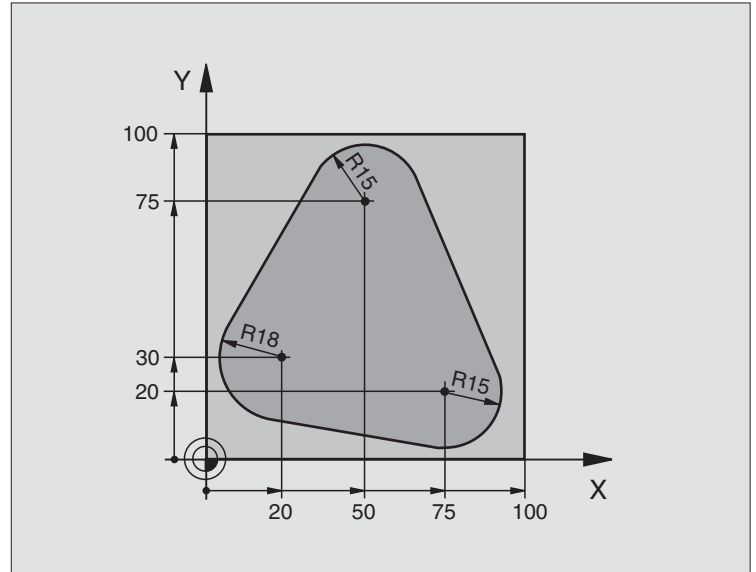
Program execution

- 1st step: Main program UPGREP is executed up to block 11.
- 2nd step: Subprogram 2 is called and executed.
- 3rd step: Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.
- 4th step: Main program UPGREP is executed from block 13 to block 19. 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

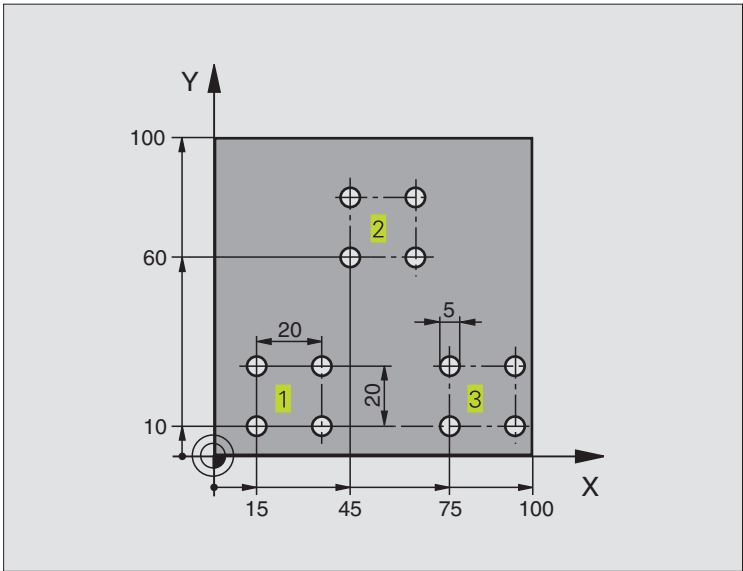


0	BEGIN PGM PGMWDH MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-40	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	TOOL DEF 1 L+0 R+10	Define the tool
4	TOOL CALL 1 Z S500	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	L X-20 Y+30 R0 F MAX	Pre-position in the working plane
7	L Z+0 R0 FMAX M3	Pre-position to the workpiece surface
8	LBL 1	Set label for program section repeat
9	L IZ-4 R0 F MAX	Infeed depth in incremental values (in the open)
10	APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach contour
11	FC DR- R18 CLSD+ CCX+20 CCY+30	Contour
12	FLT	
13	FCT DR- R15 CCX+50 CCY+75	
14	FLT	
15	FCT DR- R15 CCX+75 CCY+20	
16	FLT	
17	FCT DR- R18 CLSD- CCX+20 CCY+30	
18	DEP CT CCA90 R+5 F1000	Depart contour
19	L X-20 Y+0 R0 F MAX	Retract tool
20	CALL LBL 1 REP 4	Return jump to LBL 1; section is repeated a total of 4 times
21	L Z+250 R0 FMAX M2	Retract in the tool axis, end of program
22	END PGM PGMWDH MM	

Example: Groups of holes

Program sequence

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



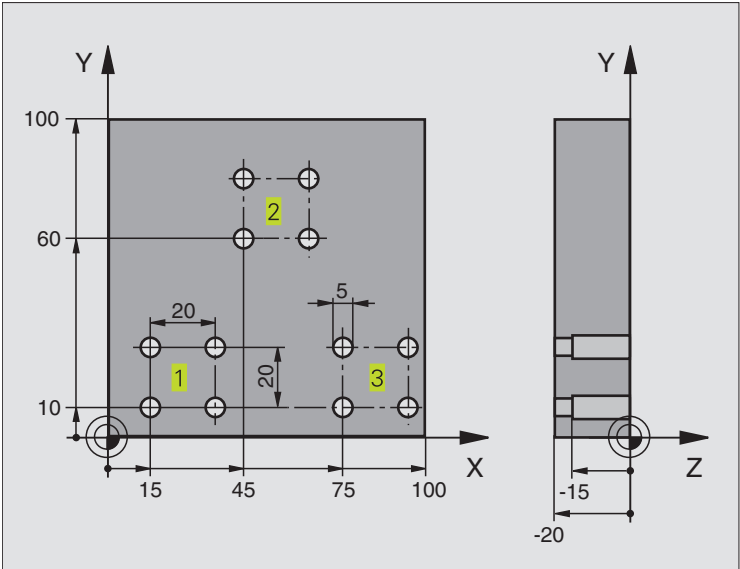
0	BEGIN PGM UP1 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	T00L DEF 1 L+0 R+2.5	Define the tool
4	T00L CALL 1 Z S5000	Call the tool
5	L Z+250 R0 FMAX	Retract the touch probe
6	CYCL DEF 200 DRILLING	
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-10 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q202=5 ;PLUNGING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	
7	L X+15 Y+10 R0 F MAX M3	Move to starting point for group 1
8	CALL LBL 1	Call the subprogram for the group
9	L X+45 Y+60 R0 F MAX	Move to starting point for group 2
10	CALL LBL 1	Call the subprogram for the group
11	L X+75 Y+10 R0 F MAX	Move to starting point for group 3
12	CALL LBL 1	Call the subprogram for the group
13	L Z+250 R0 FMAX M2	End of main program

14	LBL 1	Beginning of subprogram 1: Group of holes
15	CYCL CALL	1st hole
16	L IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
17	L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
18	L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
19	LBL 0	End of subprogram 1
20	END PGM UP1 MM	

Example: Groups of holes with several tools

Program sequence

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



0	BEGIN PGM UP2 MM	
1	BLK FORM 0.1 Z X+0 Y+0 Z-20	
2	BLK FORM 0.2 X+100 Y+100 Z+0	
3	T00L DEF 1 L+0 R+4	Define tool: center drill
4	T00L DEF 2 L+0 R+3	Define tool: drill
5	T00L DEF 3 L+0 R+3.5	Define tool: reamer
6	T00L CALL 1 Z S5000	Call tool: center drill
7	L Z+250 R0 F MAX	Retract the touch probe

8	CYCL DEF 200 DRILLING	Cycle definition: Centering
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-3 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q202=3 ;PLUNGING DEPTH	
	Q210=0 ;DWELL TIME AT TOP	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	
9	CALL LBL 1	Call subprogram 1 for the entire hole pattern
10	L Z+250 R0 FMAX M6	Tool change
11	TOOL CALL 2 Z S4000	Call the drilling tool
12	FN 0: Q201 = -25	New depth for drilling
13	FN 0: Q202 = +5	New plunging depth for drilling
14	CALL LBL 1	Call subprogram 1 for the entire hole pattern
15	L Z+250 R0 FMAX M6	Tool change
16	TOOL CALL 3 Z S500	Tool call: reamer
17	CYCL DEF 201 REAMING	Cycle definition: REAMING
	Q200=2 ;SET-UP CLEARANCE	
	Q201=-15 ;DEPTH	
	Q206=250 ;FEED RATE FOR PLUNGING	
	Q211=0.5 ;DWELL TIME AT DEPTH	
	Q208=400 ;RETRACTION FEED RATE	
	Q203=+0 ;SURFACE COORDINATE	
	Q204=10 ;2ND SET-UP CLEARANCE	
18	CALL LBL 1	Call subprogram 1 for the entire hole pattern
19	L Z+250 R0 FMAX M2	End of main program
20	LBL 1	Beginning of subprogram 1: Entire hole pattern
21	L X+15 Y+10 R0 F MAX M3	Move to starting point for group 1
22	CALL LBL 2	Call subprogram 2 for the group
23	L X+45 Y+60 R0 F MAX	Move to starting point for group 2
24	CALL LBL 2	Call subprogram 2 for the group
25	L X+75 Y+10 R0 F MAX	Move to starting point for group 3
26	CALL LBL 2	Call subprogram 2 for the group
27	LBL 0	End of subprogram 1
28	LBL 2	Beginning of subprogram 2: Group of holes
29	CYCL CALL	1st hole with active fixed cycle
30	L IX+20 R0 FMAX M99	Move to 2nd hole, call cycle
31	L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
32	L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
33	LBL 0	End of subprogram 2
34	END PGM UP2 MM	



10

Programming:

Q Parameters

10.1 Principle and Overview

You can program an entire family of parts on the TNC with 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
- RPM
- Cycle data

Q parameters also enable you to program contours that are defined through 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, global for all programs in the TNC memory. When you call OEM cycles, these parameters are only effective locally (depending on MP7251).	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, globally effective for all programs and OEM cycles that are stored in the TNC memory	Q200 to Q299

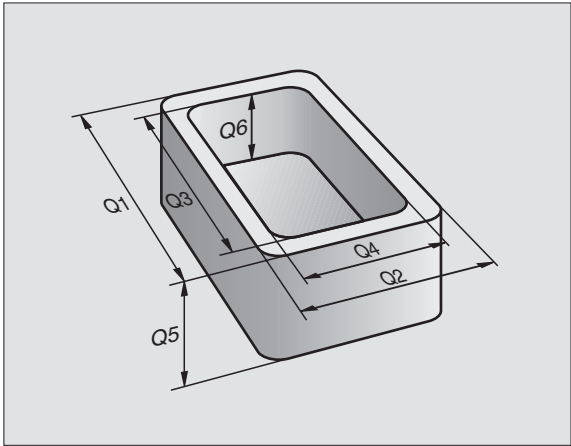
Programming notes

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

Q parameters can be assigned numerical values between -99 999.9999 and +99 999.9999.



Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius. For further information, see section "10.9 Preassigned Q Parameters."



Calling Q parameter functions

When you are writing a part program, press the “Q” key (below the +/- key in the keypad for numerical input and axis selection). 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
If/then conditions, jumps	JUMP
Other functions	DIVERSE FUNCTION
Entering Formulas Directly	FORMULA

10.2 Part Families — Q Parameters in Place of Numerical Values

The Q parameter function FN0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

15 FN0: Q10 = 25	ASSIGN:
...	Q10 contains the value 25
25 L X +Q10	Means L X +25

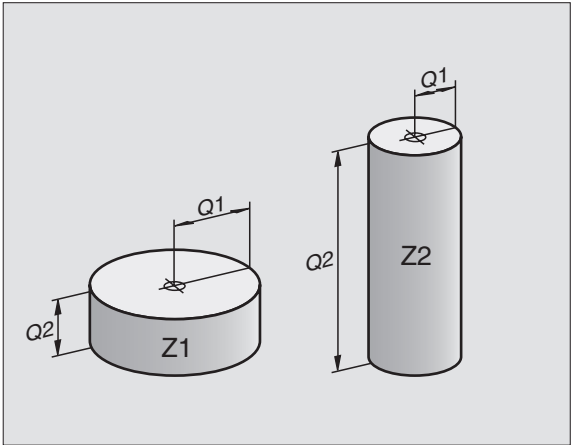
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example

Cylinder with Q parameters

Cylinder radius	R = Q1
Cylinder height	H = Q2
Cylinder Z1	Q1 = +30 Q2 = +10
Cylinder Z2	Q1 = +10 Q2 = +50



10.3 Describing Contours Through Mathematical Functions

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

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

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

“Values” in the above table are entered to the right of the “=” character and can be any of 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.

Example: Programming fundamental operations



To select Q parameter functions, press the Q key.



To select the mathematical functions: Press the BASIC ARITHMETIC



To select the Q parameter function ASSIGN, press the FN0 X = Y soft key.

Parameter number for result?

5



Enter a parameter number, for example 5.

1st value or parameter ?

10



Assign a value to Q5, for example 10.



To select Q parameter functions, press the Q key.



To select the mathematical functions: Press the BASIC ARITHMETIC



To select the Q parameter function MULTIPLICATION, press the FN3 X * Y soft key.

Parameter number for result?

12



Enter a Q parameter number, for example 12.

1st value or parameter ?

Q5



Enter Q5 for the first value.

Multiplier?

7



Enter 7 for the second value.

The TNC displays the following program blocks:

```
16 FN0: Q5 = +10
17 FN3: Q12 = +Q5 * +7
```

10.4 Trigonometric Functions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. For a right triangle, the trigonometric functions of the angle α are defined by the following equations:

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

Example:
 $a = 10 \text{ mm}$
 $b = 10 \text{ mm}$
 $\alpha = \arctan (a / b) = \arctan 1 = 45^\circ$

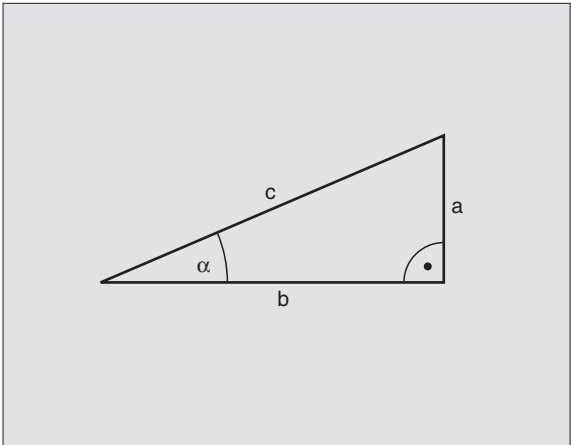
Furthermore:

$a^2 + b^2 = c^2$ (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 at right.

Programming: See „Example: Programming fundamental operations“.



Function	Soft key
FN6: SINE Example: FN6: Q20 = SIN-Q5 Calculate the sine of an angle in degrees (°) and assign it to a parameter.	FN6 SIN(X)
FN7: COSINE Example: FN7: Q21 = COS-Q5 Calculate the cosine of an angle in degrees (°) and assign it to a parameter.	FN7 COS(X)
FN8: ROOT-SUM OF SQUARES Example: FN8: Q10 = +5 LEN +4 Calculate and assign length from two values	FN8 X LEN Y
FN13: ANGLE Example: FN13: Q20 = +10 ANG-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.	FN13 X ANG Y

10.5 If-Then Decisions with Q Parameters

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 section 9 “Subprograms and Program Section Repeats”). If it is not fulfilled, the TNC continues with the next block.

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

Unconditional jumps

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

FN9: IF+10 EQU+10 GOTO LBL1

Programming If-Then decisions

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

Function	Soft key
FN9: IF EQUAL, JUMP Example: FN9: IF +Q1 EQU +Q3 GOTO LBL 5 If the two values or parameters are equal, jump to the given label.	<div>FN9 IF X EQ V GOTO</div>
FN10: IF NOT EQUAL, JUMP Example: FN10: IF +10 NE -Q5 GOTO LBL 10 If the two values or parameters are not equal, jump to the given label.	<div>FN10 IF X NE V GOTO</div>
FN11: IF GREATER THAN, JUMP Example: FN11: IF+Q1 GT+10 GOTO LBL 5 If the first parameter or value is greater than the second value or parameter, jump to the given label.	<div>FN11 IF X GT V GOTO</div>
FN12: IF LESSTHAN, JUMP Example: FN12: IF+Q5 LT+0 GOTO LBL 1 If the first value or parameter is less than the second value or parameter, jump to the given label.	<div>FN12 IF X LT V GOTO</div>

Abbreviations used:	
IF	If
EQU	Equals
NE	Not equal
GT	Greater than
LT	Less than
GOTO	Go to

10.6 Checking and Changing Q Parameters

During a program run or test run, you can check or change Q parameters if necessary.

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



- ▶ To call the Q parameter table, press the Q key.
- ▶ Using the arrow keys you can select a Q-parameter on the current screen page. You can go to the next or the previous screen page using the PAGE soft keys.
- ▶ If you wish to change the value of a parameter, enter a new value, confirm it with the ENT key and conclude your entry with the END key.

To leave the value unchanged, terminate the dialog with the END key.

Test run									
Q0	=	+0							
Q1	=	+0.5							
Q2	=	+32							
Q3	=	+16							
Q4	=	+24							
Q5	=	+10							
Q6	=	+6							
Q7	=	+12							
Q8	=	+6							
Q9	=	+0							
Q10	=	+0.5							
Q11	=	+80							
ACTL.		X		+0.195					
		Y		-11.000					
		Z		+136.000					
		T		0					
		F		1000					
		S						M5/9	
PAGE		PAGE							
↓		↑							

10.7 Additional Functions

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

Function	Soft key
FN14:ERROR Display error messages	FN14 ERROR*
FN15:PRINT Unformatted output of texts or Q parameter values	FN15 PRINT
FN18:SYS-DATUM READ Read system data	FN18 SYS-DATUM READ
FN19:PLC Transfer values to the PLC	FN19 PLC*

FN 14: ERROR Displaying error messages

With the function FN14: 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 FN 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error numbers and the associated texts are listed in the table at right.

Resulting NC block

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

180 FN14: ERROR = 254

Range of error numbers	Standard dialog text
0 ... 299	FN 14: ERROR CODE 0 299
300 ... 999	No standard dialog text prepared
1000 ... 1099	Internal error messages (see table at right)

Error code and 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	Incorrect RPM
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive nesting
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too large
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	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted

FN 15: PRINT**Output of texts or Q parameter values**

Setting the data interface: In the menu option RS 232 INTERFACE, you must enter where the texts or Q parameters are to be stored. See section "14.4 MOD Functions, Setting the Data Interface."

The function FN15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer. When you transfer the data to a PC, the TNC stores the data in the file %FN15RUN.A (output in program run mode) or in the file %FN15SIM.A (output in test run mode).

To output dialog texts and error messages with FN15: 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

```
67 FN15:PRINT 20
```

To output dialog texts and error messages with FN15: PRINT "Q parameter"

Application example: Recording workpiece measurement.

You can transfer up to six Q parameters and numerical values simultaneously. The TNC separates them with slashes.

Example: Output of dialog text 1 and numerical value for Q1

```
70 FN15:PRINT 1/Q1
```

FN 18: SYS-DATUM READ

Reading system data

With the function FN18: SYS-DATUM READ you can read system data and store them in Q parameters. You select the system data through a group number (ID number),and additionally through a number and an index.

Group name, ID No.	Number	Index	System data
Program information, 10	1	–	MM/inch condition
	2	–	Overlap factor for pocket milling
	3	–	Number of active fixed cycle
Machine status, 20	1	–	Active tool number
	2	–	Prepared tool number
	3	–	Active tool axis 0=X, 1=Y, 2=Z
	4	–	Programmed spindle rpm
	5	–	Active spindle status: 0=off, 1=on
	6	–	Active spindle orientation angle
	7	–	Active gear range
	8	–	Coolant status: 0=off, 1=on
	9	–	Active feed rate
	10	–	Active feed rate for transition arc
Data from the tool table, 50	1	–	Tool length
	2	–	Tool radius
	4	–	Oversize for tool length DL
	5	–	Oversize for tool radius DR
	7	–	Tool inhibited (0 or 1)
	8	–	Number of replacement tool
	9	–	Maximum tool age TIME1
	10	–	Maximum tool age TIME2
	11	–	Current tool age CUR. TIME
	12	–	PLC status
	13	–	Maximum tooth length LCUTS
	14	–	Maximum plunge angle ANGLE
	15	–	TT: Number of teeth CUT
	16	–	TT: Wear tolerance for length LTOL
	17	–	TT: Wear tolerance for radius RTOL
	18	–	TT: Rotational direction DIRECT (3 or 4)
	19	–	TT: Offset for radius R-OFFS
	20	–	TT: Offset for length L-OFFS
	21	–	TT: Breakage tolerance in length LBREAK
	22	–	TT: Breakage tolerance in radius RBREAK

Group name, ID No.	Number	Index	System data
Pocket table data, 51	1	–	Tool pocket location number
	2	–	Fixed pocket: 0=no, 1=yes
	3	–	Pocket locked: 0=no, 1= yes
	4	–	Tool is a special tool: 0=no, 1=yes
	5	–	PLC status
Pocket number for active tool, 52	1	–	Pocket number in tool magazine
Compensation data, 200	1	–	Programmed tool radius
	2	–	Programmed tool length
	3	–	Oversize for tool radius DR from TOOL CALL
	4	–	Oversize for tool length DL from TOOL CALL
Active transformations, 210	1	–	Basic rotation in MANUAL OPERATION mode
	2	–	Programmed rotation with Cycle 10
	3	–	Active mirror axis
			0: mirroring not active
			+1: X axis mirrored
			+2: Y axis mirrored
			+4: Z axis mirrored
			+8: IVth axis mirrored
			Combinations = sum of individual axes
	4	1	Active scaling factor in X axis
	4	2	Active scaling factor in Y axis
	4	3	Active scaling factor in Z axis
	4	4	Active scaling factor in IVth axis
Active coordinate system, 211	1	–	Input system
	2	–	M91 system (see section "7.3 Miscellaneous Functions for Coordinate Data)."
	3	–	M92 system (see section "7.3 Miscellaneous Functions for Coordinate Data)."
Datums, 220	1	1 to 4	Datum set manually in M91 system Index 1 to 4: X-axis to IVth axis
	2	1 to 4	Programmed datum Index 1 to 4: X-axis to IVth axis
	3	1 to 4	Active datum in M91 system Index 1 to 4: X-axis to IVth axis
	4	1 to 4	PLC datum shift

Group name, ID No.	Number	Index	System data
Limit switch, 230	1	–	Number of the active limit switch range
	2	1 to 4	Negative coordinate limit switch in M91 system Index 1 to 4: X-axis to IVth axis
	3	1 to 4	Positive coordinate limit switch in M91 system Index 1 to 4: X-axis to IVth axis
Positions in M91 system, 240	1	1 to 4	Nominal position; Index 1 to 4: X-axis to IVth axis
	2	1 to 4	Last touch point Index 1 to 4: X-axis to IVth axis
	3	1 to 4	Active pole; Index 1 to 4: X-axis to axis IV axis
	4	1 to 4	Center point of circle; Index 1 to 4: X-axis to IVth axis
	5	1 to 4	Center point of circle for the last RND block Index 1 to 4: X-axis to IVth axis
Positions in the input system, 270	1	1 to 4	Nominal position; Index 1 to 4: X-axis to IVth axis
	2	1 to 4	Last touch point Index 1 to 4: X-axis to IVth axis
	3	1 to 4	Active pole; Index 1 to 4: X-axis to axis IV axis
	4	1 to 4	Center point of circle; Index 1 to 4: X-axis to IVth axis
	5	1 to 4	Center point of circle for the last RND block Index 1 to 4: X-axis to IVth axis
TT 120 calibration data, 350	20	1	Center of probe contact in X-axis
		2	Center of probe contact in Y-axis
		3	Center of probe contact in Z axis
	21	–	Probe contact radius

Example: Assign the value of the active scaling factor for the Z axis to Q25.

```
55 FN18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN19: PLC**Transferring values to the PLC**

The function FN19: PLC transfers up to two numerical values or Q parameter contents to the PLC.

Resolution and units: 1 µm and 0.001° or
 0,1 µm and 0.0001°



The resolution depends on the machine parameter 4020 (default setting is 1 µm and 0.001°).

Example: Numerical value 10 (meaning 10 µm or 0,01°) is sent to the PLC.

56 FN19:PLC=+10/+Q3

10.8 Entering Formulas Directly

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Entering formulas

Press the FORMULA soft key to call the formula functions.
The TNC displays the following soft keys in several soft-key rows:

Mathematical function	Soft key
Addition Example: Q10 = Q1 + Q5	<div>+</div>
Subtraction Example: Q25 = Q7 – Q108	<div>-</div>
Multiplication Example: Q12 = 5 * Q5	<div>*</div>
Division Example: Q25 = Q1 / Q2	<div>/</div>
Open parentheses Example: Q12 = Q1 * (Q2 + Q3)	<div>(</div>
Close parentheses Example: Q12 = Q1 * (Q2 + Q3)	<div>)</div>
Square Example: Q15 = SQ 5	<div>SQ</div>
Square root Example: Q22 = SQRT 25	<div>SQRT</div>
Sine of an angle Example: Q44 = SIN 45	<div>SIN</div>
Cosine of an angle Example: Q45 = COS 45	<div>COS</div>
Tangent of an angle Example: Q46 = TAN 45	<div>TAN</div>

Mathematical function	Soft key
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: Q11 = ACOS Q40	ACOS
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 Example: Q15 = 3^3	^
Constant "pi" (3.14159) e.g. 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, 2.7183ⁿ Example: Q1 = EXP Q12	EXP
Negate (multiplication by -1) Example: Q2 = NEG Q1	NEG
Drop places after the decimal point (form an integer) Example: Q3 = INT Q42	INT
Absolute value Example: Q4 = ABS Q22	ABS
Drop places before the decimal point (form a fraction) Example: Q5 = FRAC Q23	FRAC

Mathematical function	Soft key
Check the sign of a number e.g. Q12 = SGN Q50 If result for Q12 = 1: Q50 >= 0 If result for Q12 = -1: Q50 < 0	SGN

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)

12 Q1 = 5 * 3 + 2 * 10 = 35

1st step: 5 * 3 = 15
2nd step: 2 * 10 = 20
3rd step: 15 + 20 = 35



13 Q2 = SQ 10 - 3^3 = 73

1st step: 10² = 100
2nd step: 3³ = 27
3rd step: 100 - 27 = 73

- **Distributive law**
for calculating with parentheses
 $a * (b + c) = a * b + a * c$

Programming example


Calculate an angle with arc tangent as opposite side (Q12) and adjacent side (Q13); then store in Q25.



To select the formula entering function, press the Q key and FORMULA soft key.

Parameter number for result?



25




Enter the parameter number.

Shift the soft-key row and select the arc tangent function.





Shift the soft-key row and open parentheses.




12

Enter Q parameter number 12.





Select division.



13

Enter Q parameter number 13.

Close parentheses and conclude formula entry.

Resulting NC block

37 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.

Tool radius: Q108

The current value of the tool radius is assigned to Q108.

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = -1
Z axis	Q109 = 2
Y axis	Q109 = 1
X axis	Q109 = 0

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 PGM CALL) 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.

Coordinates after probing during program run

The parameters Q115 to Q118 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis	Q118

Deviation between actual value and nominal value during automatic tool measurement with the TT 120

Actual-nominal deviation	Parameter
Tool length	Q115
Tool radius	Q116

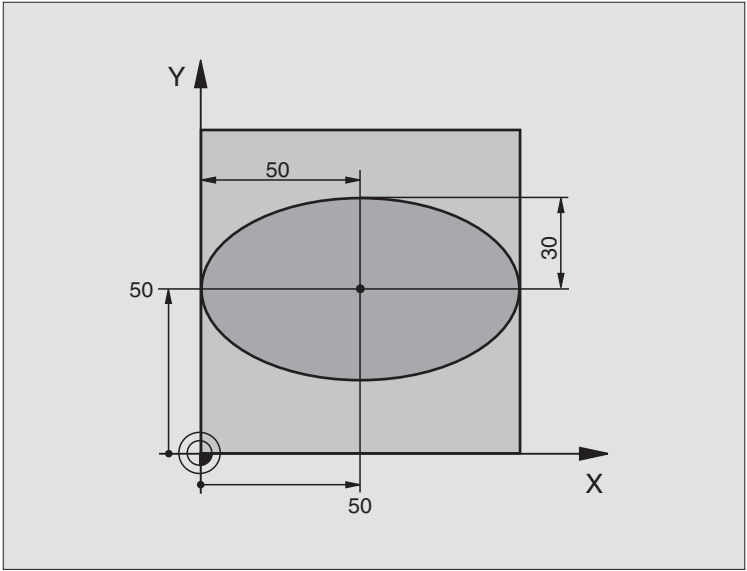
Active tool radius compensation

Active radius compensation	Parameter value
R0	Q123 = 0
RL	Q123 = 1
RR	Q123 = 2
R+	Q123 = 3
R-	Q123 = 4

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculating 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.



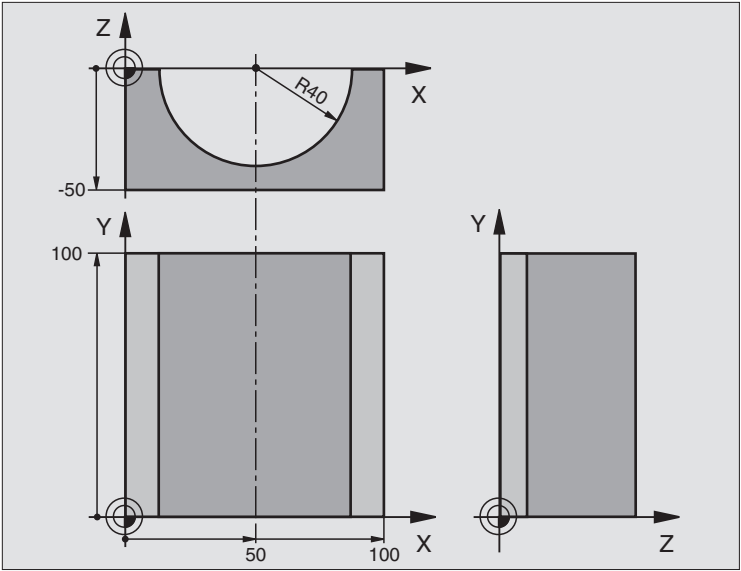
0	BEGIN PGM ELLIPSE MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +50	Center in Y axis
3	FN 0: Q3 = +50	Semiaxis in X
4	FN 0: Q4 = +30	Semiaxis in Y
5	FN 0: Q5 = +0	Starting angle in the plane
6	FN 0: Q6 = +360	End angle in the plane
7	FN 0: Q7 = +40	Number of calculating steps
8	FN 0: Q8 = +0	Rotational position of the ellipse
9	FN 0: Q9 = +5	Milling depth
10	FN 0: Q10 = +100	Feed rate for plunging
11	FN 0: Q11 = +350	Feed rate for milling
12	FN 0: Q12 = +2	Setup clearance for pre-positioning
13	BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+2.5	Define the tool
16	TOOL CALL 1 Z S4000	Call the tool
17	L Z+250 R0 F MAX	Retract the touch probe
18	CALL LBL 10	Call machining operation
19	L Z+100 R0 FMAX M2	Retract in the tool axis, end of program

20	LBL 10	Subprogram 10: Machining operation
21	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of ellipse
22	CYCL DEF 7.1 X+Q1	
23	CYCL DEF 7.2 Y+Q2	
24	CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
25	CYCL DEF 10.1 ROT+Q8	
26	Q35 = (Q6 - Q5) / Q7	Calculate angle increment
27	Q36 = Q5	Copy starting angle
28	Q37 = 0	Set counter
29	Q21 = Q3 * COS Q36	Calculate X coordinate for starting point
30	Q22 = Q4 * SIN Q36	Calculate Y coordinate for starting point
31	L X+Q21 Y+Q22 R0 F MAX M3	Move to starting point in the plane
32	L Z+Q12 R0 F MAX	Pre-position in tool axis to setup clearance
33	L Z-Q9 R0 FQ10	Move to working depth
34	LBL 1	
35	Q36 = Q36 + Q35	Update the angle
36	Q37 = Q37 + 1	Update the counter
37	Q21 = Q3 * COS Q36	Calculate the current X coordinate
38	Q22 = Q4 * SIN Q36	Calculate the current Y coordinate
39	L X+Q21 Y+Q22 R0 FQ11	Move to next point
40	FN 12: IF +Q37 LT +Q7 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
41	CYCL DEF 10.0 ROTATION	Reset the rotation
42	CYCL DEF 10.1 ROT+0	
43	CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
44	CYCL DEF 7.1 X+0	
45	CYCL DEF 7.2 Y+0	
46	L Z+Q12 R0 F MAX	Move to setup clearance
47	LBL 0	End of subprogram
48	END PGM ELLIPSE MM	

Example: Concave cylinder machined with spherical cutter

Program sequence

- This program requires a spherical cutter.
- 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.
- Tool length with respect to the ball center.



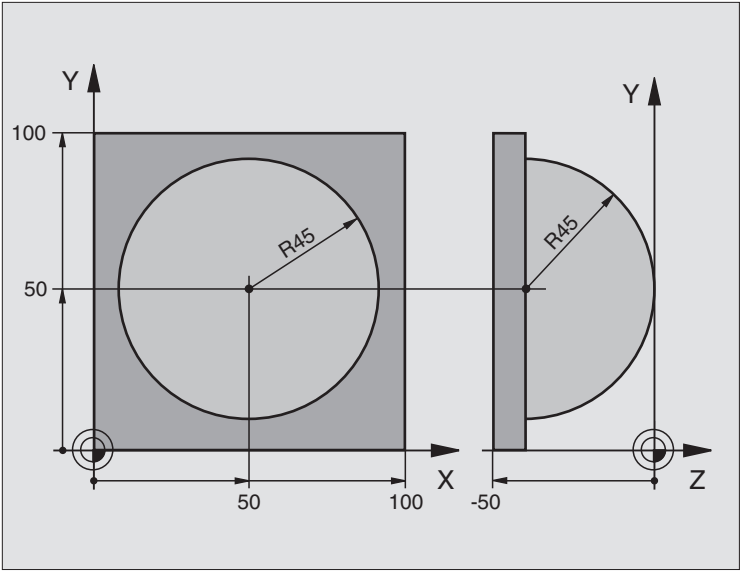
0	BEGIN PGM CYLIN MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +0	Center in Y axis
3	FN 0: Q3 = +0	Center in Z axis
4	FN 0: Q4 = +90	Starting angle in space (Z/X plane)
5	FN 0: Q5 = +270	End angle in space (Z/X plane)
6	FN 0: Q6 = +40	Radius of the cylinder
7	FN 0: Q7 = +100	Length of the cylinder
8	FN 0: Q8 = +0	Rotational position in the X/Y plane
9	FN 0: Q10 = +5	Allowance for cylinder radius
10	FN 0: Q11 = +250	Feed rate for plunging
11	FN 0: Q12 = +400	Feed rate for milling
12	FN 0: Q13 = +90	Number of cuts
13	BLK FORM 0.1 Z X+0 Y+0 Z-50	Define the blank form
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+3	Define the tool
16	TOOL CALL 1 Z S4000	Call the tool
17	L Z+250 R0 F MAX	Retract the touch probe
18	CALL LBL 10	Call machining operation
19	FN 0: Q10 = +0	Reset allowance
20	CALL LBL 10	Call machining operation
21	L Z+100 R0 FMAX M2	Retract in the tool axis, end of program

22	LBL 10	Subprogram 10: Machining operation
23	Q16 = Q6 - Q10 - Q108	Account for allowance and tool, based on the cylinder radius
24	FN 0: Q20 = +1	Set counter
25	FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
26	Q25 = (Q5 - Q4) / Q13	Calculate angle increment
27	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of cylinder (X axis)
28	CYCL DEF 7.1 X+Q1	
29	CYCL DEF 7.2 Y+Q2	
30	CYCL DEF 7.3 Z-Q3	
31	CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
32	CYCL DEF 10.1 ROT+Q8	
33	L X+0 Y+0 R0 F MAX	Pre-position in the plane to the cylinder center
34	L Z+5 R0 F1000 M3	Pre-position in the tool axis
35	CC Z+0 X+0	Set pole in the Z/X plane
36	LP PR+Q16 PA+Q24 FQ11	Move to starting position on cylinder, plunge-cutting obliquely into the material
37	LBL 1	
38	L Y+Q7 R0 FQ11	Longitudinal cut in Y+ direction
39	FN 1: Q20 = +Q20 + +1	Update the counter
40	FN 1: Q24 = +Q24 + +Q25	Update solid angle
41	FN 11: IF +Q20 GT +Q13 GOTO LBL 99	Finished? If finished, jump to end.
42	LP PR+Q16 PA+Q24 FQ12	Move in an approximated "arc" for the next longitudinal cut
43	L Y+0 R0 FQ11	Longitudinal cut in Y- direction
44	FN 1: Q24 = +Q24 + +Q25	Update the counter
45	FN 1: Q20 = +Q20 + +1	Update solid angle
46	FN 12: IF +Q20 LT +Q13 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
47	LBL 99	
48	CYCL DEF 10.0 ROTATION	Reset the rotation
49	CYCL DEF 10.1 ROT+0	
50	CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
51	CYCL DEF 7.1 X+0	
52	CYCL DEF 7.2 Y+0	
53	CYCL DEF 7.3 Z+0	
54	LBL 0	End of subprogram
55	END PGM CYLIN MM	

Example: Convex sphere machined with end mill

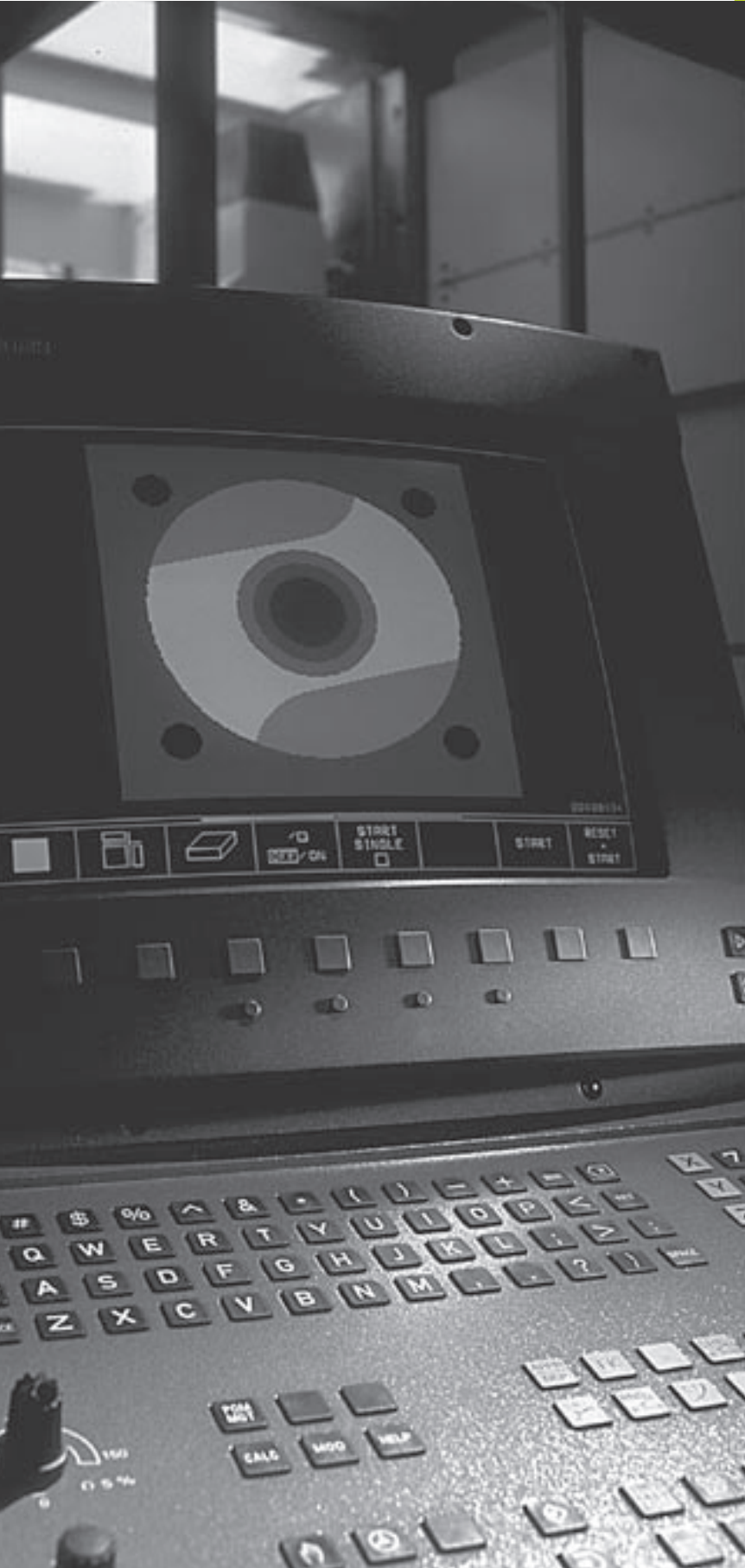
Program sequence

- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined via 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.



0	BEGIN PGM BALL MM	
1	FN 0: Q1 = +50	Center in X axis
2	FN 0: Q2 = +50	Center in Y axis
3	FN 0: Q4 = +90	Starting angle in space (Z/X plane)
4	FN 0: Q5 = +0	End angle in space (Z/X plane)
5	FN 0: Q14 = +5	Angle increment in space
6	FN 0: Q6 = +45	Radius of the sphere
7	FN 0: Q8 = +0	Starting angle of rotational position in the X/Y plane
8	FN 0: Q9 = +360	End angle of rotational position in the X/Y plane
9	FN 0: Q18 = +10	Angle increment in the X/Y plane for roughing
10	FN 0: Q10 = +5	Allowance in sphere radius for roughing
11	FN 0: Q11 = +2	Setup clearance for pre-positioning in the tool axis
12	FN 0: Q12 = +350	Feed rate for milling
13	BLK FORM 0.1 Z X+0 Y+0 Z-50	Define the blank form
14	BLK FORM 0.2 X+100 Y+100 Z+0	
15	TOOL DEF 1 L+0 R+7.5	Define the tool
16	TOOL CALL 1 Z S4000	Call the tool
17	L Z+250 R0 F MAX	Retract the touch probe
18	CALL LBL 10	Call machining operation
19	FN 0: Q10 = +0	Reset allowance
20	FN 0: Q18 = +5	Angle increment in the X/Y plane for finishing
21	CALL LBL 10	Call machining operation
22	L Z+100 R0 FMAX M2	Retract in the tool axis, end of program

23	LBL 10	Subprogram 10: Machining operation
24	FN 1: Q23 = +Q11 + +Q6	Calculate Z coordinate for pre-positioning
25	FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
26	FN 1: Q26 = +Q6 + +Q108	Compensate sphere radius for pre-positioning
27	FN 0: Q28 = +Q8	Copy rotational position in the plane
28	FN 1: Q16 = +Q6 + -Q10	Account for allowance in the sphere radius
29	CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of sphere
30	CYCL DEF 7.1 X+Q1	
31	CYCL DEF 7.2 Y+Q2	
32	CYCL DEF 7.3 Z-Q16	
33	CYCL DEF 10.0 ROTATION	Account for starting angle of rotational position in the plane
34	CYCL DEF 10.1 ROT+Q8	
35	CC X+0 Y+0	Set pole in the X/Y plane for pre-positioning
36	LP PR+Q26 PA+Q8 R0 FQ12	Pre-position in the plane
37	LBL 1	Pre-position in the tool axis
38	CC Z+0 X+Q108	Set pole in the Z/X plane, offset by the tool radius
39	L Y+0 Z+0 FQ12	Move to working depth
40	LBL 2	
41	LP PR+Q6 PA+Q24 R0 FQ12	Move upward in an approximated "arc"
42	FN 2: Q24 = +Q24 - +Q14	Update solid angle
43	FN 11: IF +Q24 GT +Q5 GOTO LBL 2	Inquire whether an arc is finished. If not finished, return to LBL 2.
44	LP PR+Q6 PA+Q5	Move to the end angle in space
45	L Z+Q23 R0 F1000	Retract in the tool axis
46	L X+Q26 R0 F MAX	Pre-position for next arc
47	FN 1: Q28 = +Q28 + +Q18	Update rotational position in the plane
48	FN 0: Q24 = +Q4	Reset solid angle
49	CYCL DEF 10.0 ROTATION	Activate new rotational position
50	CYCL DEF 10.1 ROT+Q28	
51	FN 12: IF +Q28 LT +Q9 GOTO LBL 1	
52	FN 9: IF +Q28 EQU +Q9 GOTO LBL 1	Unfinished? If not finished, return to label 1
53	CYCL DEF 10.0 ROTATION	Reset the rotation
54	CYCL DEF 10.1 ROT+0	
55	CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
56	CYCL DEF 7.1 X+0	
57	CYCL DEF 7.2 Y+0	
58	CYCL DEF 7.3 Z+0	
59	LBL 0	End of subprogram
60	END PGM BALL MM	



11

**Test Run
and Program Run**

11.1 Graphics

In the Test Run mode of operation the TNC graphically simulates the machining of the workpiece. 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.

The TNC will not show a graphic if

- the current program has no valid blank form definition
- no program is selected




A graphic simulation is not possible for program or program sections that include rotary-axis traverse. In such cases the TNC displays an error messages.

Overview of display modes

After you have selected the GRAPHICS or PROGRAM + GRAPHICS screen layout in the Test Run operating mode, the TNC displays the following soft keys:

Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	


Plan view





 ► Press the soft key for plan view.

Projection in 3 planes

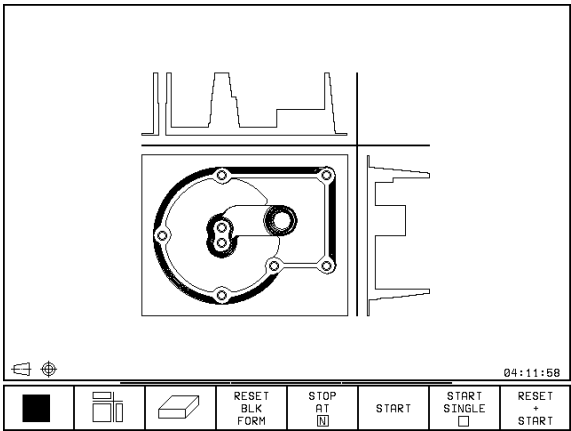
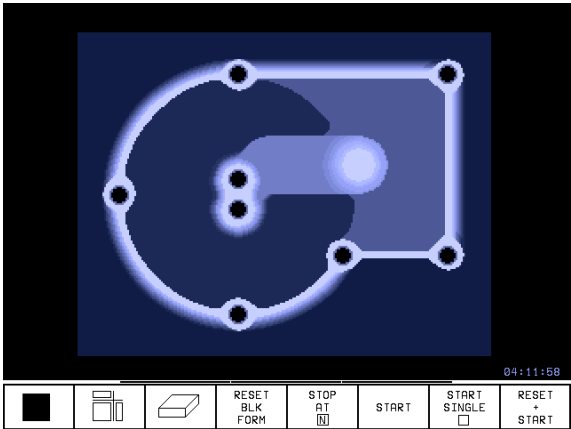
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

In addition, you can shift the sectional planes with the corresponding soft keys:

-  ► Press the soft key for projection in three planes.
- Shift the soft-key row until the TNC displays the following soft keys:

Function	Soft keys	
Shift the vertical sectional plane to the right or left		
Shift the horizontal sectional plane upwards or downwards		

The positions of the sectional planes are visible during shifting.



3-D view

The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

In the Test Run mode of operation you can isolate details for magnification (see “Magnifying details”).



► Press the soft key for 3-D view.

To rotate the 3-D view

Shift the soft-key row until the following soft keys appear:

Function	Soft keys	
Rotate the workpiece in 27° steps about the vertical axis		

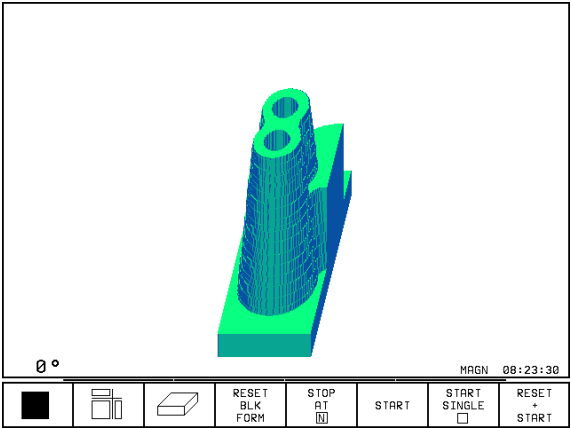
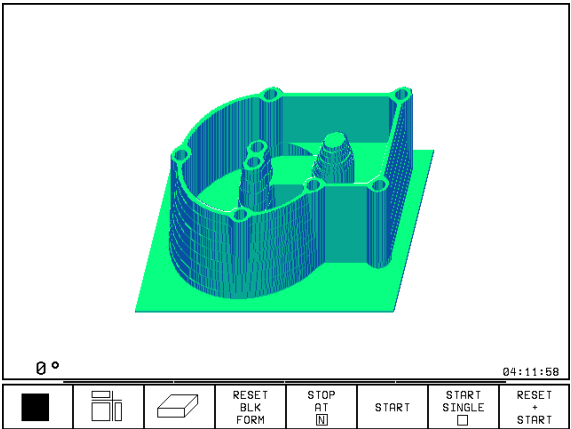
Magnifying details

You can isolate a detail in the Test Run operating mode, when the 3-D display mode is selected.

The graphic simulation must first have been stopped. A detail magnification is always effective in all display modes.

Shift the soft-key row in the Test Run mode of operation until the following soft keys appear:

Function	Soft keys	
Select the workpiece surface to be trimmed: Press the soft key several times		
Shift the sectional plane to reduce or magnify the blank form		
Select the isolated detail		



To change the detail magnification:

The soft keys are listed in the table above.

- ▶ Interrupt the graphic simulation, if necessary.
- ▶ Use a soft key to select the workpiece surface
- ▶ To reduce or magnify the blank form, press and hold the minus or plus soft key, respectively.
- ▶ To select the isolated detail, press the TRANSFER soft key.
- ▶ Restart the test run by pressing the START soft key (RESET + START returns the workpiece blank to its original state).

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	<div>RESET BLK FORM</div>
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM	<div>WINDOW BLK FORM</div>



With the WINDOW BLK FORM soft key, you return the displayed workpiece blank to its originally programmed dimensions, even after isolating a detail TRANSFER 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 which 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.

To activate 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	<div>STORE ⏸</div>
Display the sum of stored time and displayed time	<div>ADD ⏸ + ⏸</div>
Clear displayed time	<div>RESET 00:00:00 ⏸</div>

11.2 Test run

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 errors:

- 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
- Additional status display

Test run									
BEGIN PGM 3507 MM									
1	BLK	FORM	0.1	Z	X-20	Y-20	Z-20		
2	BLK	FORM	0.2		X+20	Y+20	Z+0		
3	TOOL	DEF	1	L+0	R+3				
4	TOOL	CALL	1	Z	S1000				
5	L	Z+50	R0	FMAX	M3				
6	L	X+50	Y+50	R0	FMAX	M8			
7	L	Z-5	R0	FMAX	M1				
8	CC	X+0	Y+0						
9	LP	PR+14	PA+45	RR	F500				
10	RND	R1							
11	FC	DR+	R2.5	CLSD+					
ACTL.		X			+0.195				
		Y			-11.000				
		Z			+136.000				
							T		
							F	0	
							S	1000	M5/9
STORE	ADD	RESET							
⏸	⏸ + ⏸	00:00:00 ⏸							


Running a program test



- Select the Test Run mode of operation.
- 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

Test run	
<pre> BEGIN PGM SLOLD MM 1 FN 0: 01 =+0.5 2 FN 0: 02 =+32 3 FN 0: 03 =+16 4 FN 0: 04 =+24 5 FN 0: 05 =+10 6 FN 0: 06 =+6 7 FN 0: 07 =+12 8 FN 0: 08 =+6 9 FN 0: 010 =+0.5 10 FN 0: 011 =+80 11 FN 0: 012 =+45.8 </pre>	 <p>0° 00:00:00</p>
<pre> NOML. X -219.210 Y +0.795 Z +212.795 </pre>	<p>T 1 Z</p> <p>F 0</p> <p>S M5/9</p>
<div> </div>	<div> <div>RESET BLK FORM</div> <div>STOP AT</div> <div>START</div> <div>START SINGLE</div> <div>RESET + START</div> </div>

Running 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. If you have selected a screen layout showing a graphic, the graphic will also be brought up to block 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.



- Up to block number: Enter the number of the block at which the test run should stop.
- Program: If you wish to go into a program that you call with CALL PGM, enter the name of the program containing the block with the selected block number.
- 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.

Test run	
<pre> BEGIN PGM 3507 MM 1 BLK FORM 0.1 Z X-20 Y-20 Z-20 2 BLK FORM 0.2 X+20 Y+20 Z+0 3 TOOL DEF 1 L+0 R+3 4 TOOL CALL 1 Z S1000 5 L Z+50 R0 FMAX M3 6 L X+50 Y+50 R0 F 7 L Z-5 R0 FMAX M1 8 CC X+0 Y+0 9 LP PR+14 PA+45 RR F500 10 RND R1 11 FC DR+ R2.5 CLSD+ </pre>	<div> To block number = 12 Program = 3507 Repetitions = 1 </div>
<pre> ACTL. X +0.195 Y -11.000 Z +136.000 </pre>	<p>T F</p> <p>S 1000 M5/9</p>
<div> <div>STOP AT N</div> <div>START</div> <div>END</div> </div>	

11.3 Program run

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the Program Run, Single Block mode of operation you must start each block separately by pressing the machine START button.


The following TNC functions can be used in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Optional Block Skip
- Checking and changing Q parameters
- Additional status display

To run a part program:

Preparation

- 1 Clamp the workpiece to the machine table.
- 2 Set the datum
- 3 Select the part program (status M).




You can adjust the feed rate and spindle speed with the override knobs.

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.

Program run, full sequence									
BEGIN PGM 3507 MM					Programs 3507 / 0				
1 BLK FORM 0.1 Z X-20 Y-20 Z-20					ACTL. X -219.210				
2 BLK FORM 0.2 X+20 Y+20 Z+0					Y +0.795				
3 TOOL DEF 1 L+0 R+3					Z +212.795				
4 TOOL CALL 1 Z S1000									
5 L Z+50 R0 FMAX M3					<div> Basic rotation +12.357</div>				
6 L X+50 Y+50 R0 FMAX M8									
7 L Z-5 R0 FMAX M1									
8 CC X+0 Y+0									
9 LP PR+14 PA+45 RR F500									
10 RND R1									
11 FC DR+ R2.5 CLSD+									
NOML. X -219.210									
Y +0.795									
Z +212.795									

Running a part program which contains coordinates for non-controlled axes

The TNC can also run programs in which you have programmed non-controlled axes.

If the TNC arrives at a block in which you have programmed a non-controlled axis, it stops program run. At the same time it superimposes a window showing the distance-to-go to the target position (see figure at top right). Proceed as follows:

- Move the axis manually to the target position. The TNC constantly updates the distance-to-go window, and always shows the distance remaining to reach the target position.
- Once you have reached the target position, press the NC START key to continue program run. If you press the NC START key before you have arrived at the target position, the TNC will output an error message.



Machine parameter 1030.x determines how accurately you need to approach the target position (possible input values: 0.001 to 2 mm).

Non-controlled axes must be programmed in separate positioning blocks, otherwise the TNC will output an error message.

Program run, full sequence									
2	BLK	FORM	0.2	X+100	Y+100	Z+0			
3	TOOL	DEF	211	L+0	R+3				
4	TOOL	DEF	212	L+0	R+3				
5	TOOL	CALL	211	Z	S5000				
6	L	Z+50	R0	F5000					
7	L	X+50	Y+50	R0					
8	SEL	PATTERN		"MUSTPKT"					
9	CYCL	DEF	4.0	POCKET	MTLT	ING			
10	CYCL	DEF							
11	CYCL	DEF		Z	+33.764				
12	CYCL	DEF	4.3	PLUNGING	PLSD				
13	CYCL	DEF	4.4	X+25					
ACTL.		X		+0.735					
*		Y		-125.145					
		+Z		+16.235					
							T 211 Z		
							F 0		
							S 2600	M5/9	
									INTERNAL STOP

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:

- STOP (with and without a miscellaneous function)
- Miscellaneous function M0, M1 (see section "11.6 Optional Program Run Interruption"), M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

To interrupt machining 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.

Interruption of machining 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 Program Run, Single Block. 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.

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.

Use the function „Returning to the Contour“ (see below) to return to a contour at the point of interruption.

Program run, full sequence									
3	T00L	DEF	1	L+0	R+5				
4	T00L	CALL	1	Z	S1000				
5	L	Z+100	R0	FMAX					
6	M112	T0.3	A+10						
7	L	X+20	Y+0	R0	F500				
8	L	X+48	Y+79						
9	L	X+49	Y+79						
10	L	X+50	Y+80						
11	L	X+51	Y+79						
12	L	X+52	Y+79						
13	L	X+80	Y+0	FMAX					
14	L	X+100	Y+0	R0	FMAX				
ACTL.		X		+78.415					
*		Y		+0.000					
		Z		+100.000					
						T	1	Z	
						F	0		
						S	1000		M5 / 9
MANUAL OPERATION									INTERNAL STOP

Resuming program run after an interruption



If you interrupt a program run during a fixed cycle, you must resume the program from the beginning of the cycle. This means that some machining operations will be repeated.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations
- The coordinates of the circle center that was last defined
- The current count of program section repeats
- The number of the block where a subprogram or a program section repeat was last called

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
- A programmed interruption
- The EMERGENCY STOP button was pressed (machine-dependent function)



If you interrupted program run with the INTERNAL STOP soft key, you can select another block with the GOTO key and resume machining there.

If you select the BEGIN PGM block (block 0), the TNC resets all stored data (tool data, etc.)


If you interrupted program run during a program section repeat, you can only use GOTO to select other blocks within the program section repeat.

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 at the place at which 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 block-scan function can vary depending on the individual machine. Refer to your machine tool 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.



Always begin a block scan from the beginning of the program.

If the program contains a programmed interruption before the startup block, the TNC interrupts the block scan. Press the RESTORE POS. AT and START soft keys once again to continue the block scan.


After a block scan, use the returning-to-contour function to move the tool to mid-program startup position (see following page).

Tool length compensation only becomes effective after the tool has been called and positioned in the tool axis. This also applies to tool lengths that have changed.

- ▶ To go to the first block of the current program to start a block scan, enter GOTO "0".
- ▶ To select block scan, press the RESTORE POS AT N soft key. The TNC displays an input window:



- ▶ 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 all 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. Make sure that the tool in the spindle corresponds to the tool called in the program.
- ▶ To start the block scan, press the START soft key.
- ▶ To return to contour: See following section "Returning to the Contour."



You can move the input window for mid-program startup. Press the screen layout key and use the displayed soft keys.

Program run, full sequence							
BEGIN PGM 3507 MM							
1	BLK FORM	0.1	Z	X-20	Y-20	Z-20	
2	BLK FORM	0.2	X+20	Y+20	Z+0		
3	TOOL DEF	1	L+0	R+3			
4	TOOL CALL	1	Z	S100			
5	L	Z+50	R0	FMAX	M3		
6	L	X+50	Y+50	R0	F		
7	L	Z-5	R0	FMAX	M1		
8	CC	X+0	Y+0				
9	LP	PR+14	PA+45	RR	F500		
10	RND	R1					
11	FC	DR+	R2.5	CLSD+			
ACTL.							
X				+0.195			
Y				-11.000			
Z				+136.000			
				T			
				F	0		
				S	1000 M5/9		
					START		END

Returning to the contour

With the RESTORE POSITION function, the TNC returns the tool to the workpiece contour after you moved the machine axes during a program interruption with the MANUAL OPERATION soft key:

- ▶ To select a return to contour, press the RESTORE POSITION soft key (not necessary with block scan). In the displayed window the TNC shows 1 the position to which it moves the tool.
- ▶ To move the axes in the sequence that the TNC suggests 1 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

3 TOOL DEF 1 L+0 R+5

4 TOOL CALL 1 Z S1000

5 L Z+100 R0 FMAX

6 M112 T0.3 A+10

7 L X+20 Y+0 R0 F50

8 L X+48 Y+79

9 L X+49 Y+79

10 L X+50 Y+80

11 L X+51 Y+79

12 L X+52 Y+79

13 L X+80 Y+0 FMAX

14 L X+100 Y+0 R0 FMAX

ACTL. X +0.195

* Y -11.000

Z +136.000

T 1 Z

F 0

S 1000 M5/9

RESTORE X

RESTORE Y

RESTORE Z

MANUAL OPERATION

INTERNAL STOP

11.4 Blockwise Transfer: Running Longer Programs

Machine programs that require more memory space than is available in the TNC can be transferred „blockwise“ from an external memory.

The program blocks are read in via data interface and are then deleted immediately after being executed. In this way programs of unlimited length can be executed.



The program may have a maximum of 20 TOOL DEF blocks. If you require more tools then use a tool table.

If the program contains a PGM CALL block, the called program must be stored in the TNC memory.

The program may not include:

- Subprograms
- Program section repeats
- Function FN15:PRINT

Blockwise program transfer

Configure the data interface with the "Set block buffer" MOD function (see „14.4 Setting the External Data Interface“).



- ▶ Select the Program Run, Full Sequence mode or the Program Run, Single Block mode.
- ▶ Begin blockwise transfer: Press the BLOCKWISE TRANSFER
- ▶ Enter the program name, if necessary change the program type via soft key, and confirm with ENT. The TNC reads in the selected program via data interface. If you do not indicate the type of program, the TNC will read in the type you have defined in the MOD function „Program input“.
- ▶ Start the part program with the machine start button. If you have defined a block buffer greater than 0, the TNC waits with the program start until the defined number of NC blocks has been read in.

11.5 Optional Block Skip

In a test run or program run, the TNC can skip over blocks that begin with a slash (/):



- ▶ To run or test the program with the blocks preceded by a slash, set the soft key to OFF.



- ▶ To run or test the program without the blocks preceded by a slash, set the soft key to ON.



This function does not work for TOOL DEF blocks.

11.6 Optional Program Run Interruption

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



12

3-D Touch Probes

12.1 Touch Probe Cycles in the Manual and Electronic Handwheel modes.



The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.



If you are carrying out measurements during program run, be sure that the tool data (length, radius, axis) can be used from the calibrated data or from the last TOOL CALL block (selected with MP7411).

After you press the machine START button, the touch probe approaches the workpiece in a paraxial movement in the selected probe function. The machine tool builder sets the probe feed rate (see figure at right). When the probe contacts the workpiece, it

- the 3-D touch probe system sends a signal to the TNC: the coordinates of the probed position are saved
- stops moving, and
- returns to its starting position in rapid traverse.

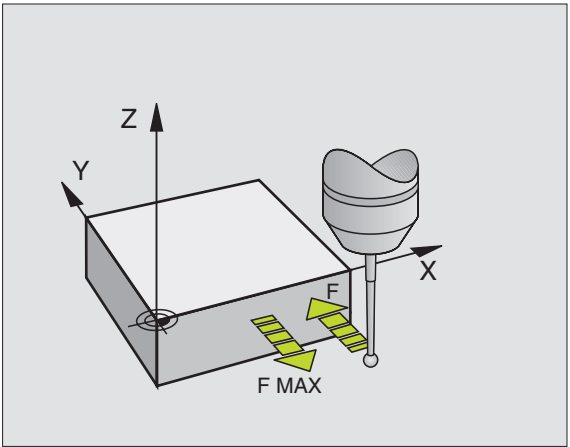
If the stylus is not deflected within a defined distance, the TNC displays an error message (MP 6130).

To select the touch probe functions:

- Select the Manual Operation or Electronic Handwheel mode of operation



- To choose the touch probe functions, press the TOUCH PROBE soft key. The TNC displays additional soft keys — see table at right.



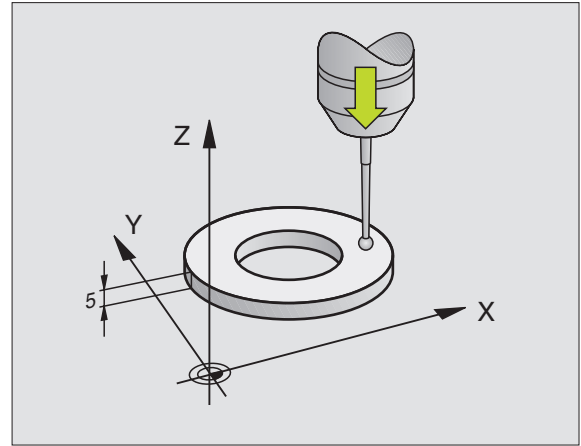
Function	Soft key
Calibrate the effective length	
Calibrate the effective radius	
Basic rotation	
Set the datum	
Set the datum at a corner	
Set the datum at a circle center	

Calibrating a triggering touch probe

The touch probe must be calibrated

- its commissioning
- when the stylus breaks
- when the stylus is changed
- when the probe feed rate is changed
- in case of irregularities such as those resulting from thermal changes in the machine

During calibration, the TNC finds the “effective” length of the stylus and the “effective” radius of the ball tip. To calibrate the touch probe, clamp a ring gauge of known height and known internal radius to the machine table.



To calibrate the effective length:

- ▶ Set the datum in the tool axis such that for the machine tool table $Z=0$.



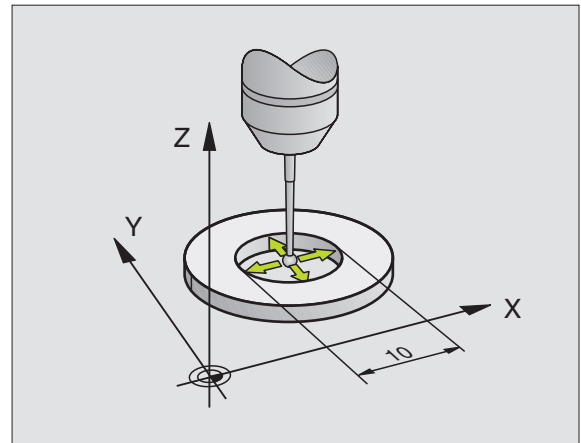
- ▶ To select the calibration function for the touch probe length, press the TOUCH PROBE and CAL L soft keys. The TNC then displays a menu window with four input fields.
- ▶ Select the tool axis via soft key.
- ▶ Datum: Enter the height of the ring gauge.
- ▶ The menu items Effective ball radius and Effective length do not require input.
- ▶ Move the touch probe to a position just above the ring gauge.
- ▶ To change the displayed traverse direction (if necessary), press an arrow key.
- ▶ To probe the upper surface of the ring gauge, press the machine START button.

Calibrating the effective radius and compensating center misalignment

After the touch probe is inserted it normally needs to be exactly aligned with the spindle axis. The misalignment is measured with this calibration function and compensated electronically.

For this operation the TNC rotates the 3-D touch probe by 180°. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in the machine parameter 6160.

The center misalignment is measured after the effective ball tip radius is calibrated.



- In the Manual Operation mode, position the ball tip in the bore of the ring gauge.



- To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL R soft key.

- Select the tool axis, enter the radius of the ring gauge.

- To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the effective ball-tip radius.

- If you want to terminate the calibration function at this point, press the END soft key.



- If you want to determine the ball-tip center misalignment, press the 180° soft key. The TNC rotates the touch probe by 180°.

- To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the ball-tip center misalignment.

Displaying calibration values

The TNC stores the effective length and radius, and the center misalignment for use when the touch probe is needed again. You can display the values on the screen with the soft keys CAL L and CAL R.

Compensating workpiece misalignment

The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

For this purpose, the TNC sets the rotation angle to the desired angle with respect to the reference axis in the working plane. See figure at lower right.



Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.

To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.



- Select the touch probe function: Press the PROBING ROT soft key

- Position the ball tip at a starting position near the first touch point.

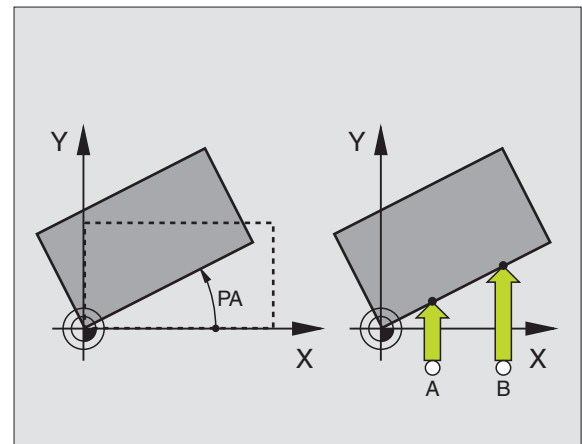
- Select the probe direction perpendicular to the angle reference axis: Select the axis with an arrow key.

- To probe the workpiece, press the machine START button.

- Position the ball tip at a starting position near the second touch point.

- To probe the workpiece, press the machine START button.

Calibration effective radius							
X+ X- Y+ Y-							
Tool axis = Z Radius ring gauge = 24.998 Effect. probe radius = 2.335 Effective length = +0 Styl.tip center offset X+0 Styl.tip center offset Y+0							
ACTL.		X	+0.195				
		Y	-11.000				
		Z	+136.000				
		T	0				
		S	1000	M5/9			
X	Y	Z					END



The TNC saves the basic rotation in non-volatile memory. The basic rotation is effective for all subsequent program runs and test runs.

Displaying a basic rotation

The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. The TNC also displays the rotation angle in the additional status display (screen layout: PROGRAM + STATUS POSITION DISPLAY).

In the status display a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.

To cancel a basic rotation:

- ▶ Select the touch probe function: Press the PROBING ROT soft key
- ▶ Enter a rotation angle of zero and confirm with the ENT key.
- ▶ To terminate the probe function, press the END key.

Basic rotation									
X+ X- Y+ Y-									
Rotation angle = +12.357									
ACTL. X -219.715				T 2 Z F 0 S <div>ROT</div> <div>M5/9</div>					
Y +0.285									
Z +212.680									
									END

12.2 Setting the Datum with a 3-D Touch Probe

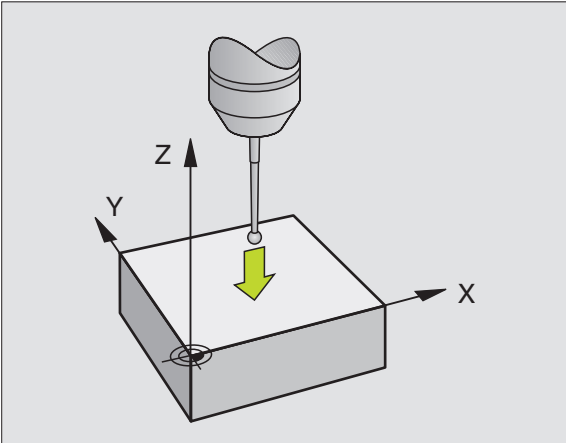
The following functions are available for setting the datum on an aligned workpiece:

- Datum setting in any axis with PROBING POS
- Defining a corner as datum with PROBING P
- Setting the datum at a circle center with PROBING CC

Setting the datum in any axis (see figure at bottom right)



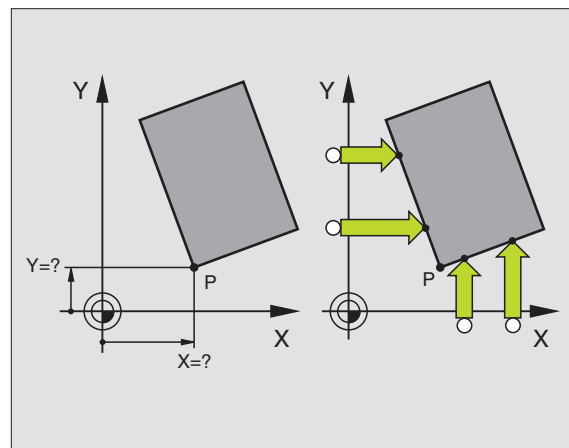
- ▶ Select the touch probe function: Press the PROBING POS soft key
- ▶ Move the touch probe to a starting position near the first touch point
- ▶ Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z-. Selection is made with the arrow keys.
- ▶ To probe the workpiece, press the machine START button.
- ▶ Datum: Enter the nominal coordinate and confirm you entry with ENT.



Corner as datum — using points that were already probed for a basic rotation (see figure at upper right)



- ▶ Select the touch probe function: Press the PROBING P soft key
- ▶ Touch points of basic rotation?: Press ENT to transfer the touch point coordinates to memory.
- ▶ Position the touch probe at a starting position near the first touch point of the side that was not probed for basic rotation.
- ▶ Select the probe direction: Select the axis with an arrow key.
- ▶ To probe the workpiece, press the machine START button.
- ▶ Position the touch probe near the second touch point on the same side.
- ▶ To probe the workpiece, press the machine START button.
- ▶ Datum: Enter both datum coordinates into the menu window, and confirm your entry with the ENT key.
- ▶ To terminate the probe function, press the END key.



Corner as datum — without using points that were already probed for a basic rotation

- ▶ Select the touch probe function: Press the PROBING P soft key
- ▶ Touch points of basic rotation?: Press NO ENT to ignore the previous touch points. (The dialog question only appears if a basic rotation was made previously.)
- ▶ Probe both workpiece sides twice.
- ▶ Enter the datum coordinates and confirm your entry with ENT.
- ▶ To terminate the probe function, press the END key.

Circle center as datum

With this function, the centers of bore holes, circular pockets, cylinders, studs, circular islands, etc. can be defined as datums.

Inside circle

The TNC automatically probes the inside wall in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- Position the touch probe approximately in the center of the circle.

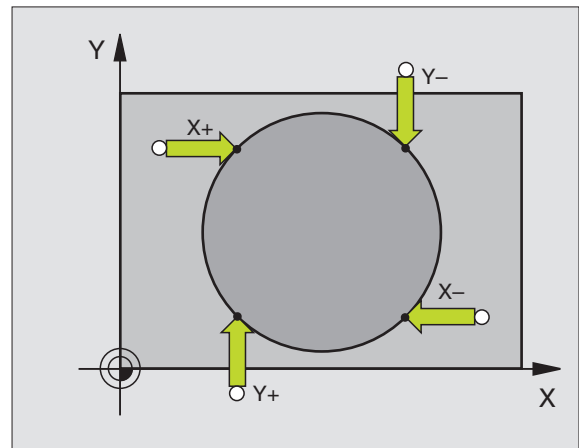
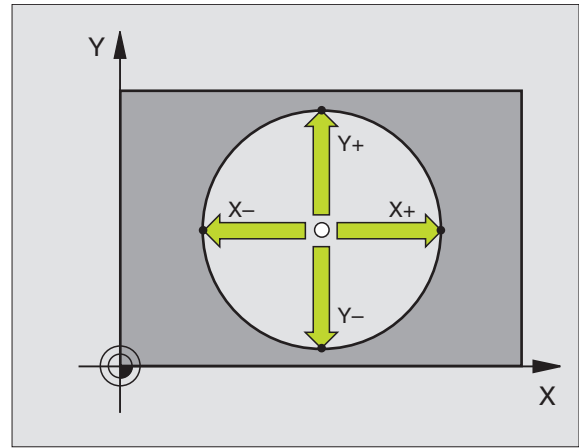


- Select the touch probe function: Press the PROBING CC soft key
- To probe the workpiece, press the machine START button four times. The touch probe touches four points on the inside of the circle.
- If you are probing to find the stylus center (only available on machines with spindle orientation, depending on MP6160), press the 180° soft key and probe another four points on the inside of the circle.
- If you are not probing to find the stylus center, press the END key.
- Datum: Enter both circle center coordinates into the menu window, and confirm your entry with ENT.
- To terminate the probe function, press the END key.

Outside circle

- Position the touch probe at the starting position for the first touch point outside of the circle.
- Select the probe direction with a soft key.
- To probe the workpiece, press the machine START button.
- Repeat the probing process for the remaining three points. See figure at lower right.
- Enter the datum coordinates and confirm your entry with ENT.

After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.



12.3 Measuring Workpieces with a 3-D Touch Probe

With a 3-D touch probe you can determine:

- position coordinates, and from them,
- dimensions and angles on the workpiece.

To find the coordinate of a position on an aligned workpiece:



- ▶ Select the touch probe function: Press the PROBING POS soft key
- ▶ Move the touch probe to a starting position near the first touch point
- ▶ Select the probe direction and axis of the coordinate. Use the arrow keys for selection.
- ▶ To probe the workpiece, press the machine START button.

The TNC shows the coordinates of the touch point as datum.

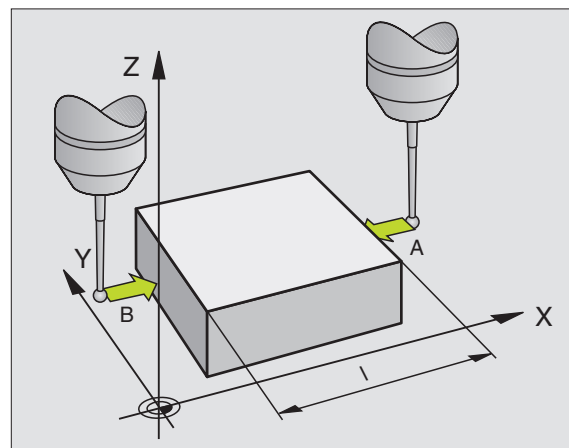
Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point as described under “Corner as datum.” The TNC displays the coordinates of the probed corner as datum.

Measuring workpiece dimensions



- ▶ Select the touch probe function: Press the PROBING POS soft key
- ▶ Position the touch probe at a starting position near the first touch point A.
- ▶ Select the probe direction with the arrow keys.
- ▶ To probe the workpiece, press the machine START button.
- ▶ If you will need the current datum later, write down the value that appears in the Datum display.
- ▶ Datum: Enter “0”.
- ▶ To terminate the dialog, press the END key.
- ▶ Select the touch probe function: Press the PROBING POS soft key



- ▶ Position the touch probe at a starting position near the second touch point B.
- ▶ Select the probe direction with the arrow keys: Same axis but opposite direction as for A.
- ▶ To probe the workpiece, press the machine START button.

The value displayed as DATUM is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

- ▶ Select the touch probe function: Press the PROBING POS soft key
- ▶ Probe the first touch point again.
- ▶ Set the Datum to the value that you wrote down previously.
- ▶ To terminate the dialog, press the END key.

Measuring angles

You can use the 3-D touch probe to measure angles in the working plane. You can measure

- the angle between the angle reference axis and a workpiece side, or
- the angle between two sides.

The measured angle is displayed as a value of maximum 90°.

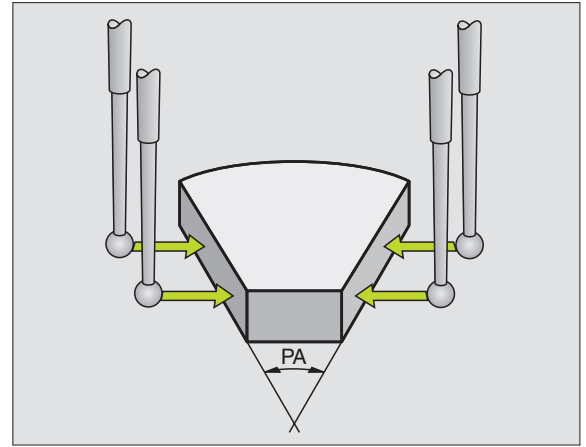
To find the angle between the angle reference axis and a side of the workpiece:



- ▶ Select the touch probe function: Press the PROBING ROT soft key
- ▶ Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- ▶ Make a basic rotation with the side of the workpiece (see "Compensating workpiece misalignment").
- ▶ Press the PROBING ROT soft key to display the angle between the angle reference axis and the side of the workpiece as the rotation angle.
- ▶ Cancel the basic rotation, or restore the previous basic rotation:
- ▶ This is done by setting the Rotation angle to the value that you wrote down previously.

To measure the angle between two workpiece sides:

- ▶ Select the touch probe function: Press the PROBING ROT soft key
- ▶ Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- ▶ Make a basic rotation for the first side (see "Compensating workpiece misalignment").
- ▶ Probe the second side as for a basic rotation, but do *not* set the Rotation angle to zero!
- ▶ Press the PROBING ROT soft key to display the angle PA between the two sides as the Rotation angle.
- ▶ Cancel the basic rotation, or restore the previous basic rotation by setting the Rotation angle to the value that you wrote down previously.

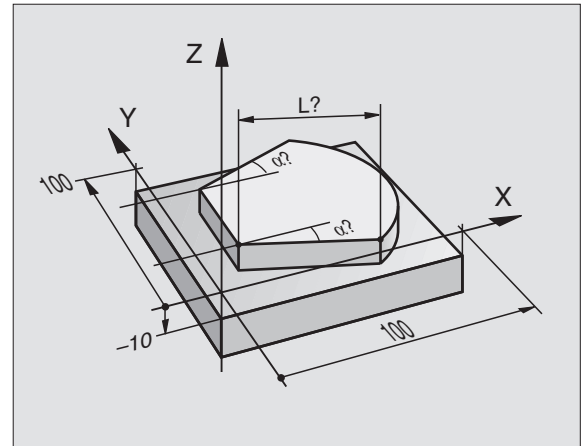


Measuring with the 3-D touch probe during program run

The 3-D touch probe can measure positions on the workpiece while the program is being run — even if the working plane is tilted. Applications:

- Measuring differences in the height of cast surfaces
- Tolerance checking during machining

To program the use of a touch probe, press the TOUCH PROBE key and REF PLANE soft key in the Programming and Editing mode of operation. The TNC pre-positions the touch probe to automatically probe the desired position. During probing, the TNC moves the touch probe parallel to the machine axis that was defined in the touch probe cycle. The TNC takes an active basic rotation or rotation only into account for calculating the touch point. The coordinate of the touch point is stored in a Q parameter. The TNC interrupts the probing process if the stylus is not deflected within a certain distance (selectable via MP 6130). Upon contact, the position coordinates of the south pole of the probe tip are also stored in the parameters Q115 to Q118. The values in these parameters do not include the stylus length and radius.





Pre-position the touch probe manually to avoid a collision when the programmed pre-positioning point is approached.

Be sure to use the tool data (length, radius, axis) either from the calibrated data or from the last TOOL CALL block. Selection is made with machine parameter MP7411.

- Press the TOUCH PROBE key in the Programming and Editing mode of operation.



- Select the touch probe function: Press the REF PLANE soft key
- Parameter number for result: Enter the number of the Q parameter to which you want to assign the coordinate.
- Probing axis/Probing direction: Enter the probing axis with the axis selection key and the algebraic sign for the probing direction. Confirm your entries with the ENT key.
- Coordinates: Use the axis selection keys to enter all coordinates of the nominal pre-positioning point values for the touch probe.
- To conclude input, press the ENT key.

Example NC blocks

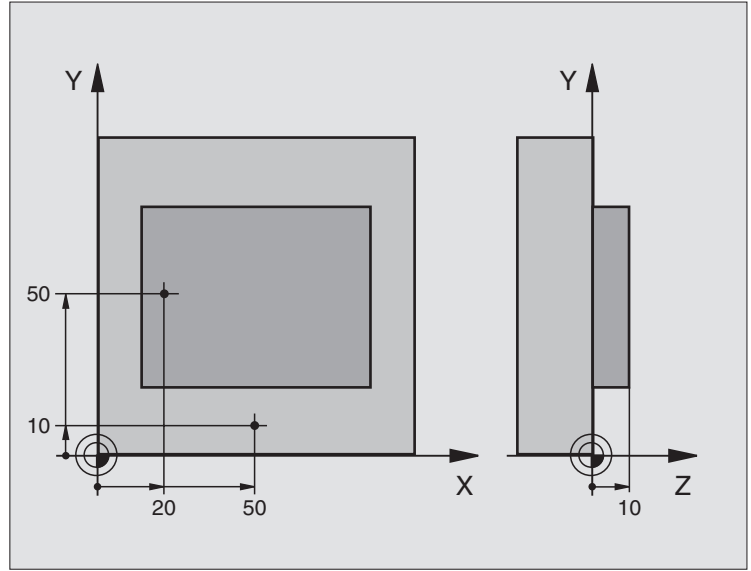
```
67 TCH PROBE 0.0 ref. plane Q5 X-
```

```
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

Example: Measuring the height of an island on a workpiece

Program sequence

- Assign the program parameters
- Measure the height with the REF PLANE cycle
- Calculate the height



BEGIN PGM 3DPROBE MM	
1 FN 0: Q11 = +20	1st touch point: X coordinate
2 FN 0: Q12 = +50	1st touch point: Y coordinate
3 FN 0: Q13 = +10	1st touch point: Z coordinate
4 FN 0: Q21 = +50	2nd touch point: X coordinate
5 FN 0: Q22 = +10	2nd touch point: Y coordinate
6 FN 0: Q23 = +0	2nd touch point: Z coordinate
7 TOOL CALL 0 Z	Call touch probe
8 L Z+250 R0 F MAX	Retract touch probe
9 L X+Q11 Y+Q12 R0 FMAX	Pre-position for first measurement
10 TCH PROBE 0.0 REF. PLANE Q10 Z-	Measure upper surface of the workpiece
11 TCH PROBE 0.1 Z+Q13	
12 L X+Q21 Y+Q22 R0 FMAX	Pre-position for second measurement
13 TCH PROBE 0.0 REF. PLANE Q20 Z-	Measure the depth
14 TCH PROBE 0.1 Z+Q23	
15 FN 2: Q1 = +Q20 - +Q10	Calculate the absolute height of the island
16 STOP	Interrupt program run: check Q1
17 L Z+250 R0 FMAX M2	Retract the tool, end of program
END PGM 3DPROBE MM	



13

Digitizing

13.1 Digitizing with a Triggering Touch Probe (Optional)

The digitizing option enables you to reduce a three-dimensional part into discrete digital information by scanning it with a touch probe.

The following components are required for digitizing:

- Touch probe
- "Digitizing option" software module in the TNC
- SUSA evaluation software from HEIDENHAIN for further processing of the data digitized in the MEANDER cycle.

The following cycles are available for digitizing:

- cycle
- MEANDER
- CONTOUR LINES



The TNC and the machine tool must be specially prepared by the machine tool builder for the use of a touch probe.

The touch probe must be calibrated before digitizing.

Function

The touch probe scans a 3-D surface point for point in a selectable grid. The scanning speeds vary from 200 to 800 mm/min with a probe point interval (PP.INT) of 1 mm (machine-dependent value).

The TNC reads out the digitized positions via data interface, usually to a PC. You need to configure your data interface accordingly (see „14.4 Setting the External Data Interface“).

If you want to mill the digitized data subsequently using a tool whose radius equals the radius of the ball tip, you can machine the digitized data directly from the file without any further evaluation.



Digitizing cycles are programmed only for the main axes X, Y and Z.

Coordinate transformations or a basic rotation must not be active during digitizing.

The TNC also transfers the BLK FORM to the digitizing data file.

13.2 Programming Digitizing Cycles

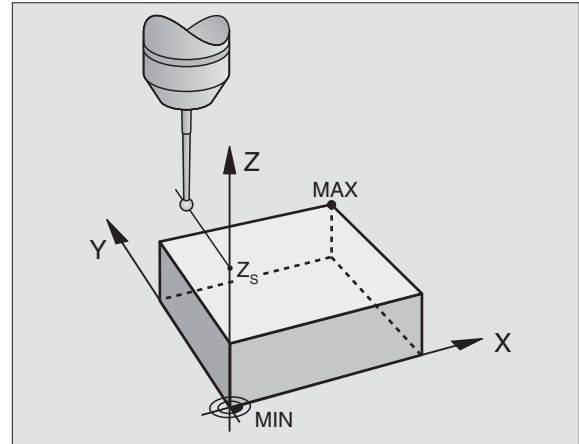
- ▶ Press the TOUCH PROBE key.
- ▶ Use soft keys to select the desired digitizing cycle
- ▶ Answer the dialog questions on the TNC screen: Enter the appropriate values with the keyboard and confirm each input with the ENT key. When all the required data have been entered, the TNC terminates the cycle definition automatically. For information on the individual input parameters refer to the cycle descriptions in this chapter.

Defining the digitizing range

Cycle 5 RANGE is available for defining the digitizing range. This cycle defines a cuboid range within which the touch probe scans the contour.

Similar to defining the workpiece blank, the digitizing range is programmed by entering the MIN and MAX point coordinates of the three main axes X, Y and Z (see figure at right).

- ▶ PGM Name digitizing data: Enter the name of the file in which the digitizing data is to be stored.
- ▶ TCH PROBE axis: Enter the touch probe axis.
- ▶ Min. point range: Lowest coordinates in the range to be digitized.
- ▶ Max. point range: Highest coordinates in the range to be digitized.
- ▶ Clearance height: Position in probe axis at which the stylus cannot collide with the model.



Example NC blocks

```

50 TCH PROBE 5.0 RANGE
51 TCH PROBE 5.1 PGM NAME: DATA
52 TCH PROBE 5.2 Z X+0 Y+0 Z+0
53 TCH PROBE 5.3 X+10 Y+10 Z+20
54 TCH PROBE 5.4 HEIGHT: + 100

```


13.3 Meander Digitizing

Digitizing cycle 6 MEANDER

The MEANDER cycle scans and digitizes a 3-D surface in a back-and-forth series of parallel lines. This method is best suited for digitizing relatively flat surfaces. If you want to use the SUSA evaluation software to further process the data, you must use the MEANDER cycle.

Before starting the digitizing process, select an axis in the working plane. During digitizing, the touch probe moves in the positive direction of the selected axis, starting at the MIN point of the working plane. When it reaches the range limit, it moves by the line spacing in the positive direction of the other axis and then travels back along this line. At the other end of this line, the touch probe moves once again by the line spacing. This process is repeated until the entire range has been scanned.

Once the entire range has been scanned, the touch probe returns to the clearance height.

Starting position

- MIN point coordinates from Cycle 5 RANGE in the working plane, spindle axis coordinate = clearance height.
- Automatically move to the starting position:
First in the probe axis to clearance height, then in the working plane.

Contour approach

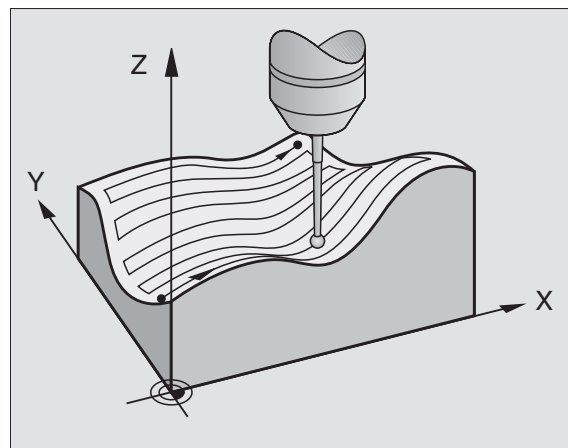
The touch probe moves in the negative probe axis direction toward the model. When it makes contact, the TNC stores the position coordinates.



Before the MEANDER cycle, the part program must have a range defined in the RANGE cycle.

Digitizing parameters

- ▶ Line direction: Coordinate axis in whose positive direction the touch probe moves from the first contour point.
- ▶ Limit in normal lines direction: Distance by which the touch probe lifts off the model after each deflection of the stylus during scanning. Input range: 0 to 5 mm. Recommended input value: Enter an input value between 50% and 100% of the probe point interval. The smaller the ball-tip radius, the larger the Limit in Normal lines direction should be programmed.
- ▶ Line spacing: The offset by which the probe moves in the probe axis at the end of each line before scanning the next line. Input range: 0 to 5 mm
- ▶ Max. probe point interval: maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners. Input range: 0.02 to 5 mm



Example NC blocks

```
60 TCH PROBE 6.0 MEANDER
61 TCH PROBE 6.1 DIRECTN: X
62 TCH PROBE 6.2 TRAVEL: 0.5 L.SPAC: 0.2
   PP.INT: 0.5
```

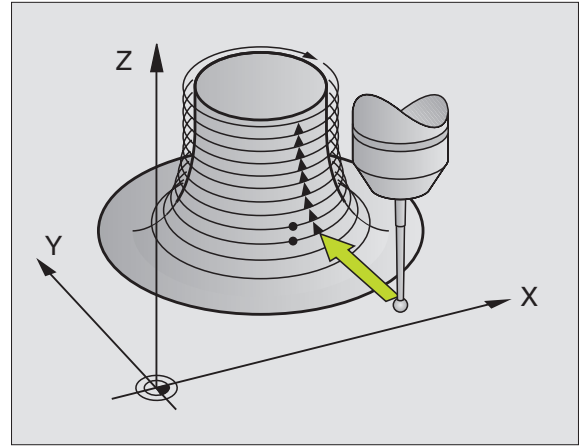
13.4 Contour Line Digitizing

Digitizing Cycle 7 CONTOUR LINES

The CONTOUR LINES cycle scans a 3-D contour by circling around the model in a series of upwardly successive levels. This cycle is particularly useful for digitizing steep contours, such as the gating of injection molding dies, or for digitizing a single contour line, such as the contour of a cam disk.

After first making contact with the form, the probe starts scanning at constant height around the form. After it has orbited the form and returned to the first probe point, it then moves in the positive or negative direction of the probe axis by the value of the LINE SPACING. It then once again orbits the form at constant height. This process is repeated until the entire range has been scanned.

Once the entire range has been scanned, the touch probe returns to the clearance height and the programmed starting point.



Requirements for the scanning range

- In the touch probe axis: The defined RANGE must be lower than the highest point of the 3-D model by at least the radius of the probe tip.
- In the plane perpendicular to the touch probe: The defined RANGE must be larger than the 3-D model by at least the radius of the probe tip.

Starting position

- Probe axis coordinate of the MIN point from Cycle 5 RANGE if the line spacing is entered as a positive value.
- Probe axis coordinate of the MAX point from Cycle 5 RANGE if the line spacing is entered as a negative value.
- Define the working plane coordinates in the CONTOUR LINES cycle.
- Automatically move to the starting position:
First in the probe axis to clearance height, then in the working plane.

Contour approach

The touch probe moves toward the surface in the programmed direction. When it makes contact, the TNC stores the position coordinates.



Before the CONTOUR LINES cycle, the part program must have a range defined in the RANGE cycle.

Digitizing parameters

- ▶ Time limit: Time within which the touch probe must orbit the model and reach the first touch point. If the entered time limit is exceeded, the TNC interrupts the digitizing process. Input range: 0 to 7200 seconds. The input value 0 means there is no time limit.
- ▶ Starting point: Coordinates of the starting position in the working plane.
- ▶ Axis and direction of approach: Coordinate axis and direction in which the touch probe approaches the model.
- ▶ Starting probe axis and direction: Coordinate axis and direction in which the probe begins scanning the model. Defining the direction at the same time determines the whether the subsequent machining operation is performed by up-cut or climb milling.
- ▶ Limit in normal lines direction: Distance by which the touch probe lifts of the model after each deflection of the stylus during scanning. Input range: 0 to 5 mm. Recommended input value: Enter an input value between 50% and 100% of the probe point interval. The smaller the ball-tip radius, the larger the Limit in Normal lines direction should be programmed.
- ▶ Line spacing and direction: The offset by which the touch probe moves to start a new contour line after completing the previous one. The algebraic sign determines the direction in which the probe is offset. Input range: -5 to +5 mm



If you only want to digitize a single contour line, enter zero for the line spacing.

- ▶ Max. probe point interval: maximum spacing between consecutive digitized positions. The TNC also accounts for important points that define the contour of the model, for example at inside corners. Input range: 0.02 to 5 mm

Example NC blocks

```
60 TCH PROBE 7.0 CONTOUR LINES
```

```
61 TCH PROBE 7.1 TIME: 0 X+0 Y+0
```

```
62 TCH PROBE 7.2 ORDER: Y- / X-
```

```
63 TCH PROBE 7.2 TRAVEL: 0.5 L.SPAC+: +0.2
```

```
PP.INT: 0.5
```

13.5 Using Digitized Data in a Part Program

Resulting NC blocks of a file containing data that were digitized with the CONTOUR LINES cycle.

BEGIN PGM DATA MM	Program name DATA is entered in the RANGE cycle
1 BLK FORM 0.1 Z X-40 Y-20 Z+0	Define workpiece blank: The size is determined by the TNC
2 BLK FORM 0.2 X+40 Y+40 Z+25	
3 L Z+250 FMAX	Clearance height in Z is entered in the RANGE cycle
4 L X+0 Y-25 FMAX	Starting point in X/Y is entered in the CONTOUR LINES cycle
5 L Z+25	Starting height in Z is entered in the CONTOUR LINES cycle and depends on the sign of the LINE SPACING
6 L X+0.002 Y-12.358	First digitized position
7 L X+0.359 Y-12.021	Second digitized position
...	
253 L X+0.003 Y-12.390	A contour line has been completed: probe has returned to the first digitized position
254 L Z+24.5	Move to next contour line
...	
2597 L X+0.093 Y-16.390	Last digitized position in the range
2598 L X+0 Y-25 FMAX	Back to the starting point in X/Y
2599 L Z+250 FMAX	Back to the clearance height in Z
END PGM DATA MM	End of program

To execute the digitized data, create the following program:

BEGIN PGM MILLING MM	Tool definition: tool radius = stylus radius
1 TOOL DEF 1 L+0 R+4	Tool call
2 TOOL CALL 1 Z S4000	Define feed rate for milling, spindle and coolant ON
3 L RO F1500 M13	Call up digitized data that are stored externally
4 CALL PGM EXT:DATA	
END PGM MILLING MM	



14

MOD Functions

14.1 Selecting, Changing and Exiting the MOD Functions

The MOD functions provide additional displays and input possibilities.

To select the MOD functions

Call the mode of operation in which you wish to change the MOD function.



- ▶ To select the MOD functions, press the MOD key. The figure at upper right shows the „MOD screen“.

You can make the following changes:

- Select position display
- Unit of measurement (mm/inches)
- Programming language for MDI
- Enter code number
- Set data interface
- Machine-Specific User Parameters
- Axis traverse limits
- Display NC software number
- Display PLC software number

To change the MOD functions:

- ▶ Select the desired MOD function in the displayed menu with the arrow keys.
- ▶ Press the ENT key repeatedly until the desired function is highlighted, or enter the appropriate numbers and confirm your entry with ENT.

To exit the MOD functions:


- ▶ Close the MOD functions with the END soft key or key.

14.2 System Information

You can use the soft key INFO SYSTEM to display the following information:

- Free program memory
- NC software number
- PLC software number
- DSP software number
- Options (if present), e.g. digitizing

are displayed on the TNC screen after the functions have been selected.

Programming and editing									
Position display 1 ACTL.									
Position display 2 DIST.									
Change MM/INCH MM									
Program input HEIDENHAIN									
ACTL. X +0.195									
Y -11.000									
Z +136.000									
					T				
					F 0				
					S 1000 M5/9				
		RS 232 SETUP		USER PARAMETER		TRAVERSE RANGE		INFO SYSTEM	
								HELP	
								END	

14.3 Code Number

To enter the code number, press the soft key with the key symbol.
The TNC requires a code number for the following functions:

Function	Code number
Selecting user parameters	123
Canceling program protection	86357
Operating hours counter for:	
Control On	
Program Run	
Spindle On	857282

You can reset the individual times by pressing the ENT key (must be enabled by machine parameter)

14.4 Setting the Data Interface

Press the soft key marked RS 232 SETUP to call a menu for setting the data interfaces:

Setting the OPERATING MODE of the external device

External device	RS232 INTERFACE
HEIDENHAIN floppy disk unit FE 401 and FE 401B	FE
Non-HEIDENHAIN devices such as tape punchers, PC without TNCremo	EXT1, EXT2
PC with HEIDENHAIN software TNCremo	FE
No data transfer; e.g. digitizing without position value capture, or working without an external device	NONE

Setting the BAUD RATE

You can set the BAUD-RATE (data transfer speed) from 110 to 115,200 baud. For each operating mode (FE, EXT1 etc.), the TNC stores and individual BAUD RATE.

Programming and editing							
RS232 interface							
FE							
Baud rate							
57600							
Memory for blockwise transfer							
Available [KB]							
90							
Reserved [KB]							
20							
Block buffer							
1000							

Creating the memory for blockwise transfer

In order to be able to edit other programs while blockwise execution is in progress, you need to create a memory for blockwise transfer.

The TNC shows the available free memory space. The reserved memory space should be less than the total free memory space available.

Setting the block buffer

To ensure a continuous program run during blockwise transfer, the TNC needs a certain quantity of blocks stored in program memory.

In the block buffer you define how many NC blocks are read in through the data interface before the TNC begins the program run. The input value for the block buffer depends on the point intervals in the part program. For very small point intervals, enter a large block buffer. For large point intervals, enter a small block buffer. Proposed value: 1000

Software for data transfer

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremo data transfer software. With TNCremo, data transfer is possible with all HEIDENHAIN controls via serial interface.



Please contact your HEIDENHAIN agent if you would like to receive the TNCremo data transfer software for a nominal fee.

System requirements for TNCremo

- AT personal computer or compatible system
- 640 KB working memory
- 1 MB free memory space on your hard disk
- One free serial interface
- Operating system MS-DOS/PC-DOS 3.00 or later, Windows 3.1 or later, OS/2
- A Microsoft-compatible mouse (for ease of operation, not essential)

Installation under Windows

- ▶ Start the SETUP.EXE installation program in the file manager (explorer)
- ▶ Follow the instructions of the setup program

Starting TNCremo under Windows

Windows 3.1, 3.11, NT:

- Doubleclick on the icon in the program group HEIDENHAIN Applications

Windows95:

- Click on <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremo>

When you start TNCremo for the first time, you will be asked for the type of control you have connected, the interface (COM1 or COM2) and the data transfer speed. Enter the necessary information.

Data transfer between the TNC 410 and TNCremo

Ensure that:

- The TNC 410 is connected to the correct serial interface of your computer
- The data transfer speed set on the TNC is the same as that set on TNCremo

Once you have started TNCremo, you will see a list of all of the files that are stored in the active directory on the left of the window. Using the menu items <Directory>, <Change>, you can change the active directory or select another directory. To start data transfer from the TNC (see „4.2 File Management“), select <Connect>, <File server>. TNCremo is now ready to receive data.

End TNCremo

Select the menu items <File>, <Exit>, or press the key combination ALT+X



Refer also to the TNCremo help texts where all of the functions are explained in more detail.

14.5 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 user parameters. Refer to your machine manual.

14.6 Position Display Types

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:

- 1 Starting position
- 2 Target position of the tool
- 3 Workpiece datum
- 4 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 as referenced 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	LAG

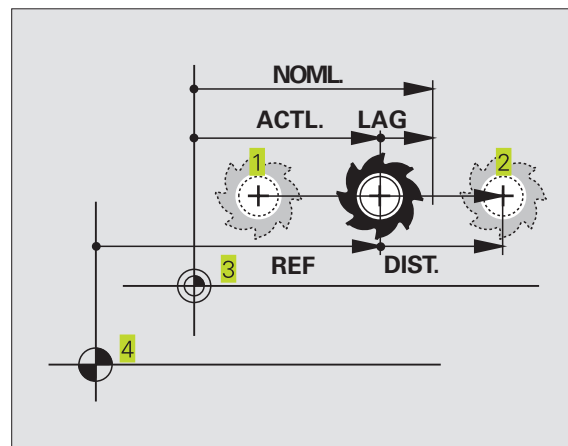
With the MOD function Position display 1 you can select the position display in the status display. With Position display 2 you can select the position display in the additional status display.

14.7 Unit of Measurement

The MOD function Change MM/INCHES determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm) set the Change MM/INCH function to mm. The value is displayed with 3 digits after the decimal point.
- To select the inch system (e.g. X = 0.6216 inch) set the Change MM/inches function to inches. The value is displayed to 4 decimal places.

This MOD function also determines the unit of measurement when you open a new program.



14.8 Select the Programming Language

The PROGRAM INPUT MOD function lets you decide whether to program the MDI file in HEIDENHAIN conversational dialog or in ISO format.

- For conversational programming, set to HEIDENHAIN
- For ISO programming, set to ISO

This MOD function also determines the programming language when you open a new program.



If you wish to switch between conversational and ISO input (and back again), you need to delete the last active \$MDI file in the Programming mode.

14.9 Enter Axis Traverse Limits

The AXIS LIMIT mod function allows you to set limits to axis traverse within the machine's actual working envelope.

Possible application:

to protect an indexing fixture against tool collision.

Axis Traverse Limits for Program Run

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the MOD function TRAVERSE RANGE MACHINE. With this function, you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum.

Working without additional traverse limits

To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 30000 mm) as the traverse range.

To find and enter the maximum traverse:

- ▶ Select REF position display
- ▶ 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.



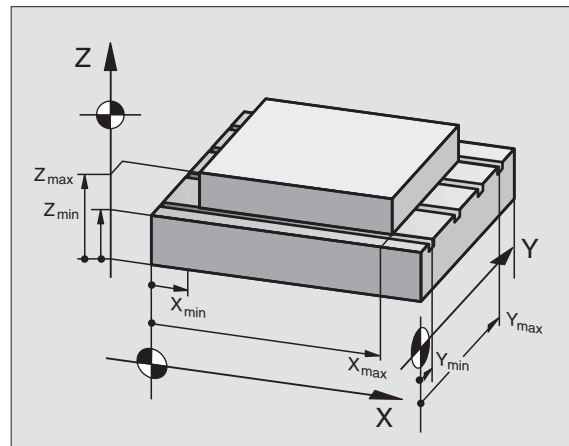
- ▶ Enter the limits for axis traverse: Press the TRAVERSE RANGE MACHINE soft key. Enter the noted values for the axes as limits, confirming each value with the ENT key.

- ▶ To exit the MOD function, press the END key.



The tool radius is not automatically compensated in the axis traverse limit value.

The traverse range limits and software limit switches become active as soon as the reference points are traversed.



Axis Traverse Limits for Test Run

It is possible to define a separate „traverse range“ for the Test Run and the programming graphics. Press the soft key TRAVERSE RANGE TEST, after you have activated the MOD function. Then enter the desired values and confirm each value with the ENT key.

In addition to the axis traverse limits, you can also define the position of the workpiece datum referenced to the machine datum.

14.10 The HELP Function



The HELP function is not available on every machine. Refer to your machine tool builder for more information on the HELP function.

The HELP function 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 and executed in a help file.

Selecting and executing a HELP function

- ▶ Select the MOD function with the MOD key.



- ▶ Select the HELP function with the HELP soft key
- ▶ Use the up and down arrow keys to select a line in the HELP file, which is marked with an #
- ▶ Use the NC start key to execute the selected HELP function



15

Tables and Overviews

15.1 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 typical 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 (for bit-coded machine parameters)**
Enter a percent sign (%) before the number
- **Hexadecimal numbers (for bit-coded machine parameters)**
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 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

Determining the control character for blockwise 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
1 1/2 stop bits: +0
2 stop bits: +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**: $1+0+8+0+32+64 = 105$

Integrating TNC interfaces EXT1 (5030.0) and EXT2 (5030.1) to external device

MP5030.x
Standard transmission: 0
Interface for blockwise transfer: 1

3-D touch probes and digitizing

Select signal transmission	<div>MP6010</div> <div>Touch probe with cable transmission: 0</div> <div>Touch probe with infrared transmission: 1</div>
Probing feed rate for triggering touch probes	<div>MP6120</div> <div>80 to 3000 [mm/min]</div>
Maximum traverse to first probe point	<div>MP6130</div> <div>0.001 to 30 000 [mm]</div>
Safety clearance to probing point during automatic measurement	<div>MP6140</div> <div>0.001 to 30 000 [mm]</div>
Rapid traverse for triggering touch probes	<div>MP6150</div> <div>1 to 300 000 [mm/min]</div>
Measure center misalignment of the stylus when calibrating a triggering touch probe	<div>MP6160</div> <div>No 180° rotation of the 3-D touch probe during calibration: 0</div> <div>M function for 180° rotation of the 3-D touch probe during calibration: 1 to 88</div>
Radius measurement with the TT 120 touch probe: Probing direction	<div>MP6505</div> <div>Positive probing direction in the angle reference axis (0° axis): 0</div> <div>Positive probing direction in the +90° axis: 1</div> <div>Negative probing direction in the angle reference axis (0° axis): 2</div> <div>Negative probing direction in the +90° axis: 3</div>
Probing feed rate for second measurement with TT 120, stylus shape, corrections in TOOL.T	<div>MP6507</div> <div>Calculate feed rate for second measurement with TT 120, with constant tolerance: +0</div> <div>Calculate feed rate for second measurement with TT 120, with variable tolerance: +1</div> <div>Constant feed rate for second measurement with TT 120: +2</div>
Maximum permissible measuring error with TT 120 during measurement with rotating tool	<div>MP6510</div> <div>Required for calculating the probing feed rate in connection with MP6570</div> <div>0.002 to 0.999 [mm] (recommended input value: 0.005 mm)</div>
Feed rate for probing a stationary tool with the TT 120	<div>MP6520</div> <div>80 to 3000 [mm/min]</div>

Radius measurement with the TT 120: Distance from lower edge of tool to upper edge of stylus	MP6530 0.001 to 30 000.000 [mm]
Set-up clearance in the tool axis above the stylus of the TT 120 for pre-positioning	MP6540.0 0.001 to 30 000.000 [mm]
Clearance zone in the machining plane around the stylus of the TT 120 for pre-positioning	MP6540.1 0.001 to 30 000.000 [mm]
Rapid traverse for TT 120 in the probe cycle	MP6550 10 to 20 000 [mm/min]
M function for spindle orientation when measuring individual teeth	MP6560 -1 to 88
Measuring rotating tools: Permissible rotational speed at the circumference of the milling tool Required for calculating rpm and probe feed rate	MP6570 40.000 to 120.000 [m/min]
REF coordinates of the TT-120 probe contact center point	MP6580.0 X axis: -30 000.000 to 30 000.000 MP6580.1 Y axis: -30 000.000 to 30 000.000 MP6580.2 Z axis: -30 000.000 to 30 000.000

TNC displays, TNC editor

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 250
Dialog language	MP7230 German: 0 English: 1
Configure tool tables	MP7260 Inactive: 0 Number of tools per tool table: 1 to 254
Configure pocket tables	MP7261 Inactive: 0 Number of pockets in the pocket table: 1 to 254

Configure tool table (To omit from the table: enter 0);
Column number in the tool table for

MP7266.0	Tool name – NAME: 0 to 22
MP7266.1	Tool length – L: 0 to 22
MP7266.2	Tool radius – R: 0 to 22
MP7266.3	Reserved
MP7266.4	Oversize length – DL: 0 to 22
MP7266.5	Oversize radius – DR: 0 to 22
MP7266.6	Reserved
MP7266.7	Tool locked – TL: 0 to 22
MP7266.8	Replacement tool – RT: 0 to 22
MP7266.9	Maximum tool life — TIME1: 0 to 22
MP7266.10	Maximum tool life for TOOL CALL — TIME2: 0 to 22
MP7266.11	Current tool life — CUR. TIME: 0 to 22
MP7266.12	Tool comment – DOC: 0 to 22
MP7266.13	Number of teeth – CUT.: 0 to 22
MP7266.14	Tolerance for wear detection in tool length – LTOL: 0 to 22
MP7266.15	Tolerance for wear detection in tool radius – RTOL: 0 to 22
MP7266.16	Cutting direction – DIRECT.: 0 to 22
MP7266.17	PLC status – PLC: 0 to 22
MP7266.18	Offset of the tool in the tool axis in addition to MP6530 _ TT:L-OFFS: 0 to 22
MP7266.19	Offset of the tool between stylus center and tool center — TT:R-OFFS: 0 to 22
MP7266.20	Tolerance for break detection in tool length — LBREAK.: 0 to 22
MP7266.21	Tolerance for break detection in tool radius — RBREAK: 0 to 22

Configure pocket tables (to omit from table enter 0); Column number in the tool table for	MP7267.0 Tool number — T: 0 to 5
	MP7267.1 Special tool — ST: 0 to 5
	MP7267.2 Fixed pocket — F: 0 to 5
	MP7267.3 Pocket locked — L: 0 to 5
	MP7267.4 PLC status — PLC: 0 to 5
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 of the „slowest“ axis): +1 Spindle speed S and miscellaneous function M effective after STOP: +0 Spindle speed S and miscellaneous function M no longer effective after STOP: +2
Display of gear range	MP7274 Do not display current gear range:0 Display current gear range: 1
Decimal character	MP7280 The decimal character is a comma: 0 The decimal character is a point: 1
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

Display step for the X axis	<div>MP7290.0</div> <div>0.1 mm or 0.1°: 0</div> <div>0.05 mm or 0.05°: 1</div> <div>0.01 mm or 0.01°: 2</div> <div>0,005 mm or 0.005°: 3</div> <div>0.001 mm or 0.001°: 4</div>
Display step for the Y axis	<div>MP7290.1</div> <div>0.1 mm or 0.1°: 0</div> <div>0.05 mm or 0.05°: 1</div> <div>0.01 mm or 0.01°: 2</div> <div>0.005 mm or 0.005°: 3</div> <div>0.001 mm or 0.001°: 4</div>
Display step for the Z axis	<div>MP7290.2</div> <div>0.1 mm or 0.1°: 0</div> <div>0.05 mm or 0.05°: 1</div> <div>0.01 mm or 0.01°: 2</div> <div>0.005 mm or 0.005°: 3</div> <div>0.001 mm or 0.001°: 4</div>
Display step for the IVth axis	<div>MP7290.3</div> <div>0.1 mm or 0.1°: 0</div> <div>0.05 mm or 0.05°: 1</div> <div>0.01 mm or 0.01°: 2</div> <div>0.005 mm or 0.005°: 3</div> <div>0.001 mm or 0.001°: 4</div>
Disable datum setting	<div>MP7295</div> <div>Do not disable datum setting: +0</div> <div>Disable datum setting in the X axis: +1</div> <div>Disable datum setting in the Y axis: +2</div> <div>Disable datum setting in the Z axis: +4</div> <div>Disable datum setting in the IV axis: +8</div> <div>Disable datum setting in the V axis: +16</div> <div>Disable datum setting in the 6th axis: +32</div> <div>Disable datum setting in the 7th axis: +64</div> <div>Disable datum setting in the 8th axis: +128</div> <div>Disable datum setting in the 9th axis: +256 Lock axis: +8</div>
Disable datum setting with the orange axis keys	<div>MP7296</div> <div>Do not inhibit datum setting: 0</div> <div>Disable datum setting with the orange axis keys: 1</div>

Reset status display, Q parameters and tool data at program end	MP7300 Do not delete status display: +0 Delete status display: +1 Delete Q parameter: +0 Do not delete Q parameter: +2 Delete tool number, axis and data: +0 Do not delete tool number, axis and data: +4
Graphic display mode	MP7310 Projection in three planes according to ISO 6433, projection method 1: +0 Projection in three planes according to ISO 6433, projection method 2: +1 Do not rotate coordinate system for graphic display: +0 Rotate coordinate system for graphic display by 90°: +2 Simulation for fixed cycles, draw only the last infeed: +0 Simulation for fixed cycles, draw all infeeds: +16
Programming graphic display mode	MP7311 Do not show penetration points as circle: +0 Show penetration points as circle: +1 Do not show meander paths in cycles: +0 Show meander paths in cycles: +2 Do not show compensated paths: +0 Show compensated paths: +3

Machining and program run

Cycle 17: Oriented spindle stop at beginning of cycle

MP7160
Oriented spindle stop: **0**
No oriented spindle stop: **1**

Effect of Cycle 11 SCALING FACTOR

MP7410
SCALING FACTOR effective in 3 axes: **0**
SCALING FACTOR effective in the working plane only: **1**

Tool data in programmable probe cycleTOUCH-PROBE 0

MP7411
Overwrite current tool data by the calibrated data from the 3-D touch probe system: **0**
Current tool data are retained: **1**

Contour element connection mode for milling

MP7415.0
Insert rounding arc: **0**
Insert third-degree polynomial (cubic spline, curve without sudden change in velocity): **1**
Insert fifth-degree polynomial (curve without sudden change in acceleration): **2**
Seventh-degree polynomial (curve without sudden change in acceleration rate of change): **3**

Settings for contour milling

MP7415.1
Do not smoothen the contours: **+0**
Smoothen the contours: **+1**
Do not smoothen the velocity profile if a short straight-line segment lies between contour transitions: **+0**
Smoothen the velocity profile if a short straight-line segment lies between contour transitions: **+2**

SL cycles, machining sequence

MP7420.0

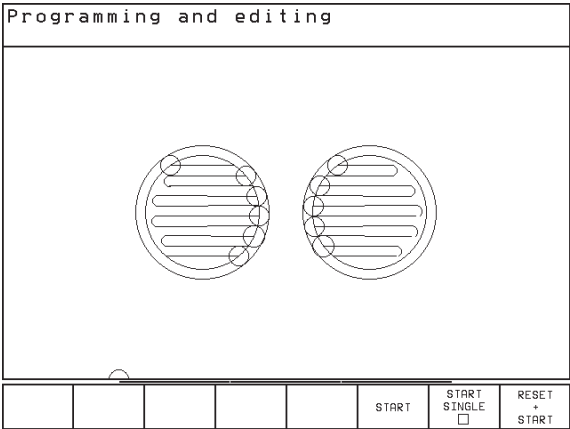
Mill channel around the contour — clockwise for islands and counterclockwise for pockets: **+0**
Mill channel around the contour — clockwise for pockets and counterclockwise for islands: **+1**
First mill the channel, then rough out the contour: **+0**
First rough out the contour, then mill the channel: **+2**
Combine compensated contours: **+0**
Combine uncompensated contours: **+4**
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 applies to the Cycles 6, 15, 16:
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 spindle axis only: **+16**

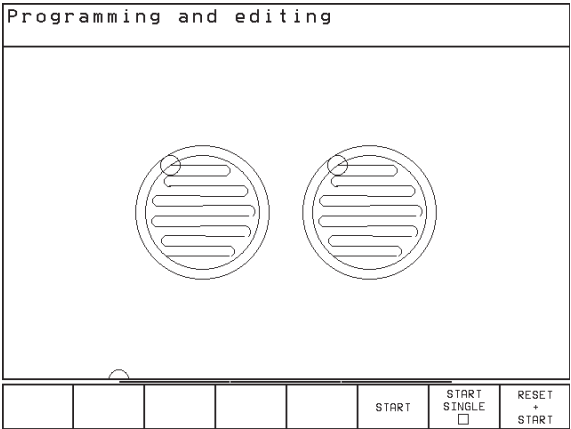
SL cycles, machining sequence

MP7420.1

Rough-out separate areas together, plunging with every pass: **+0**
Rough-out separate areas separately, plunging only once for each area **+1**
Bit 1 to bit 7: reserved



MP7420.1 = 0
(small circles = penetration)



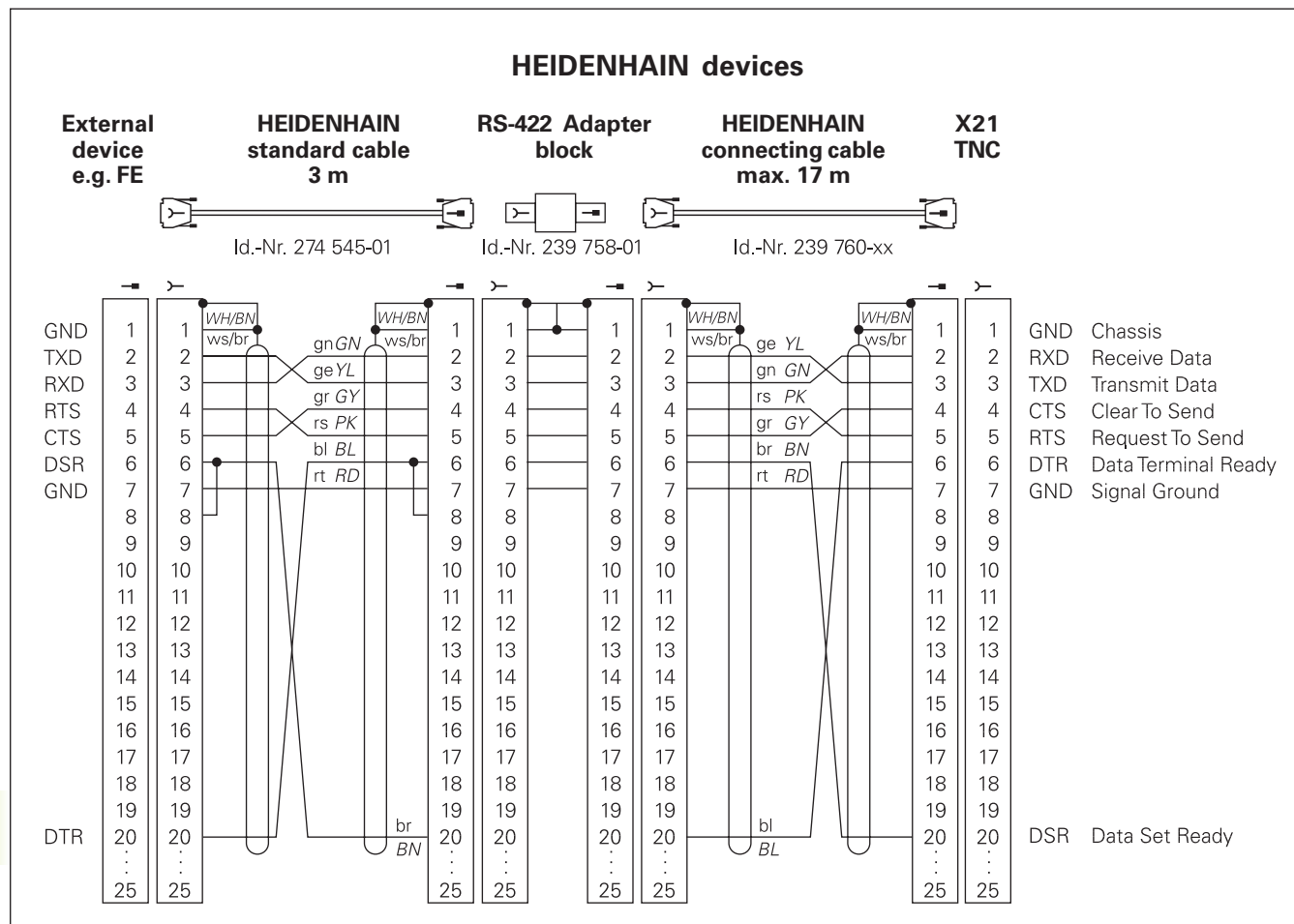
MP7420.1 = 1

Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING: Overlap factor	
MP7430	
0.1 to 1.414	
Behavior of M functions	
MP7440	
Program stop with M06: +0	
No program stop with M06: +1	
No cycle call with M89: +0	
Cycle call with M89: +2	
Program stop for M functions: +0	
No program stop for M functions: +4	
Feed rate in tool axis with M103 F.	
Reduction not effective: +0	
Feed rate in tool axis with M103 F.	
Reduction effective: +16	
Do not set axis-in-position marker for waiting time between two NC blocks: +0	
Set axis-in-position marker for waiting time between two NC blocks: +32	
Maximum permissible angle of directional change for constant contouring speed (effective for corners with R0 and for all inside corners).	
This feature works both during operation with servo lag as well as with velocity feedforward.	
MP7460	
0.000 to 179.999 [°]	
Maximum contouring speed at a feed rate override setting of 100% in the program run modes	
MP7470	
0 to 99 999 [mm/min]	
Datums from a datum table are referenced to the	
MP7475	
Workpiece datum: +0	
Machine datum: +1	
Electronic handwheels	
Handwheel type	
MP7640	
Machine without handwheel: 0	
Multiple handwheels with supplementary buttons:5	
HR 410 portable handwheel with additional functions: 6	
Handwheel functions	
MP7641	
Interpolation factor can be entered with the keyboard: +0	
Define interpolation factor via PLC module:+1	
Handwheel not active in the Programming and Editing mode: +0	
Handwheel active in the Programming and Editing mode: +2	

15.2 Pin Layout and Connecting Cable for the Data Interface

RS-232-C/V.24 Interface

HEIDENHAIN devices

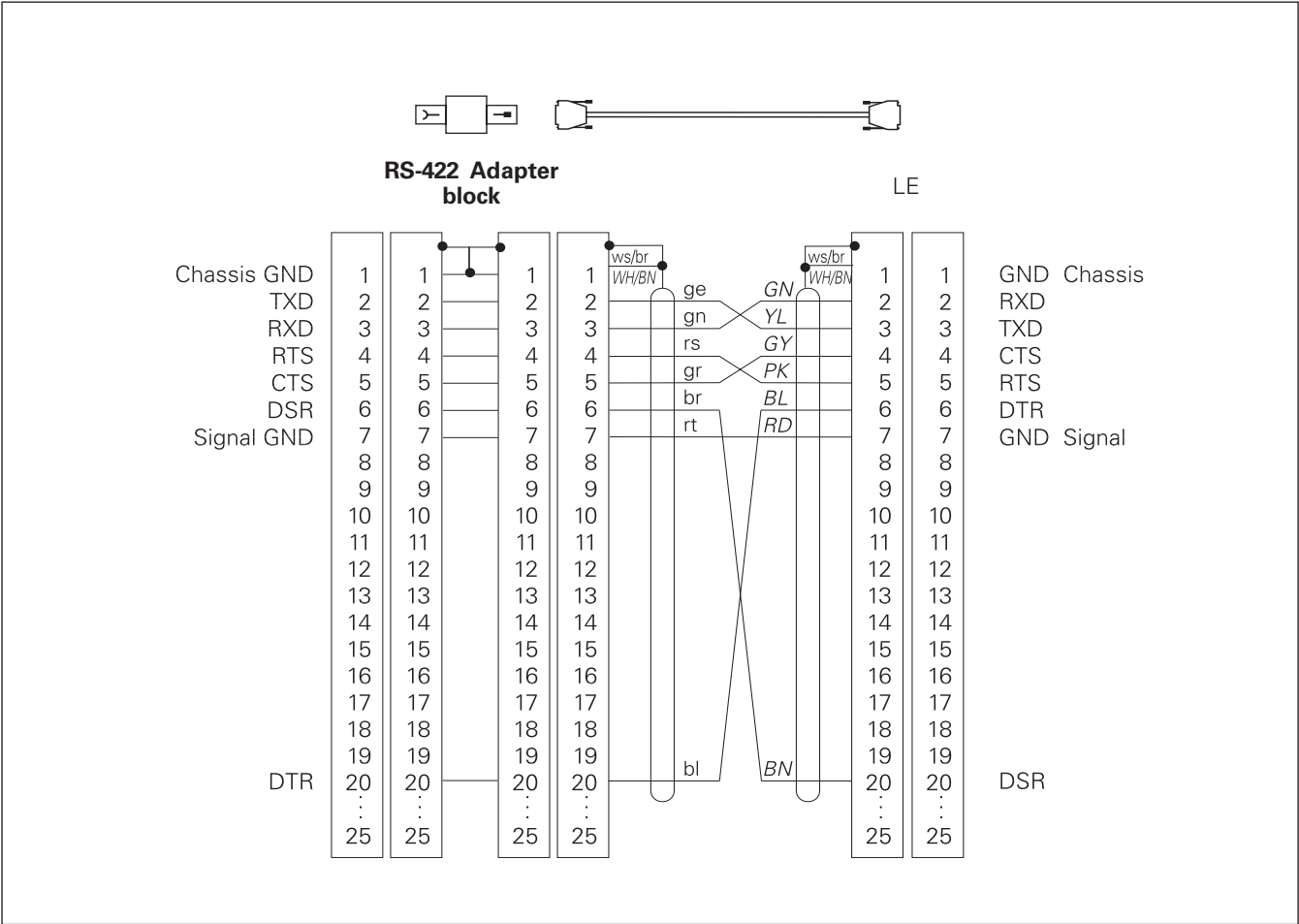


The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

Non-HEIDENHAIN devices

The connector pin layout of a non-HEIDENHAIN device may differ considerably from that of a HEIDENHAIN device.

This often depends on the unit and type of data transfer. The figure below shows the connector pin layout on the adapter block.



15.3 Technical Information

TNC features

Description	Contouring control for machines with up to 4 axes; with oriented spindle stop.
Components	<ul style="list-style-type: none">■ Logic unit■ Keyboard■ Color visual display unit with soft keys
Data interface	<ul style="list-style-type: none">■ RS-232-C /V.24
Simultaneous axis control for contour elements	<ul style="list-style-type: none">■ Straight lines: up to 3 axes■ Circles: up to 2 axes■ Helixes: 3 axes
Look Ahead	<ul style="list-style-type: none">■ Defined rounding of discontinuous contour transitions (such as for 3-D surfaces)■ Geometry precalculation of radius-compensated positions for feed rate adaptation with M120
Background programming	One part program can be edited while the TNC runs another program
Graphics	<ul style="list-style-type: none">■ Programming graphics■ Test graphics
File types	<ul style="list-style-type: none">■ HEIDENHAIN dialog programs■ ISO programs■ Datum tables■ Tool tables■ Pocket table
Program memory	<ul style="list-style-type: none">■ Buffer battery backup for approx. 10 000 NC blocks (depending on length of block), 256 KB■ Up to 64 files possible
Tool definitions	Up to 254 tools in the program or in tool tables
Programming support	<ul style="list-style-type: none">■ Functions for approaching and departing the contour■ HELP

Programmable functions

Contour elements	<ul style="list-style-type: none"> ■ Straight line ■ Chamfer ■ Circular arc ■ Circle center ■ Circle radius ■ Tangentially connecting arc ■ Corner rounding ■ Straight lines and circular arcs for contour approach and departure
FK free contour programming	For all contour elements not dimensioned for conventional NC programming
Program jumps	<ul style="list-style-type: none"> ■ Subprograms ■ Program section repeats ■ Main program as subprogram
Fixed cycles	<ul style="list-style-type: none"> ■ Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping ■ Milling and finishing rectangular and circular pockets ■ Cycles for milling linear and circular slots ■ Regular linear and circular point patterns ■ Irregular linear and circular hole patterns ■ Cycles for multipass-milling of flat and twisted surfaces ■ Milling pockets and islands from a list of subcontour elements
Coordinate transformations	<ul style="list-style-type: none"> ■ Datum shift ■ Mirroring ■ Rotation ■ Scaling factor
3-D touch probe applications	<ul style="list-style-type: none"> ■ Touch probe functions for setting datums and for automatic workpiece measurement ■ Digitizing 3-D surfaces with triggering touch probe (optional) ■ Automatic workpiece measurement with TT 120

Mathematical functions	<ul style="list-style-type: none">■ Basic arithmetic +, −, x and ÷■ Trigonometry sin, cos, tan, arcsin, arccos, arctan■ Square root (\sqrt{a}) and root sum of squares ($\sqrt{a^2 + b^2}$)■ Squaring (SQ)■ Powers (^)■ pi (3.14)■ Logarithms■ Exponential function■ Negation (NEG)■ Forming an integer (INT)■ Forming absolute numbers (ABS)■ Dropping the values before the decimal point (FRAC)■ Logical comparisons (greater than, less than, equal to, not equal to)
TNC Specifications	
Block processing time	6 ms/block 20 ms/block for blockwise execution via data interface
Control loop cycle time	Contouring interpolation: 6 ms
Data transfer rate	Max. 115 200 baud
Ambient temperature	<ul style="list-style-type: none">■ Operation: 0°C to +45°C (32° to 113° F)■ Storage: −30° C to +70° C (−22° to 158° F)
Traverse range	Maximum 30 m (11 81 inches)
Traversing speed	Maximum 300 m/min (11 811 inches/min)
Spindle speed	Maximum 99 999 rpm
Input range	<ul style="list-style-type: none">■ Minimum 1 µm (0.0001 inch) or 0.001°■ Maximum 30 000.000 mm (1,181 inches) or 30 000.000°

15.4 TNC 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

Some of the more frequent TNC error messages are explained in the following list.

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. To clear the TNC error message, first correct the error and then press the CE key.

TNC error messages during programming

Further file entry impossible	Erase some old files to make room for new ones.
Entry value incorrect	<ul style="list-style-type: none">■ Enter a correct label number.■ Note the input limits.
Ext. in-/output not ready	<ul style="list-style-type: none">■ Connect the data transfer cable.■ Transfer cable is defective or not soldered properly.■ Switch on the connected device (PC, printer).■ The data transfer speeds (baud rates) are not identical.
Protected file!	Cancel edit protection if you wish to edit the file.
Label number already assigned	A given label number can only be entered once in a program.
Jump to label 0 not permitted	Do not program CALL LBL 0.

TNC error messages during test run and program run

Axis double programmed	Each axis can have only one value for position coordinates.
Selected block not addressed	Before a test run or program run, you must enter GOTO 0.
Touch point inaccessible	<ul style="list-style-type: none">■ Pre-position the 3-D touch probe to a position nearer the model.■ The machine parameters that define the position of the TT touch probe do not correspond to the actual position of the TT touch probe.
Arithmetical error	<p>You have calculated with non-permissible values.</p> <ul style="list-style-type: none">■ Define values within the range limits.■ Choose probe positions for the 3-D touch probe that are farther apart.■ Enter a value other than 0 for the number of teeth in the tool table when you are measuring the individual teeth with the TT touch probe.■ Calibrate TCH PROBE 30 (calibrate TT) before measuring the tool length or radius.■ All calculations must be mathematically possible.
Path comp. wrongly ended	Do not cancel tool radius compensation in a block with a circular path.
Path comp. wrongly started	<ul style="list-style-type: none">■ Use the same radius compensation before and after an RND and CHF block.■ Do not begin tool radius compensation in a block with a circular path.
CYCL DEF incomplete	<ul style="list-style-type: none">■ Define the cycles with all data in the proper sequence.■ Do not call the coordinate transformation cycles.■ Define a cycle before calling it.■ Enter a pecking depth other than 0.
Plane wrongly defined	<ul style="list-style-type: none">■ Do not change the tool axis while a basic rotation is active.■ Correctly define the main axes for circular arcs.■ Define both main axes for CC.
Wrong axis programmed	<ul style="list-style-type: none">■ Do not attempt to program locked axes.■ Program a rectangular pocket or slot in the working plane.■ Do not mirror rotary axes.■ Enter a positive chamfer length.

Wrong RPM	Program a spindle speed within the programmed range.
Chamfer not permitted	A chamfer block must be located between two straight-line blocks with identical radius compensation.
Faulty program data	The program that was read in through the data interface contains incorrect block formats.
No editing of running program	A program cannot be edited while it is being transmitted or executed.
Circle end pos. incorrect	<ul style="list-style-type: none"> ■ Enter complete information for connecting arc. ■ Enter end points that lie on the circular path.
Circle center undefined	<ul style="list-style-type: none"> ■ Define a circle center with the CC key. ■ Define a pole with the CC key.
Label number not found	Only call label numbers that have been set.
Scaling factor not permitted	Enter identical scaling factors for coordinate axes in the plane of the circle.
PGM section cannot be shown	<ul style="list-style-type: none"> ■ Enter a smaller tool radius. ■ 4-D movements cannot be graphically simulated. ■ Enter a tool axis for simulation that is the same as the axis in the BLK FORM.
Radius comp. undefined	Enter radius compensation RR or RL in the first subprogram for Cycle 14 CONTOUR GEOMETRY.
Rounding-off undefined	Enter tangentially connecting arcs and rounding arcs correctly.
Rounding radius too large	Rounding arcs must fit between contour elements.
Key non-functional	This message always appears when you press a key that is not needed for the current dialog.
Stylus already in contact	Before probing, pre-position the stylus where it is not touching the workpiece surface.

Calibrating a touch probe	<ul style="list-style-type: none"> ■ Recalibrate the TT, machine parameters for the TT have been changed.
Probe system not ready	<ul style="list-style-type: none"> ■ Be sure that the transmitting/receiving window of the TS 630 is oriented to the receiving unit ■ Check whether the touch probe is ready for operation
TOOL CALL is missing	<ul style="list-style-type: none"> ■ Call only defined tools ■ Run a block scan with PLC = ON
Program start undefined	<ul style="list-style-type: none"> ■ Begin the program only with a TOOL DEF block. ■ Do not resume an interrupted program at a block with a tangential arc or if a previously defined pole is needed.
Feed rate is missing	<ul style="list-style-type: none"> ■ Enter the feed rate for the positioning block. ■ Enter FMAX in each block. When working with point tables: program feed rate with numerical values.
Wrong sign programmed	Enter the correct sign for the cycle parameter.
Tool radius too large	<p>Enter a tool radius that</p> <ul style="list-style-type: none"> ■ lies within the given limits ■ permits the contour elements to be calculated and machined.
Tool life expired	TIME1 or TIME2 of TOOL.T has been exceeded with no replacement tool defined in the tool table.
Angle reference missing	<ul style="list-style-type: none"> ■ Complete your definition of the arc and its end points. ■ If you enter polar coordinates, define the polar angle correctly.
Excessive subprogramming	<ul style="list-style-type: none"> ■ Conclude subprograms with LBL0. ■ Program CALL LBL for subprograms without REP. ■ Program CALL LBL for program section repeats to include the repetitions (REP). ■ Subprograms cannot call themselves. ■ Subprograms cannot be nested in more than eight levels. ■ Main programs cannot be nested as subprograms in more than four levels.

TNC error messages during digitizing

Axis double programmed	Program two different axes for the coordinates of the starting point (CONTOUR LINES cycle).
Start position incorrect	Program the starting point coordinates for the CONTOUR LINES cycle within the RANGE.
Touch point inaccessible	<ul style="list-style-type: none"> ■ The stylus must not be deflected before it reaches the RANGE. ■ The stylus must be deflected somewhere within the RANGE.
Range exceeded	Enter a RANGE that includes the entire 3-D surface to be scanned.
Faulty range data	<ul style="list-style-type: none"> ■ Enter MIN coordinates that are smaller than their MAX coordinates. ■ Define the RANGE within the limits through software limit switches. ■ Define the range for the MEANDER and CONTOUR LINES cycles.
Rotation not permitted	Reset all coordinate transformations before digitizing.
Height axis not permitted here	Define the starting position coordinate (CONTOUR LINES cycle) in axes different from the stylus axis.
Wrong axis programmed	<ul style="list-style-type: none"> ■ Enter calibrated touch probe axis in the RANGE cycle. ■ Do not program the same axes twice in the RANGE cycle
Scaling factor not permitted	Reset all coordinate transformations before digitizing.
Mirroring not permitted	Reset all coordinate transformations before digitizing.
Stylus already in contact	Pre-position the stylus so that it cannot be deflected outside the RANGE.

Probe system not ready	<ul style="list-style-type: none">■ Be sure that the transmitting/receiving window of the TS 630 is oriented to the receiving unit■ Check whether the touch probe is ready for operation■ Touch probe cannot be retracted
Exchange touch probe battery	<ul style="list-style-type: none">■ Change the battery in the probe head (TS 630).■ This message is output when the touch probe reaches the end of a line.
Time limit exceeded	Enter a time limit that is appropriate for the 3-D surface to be scanned (CONTOUR LINES cycle).

15.5 Changing 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. The buffer batteries are located next to the power supply unit in the logic unit (round, black case). The TNC also has an power storage device that provide the control with current while you are exchanging the batteries (for a maximum of 24 hours).



To exchange the buffer battery, first switch off the TNC!
The buffer battery must be exchanged only by trained service personnel!

Battery type: Three AA-size cells, leak-proof, IEC designation „LR6“

SYMBOLS

3-D touch probe
 calibrating
 triggering touch probe ... 249
 compensating center misalignment ... 249
 measuring during program run 256
 3-D view ... 234

A

Accessories ... 12
 Actual position capture ... 77
 Adding comments ... 40
 Angular functions ... 210
 Approach contour ... 68
 Auxiliary axes ... 27

B

Back boring ... 131
 Ball ... 228
 Blank definition ... 34
 Block memory ... 270
 Blocks
 deleting ... 38
 editing ... 38
 inserting ... 18
 Boring ... 128

C

Chamfer ... 77
 Circle center CC ... 78
 Circular hole pattern ... 159
 Circular path ... 79, 80, 81, 87, 88
 Circular pocket
 finishing ... 146
 roughing ... 144
 Circular stud finishing ... 147
 Constant contouring speed :M90 107

HEIDENHAIN TNC 410

C

Constant feed rate at the cutting edge ... 115
 Contour cycles. *See* SL cycles
 Contour filter: M124 ... 110
 Contour transition
 M112 ... 108
 M124 ... 110
 Conversational dialog ... 37
 Conversational format ... 37
 Coordinate transformation
 overview ... 181
 Corner rounding ... 82
 Cycle
 calling ... 121, 123
 defining ... 120
 groups ... 120
 with point tables ... 122
 Cylinder ... 228

D

Data interface
 pin layout ... 290
 setting ... 269
 Data transfer
 speed ... 269
 Data transfer software ... 270
 Datum selection ... 30
 Datum shift ... 182
 with datum tables ... 182
 Depart contour ... 68
 Digitized data
 executing ... 265
 Digitizing
 defining the range ... 261
 in contour lines ... 263
 meander digitizing ... 262
 programming digitizing cycles ... 261

D

Drilling ... 126, 129
 Drilling and boring cycles ... 124
 Dwell time ... 190

E

Ellipsis ... 224
 Error messages
 during digitizing ... 299
 during programming ... 295
 during test run and program run 296
 output ... 213
 Exchanging the buffer battery ... 300

F

Feed rate, changing ... 18
 File management
 calling ... 31
 copying files ... 32
 deleting files ... 32
 file name ... 31
 file type ... 31
 protecting files ... 32
 reading in files ... 33
 renaming files ... 32
 FK programming
 auxiliary points ... 96
 begin dialog ... 93
 circular paths ... 94
 closed contours ... 97
 converting FK program ... 32
 fundamentals ... 92
 graphics ... 92
 relative references ... 97
 straight line ... 94
 Full circle ... 79

G

Graphic simulation ... 235

Graphics

- detail magnification ... 234
- displays ... 232
- during programming ... 39

H

Helical interpolation ... 88

Helix ... 88

HELP files

- running ... 275

HELP function

- displaying ... 41

Hole patterns

- on a circle ... 159
- on a line ... 160
- overview ... 158

I

Insert rounding arc
between straight lines: M112 108

Interrupting machining ... 240

K

Keyboard ... 5

L

Look ahead ... 115

M

Machine parameters

- for 3-D touch probes ... 280
- for external data
transfer ... 279
- for TNC displays
and TNC editor 281

Machine-referenced
coordinates: M91/M92 105

M

Machining small contour steps: M97
112

Main axes ... 27

Mid-program startup ... 243

Mirror image ... 184

Miscellaneous functions

- entering ... 104
- for contouring behavior ... 107
- for coordinate data ... 105
- for program-run control ... 105
- for rotary axes ... 117
- for the spindle ... 105

MOD functions

- editing ... 268
- exiting ... 268
- selecting ... 268

Modes of operation 5

Moving the machine axes

- with electronic handwheel ... 16
- with incremental jog
positioning 17
- with machine direction keys ... 15

Multipass milling ... 176

N

Nesting ... 197

Non-controlled axes
in the NC program ... 239

O

Open contours: M98 113

P

Paraetric programming. *See* Q
parameter programming

Parenthetical calculation ... 219

Part families ... 207

P

Path contours

- Cartesian coordinates ... 76
 - circular path around circle
center ... 79
 - circular path with
defined radius 80
 - circular path with tangential
connection 81
 - overview ... 76
 - straight line ... 77
- FK free contour programming. *See*
FK programming
- Polar coordinates ... 86
 - circular path around
pole CC 87
 - circular path with tangential
connection 88
 - overview ... 86
 - straight line ... 87

Path functions

- fundamentals ... 65
 - circles and
circular arcs ... 66
 - pre-positioning ... 66

Pecking ... 125

Plan view ... 233

Pocket table ... 50

Point tables ... 122

- Programming example ... 136, 157

Polar coordinates

- fundamentals 28
- setting the pole ... 28

Positioning with manual
data input ... 22

POSITIP operation ... 239

Probe cycles ... 248

P

Program blocks
 editing ... 38
 opening ... 35
 structure ... 34

Program call
 calling any program as subprogram 196
 via cycle ... 190

Program management. *See* File management

Program name *See* File management: File name

Program run
 block skipping ... 246
 executing ... 238
 interrupting ... 240
 mid-program startup 243
 overview ... 238
 resuming after an interruption ... 241

Program section repeat
 calling ... 196
 operating sequence ... 195
 programming ... 196
 programming notes ... 195

Programming graphics ... 39

Projection in 3 planes 233

Q

Q parameters
 controlling ... 212
 pre-assigned ... 222, 223
 transferring values to the PLC 218

Q-parameter programming
 additional functions ... 213
 angular functions ... 210
 basic mathematical functions ... 208
 entering formula ... 219
 if/then decisions ... 211
 programming notes ... 206

R

Radius compensation ... 52
 entering ... 54
 inside corners ... 55
 machining corners ... 55
 outside corners ... 55

Rapid traverse ... 44

Read system data ... 215

Reaming ... 127

Rectangular pocket
 finishing ... 141
 roughing ... 140

Rectangular stud finishing ... 143

Reference system ... 27

Reference-point traverse ... 14

Returning to the contour 244

Rotary axis
 reducing the display ... 117
 shorter-path traverse ... 117

Rotation ... 185

Rough out. *See* SL cycles: Rough-out

Round slot milling ... 152

Ruled surface ... 178

S

Scaling factor ... 186

Scaling factor, axis-specific ... 187

Screen layout 4

Set the datum
 with a 3-D touch probe ... 251
 circle center as datum ... 253
 corner as datum ... 252
 in any axis ... 251
 without a 3-D touch probe ... 19

Setting the BAUD RATE ... 269

SL cycles
 cycle contour ... 165
 operating sequence ... 288
 overview ... 164
 pilot drilling ... 168
 roughing ... 169
 superimposed contours ... 166

Slot milling
 with reciprocating plunge cut ... 150

Slot milling ... 150

Spindle orientation ... 191

Spindle speed
 changing 18
 entering ... 18, 44

Status display
 enhanced ... 9
 general ... 9

Straight line ... 77, 87

Subprogram
 calling ... 195
 operating sequence ... 194
 programming ... 195
 programming notes ... 194

Switch on ... 14

T

- Tapping
 - rigid tapping ... 134
 - with floating tap holder ... 133
- Teach-in ... 77
- Technical information ... 292
- Test run
 - executing ... 237
 - overview ... 236
 - up to a certain block 237
- TNC 410 ... 2
- TNCremo ... 270
- Tool change ... 51
 - automatic ... 51
- Tool compensation
 - length ... 52
 - radius ... 52
- Tool data
 - calling ... 51
 - delta values ... 46
 - entering in the program ... 46
 - entering into tables 47
- Tool length ... 45
- Tool measurement
 - automatic ... 56
 - tool length ... 59
 - tool radius ... 61
 - calibrating TT 120 ... 58
- Tool movements
 - overview ... 64
 - programming ... 37

T

- Tool number ... 45
- Tool radius ... 46
- Tool table
 - available input data...47
 - editing ... 49
 - exiting ... 49
 - selecting ... 49
- Trigonometry ... 210

U

- Universal drilling ... 129
- User parameters
 - general ... 278
 - for 3-D touch probes and digitizing 280
 - for external data transfer ... 279
 - for machining and program run 287
 - for TNC displays, TNC editor 282
- User parameters

W

- Workpiece measurement ... 254
- Workpiece misalignment, compensating ... 250
- Workpiece positions
 - absolute ... 29
 - incremental ... 29
 - relative ... 29
- Workspace monitoring during PGM test ... 274

M	Effect of M Function	Effective at block	start	end	Page
M00	Stop program run/Spindle STOP/Coolant OFF			■	105
M01	Optional program run interruption			■	246
M02	Stop program run/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1			■	105
M03	Spindle ON clockwise		■		
M04	Spindle ON counterclockwise		■		
M05	Spindle STOP			■	105
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP			■	105
M08	Coolant ON		■		
M09	Coolant OFF			■	105
M13	Spindle ON clockwise/Coolant ON		■		
M14	Spindle ON counterclockwise/Coolant ON		■		105
M30	Same function as M02			■	105
M89	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameter)		■	■	121
M90	Only in lag mode: Constant contouring speed at corners		■		107
M91	Within the positioning block: Coordinates are referenced to machine datum		■		105
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position		■		105
M93	Within the positioning block: Coordinates are referenced to current tool position. Valid in blocks with R0, R+, R–		■		
M94	Reduce display of rotary axis to value less than 360°		■		117
M97	Machine small contour steps			■	112
M98	Completely machine open contours			■	113
M99	Blockwise cycle call			■	121
M101	Automatic tool change with replacement tool if max. tool life has expired		■		
M102	ResetM101			■	51
M103	Reduce feed rate during plunging to factor F (percentage)		■		114
M109	Constant contouring speed at tool cutting edge on circular arcs (increase and decrease feed rate)		■		
M110	Constant contouring speed at tool cutting edge on circular arcs (feed rate decrease only)		■		
M111	Reset M109/M110			■	115
M112	Automatic insertion of rounding arcs at non-tangential straight-line transitions; Enter tolerance T for contour deviation		■		
M113	Reset M112				108
M120	Radius-compensated contour Look Ahead		■		115
M124	Ignore points for calculating rounding arc with M112		■		110
M126	Shorter-path traverse of rotary axes		■		
M127	Reset M126			■	117

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH

Dr.-Johannes-Heidenhain-Straße 5

83301 Traunreut, Germany

☎ +49 (86 69) 31-0

FAX +49 (86 69) 50 61

E-Mail: info@heidenhain.de

Technical support FAX +49 (86 69) 31-10 00

E-Mail: service@heidenhain.de

Measuring systems ☎ +49 (86 69) 31-31 04

E-Mail: service.ms-support@heidenhain.de

TNC support ☎ +49 (86 69) 31-31 01

E-Mail: service.nc-support@heidenhain.de

NC programming ☎ +49 (86 69) 31-31 03

E-Mail: service.nc-pgm@heidenhain.de

PLC programming ☎ +49 (86 69) 31-31 02

E-Mail: service.plc@heidenhain.de

Lathe controls ☎ +49 (7 11) 95 28 03-0

E-Mail: service.hsf@heidenhain.de

www.heidenhain.de