HEIDENHAIN

TNC 310

NC Software
286 140-xx
286 160-xx

User's Manual
HEIDENHAIN Conversational
Programming

English (en)
4/2003

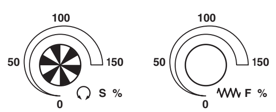
Controls on the visual display unit

- Split screen layout
- Soft keys
- Shift the soft-key rows

Machine control keys

- X+ X– Axis direction
- Rapid traverse
- Direction of spindle rotation
- Coolant
- Tool release
- I 0 Spindle ON/OFF
- NC I NC 0 NC start/NC stop

Override control knobs for feed rate/spindle speed



Mode of operation

- Manual Operation
- Positioning with Manual Data Input (MDI)
- Program Run/Test Run
- Programming and Editing

Numerical input, editing

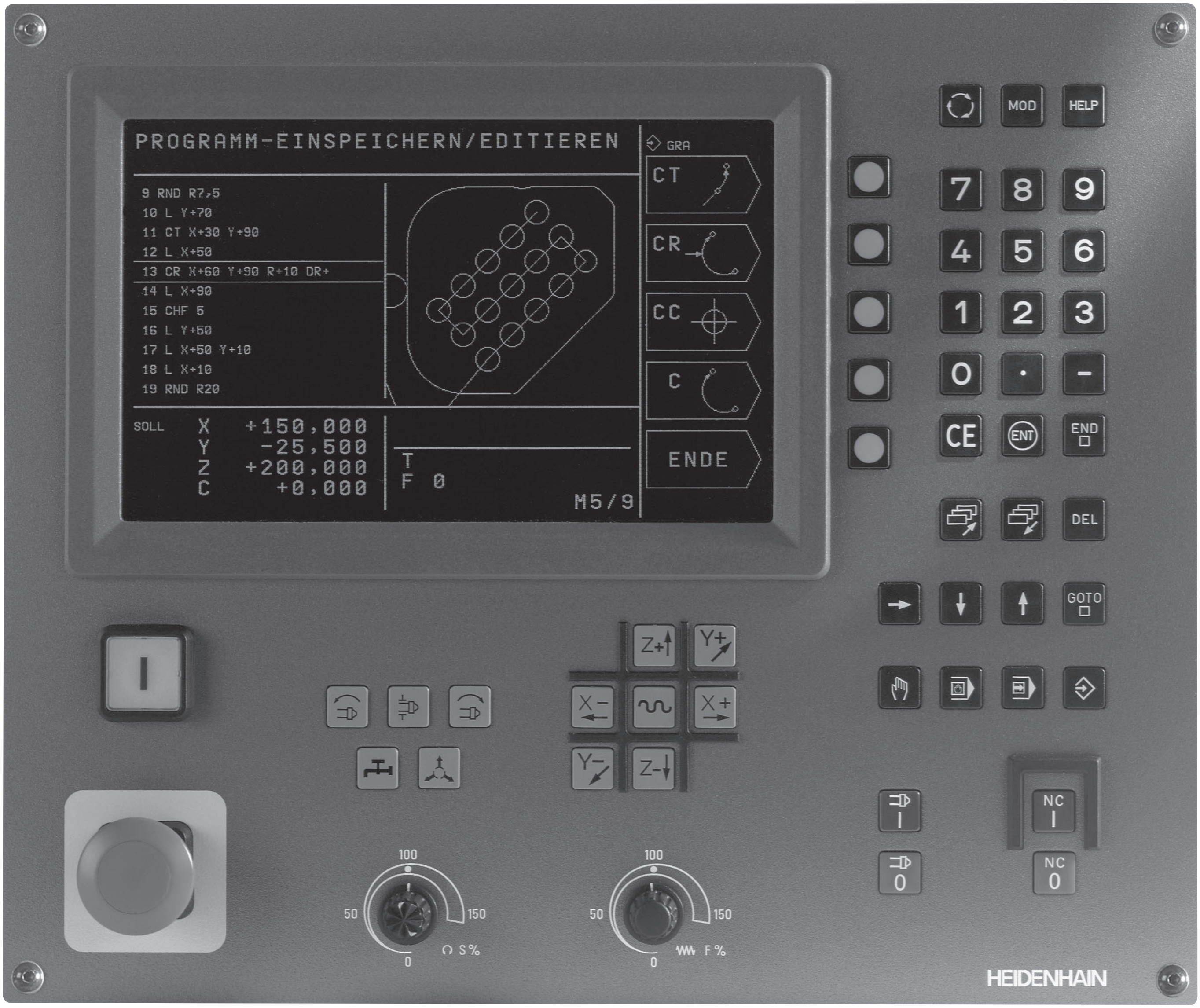
- 0 ... 9 Numbers
- Decimal point
- Change arithmetic sign
- ENT Confirm entry and resume dialog
- END Endblock
- CE Clear numerical entry or TNC error message
- DEL Abort dialog, delete program section

Programming aids

- MOD MOD functions
- HELP HELP function

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

- Move highlight
- Move highlight, skip dialog question
- GOTO Select blocks and cycles directly


PROGRAMM-EINSPEICHERN/EDITIEREN
GRA
9 RND R7,5
10 L Y+70
11 CT X+30 Y+90
12 L X+50
13 CR X+60 Y+90 R+10 DR+
14 L X+90
15 CHF 5
16 L Y+50
17 L X+50 Y+10
18 L X+10
19 RND R20
CT
CR
CC
C
ENDE
SOLL
X +150,000
Y -25,500
Z +200,000
C +0,000
T
F 0
M5/9
MOD
HELP
7 8 9
4 5 6
1 2 3
0 . -
CE
ENT
END
DEL
GOTO
Z+
Y+
X-
X+
Y-
Z-
NC
I
NC
0
100
50
150
0
S %
100
50
150
0
F %
HEIDENHAIN

TNC Models, Software and Features

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

TNC Model	NC Software No.
TNC 310	286 140-xx
TNC 310 M	286 160-xx

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

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

- Probing function for the 3-D touch probe
- Rigid tapping cycle
- Boring cycle
- Back boring cycle

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 (MDI) 3

Programming: Fundamentals, 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

MOD Functions 13

Tables and Overviews 14

Contents

1 INTRODUCTION.....1

1.1 The TNC 310.....2

1.2 Visual Display Unit and Keyboard.....3

1.3 Modes of Operation.....4

1.4 Status Displays.....7

1.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels.....11

2 MANUAL OPERATION AND SETUP.....13

2.1 Switch-On.....14

2.2 Moving the Machine Axes.....15

2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M.....18

2.4 Datum Setting (Without a 3-D Touch 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 and Writing Programs.....34

4.4 Interactive Programming Graphics.....39

4.5 HELP function.....41

5 PROGRAMMING: TOOLS.....43

5.1 Entering Tool-Related Data.....44

5.2 Tool Data.....45

5.3 Tool Compensation.....51

6 PROGRAMMING: PROGRAMMING CONTOURS.....55

6.1 Overview ofTool Movements.....56

6.2 Fundamentals of Path Functions.....57

6.3 Contour Approach and Departure.....60

Overview: Types of paths for contour approach and departure.....60

Important positions for approach and departure.....60

Approaching on a straight line with tangential connection: APPR LT.....62

Approaching on a straight line perpendicular to the first contour point: APPR LN.....62

Approaching on a circular arc with tangential connection: APPR CT.....63

Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT.....64

Departing tangentially on a straight line: DEP LT.....65

Departing on a straight line perpendicular to the last contour point: DEP LN.....65

Departing tangentially on a circular arc: DEP CT.....66

Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT.....67

6.4 Path Contours — Cartesian Coordinates.....68

Overview of path functions.....68

Straight line L.....69

Inserting a chamfer CHF between two straight lines.....69

Circle center CC.....70

Circular path C around circle center CC.....71

Circular path CR with defined radius.....72

Circular path CT with tangential connection.....73

Corner Rounding RND.....74

Example: Linear movements and chamfers with Cartesian coordinates.....75

Example: Circular movements with Cartesian coordinates.....76

Example: Full circle with Cartesian coordinates.....77

6.5 Path Contours—Polar Coordinates.....78

Polar coordinate origin: Pole CC.....78

Straight line LP.....79

Circular path CP around pole CC.....79

Circular path CTP with tangential connection.....80

Helical interpolation.....81

Example: Linear movement with polar coordinates 83

Example: Helix 84

7 PROGRAMMING: MISCELLANEOUS FUNCTIONS.....85

7.1 Entering Miscellaneous Functions M and STOP.....86

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant.....87

7.3 Miscellaneous Functions for Coordinate Data.....87

7.4 Miscellaneous Functions for Contouring Behavior.....89

7.5 Miscellaneous Function for Rotary Axes.....92

8 PROGRAMMING: CYCLES.....93

8.1 General Overview of Cycles.....94

8.2 Drilling Cycles.....96

PECKING (Cycle 1).....96

DRILLING (Cycle 200).....98

REAMING (Cycle 201).....99

BORING (Cycle 202).....100

UNIVERSAL DRILLING (Cycle 203).....101

BACK BORING (Cycle 204).....103

TAPPING with a floating tap holder (Cycle 2).....105

RIGIDTAPPING (Cycle 17).....106

Example: Drilling cycles.....107

Example: Drilling cycles 108

8.3 Cycles for Milling Pockets, Studs and Slots.....109

POCKET MILLING (Cycle 4).....110

POCKET FINISHING (Cycle 212).....111

STUD FINISHING (Cycle 213).....113

CIRCULAR POCKET MILLING (Cycle 5).....114

CIRCULAR POCKET FINISHING (Cycle 214).....116

CIRCULAR STUD FINISHING (Cycle 215) 117

SLOT MILLING (Cycle 3).....119

SLOT with reciprocating plunge-cut (Cycle 210).....120

CIRCULAR SLOT with reciprocating plunge-cut (Cycle 211) 122

Example: Milling pockets, studs and slots.....124

8.4 Cycles for Machining Hole Patterns.....126
CIRCULAR PATTERN (Cycle 220).....127
LINEAR PATTERN (Cycle 221) 128
Example: Circular hole patterns.....130
8.5 Cycles for multipass milling.....132
MULTIPASS MILLING (Cycle 230).....132
RULED SURFACE (Cycle 231).....134
Example: Multipass milling.....136
8.6 CoordinateTransformation Cycles 137
DATUM SHIFT (Cycle 7).....138
DATUM SHIFT with datum tables (Cycle 7).....138
MIRROR IMAGE (Cycle 8).....140
ROTATION (Cycle 10).....141
SCALING FACTOR (Cycle 11) 142
Example: Coordinate transformation cycles.....143
8.7 Special Cycles 145
DWELL TIME (Cycle 9) 145
PROGRAM CALL (Cycle 12).....145
ORIENTED SPINDLE STOP (Cycle 13) 146

9 PROGRAMMING: SUBPROGRAMS AND PROGRAM SECTION REPEATS.....147

9.1 Labeling Subprograms and Program Section Repeats.....148
9.2 Subprograms.....148
9.3 Program Section Repeats.....149
9.4 Nesting.....151
Subprogram within a subprogram 151
Repeating program section repeats.....152
Repeating a subprogram.....153
Example: Milling a contour in several infeeds 154
Example: Groups of holes 155
Example: Groups of holes with several tools 156

Contents

10 PROGRAMMING: Q PARAMETERS.....159

10.1 Principle and Overview.....160

10.2 Part Families — Q Parameters in Place of Numerical Values.....161

10.3 Describing Contours through Mathematical Operations.....162

10.4 Trigonometric Functions 164

10.5 If-Then Decisions with Q Parameters 165

10.6 Checking and Changing Q Parameters 166

10.7 Additional Functions 167

10.8 Entering Formulas Directly.....173

10.9 Preassigned Q Parameters.....176

10.10 Programming Examples.....178

Example: Ellipse.....178

Example: Concave cylinder machined with spherical cutter 180

Example: Convex sphere machined with end mill 182

11 TEST RUN AND PROGRAM RUN.....185

11.1 Graphics.....186

11.2 Test run.....190

11.3 Program Run.....192

11.4 Blockwise Transfer: Running Longer Programs.....199

11.5 Optional Program Run Interruption.....200

12 3-D TOUCH PROBES.....201

12.1 Touch Probe Cycles in the Manual Operation Mode.....202

Calibrating a touch trigger probe.....203

Compensating workpiece misalignment.....204

12.2 Setting the Datum with a 3-D Touch Probe.....205

12.3 Measuring Workpieces with a 3-D Touch Probe.....208

13 MOD FUNCTIONS.....211

13.1 Selecting, Changing and Exiting the MOD Functions.....212
13.2 System Information.....212
13.3 Entering the Code Number.....213
13.4 Setting the Data Interface.....213
13.5 Machine-Specific User Parameters.....216
13.6 Position DisplayTypes.....216
13.7 Unit of Measurement.....216
13.8 Axis Traverse Limits 217
13.9 Running the HELP File.....218

14 TABLES AND OVERVIEWS.....219

14.1 General User Parameters.....220
Input possibilities for machine parameters.....220
Selecting general user parameters.....220
External data transfer.....221
3-D Touch Probes.....222
TNC displays, TNC editor.....222
Machining and program run.....224
Electronic handwheels.....225
14.2 Pin Layout and Connecting Cable for the Data Interface.....226
RS-232-C/V.24 Interface 226
14.3 Technical Information.....227
TNC features.....227
Programmable functions.....228
TNC Specifications.....228
14.4 TNC Error Messages.....229
TNC error messages during programming.....229
TNC error messages during test run and program run.....229
14.5 Exchanging the Buffer Battery.....232

1

Introduction

1.1 The TNC 310

HEIDENHAIN TNC controls are shop-floor programmable contouring controls for milling, drilling and boring machines.

You can program conventional milling, drilling and boring operations right at the machine with the easily understandable interactive conversational guidance. The TNC 310 can control up to 4 axes. Instead of the fourth axis, 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 format
HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. Workpiece machining can be graphically simulated during test run.

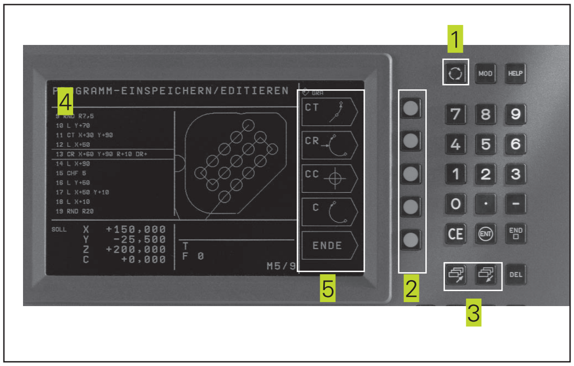
You can enter a program while the control is running another.

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

In addition, the TNC can also **run** programs with functions that cannot be programmed directly on the TNC 310 itself, such as:

- FK free contour programming
- Contour cycles
- ISO programs
- Program call with PGM CALL

1.2 Visual Display Unit and Keyboard



Visual display unit

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

1 Setting the screen layout

2 Soft key selector keys

3 Switching the soft-key rows

4 Header
When the TNC is on, the selected operating mode is shown in the screen header. Dialog prompts and TNC messages also appear here (unless the TNC is showing only graphics).

5 Soft keys
In the right margin the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately beside them 2. Directly beneath the soft-key row are rectangular boxes indicating the number of soft-key rows. These rows can be called with the 3 outside right and left. The box representing the active soft-key row is filled in.

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.

PGM + GRAPHICS

Select the desired screen layout.

Keyboard

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

1 MOD function,
HELP function

2 Numerical input

3 Dialog buttons

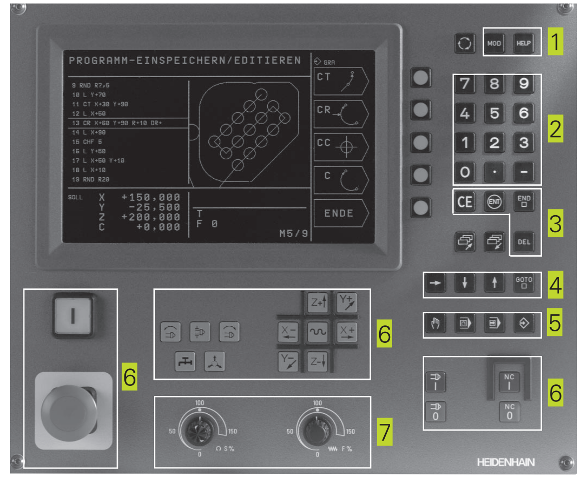
4 Arrow keys and GOTO jump command

5 Modes of Operation

6 Machine control buttons

7 Override control knobs for feed rate/spindle speed

The functions of the individual keys are described in the foldout of the front cover. The exact functioning of the machine control buttons, e.g. NC START, is described in more detail in your Machine Manual.

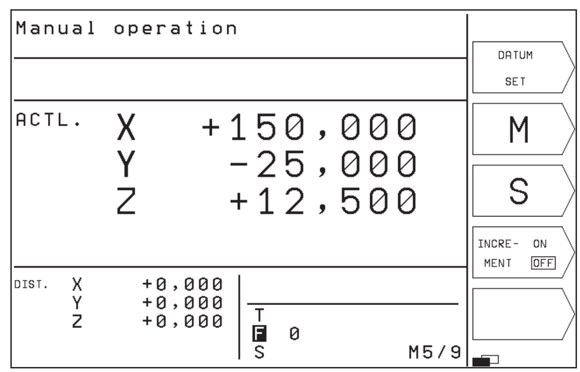


1.3 Modes of Operation

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 Handwheel

The Manual Operation mode is required for setting up the machine tool. In this operating mode, you can position the machine axes manually or by increments. Datums can be set by the usual scratching method or by using the TS 220 triggering touch probe. The TNC also supports the manual traverse of the machine axes using a HR electronic handwheel.



Soft keys for selecting the screen layout

Screen windows	Soft key
Positions	POSITION
Left: positions, right: general program information	POSITION + PGM STATUS
Left: positions, right: positions and Coordinates	POSITION + POS.DISPLAY STATUS
Left: positions, right: information on tools	POSITION + TOOL STATUS
Left: positions, right: coordinate transformations	POSITION + COORD.TRANS. STATUS

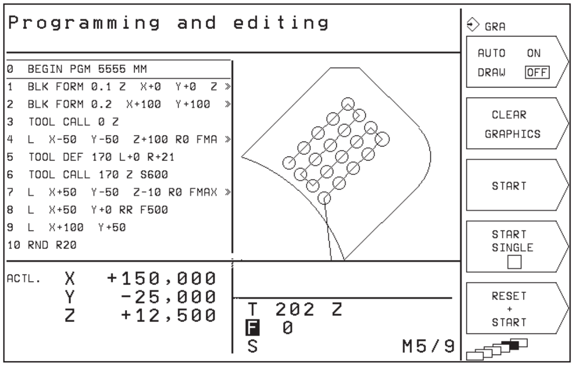
Positioning with Manual Data Input (MDI)

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

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PROGRAM
Left: program blocks, right: general program information	PGM + PGM STATUS
Left: program blocks, right: positions and Coordinates	PGM + POS. STATUS
Left: program blocks, right: tool tools	PGM + TOOL STATUS
Left: program blocks, right: coordinate transformations	PGM + COORD.TRANS. STATUS
Left: program blocks, right: help graphics for cycle programming (2nd soft-key level)	PGM + FIGURE

Programming and Editing



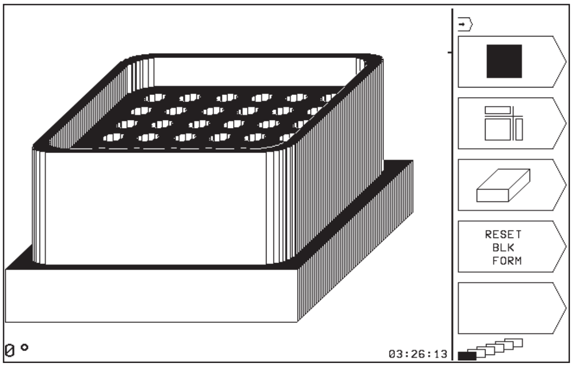
In this mode of operation you can write your part programs. The various cycles help you with programming and add necessary information. If desired, you can have the programming graphics show the individual steps.

Soft keys for selecting the screen layout

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

Test run

In the Test Run mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the work space. This simulation is supported graphically in different display modes. Use a soft key to activate the test run in the Program Run operating mode.



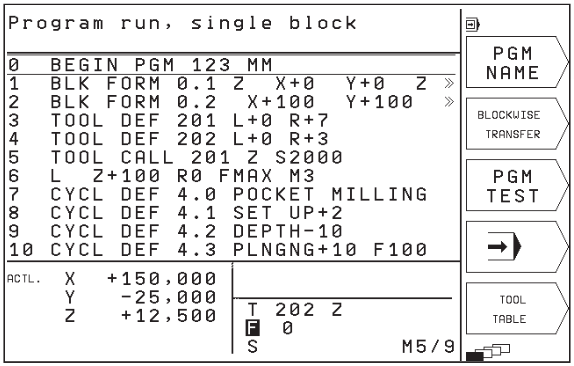
Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PROGRAM
n Test run graphics	GRAPHICS
Left: program blocks, right: general program information	PGM + PGM STATUS
Left: program blocks, right: positions and Coordinates	PGM + POS. STATUS
Left: program blocks, right: tool tools	PGM + TOOL STATUS
Left: program blocks, right: coordinate transformations	PGM + COORD.TRANS. STATUS

Program Run/Single Block and Program Run/Full Sequence

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 NC START button.



Soft keys for selecting the screen layout

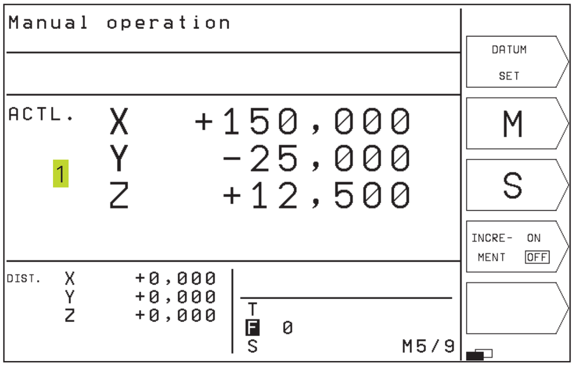
Screen windows	Soft key
Program	PROGRAM
Left: program blocks, right: general program information	PGM + PGM STATUS
Left: program blocks, right: positions and Coordinates	PGM + POS. STATUS
Left: program blocks, right: tool tools	PGM + TOOL STATUS
Left: program blocks, right: coordinate transformations	PGM + COORD.TRANS. STATUS

1.4 Status Displays

"General" status display

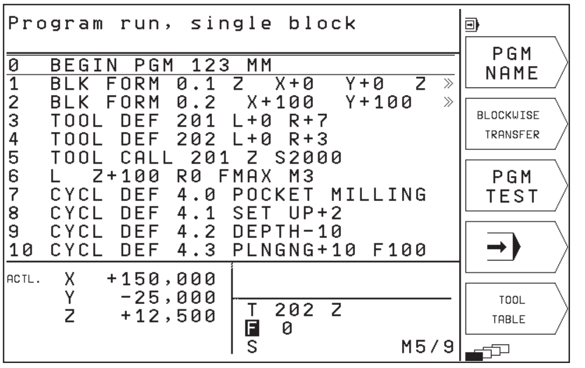
The status display informs you of the current state of the machine tool. It is displayed automatically in all modes of operation:

In the operating modes Manual Operation and Electronic Handwheel and Positioning with MDI the status display appears in the large window 1.



Information in the status display

The	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
ROT	Axes are moving plain



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 Manual Operation mode.

To switch on the additional status display:

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

You can also choose between the following additional status displays:

PGM + PGM STATUS

General program information

1 Name of main program / Active block number

2 Program called via Cycle 12

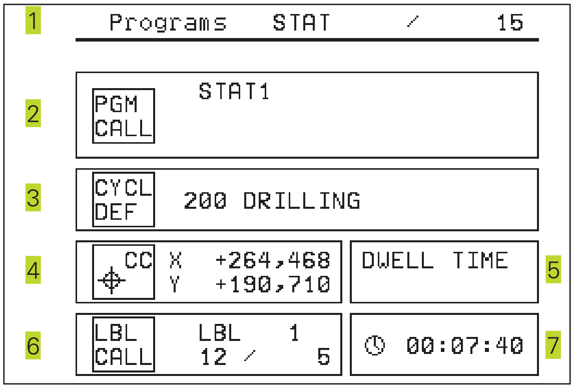
3 Active machining cycle

4 Circle center CC (pole)

5 Dwell time counter

6 Number of the active subprogram or 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

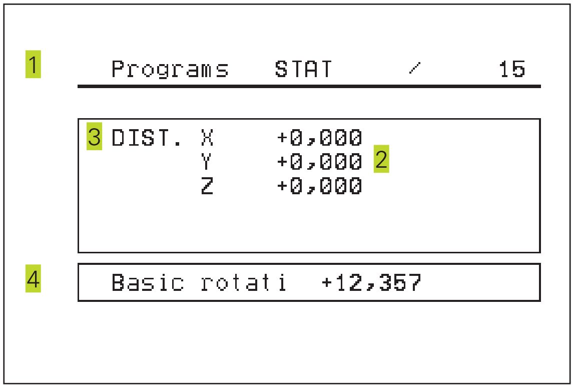
Positions and coordinates

1 Name of main program / Active block number

2 Position display

3 Type of position display, e.g. distance-to-go

4 Angle of a basic rotation



PGM + TOOL STATUS

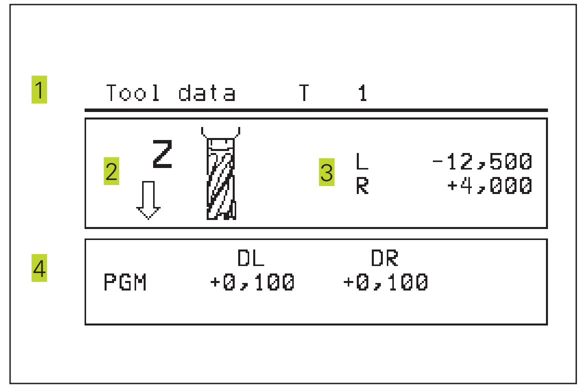
Information on tools

1 T: Tool number

2 Tool axis

3 Tool length and radius

4 Oversizes (delta values) from TOOL CALL block



PGM + COORD. TRANS. STATUS

Coordinate transformations

1 Name of main program / Active block number

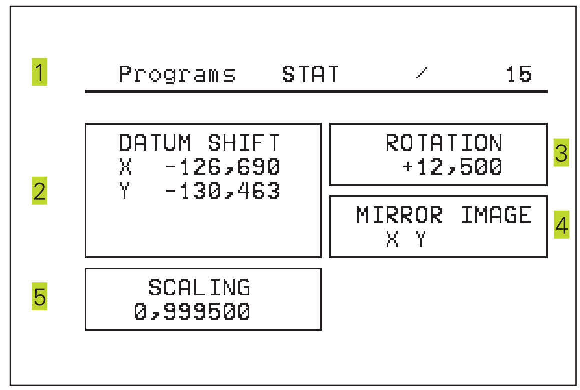
2 Active datum shift (Cycle 7)

3 Active rotation angle (Cycle 10)

4 Mirrored axes (Cycle 8)

5 Active scaling factor (Cycle 11)

For further information, refer to section 8.6 "Coordinate Transformation Cycles."



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

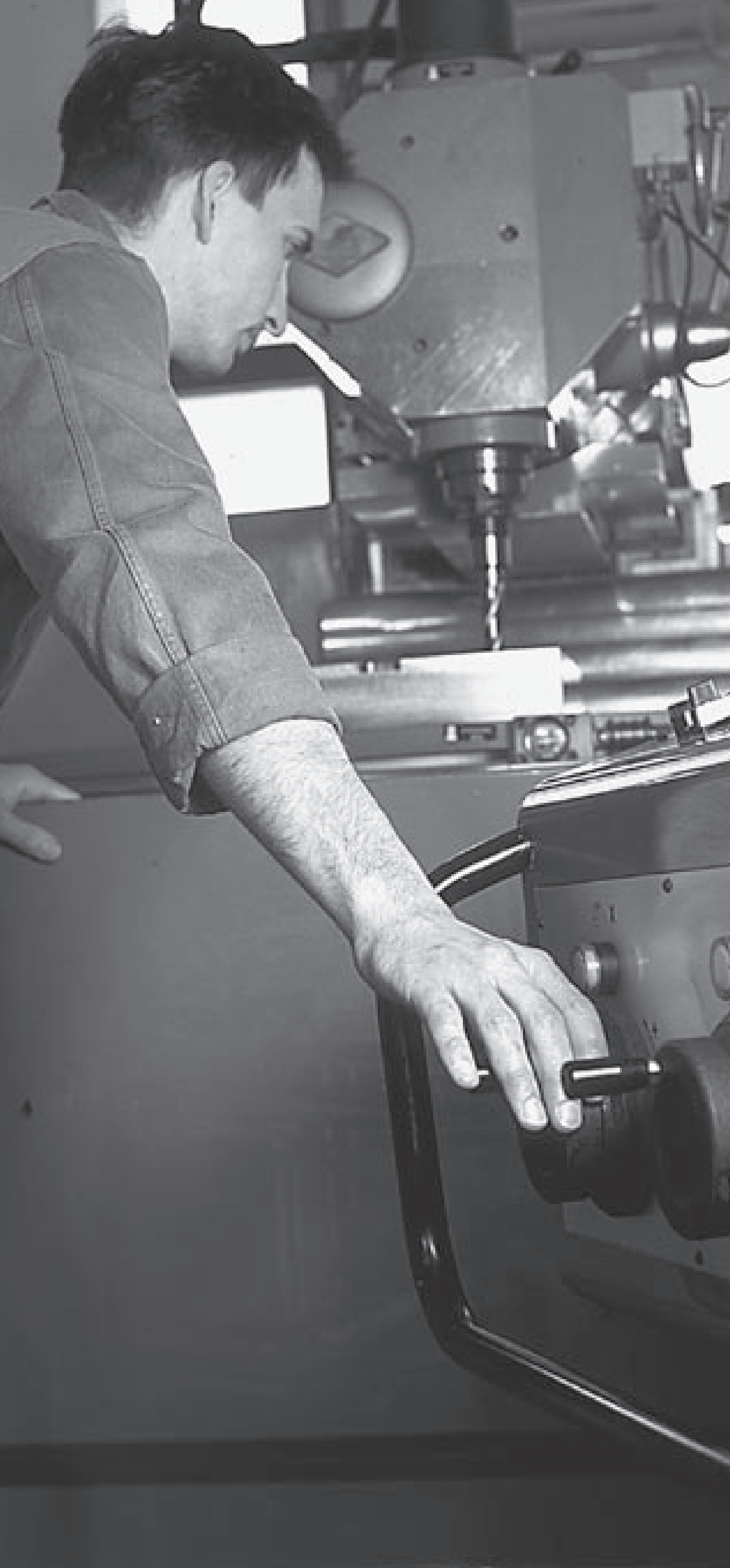
3-DTouch Probes
With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums

TS 220 touch trigger probe
This touch probe is particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable.

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

HR electronic handwheels
Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 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. Your machine manual provides more detailed information.

▶ 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`

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

`TRANSLATE PLC program`

The PLC program of the TNC is automatically compiled.

`Relay Ext. DC Voltage Missing`

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

`Traverse Reference Points`

Z+ X+	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
Z / NC I	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 NC START button.

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

2.2 Moving the Machine Axes

> Traversing the machine axes with the axis direction keys is a machine-dependent function. Refer to your machine tool manual.

Traverse the axis with the axis direction keys

	Select the Manual Operation mode.
X+	Press the axis direction button and hold it as long as you wish the axis to move.

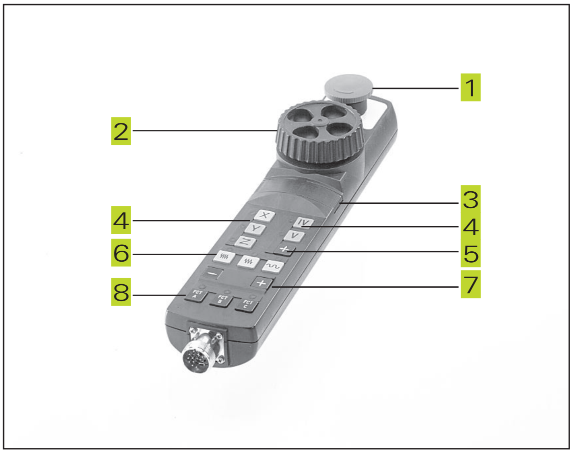
...or move the axis continuously:

X+ and NC I	Press and hold the axis direction button, then press the NC START button: The axis continues to move after you release the keys.
NC 0	Press the NC STOP key to stop the axis.

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)

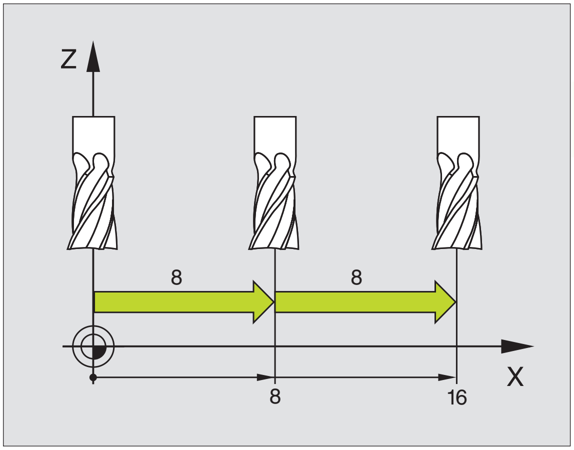
The red indicators show the axis and feed rate you have selected.

To move an axis:

	Select the Manual Operation mode.
ON OFF	Activate handwheel, set soft key to ON
	Press the permissive button.
X	Select the axis on the handwheel
	Select the feed rate.
+ or −	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 axis direction button.



	Select the Manual Operation mode.
INCRE- ON / MENT OFF	Select incremental jog positioning, set the soft key to ON

JOG INCREMENT?

8 ENT	Enter the jog increment in millimeters (here, 8 mm).
1	Select the jog increment via soft key (select 2nd or 3rd soft-key row)
Y+	Press the axis direction button to position as often as desired

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

In the Manual Operation mode, 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: Entering the spindle speed S

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

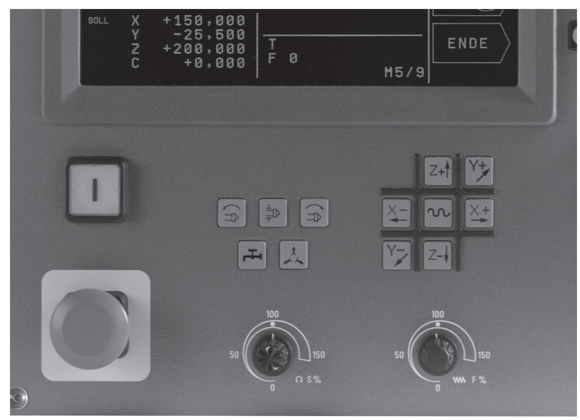
SPINDLE SPEED S=

1000	Enter the desired spindle speed,
NC I	and confirm with the NC START button

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

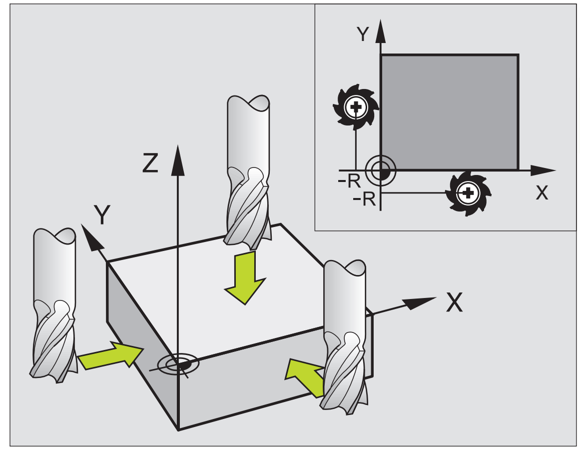
Proceed in the same way to enter the miscellaneous functions M.

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



> The 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 Datum Setting (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.

Preparation

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

Datum setting

Fragile workpiece? If the workpiece surface must not be scratched, you can lay a metal shim of know 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.
Z	Move the tool slowly until it touches the workpiece surface.
DATUM SET	Select the function for setting the datum
Z	Select the axis.

DATUM SET Z=

0 ENT	Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness *d* of the shim. In the tool axis, offset the tool radius.

Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d.

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. You can write the a short program in HEIDENHAIN conversational programming and execute it immediately. You can also call TNC cycles. The program is stored in the file $MDI. In the operating mode Positioning with MDI, the additional status displays can also be activated.

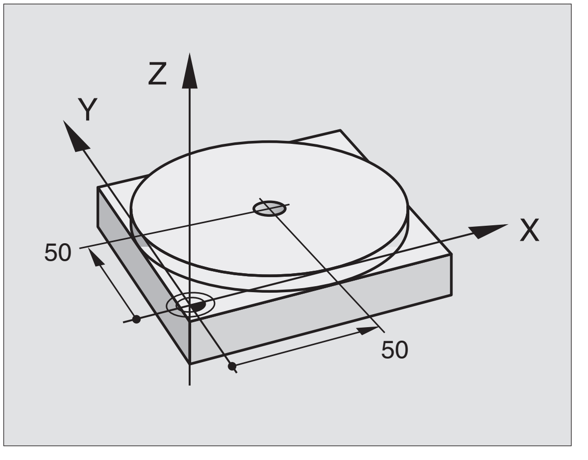
Select the Positioning with MDI mode of operation. Program the file $MDI as you wish.

To start program run, press the machine START button.

Limitations:

The following functions are not available:

- Tool radius compensation
- Programming graphics
- Programmable probing functions
- Subprograms, program section repeats
- Path functions CT, CR, RND and CHF
- Cycle 12 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`	Pos. tool aboveholeatFMAX , spindle On

`5 L Z+5 F2000`	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 R0 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."

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
PGM MGT	Call up the file manager: PGM NAME soft key
	Move the highlight to the $MDI file.
COPY ABC ⇨ XYZ	Select „Copy file“: Press the COPY soft key

Target file =

1225	Enter the name under which you want to save the current contents of the $MDI file.
ENT	Copy the file.
END	Exit the file manager: END 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.

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

4

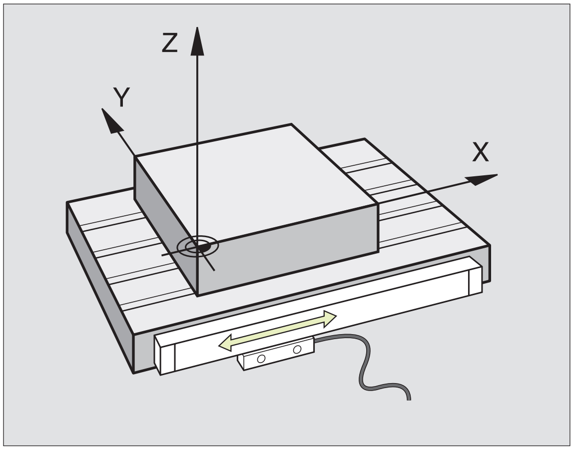
Programming:

Fundamentals of NC, File Management, Programming Aids

4.1 Fundamentals of NC

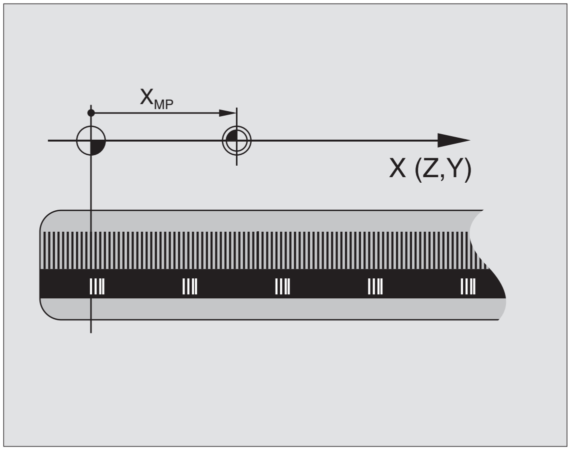
Position encoders and reference marks

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 CNC 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.

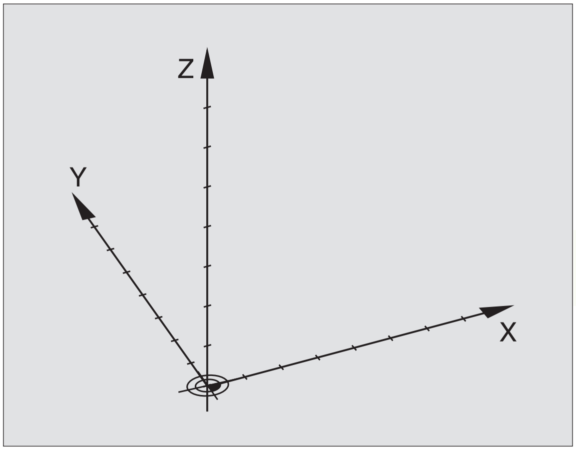


Reference system

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.

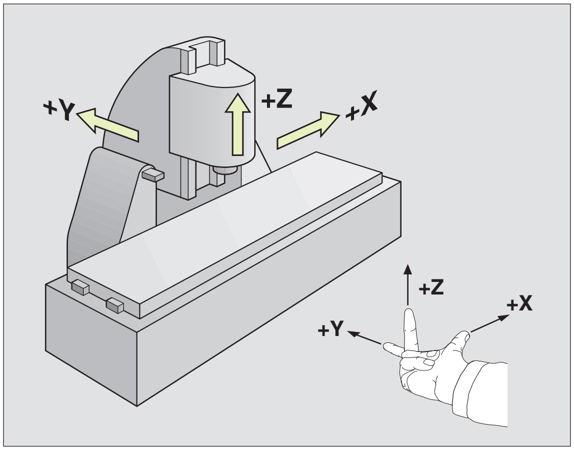
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

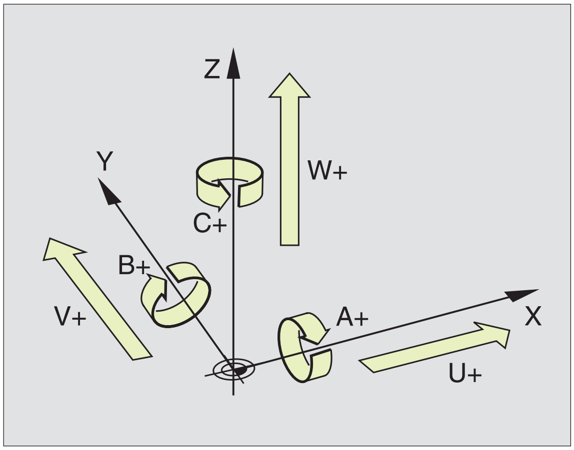


Reference systems on milling machines

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 310 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.





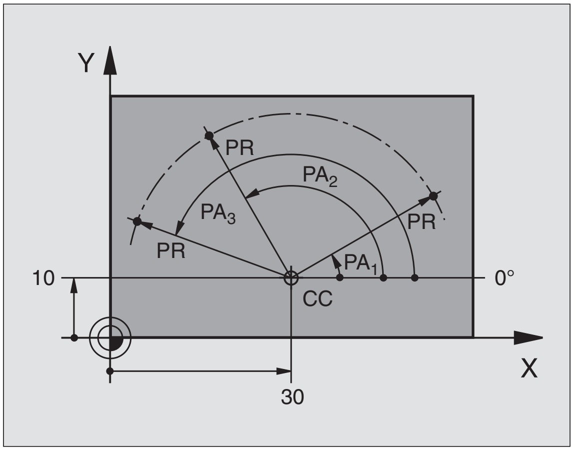
Polar coordinates

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.

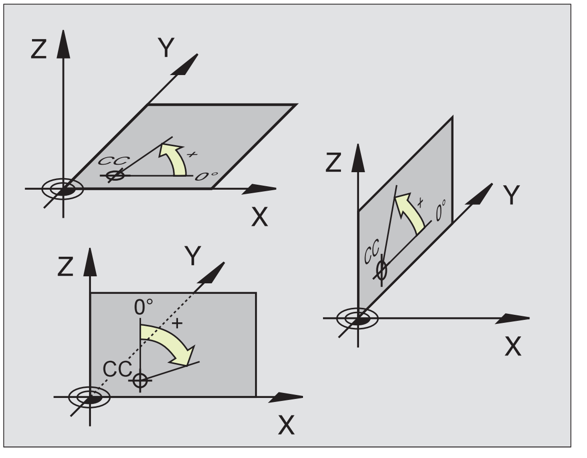
See figure at lower right.



Definition of pole and angle reference axis

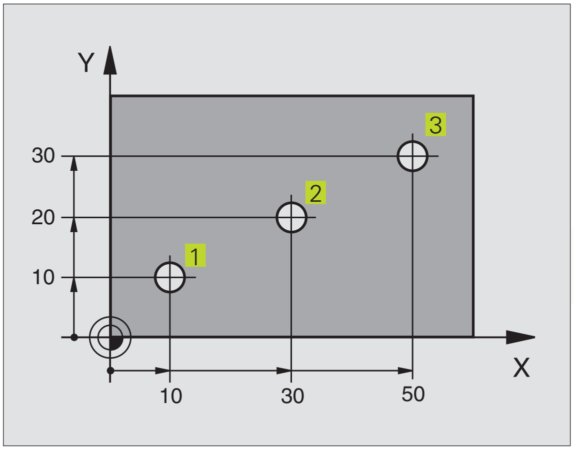
The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle PA.

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.

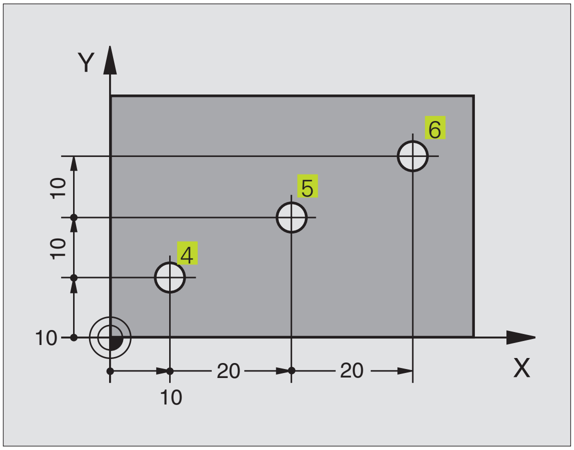


Example 1: Holes dimensioned in absolute coordinates

Hole 1	Hole 2	Hole 3
X=10 mm	X=30 mm	X=50 mm
Y=10 mm	Y=20 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"(soft key) before the axis.



Example 2: Holes dimensioned with relative coordinates

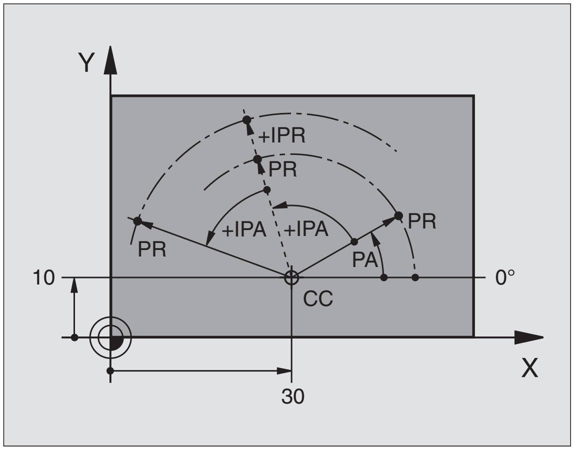
Absolute coordinates of hole 4:

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.



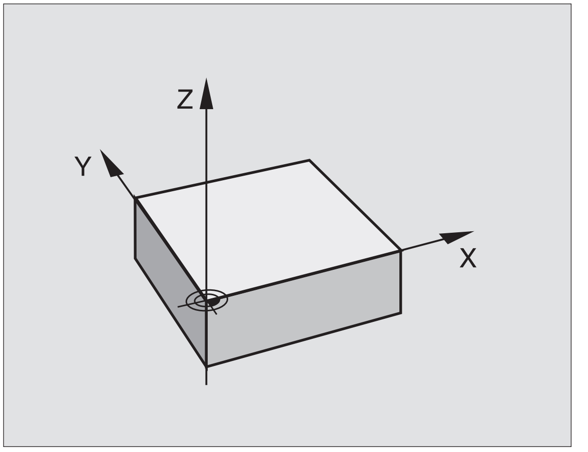
Selecting the datum

A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. Before setting the datum, you align the workpiece with the machine axes and move the tool in each axis to a known position relative to the workpiece. You then set the TNC display 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 8.6 "Coordinate Transformation Cycles."

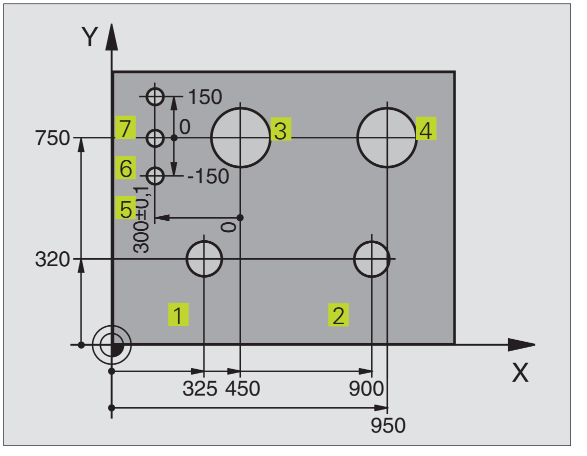
If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece, which is the most suitable for deducing the dimensions of the remaining workpiece positions.

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. 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).

35720	.H
File name	File type

The TNC can manage up to 64 files. Their total size, however, must not exceed 128 MB.

Files in theTNC	Type
Programs in HEIDENHAIN conversational format	.H
Table for Tools	.T
Table for Datums	.D

Working with the file manager

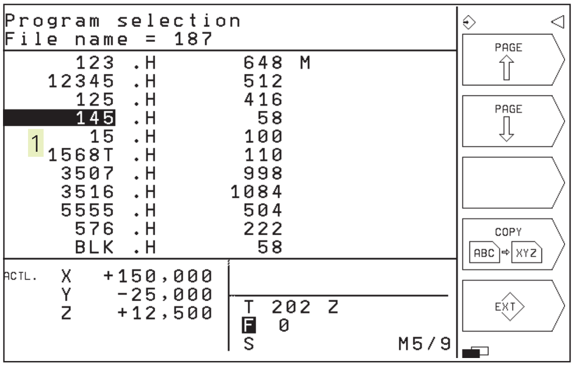
This section informs you about the meaning of the individual screen information, and describes how to select files. If you are not yet familiar with the TNC file manager, we recommend that you read this section completely and test the individual functions on your TNC.

Calling the file manager

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

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

display.	Meaning
FILE NAME	Name with up to 8 characters and file type Number following the name: File size in bytes
Status	Properties of the file:
M	Program is in a Program Run mode of operation.
P	File is protected against editing and erasure (Protected)



Selecting a file

PGM MGT — Calling 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 numbers of the file you wish to select and then press the GOTO key: The highlight moves to the first file that matches these numbers.

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

Copying a file

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

COPY ABC → XYZ

▶ 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.

Renaming a file

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

RENAME ABC = XYZ

▶ Select the renaming function.

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

▶ To execute renaming, press the ENT key.

Deleting a file

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

DELETE

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

▶ To confirm erasure: Press the YES soft key. Abort with the NO soft key if you do not wish to erase the directory

Protecting a file/Canceling file protection

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

PROTECT/ UNPROTECT

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

You also need to enter the code number 86357. To cancel file protection, enter the code number 86357.

Read in/read out files

EXT

▶ To read in or read out files: Press the ENT soft key. The TNC provides the following functions:

Functions for reading in/reading out files	Soft key
Read in all files	TRANSFER EXT → TNC
Only read in selected files; To accept a file suggested by the TNC, press the YES soft key; Press the NO soft key if you do not want to accept it.	TRANSFER EXT ? → TNC
Read in the selected file: Enter the file name	TRANSFER EXT → TNC
Read out the selected file: Move the highlight to the desired file and confirm with ENT	TRANSFER TNC → EXT
Read out all of the files in the TNC memory	TRANSFER TNC → EXT
Display the file directories of the external unit on your TNC screen	SHOW EXT DIRECTORY

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.

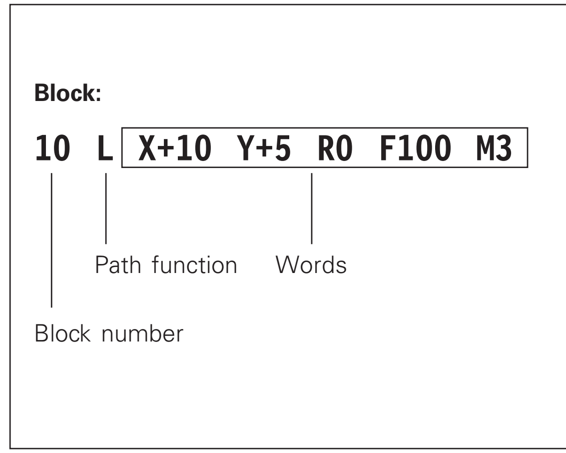
The TNC numbers the blocks in ascending sequence.

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.

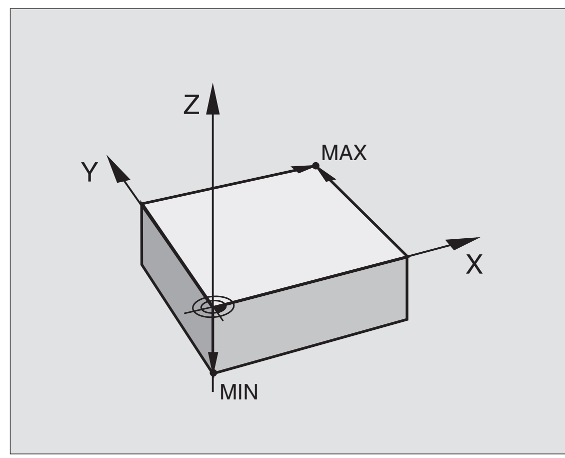


Defining the blank form — BLK FORM

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.

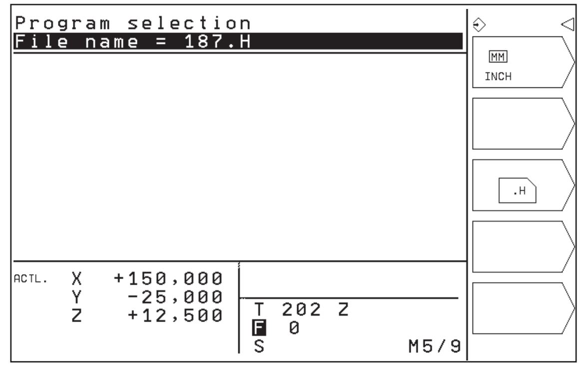
> ☞ The TNC can display the graphic only if the short side of the BLK FORM is longer than 1/64 of the long side.



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.

PGM MGT

Call up the file manager: Press the PGM NAME soft key

File name =

3056 ENT

Enter the new program number and confirm your entry with the ENT key.

File name = 3056.H

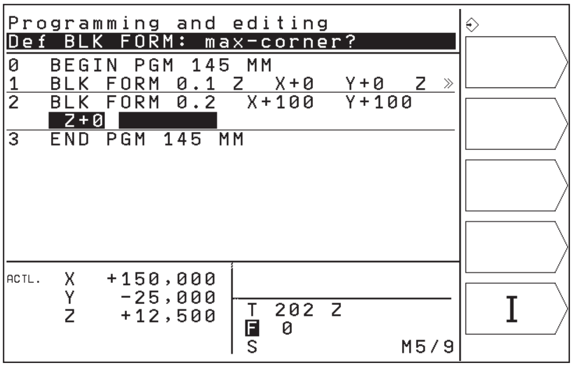
ENT

Select the default setting for unit of measurement (mm): Press the ENT key, or

MM INCH ENT

Switch to inches: Press the MM/INCH soft key and confirm with ENT.

Define the blank



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

Working spindle axis X/Y/Z ?

Z — Enter the spindle axis.

Def BLK FORM: Min corner?

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

0 ENT

-40 ENT

Def BLK FORM: Max-corner?

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

100 ENT

0 ENT

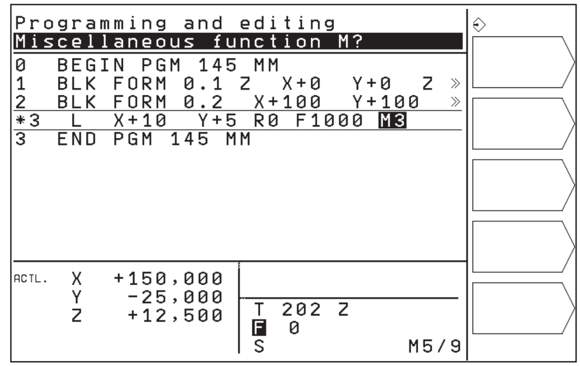
The program blocks window shows the following BLK FORM definition

0 BEGIN PGM 3056 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
3 END PGM 3056 MM	Program end, name, unit of measure

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

Programming tool movements in conversational format

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



Example of a dialog

L	Initiate the dialog.

`Coordinates ?`

X **10**	Enter the target coordinate for the X axis.
Y **5** ENT	Enter the target coordinate for the Y axis, and go to the next question with ENT.

`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 ENT	Enter the miscellaneous function M3 "spindle ON"; pressing the ENT key will terminate this dialog.

The program blocks window will display the following line:

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

Functions during the dialog	Key
Ignore the dialog question	→
End the dialog immediately	END
Abort the dialog and erase the block	DEL

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 top right).

Scrolling through the program

▶ Press the GOTO key

▶ Enter the block number and confirm with ENT, and the TNC will go to the indicated block, or

▶ Press one of the superimposed soft keys to scroll to another page (see table at top right.)

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.

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 location

▶ Select the block after which you want to insert the block you have just edited (deleted.)

▶ If you wish to insert a block you have stored in the buffer memory, press the soft key INSERT NC BLOCK

Editing and inserting words

▶ Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.

▶ To accept the change, press the END key.

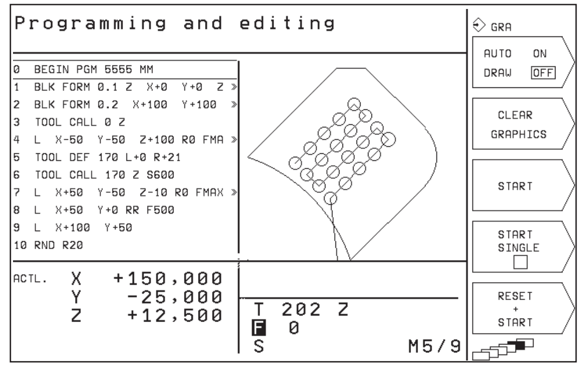
If you want to insert a word, press the horizontal arrow keys repeatedly until the desired dialog appears. You can then enter the desired value.

Selecting blocks or words	Soft keys/keys
Move from one block to the next	↓ ↑
Select individual words in a block	→
Go to the previous page	PAGE ⇧
Go to the next page	PAGE ⇩
Jump to beginning of program	BEGIN ⇧
Jump to beginning End	END ⇩

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

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.



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.

AUTO DRAW ON/OFF

▶ 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.

AUTO DRAW ON does not simulate 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.

RESET + START

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

Additional functions are listed in the table at right.

Functions	Soft key
Generate interactive graphics blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Interrupt interactive graphics This soft key only appears while the TNC generates the interactive graphics	STOP

To erase the graphic:

▷

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

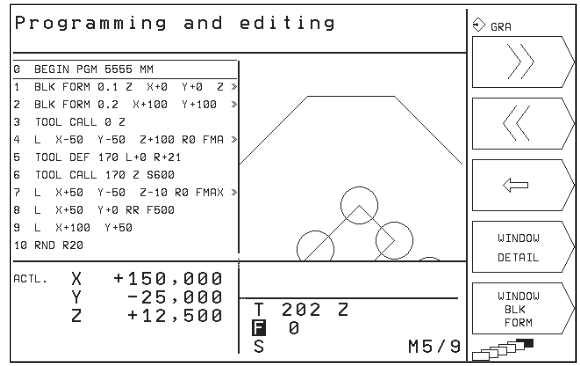
CLEAR GRAPHICS

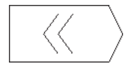
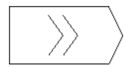
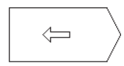
▶ Delete graphic: Press CLEAR GRAPHIC soft key

Magnifying or reducing a detail

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

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



Function	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	
Move the frame overlay to the left: Press and hold the soft key. Move the frame overlay to the right: Press and hold the arrow to the right soft key	

- Confirm the selected section with the WINDOW DETAIL soft key

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

4.5 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

HELP

▶ Press the HELP key

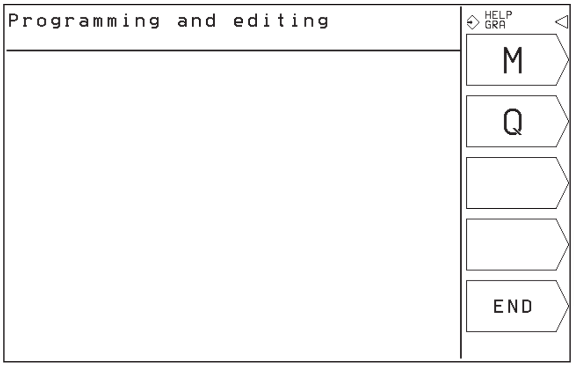
▶ Select a topic: Press one of the available soft keys

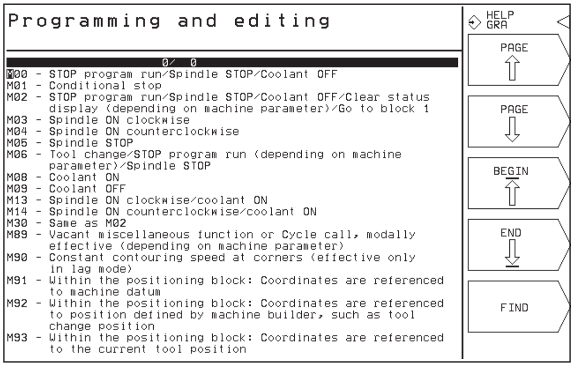
Help topics / Functions	Soft key
M functions	M
Cycle parameters	Q
HELP that is entered by the machine manufacturers (optional, not executable)	PLC
Go to previous page	PAGE ⇧
Go to next page	PAGE ⇩
Go to beginning of file	BEGIN ⇧
Go to end of file	END ⇩
Select search functions; Enter a number, Begin search with ENT key	FIND

The HELP provided by the machine manufacturer can only be displayed and not executed.

End the HELP function

Press the END key.





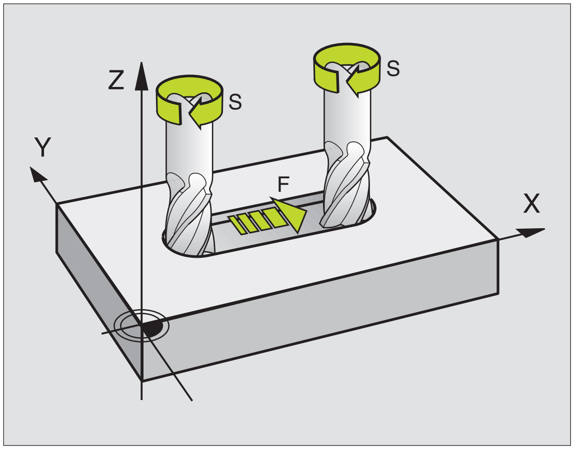
5

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 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.

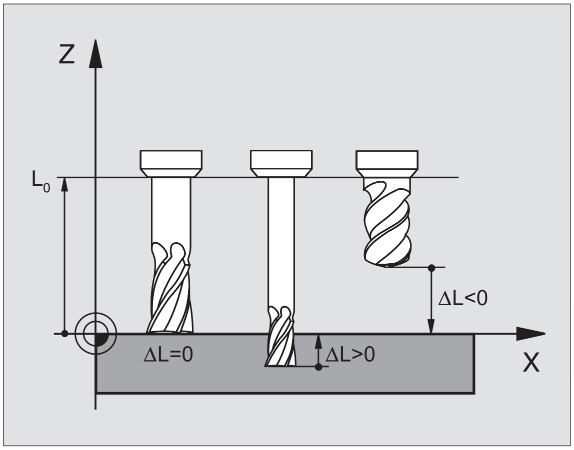
Programmed change
In the part program, you can change the spindle speed in a TOOL CALL block by entering the spindle speed only:

TOOL CALL

- To program a tool call, press the TOOL CALL soft key (3rd soft-key row)
- Ignore the dialog question for „Tool number ?" with the right arrow key
- Ignore the dialog question for „Working spindle axis X/Y/Z ?" with the right arrow key
- Enter the new spindle speed for the dialog question "Spindle speed S= ?".

Changing 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 tool tables. The TNC will consider all of the data entered 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“ 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

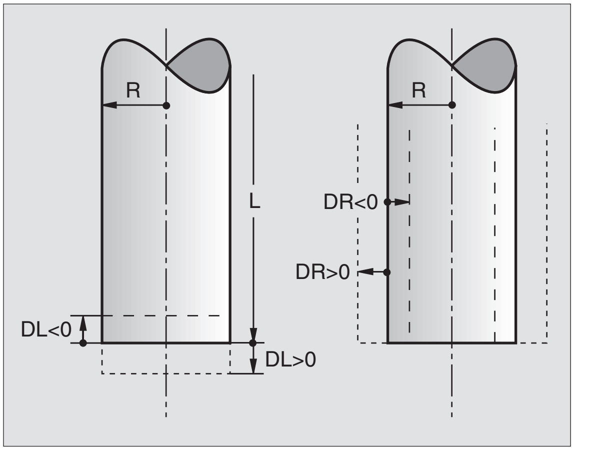
You can enter the tool radius R directly.

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), a negative delta value describes a tool undersize (DR<0). Enter the delta values when you are programming with TOOL CALL.

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.

TOOL DEF

- To select tool definition, press the TOOL DEF key.
- Enter the Tool number: Each tool is uniquely identified by its number. When the tool table is active, enter tool numbers greater than 99 (dependent on MP7260)
- Enter the tool length: Enter the compensation value for the tool length.
- Enter 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.POSY or CUR.POS Z".

Resulting NC block:

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

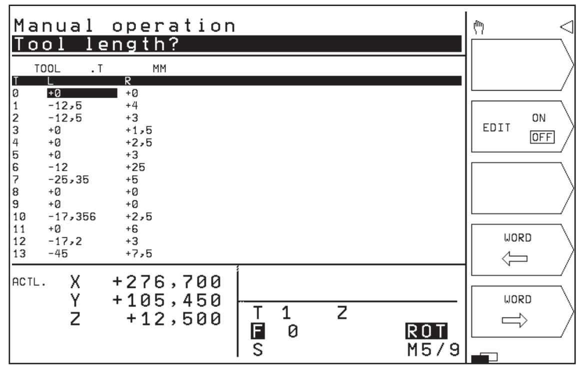
Entering tool data in tables

You can define and store up to 254 tools and their tool data in the tool table (the maximum number of tools in the table can be set in machine parameter 7260).

Tool table: Available input data

Abbr.	Input	Dialog
T	Number by which the tool is called in the program	–
L	Value for tool length compensation L	Tool length?
R	Compensation value for the tool radius R	Tool radius?

Editing the tool table



The tool table has the name TOOL.T is automatically active in a program run operating mode.

To open the tool table TOOL.T:

▶ Select any machine operating mode.

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

EDIT ON OFF ▶ Set the EDIT soft key to ON.

▶ Select the Programming and Editing mode of operation.

PGM MGT ▶ Calls the file manager.

▶ Move the highlight to TOOL.T. 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 center right). You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table on the next page.

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.

To leave the tool table:

▶ Finish editing the tool table: Press the END key.

▶ Call the file manager and select a file of a different type, e.g. a part program.

Editing functions for tool tables	Soft key
Take the value from the position display	ACTUAL POS. Z
Select previous page in table (2nd soft-key row)	PAGE ⇧
Select next page in table (2nd soft-key row)	PAGE ⇩
Move the highlight one column to the left	WORD ⇦
Move the highlight one column to the right	WORD ⇨
Delete incorrect numerical value, re-establish preset value	CE
Re-establish the last value stored	DEL
Move the highlight back to beginning of line	END □

Calling tool data

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

TOOL CALL

- ▶ 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.
- ▶ 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 change

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

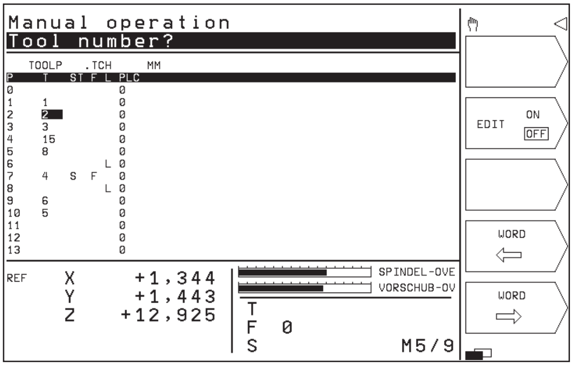
Tool change position
A tool change position must be approachable without collision. With the miscellaneous functions M91 and M92, you can enter machine-referenced (rather than workpiece-referenced) coordinates for the tool change position. If TOOL CALL 0 is programmed before the first tool call, the 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").

Pocket table for tool changer

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



To select the pocket table:

▶ In the Programming and Editing mode,

PGM MGT
- ▶ Calls the file manager.
- ▶ Move the highlight to TOOLP.TCH. 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 ON OFF
- ▶ 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.

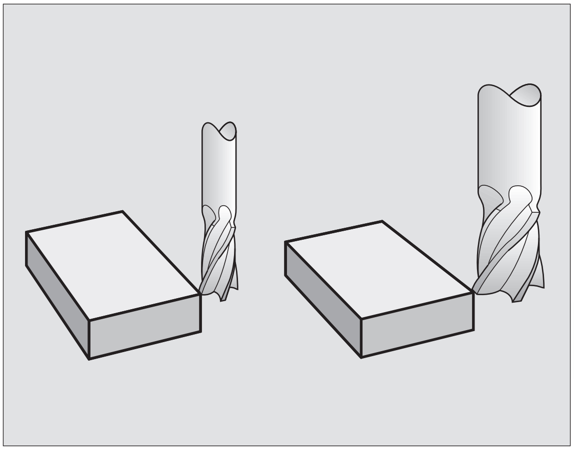
You may not use a tool number twice in the pocket table. If you do so the TNC will output an error message when you exit the table.

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

Editing functions for pocket table	Soft key
Select previous page in table (2nd soft-key row)	PAGE ⇧
Select next page in table (2nd soft-key row)	PAGE ⇩
Move the highlight one column to the left	WORD ⇦
Move the highlight one column to the right	WORD ⇨
Reset pocket table	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 large radius requiring more than one pocket (**ST**: 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?

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.

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 the TOOL CALL block into account:

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

L	is the tool length L from the TOOL DEF block or tool table
$DL_{TOOL\ CALL}$	is the oversize for length DL in the TOOL CALL block (not taken into account by the position display)

Tool radius compensation

The NC block for programming a tool movement contains:

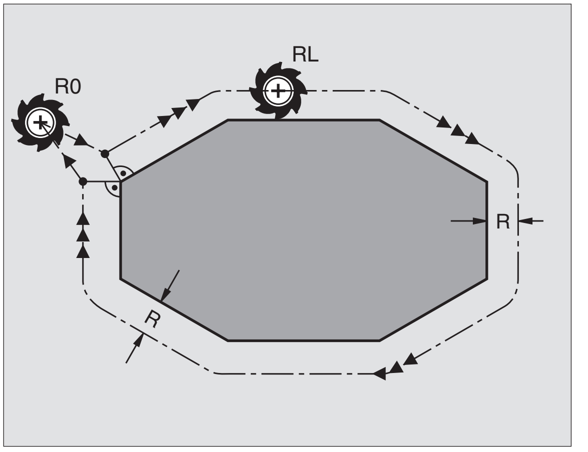
- RL or RR for compensation in the tool radius
- R+ or R– for radius compensation in single-axis movements
- R0 if no radius compensation is required

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 the TOOL CALL block into account:

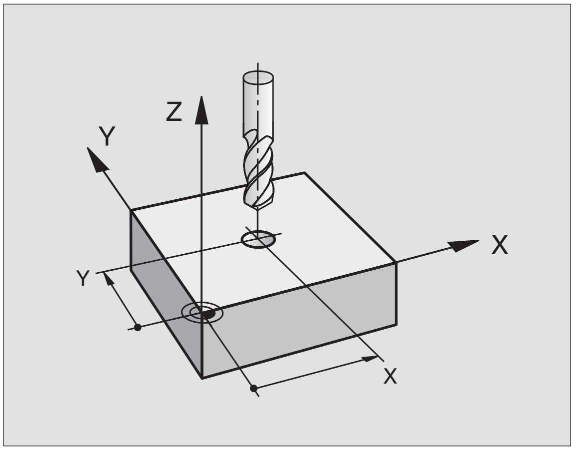
Compensation value = R + $DR_{TOOL\ CALL}$, where

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)



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.
>
> 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:

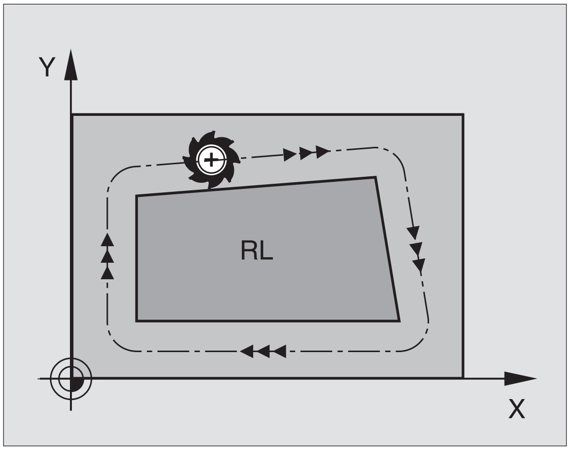
`Radius comp.: RL/RR/no comp. ?`

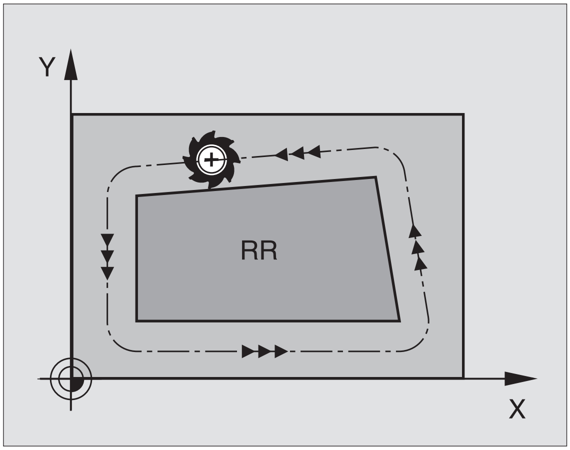
RL — To select tool movement to the left of the contour, press the RL soft key, or

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

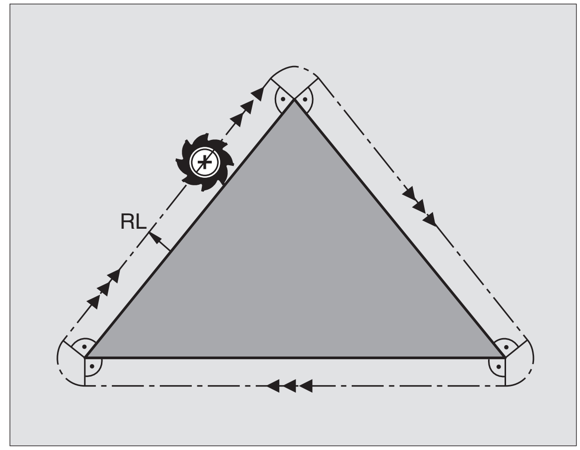
ENT — To select tool movement without radius compensation or to cancel radius compensation, press the ENT key or the R0 soft key.

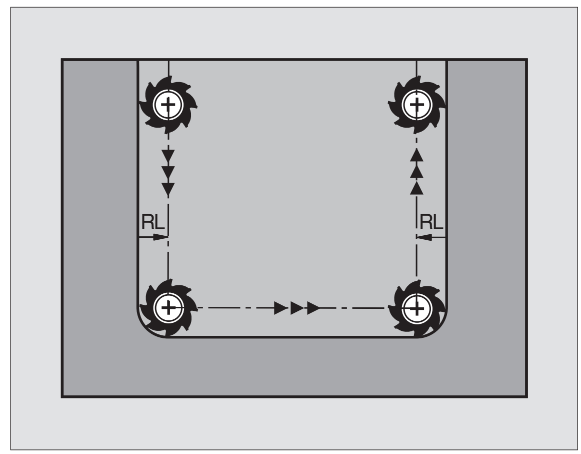
END — To terminate the dialog, press the END key.





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.

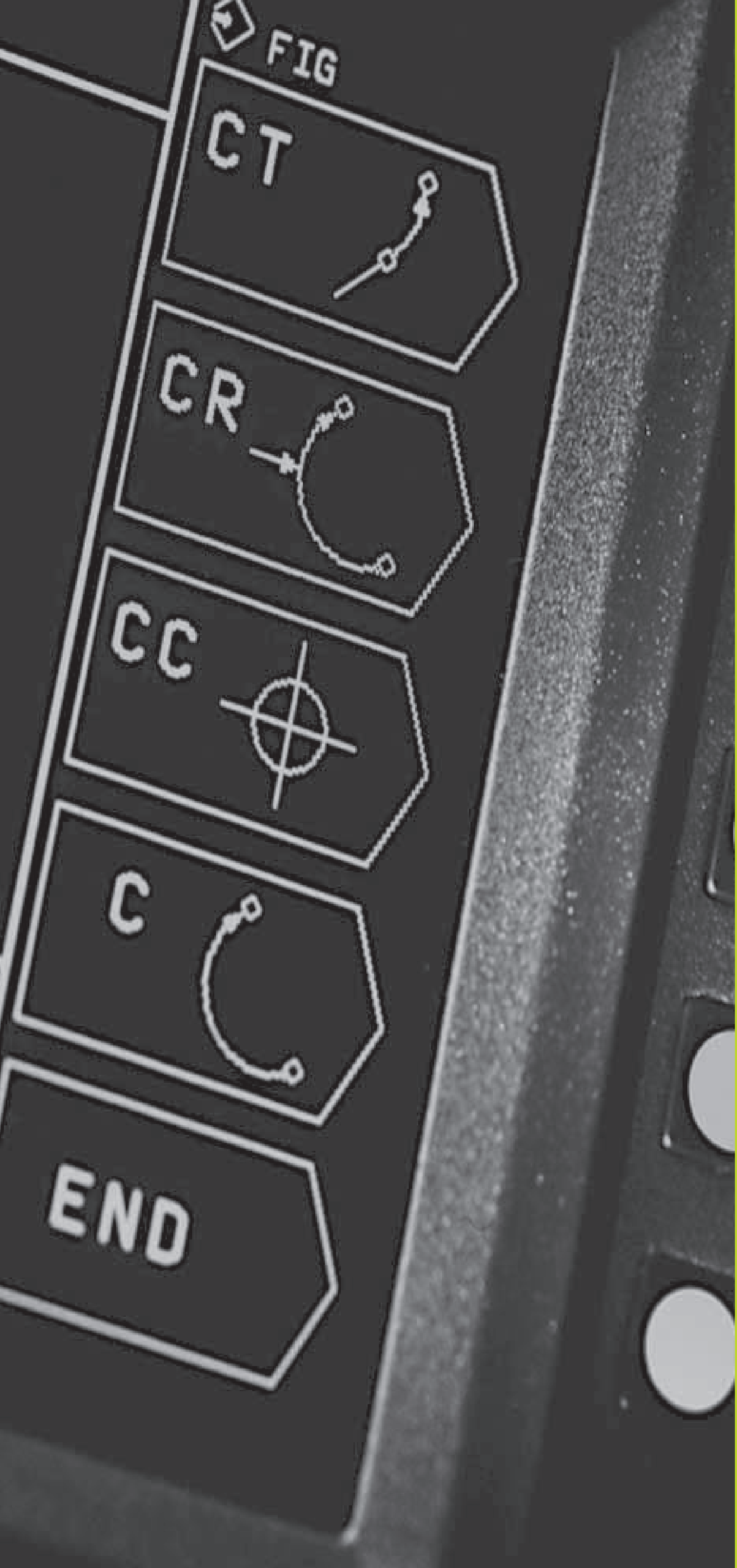
Inside corners

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 function M90. See "7.4 Miscellaneous Functions for Contouring Behavior."



6

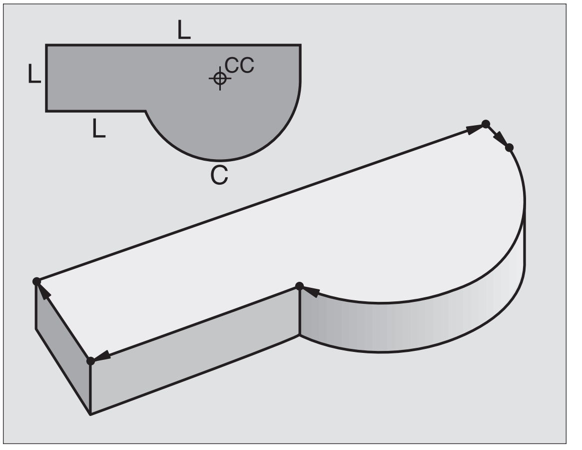
Programming:

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**.



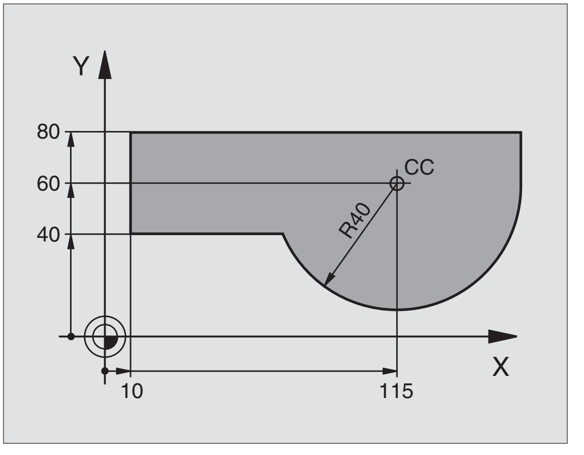
Miscellaneous functions M

With the TNC's miscellaneous functions you can affect

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

Subprograms and program section repeats

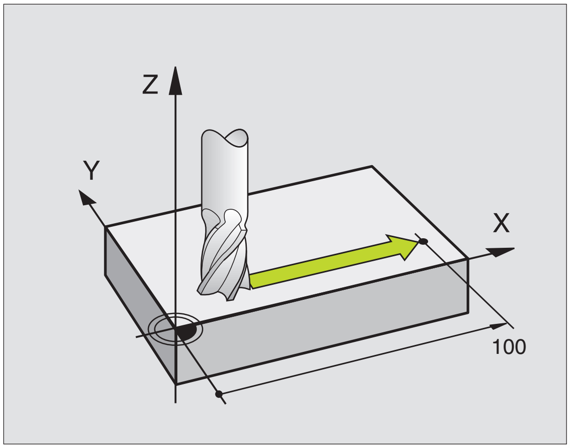
If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat. If you wish to execute a specific program section only under certain conditions, you also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



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

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.

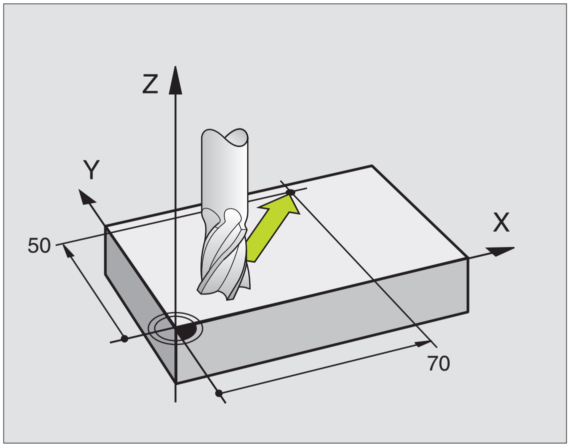
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

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.



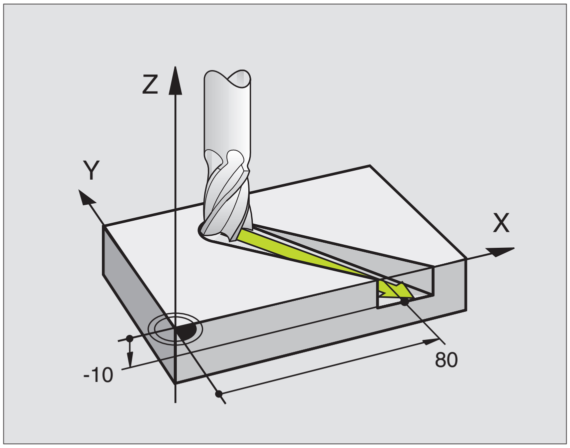
Movement in the main planes

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 XY plane to the position X=70, Y=50. See figure at center right.



Three-dimensional movement

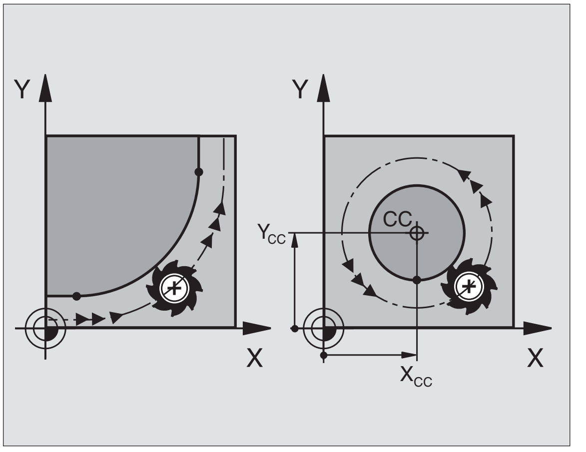
The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example:

```
L X+80 Y+0 Z-10
```

See 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.

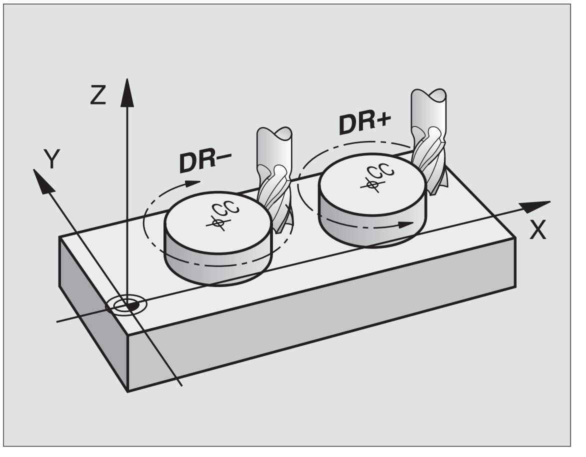


When you program a circle, the TNC assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a tool call:

Spindle axis	Main plane
Z	XY
Y	ZX
X	YZ

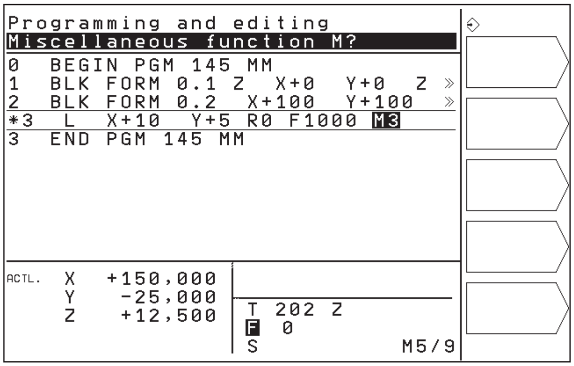
Direction of rotation DR for circular movements
When a circular path has no tangential transition to another contour element, enter the direction of rotation DR:

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



Radius compensation
Radius compensation must be programmed before the block containing the coordinates for the first contour element. You cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line 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

Use the path function keys to open a conversational dialog. The TNC asks you successively for all the necessary information and inserts the program block into the part program.

> You may not program controlled and non-controlled axes in the same block.

Example — programming a straight line:

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

Coordinates ?

X 10 / Y 5 / ENT — Enter the coordinates of the straight-line end point.

ACTUAL POSITION — Transfer the coordinates of the selected axis: Press ACTUAL POSITION soft key (second soft-key row)

Radius comp.: RL/RR/NOcomp. ?

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

Feed rate F=

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

Miscellaneous function M ?

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

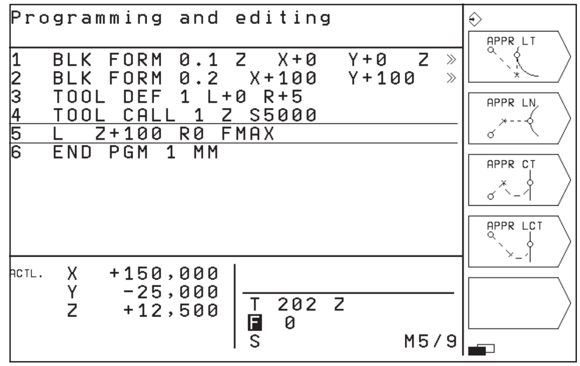
The part program now contains the following line:

```
L X+10 Y+5 RL F100 M3
```

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 following contour forms using soft keys.

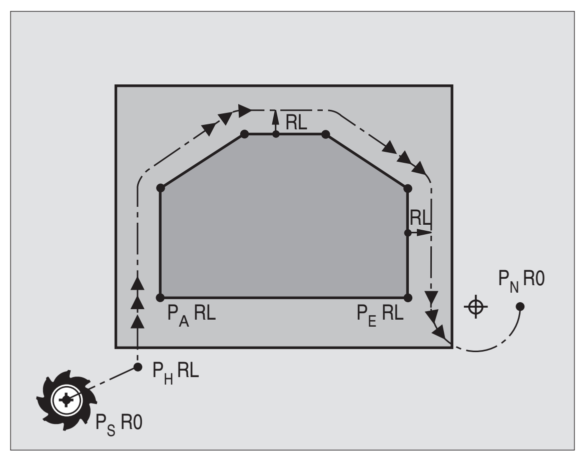


Function Soft keys:	Approach	Departure
Straight line with tangential connection	APPR LT	DEP LT
Straight line perpendicular to a contour point	APPR LN	DEP LN
Circular arc with tangential connection	APPR CT	DEP CT
Circular arc with tangential connection to the contour. Approach and departure to an auxiliary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

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.

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.

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

Radius compensation

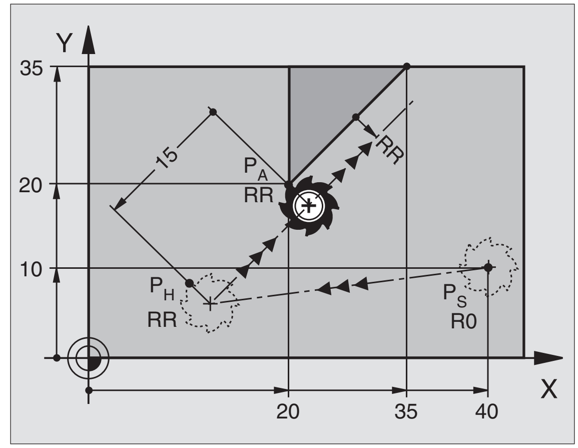
For the TNC to interpret an APPR block as an approach block you must program a change in compensation from R0 to RL/RR. The TNC automatically cancels the radius compensation in a DEP block. If you wish to program a contour element with the DEP block (no change in compensation), then you need to program 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 are 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

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.

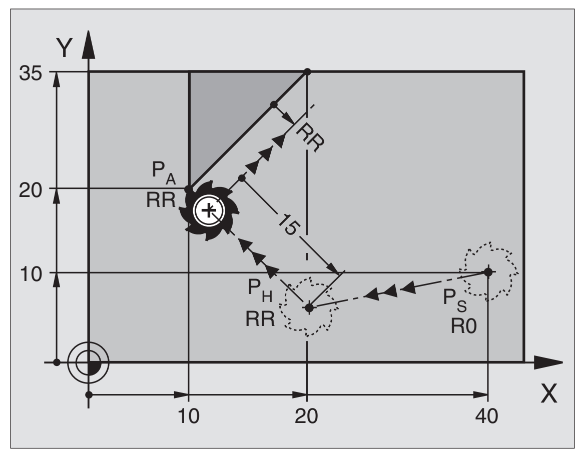
- ▶ Use any path function to approach the starting point P_S.

- ▶ 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

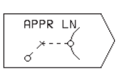
7 L X+40 Y+10 R0 FMAX M3	Approach P_S without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	P_A with radius comp. RR
9 L X+35 Y+35	End point of the first contour element
10 L ...	Next contour element

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

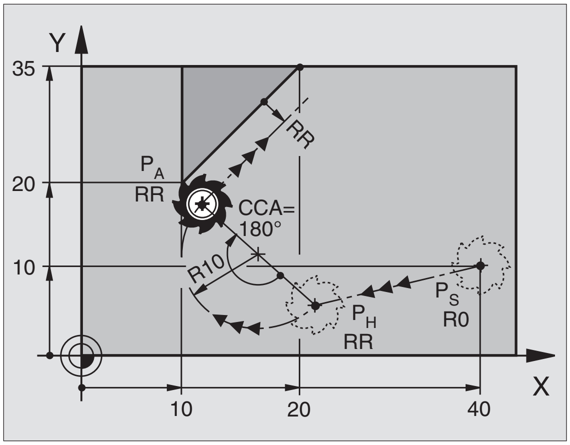
Example NC blocks

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

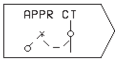
Approaching on a circular arc with tangential connection: APPR CT

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:

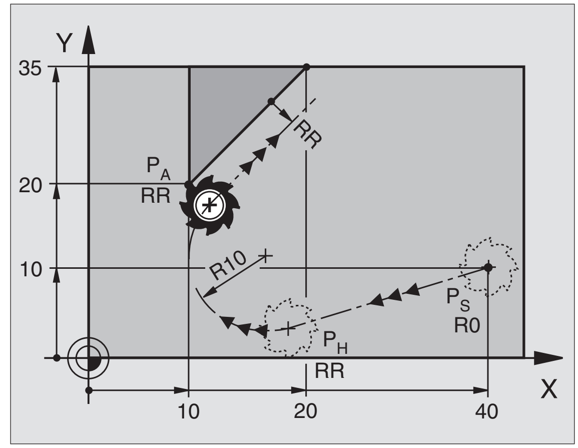


- Coordinates of the first contour point P_A
- Center angle CCA of the arc
  - CCA can be entered only as a positive value.
  - Maximum input value 360°
- Radius R of the circular arc
  - If the tool should approach the workpiece in the direction defined by the radius compensation: Enter R as a positive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

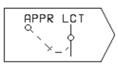
Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT



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

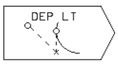
Example NC blocks

`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

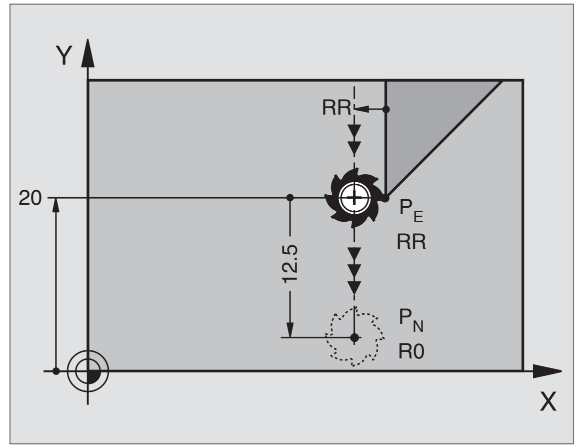
Departing tangentially on a straight line: DEP LT

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.



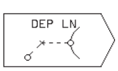
Example NC blocks

`23 L Y+20 RR F100`	Last contour element: P_E with radius compensation
`24 DEP LT LEN12.5 R0 F100`	Depart contour by LEN = 12.5 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

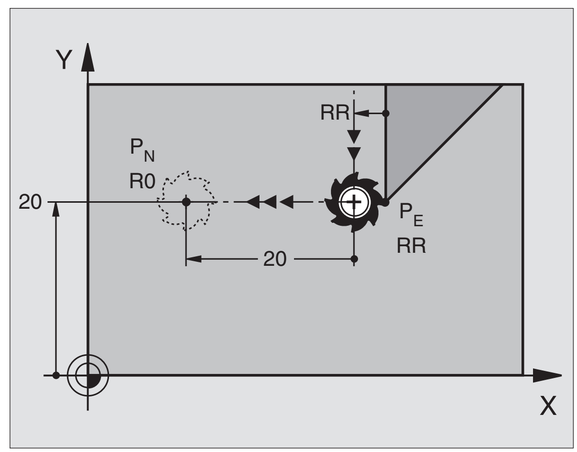
Departing on a straight line perpendicular to the last contour point: DEP LN

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

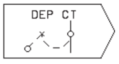
`23 L Y+20 RR F100`	Last contour element: P_E with radius compensation
`24 DEP LN LEN+20 F100`	Depart perpendicular to contour by LEN = 20 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

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.

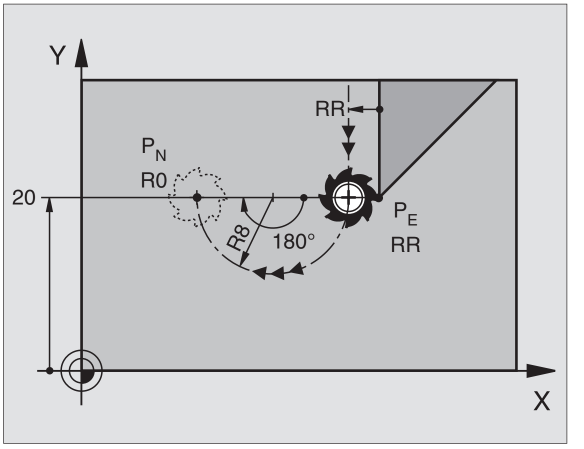
▶ Initiate the dialog with the APPR/DEP key and DEP CT soft key:



▶ Center angle CCA of the arc

▶ Radius R of the circular arc

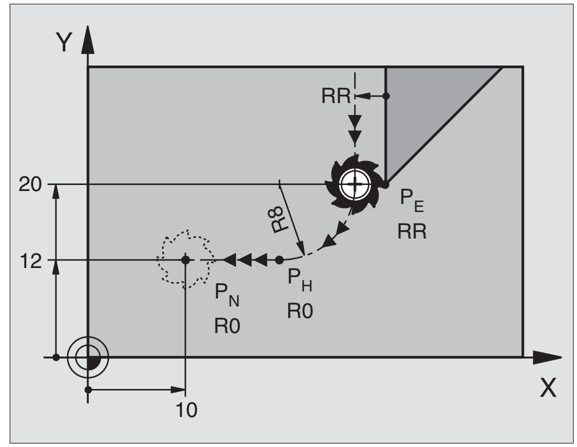
- If the tool should depart the workpiece in the direction of the radius compensation (i.e. to the right with RR or to the left with RL): Enter R as a positive value.
- If the tool should depart the workpiece on the direction **opposite** to the radius compensation: Enter R as a negative value.



Example NC blocks

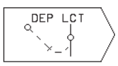
`23 L Y+20 RR F100`	Last contour element: P_E with radius compensation
`24 DEP CT CCA 180 R+8 F100`	Center angle=180°, arc radius=10 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

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:



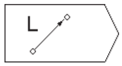
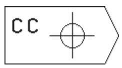
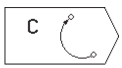
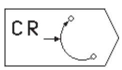
- Enter the coordinates of the end point P_N.
- Radius R of the arc
  Always enter R as a positive value

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 = 10 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

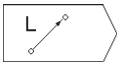
6.4 Path Contours — Cartesian Coordinates

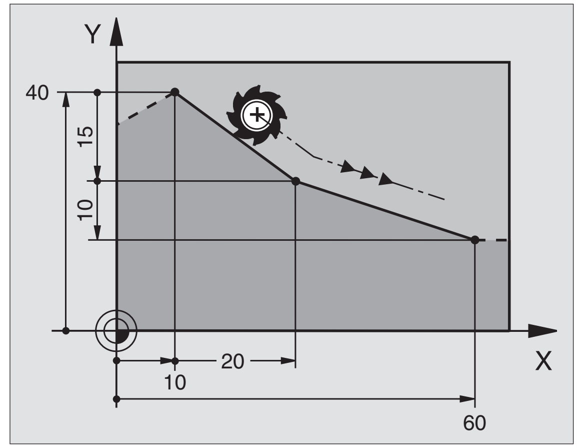
Overview of path functions

Function	Contour function soft key	Tool movement	Required input
Line **L**		Straight line	Coordinates of the straight-line end point
CHam**F**er		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 Circle **T**angential		Circular arc with tangential connection to the preceding contour element	Coordinates of the arc end point
Corner **R**ou**ND**ing		Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding-off radius R

Straight line L

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





▶ Enter the coordinates of the end point.

Further entries, if necessary:

▶ Radius compensation RL/RR/R0

▶ Feed rate F

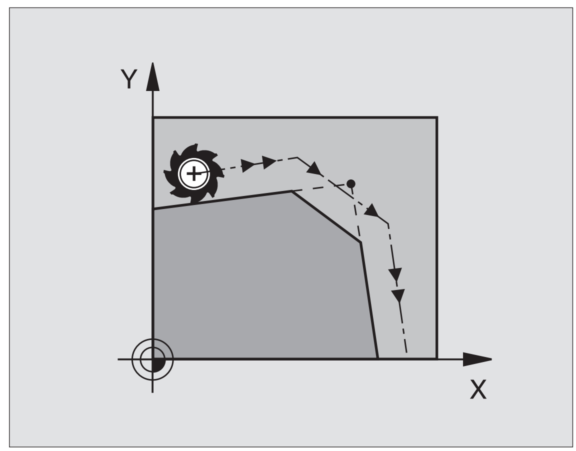
▶ Miscellaneous function M

Example NC blocks

```
7 L X+10 Y+40 RL F200 M3
8 L IX+20 IY-15
9 L X+60 IY-10
```

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.

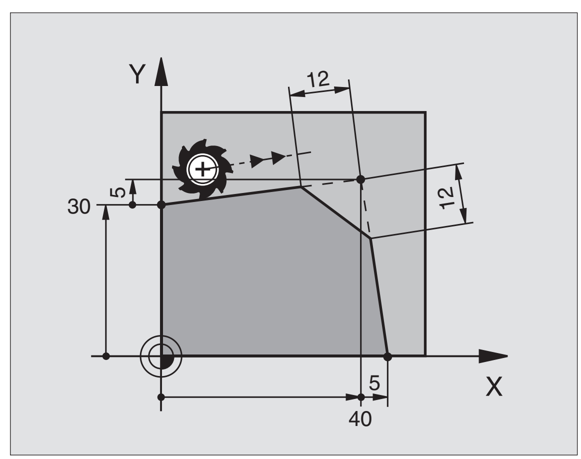
▶ Chamfer side length: Enter the length of the chamfer

Further entries, if necessary:

▶ Feed rate F (only effective in CHF block)

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.

If you have not programmed a feed rate in the CHF block, the TNC will move at the last programmed feed rate.

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

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

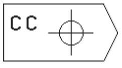
Circle center CC

You can define a circle center CC for circles that are programmed with the C soft 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“ soft key

CIRCLE

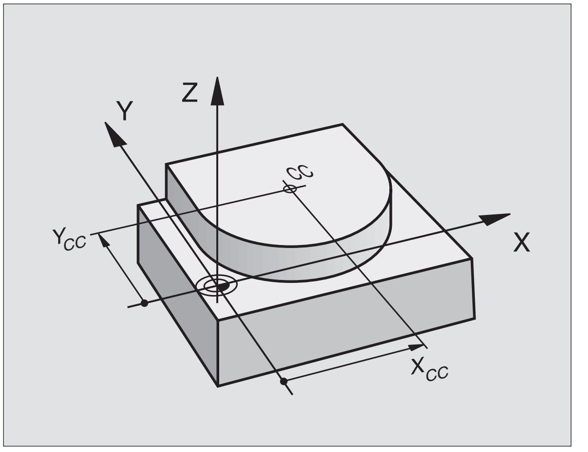
▶ Select circle functions: Press the „CIRCLE“ soft key (2nd soft-key row)



CC

▶ 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.

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 circle center: The tool does not move to this position.

The circle center is also 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.

CIRCLE
▶ Select circle functions: Press the „CIRCLE" soft key (2nd soft-key row)

CC
▶ Enter the coordinates of the circle center.

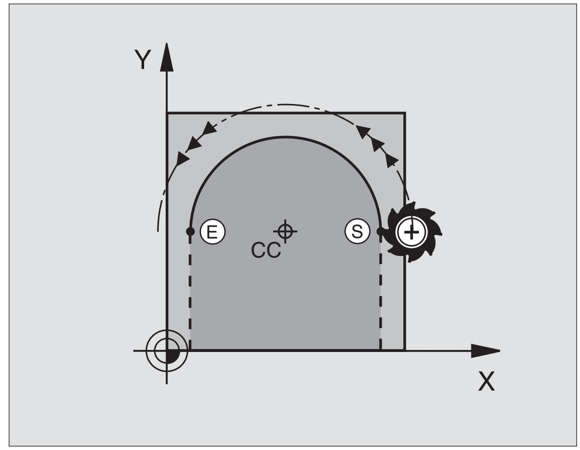
C
▶ Enter the coordinates of the arc end point

▶ Direction of rotation DR

Further entries, if necessary:

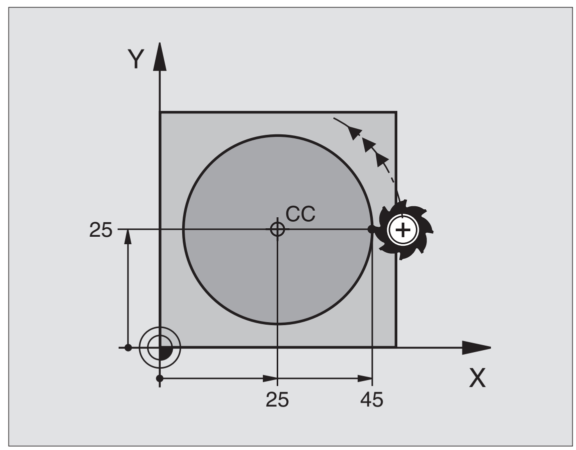
▶ Feed rate F

▶ 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+
```

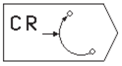


Full circle
Enter the same point you used as the starting point for the end point in a C block.

> 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

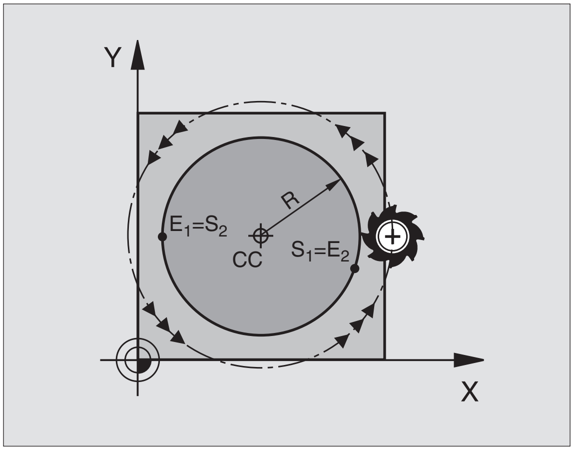
The tool moves on a circular path with the radius R.



- Select circle functions: Press the „CIRCLE" soft key (2nd soft-key row)
- 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:

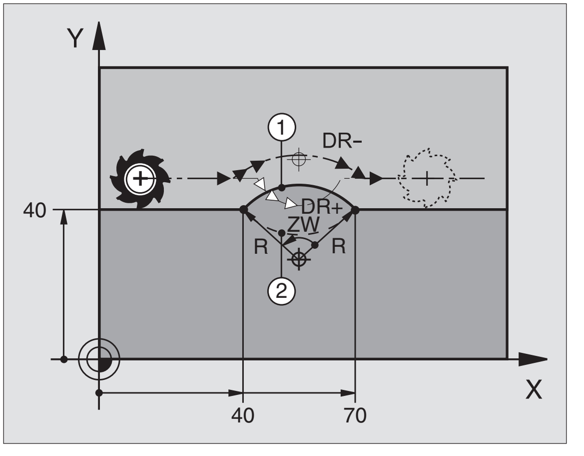
Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

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

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

Convex: Direction of rotation 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

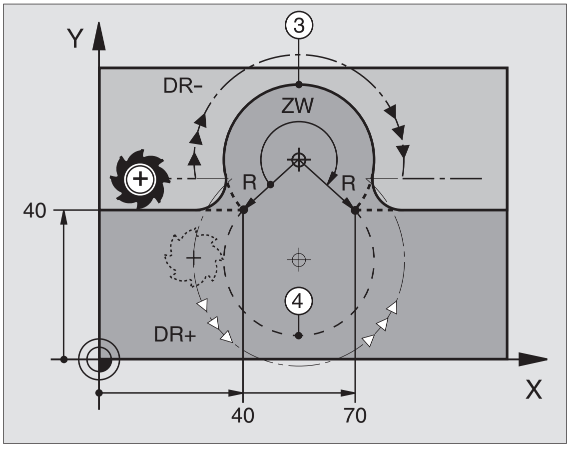
```
11 CR X+70 Y+40 R+20 DR+ (arc 2)
```

or

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

or

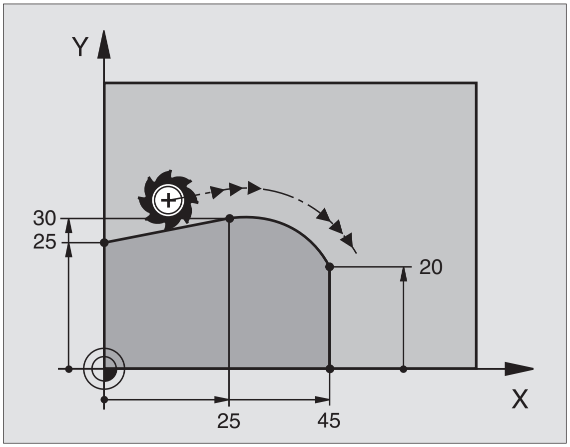
```
11 CR X+70 Y+40 R-20 DR+ (arc 4)
```



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 possible radius is 30 m.

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.

CIRCLE

- Select circle functions: Press the „CIRCLE" soft key (2nd soft-key row)

CT

- 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.

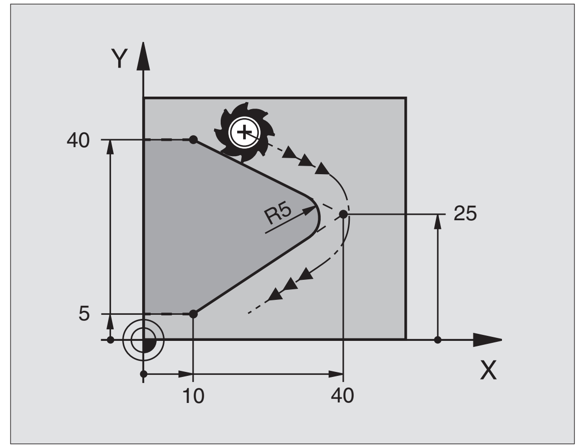
The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

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
```



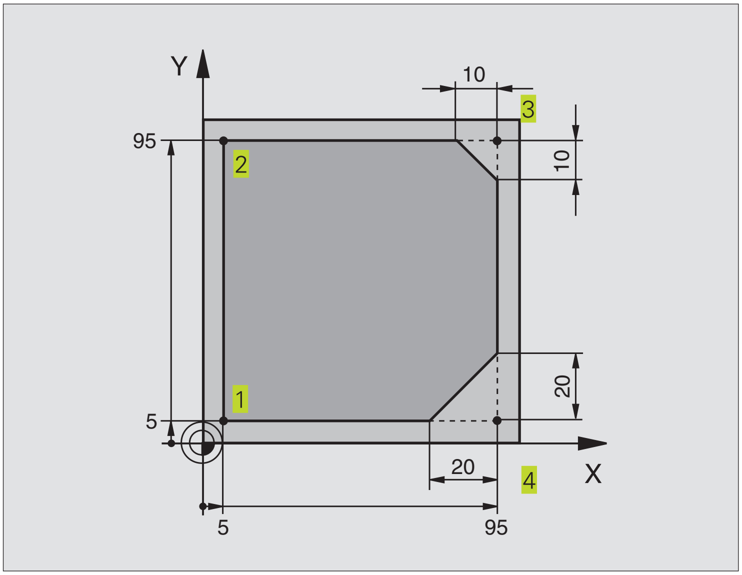
In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc.

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.

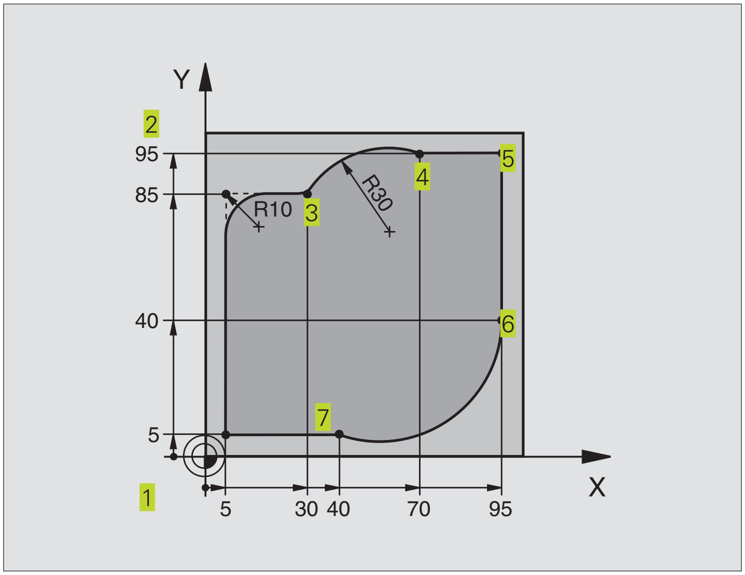
You can also use an RND block for a tangential contour approach if you do not want to use an APPR function.

Example: Linear movements and chamfers with Cartesian coordinates



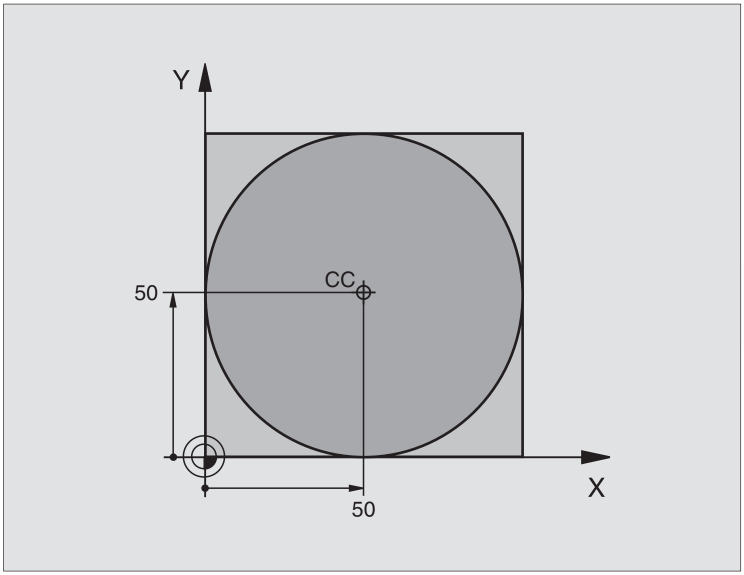
`0 BEGIN PGM 10 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define blank form for graphic workpiece simulation
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+10`	Define tool in the program
`4 TOOL CALL 1 Z S4000`	Call tool in the spindle axis and with the spindle speed S
`5 L Z+250 R0 F MAX`	Retract tool in the spindle axis at rapid traverse FMAX
`6 L X-20 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 L X+5 Y+5 RL F300`	Approach the contour at point 1
`9 RND R2`	Tangential approach to circle with R=2 mm
`10 L Y+95`	Move to point 2
`11 L X+95`	Point 3: first straight line for corner 3
`12 CHF 10`	Program chamfer with length 10 mm
`13 L Y+5`	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
`14 CHF 20`	Program chamfer with length 20 mm
`15 L X+5`	Move to last contour point 1, second straight line for corner 4
`16 RND R2`	Tangential departure from circle with R=2 mm
`17 L X-20 R0 F1000`	Retract tool in the working plane
`18 L Z+250 R0 F MAX M2`	Retract tool in the spindle axis, end of program
`19 END PGM 10 MM`	

Example: Circular movements with Cartesian coordinates



0 BEGIN PGM 20 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 TOOL DEF 1 L+0 R+10	Define tool in the program
4 TOOL CALL 1 Z S4000	Call tool in the spindle axis and with the spindle speed S
5 L Z+250 R0 F MAX	Retract tool in the spindle axis at rapid traverse FMAX
6 L X-20 Y-20 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 L X+5 Y+5 RL F300	Approach the contour at point 1
9 RND R2	Tangential approach to circle with R=2 mm
10 L Y+85	Point 2: first straight line for corner 2
11 RND R10 F150	Insert radius with R = 10 mm, feed rate: 150 mm/min
12 L X+30	Move to point 3: Starting point of the arc with CR
13 CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
14 L X+95	Move to point 5
15 L Y+40	Move to point 6
16 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
17 L X+5	Move to last contour point 1
18 RND R2	Tangential departure from circle with R=2 mm
19 L X-20 Y-20 R0 F1000	Retract tool in the working plane
20 L Z+250 R0 F MAX M2	Retract tool in the spindle axis, end of program
21 END PGM 20 MM	

Example: Full circle with Cartesian coordinates



0 BEGIN PGM 30 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
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	tool call
5 CC X+50 Y+50	Define the circle center
6 L Z+250 R0 F MAX	Retract the tool
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 L X+0 Y+50 RL F300	Approach starting point of circle
10 RND R2	Tangential approach to circle with R=2 mm
11 C X+0 DR-	Move to the circle end point (= circle starting point)
12 RND R2	Tangential departure from circle with R=2 mm
13 L X-40 Y+50 R0 F1000	Retract tool in the working plane
14 L Z+250 R0 F MAX M2	Retract tool in the spindle axis, end of program
15 END PGM 30 MM	

6.5 Path Contours— Polar 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	Contour function soft keys	Tool movement	Required input
Line **LP**	L + P	Straight line	Polar radius, polar angle of the straight-line end point
Circular arc **CP**	C + 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**	CT + P	Circular path with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helix (Helix)	C + P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis

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.

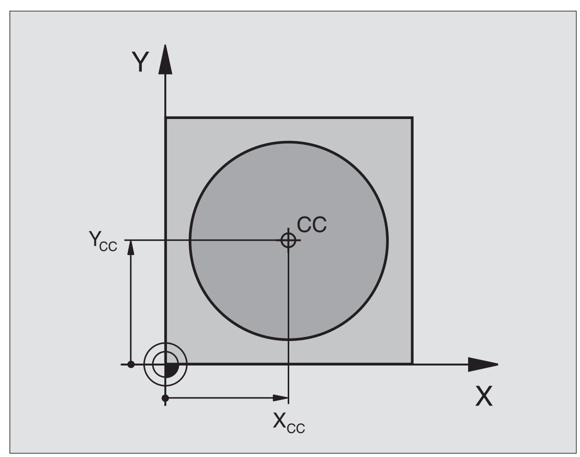
CIRCLE

▶ Select circle functions: Press the „CIRCLE" soft key

CC

▶ 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

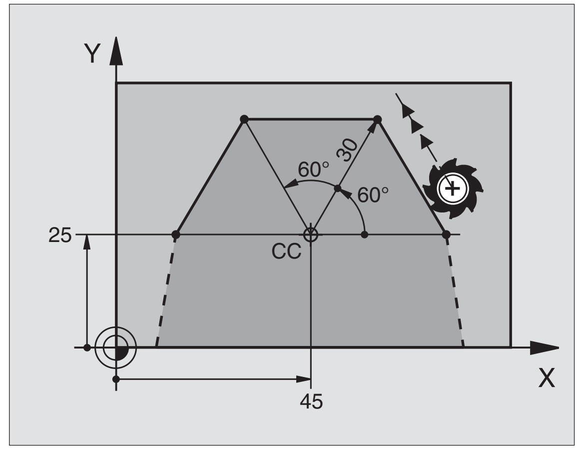
The tool moves in a straight line from its current position to the straight-line end point. The starting point is the end point of the preceding block.

▶ Select straight line function: Press the L soft key

▶ Select entry of polar coordinates: Press the P soft key (2nd soft-key row). 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°

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



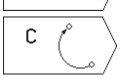
Example NC blocks

```
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
```

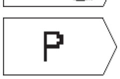
Circular path CP around pole CC

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.

▶ Select circle functions: Press the „CIRCLE“ soft key



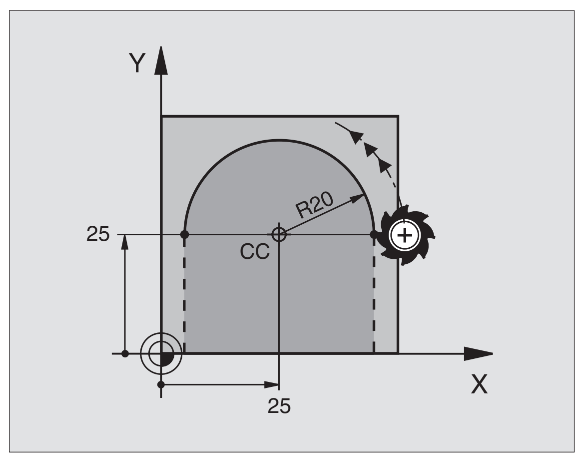
▶ Select circular path C: Press the C soft key



▶ Select entry of polar coordinates: Press the P soft key (2nd soft-key row).

▶ Polar coordinates angle PA: Angular position of the arc end point between –5400° and +5400°

▶ Direction of rotation DR



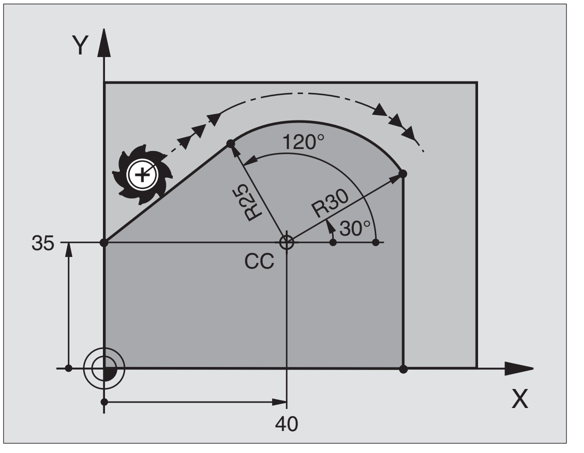
Example NC blocks

```
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.



CIRCLE
- Select circle functions: Press the „CIRCLE“ soft key

CT
- Select the circular path CT: Press the CT soft key

P
- Select entry of polar coordinates: Press the P soft key (2nd soft-key row).
- 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

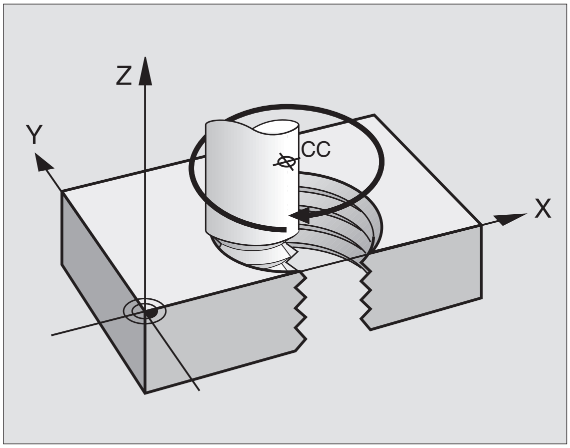
```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```

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.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

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

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

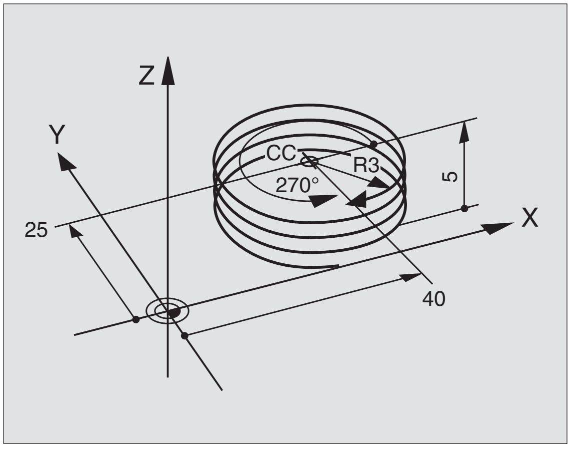
Thread revolutions n	Thread revolutions + thread overrun at the start and end of the thread
Total height h	Thread pitch P x thread revolutions n
Incremental total angle IPA	Thread revolutions x 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Thread pitch P x (thread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

Shape of the helix

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

Internal thread	Work direction	Direction	Radius compensation
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°. If the thread has of more than 15 revolutions, program the helix in a program section repeat (see section 9.2 "Program Section Repeats").

CIRCLE

▶ Select circle functions: Press the „CIRCLE" soft key

C

▶ Select circular path C: Press the C soft key

P

▶ Select entry of polar coordinates: Press the P soft key (2nd soft-key row).

▶ Polar coordinates angle: Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, identify the tool axis using a soft key.**

▶ Enter the coordinate for the height of the helix in incremental dimensions.

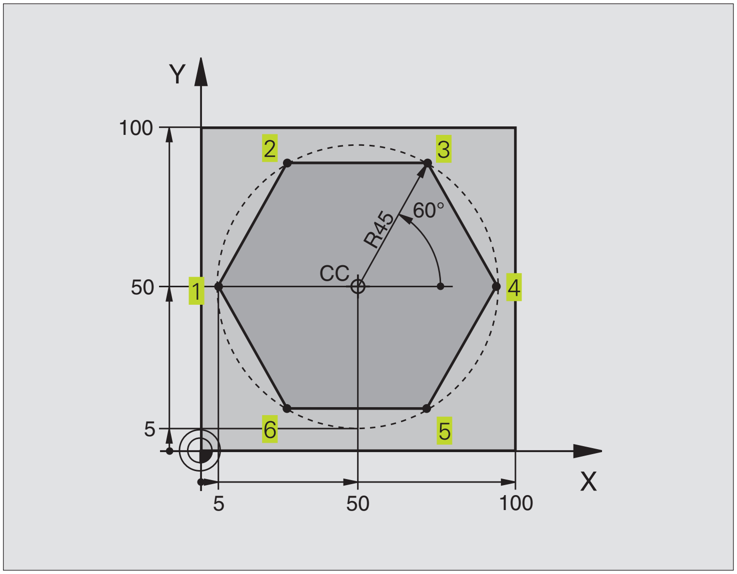
▶ Direction of rotation DR
Clockwise helix: DR–
Counterclockwise helix: DR+

▶ Radius compensation RL/RR/R0
Enter the radius compensation according to the table above.

Example NC blocks

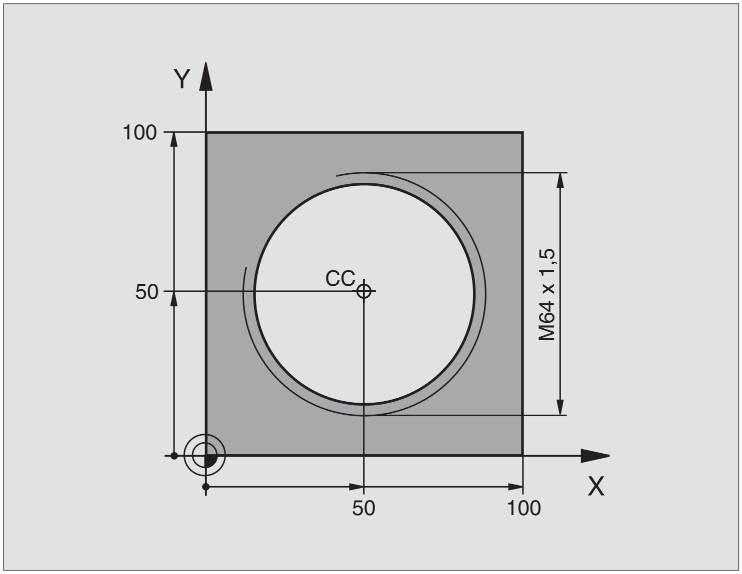
```
12 CC X+40 Y+25
13 Z+0 F100 M3
14 LP PR+3 PA+270 RL F50
15 CP IPA–1800 IZ+5 DR– RL F50
```

Example: Linear movement with polar coordinates



0 BEGIN PGM 40 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
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	tool call
5 CC X+50 Y+50	Define the datum for polar coordinates
6 L Z+250 R0 F MAX	Retract the tool
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 LP PR+45 PA+180 RL F250	Approach the contour at point 1
10 RND R1	Tangential approach to circle with R=1 mm
11 LP PA+120	Move to point 2
12 LP PA+60	Move to point 3
13 LP PA+0	Move to point 4
14 LP PA-60	Move to point 5
15 LP PA-120	Move to point 6
16 LP PA+180	Move to point 1
17 RND R1	Tangential departure from circle with R=1 mm
18 LP PR+60 PA+180 R0 F1000	Retract tool in the working plane
19 L Z+250 R0 F MAX M2	Retract tool in the spindle axis, end of program
20 END PGM 40 MM	

Example: Helix



`0 BEGIN PGM 50 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`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`	tool call
`5 L Z+250 R0 F MAX`	Retract the tool
`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 LP PR+32 PA-180 RL F100`	Approach contour
`10 RND R2`	Tangential approach to circle with R=2 mm
`11 CP IPA+3240 IZ+13,5 DR+ F200`	Helical interpolation
`12 RND R2`	Tangential departure from circle with R=2 mm
`13 L X+50 Y+50 R0 F MAX`	Retract tool in the working plane
`14 L Z+250 R0 F MAX M2`	Retract tool in the spindle axis, end of program
`15 END PGM 50 MM`	

To cut a thread with more than 16 revolutions

`...`	
`8 L Z-12.75 R0 F1000`	
`9 LP PR+32 PA-180 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)

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

> The machine tool builder may add some M functions that are not described in this User's Manual. Your machine manual provides more detailed information.

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

Miscellaneous function M ?

Only enter the number of the M function in the programming dialog.

In the MANUAL OPERATION operating mode, the M functions are entered with the M soft key.

Please note that some F functions become effective at the start of a positioning block, and others at the end.

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

Entering an M function in a 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:

STOP

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

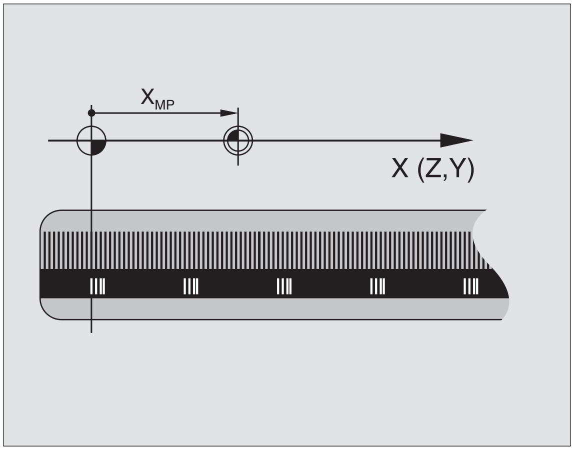
Example NC block

```
87 STOP M6
```

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.

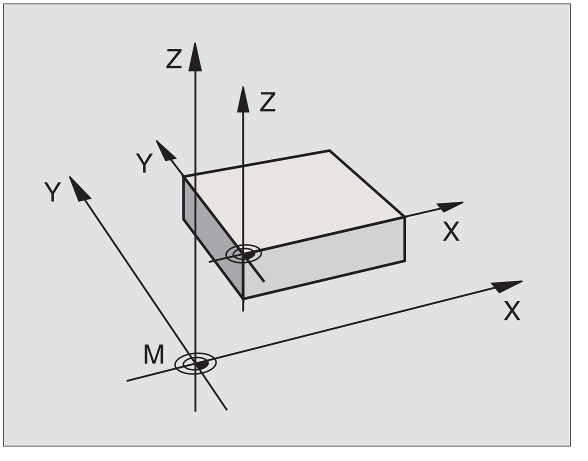
If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

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.

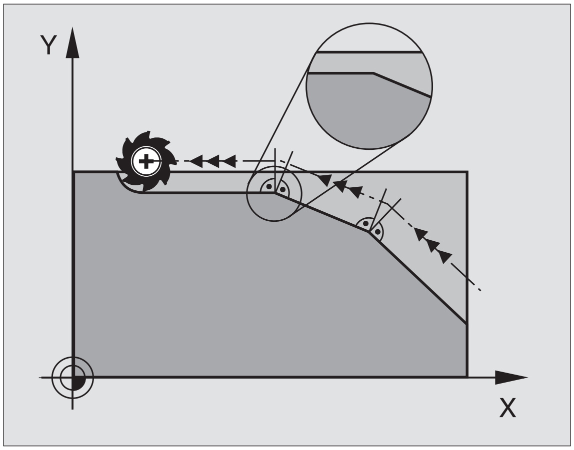


7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

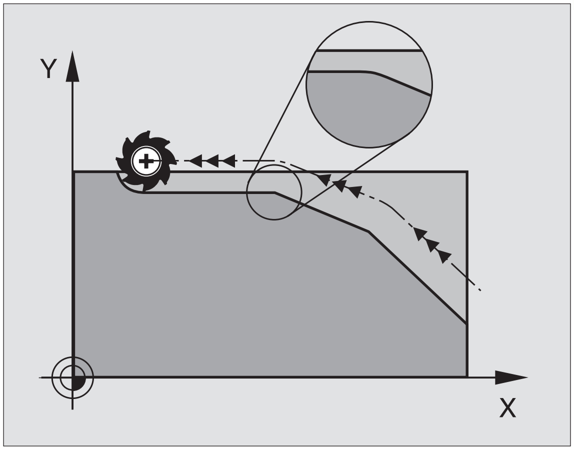
Standard behavior
The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an accurate stop.

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.

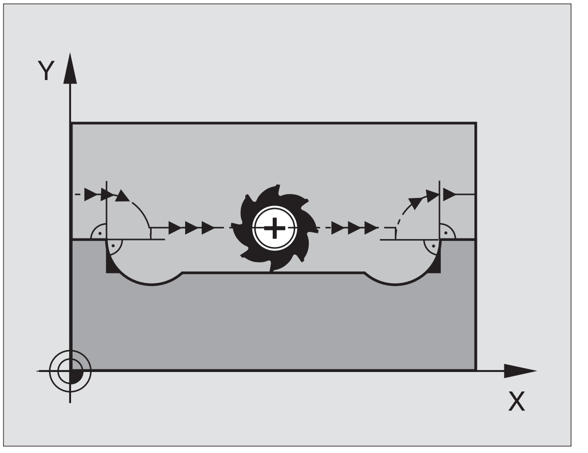
M90 becomes effective at the start of block. Operation with servo lag must be active.

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

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 TOOL LARGE."

Behavior with M97

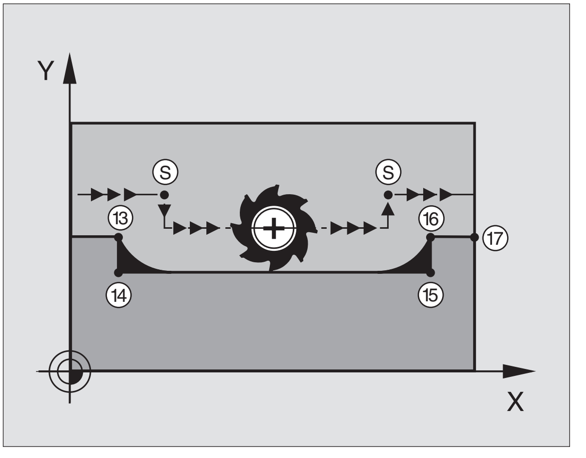
The TNC calculates the intersection of the contour elements — as at inside corners — and moves the tool over this point. See figure at lower right.

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.



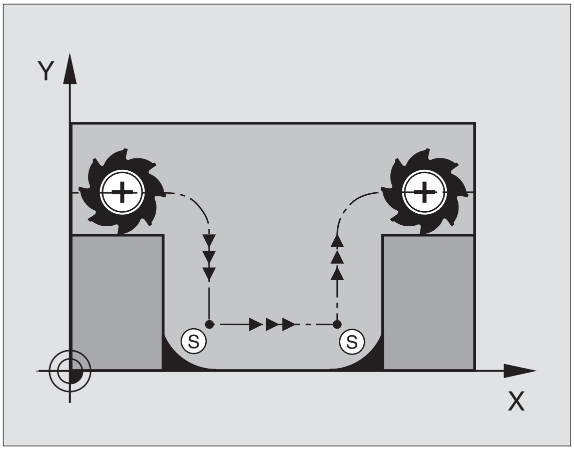
Example NC blocks

5	TOOL 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).



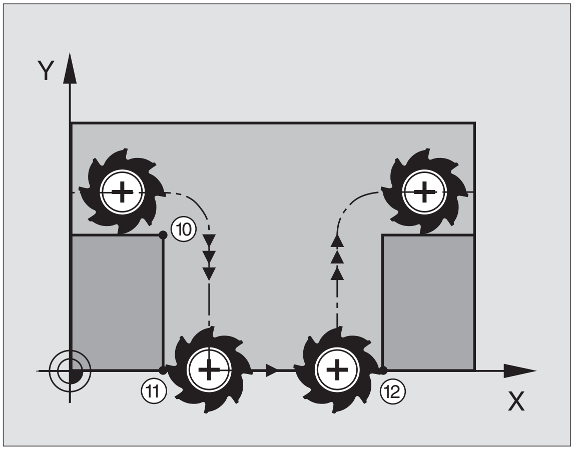
Behavior with M98
With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined (see figure at lower right).

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+ ...
```



7.4 Miscellaneous Functions for Contouring Behavior

7.5 Miscellaneous Function for Rotary Axes

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:

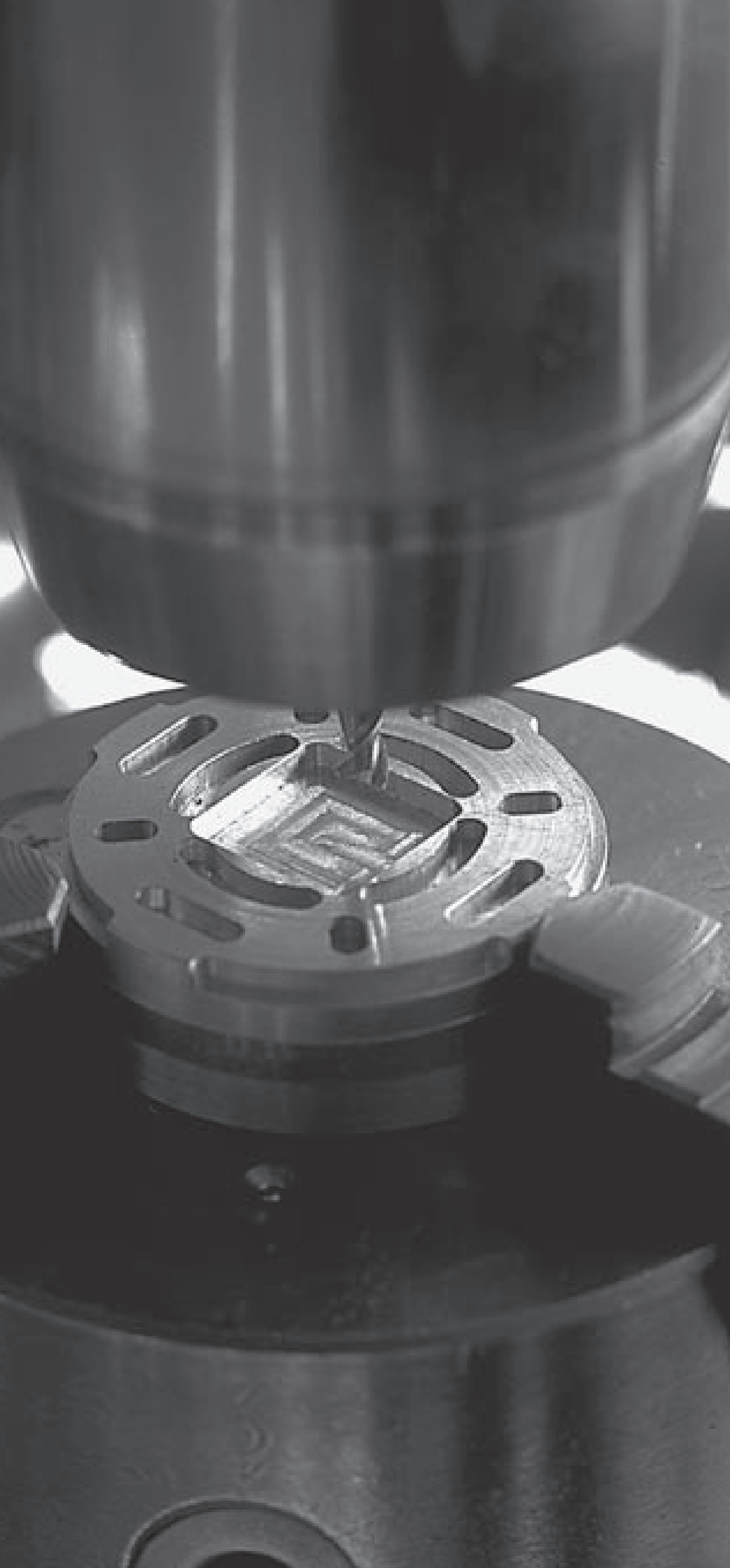
```
L M94
```

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.



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 lists 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.

Group of Cycles	Soft key
Cycles for pecking, reaming, boring and tapping	DRILLING
Cycles for milling pockets, studs and slots	POCKETS/ ISLANDS
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Cycles for producing hole patterns, such as circular or linear patterns	PATTERN
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
Special cycles such as dwell time, program call and oriented spindle stop	SPECIAL CYCLES

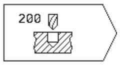
Defining a cycle

CYCL DEF

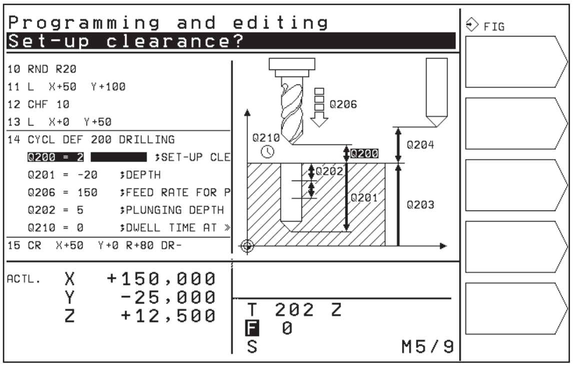
- The soft-key row shows the available groups of cycles

DRILLING

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



- Select a cycle, e.g. DRILLING The TNC initiates the programming dialog and asks all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted. 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



Example 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
```

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)
- Tool call
- 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
- 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:

CYCL CALL

- ▶ Press the CYCL CALL soft key to program a cycle call
- ▶ Enter a miscellaneous function, for example for coolant supply.

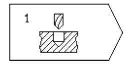
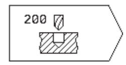
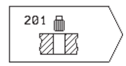
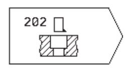
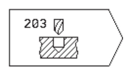
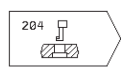
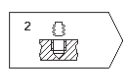
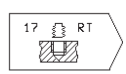
If the TNC is to execute the cycle automatically after every positioning block, program the cycle call with M89 (depending on machine parameter 7440).

To cancel M89, enter

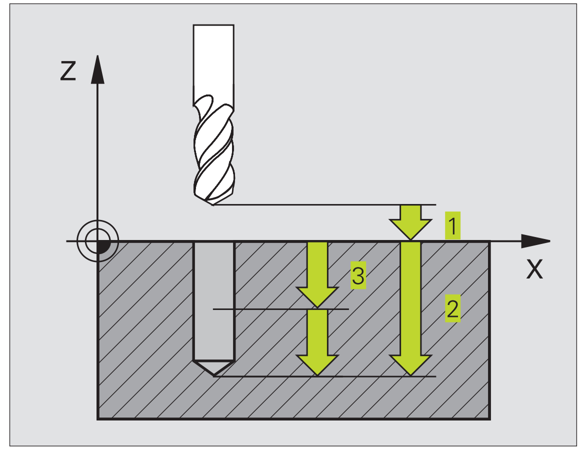
- M99 or
- CYCL CALL or
- CYCL DEF

8.2 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 setup 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 mm
- At a total hole depth exceeding 30 mm: t = hole depth / 50

Maximum advanced stop distance: 7 mm

4 The tool then advances with another infeed at the programmed feed rate F.

5 The TNC repeats this process (1 to 4) until the programmed total hole depth is reached.

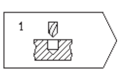
6 After a dwell time at the hole bottom, the tool is returned to the starting position in rapid traverse FMAX for chip breaking.

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:

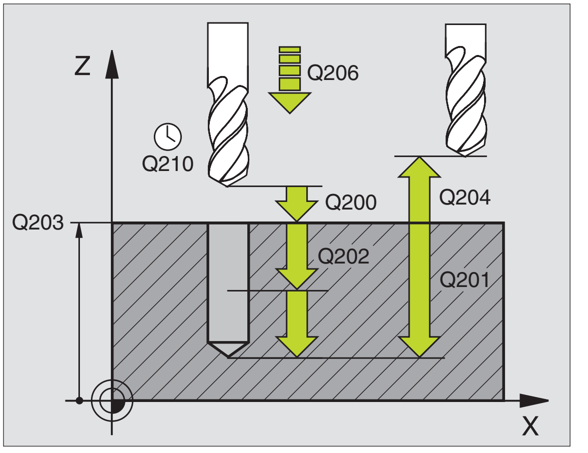
- the plunging depth is the same as the total hole depth
- the plunging depth is greater than the total hole depth

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

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.

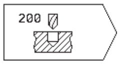
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

6 At the hole bottom, the tool is retraced 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.

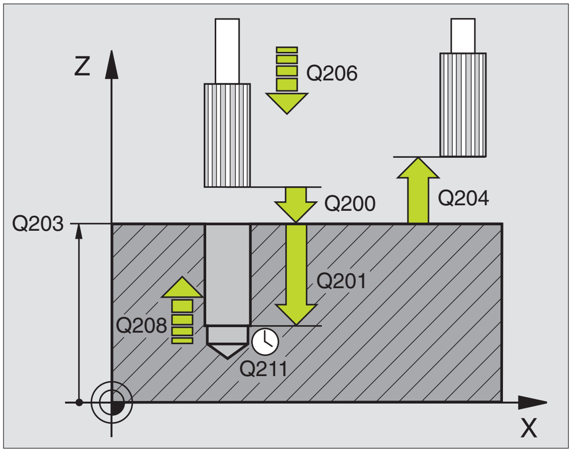
The algebraic sign for the depth parameter determines the working direction.



- 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.
- 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.

REAMING (Cycle 201)

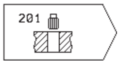


1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.
2. The tool reams to the entered depth at the programmed feed rate F.
3. If programmed, the tool remains at the hole bottom for the entered dwell time.
4. The tool then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

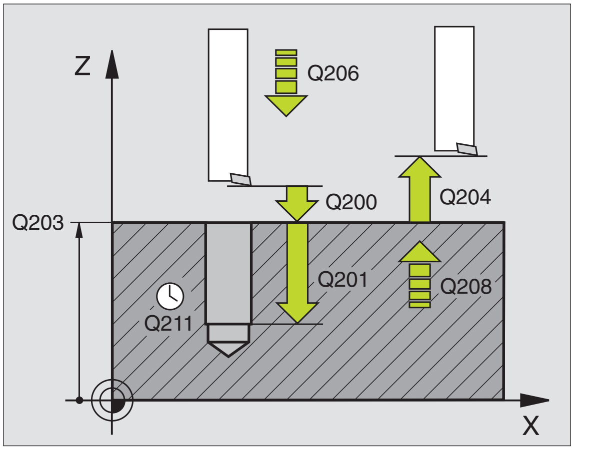
The algebraic sign for the depth parameter determines the working direction.



- ▶ 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.

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 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.

202

- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole
- Feed rate for plunging Q206: Traversing speed of the tool during boring in mm/min
- Dwell time at depth Q211: Time in seconds that the tool remains at the hole bottom
- Retraction feed rate Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at feed rate for plunging.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

- Disengaging direction (0/1/2/3/4) Q214: Determine the direction in which the TNC retracts the tool at the hole bottom (after spindle orientation).

0: Do not retract tool

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 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 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.

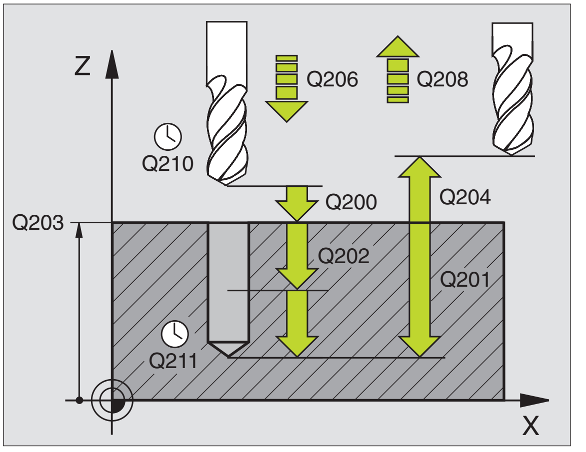
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

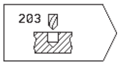
6 The tool remains at the hole bottom — if programmed — for the entered DWELL TIME to cut free, and then retracts to set-up clearance at the retraction feed rate. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in FMAX.

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.





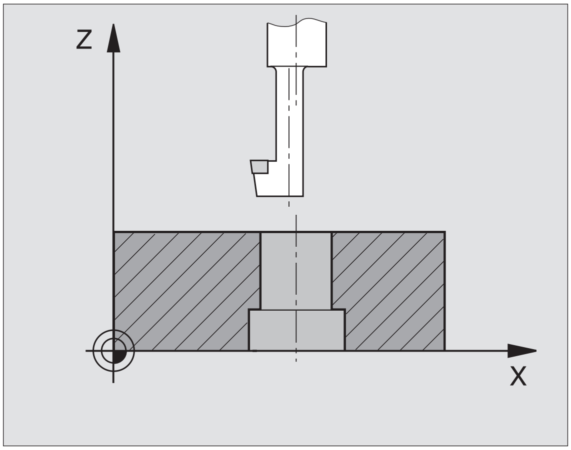
- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min
- Plunging depth 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 0.2 mm.
- 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 in FMAX.

BACK BORING (Cycle 204)

Machine and TNC must be specially prepared by the machine tool builder to perform 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:

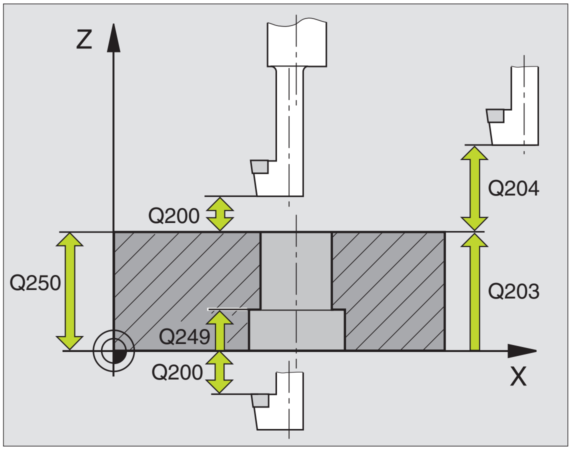
Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

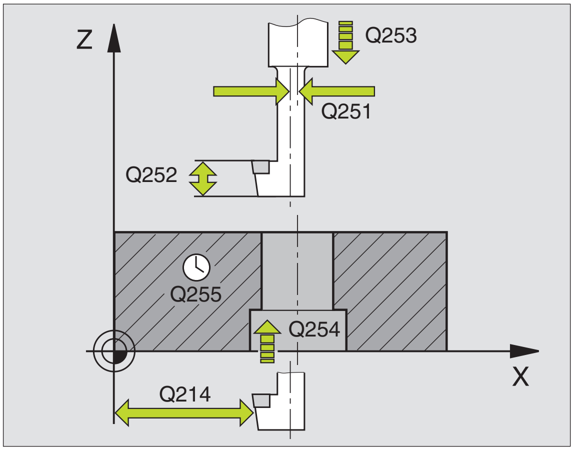
The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

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 boring, the TNC considers the tooth length of the boring bar and the thickness of the material.

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

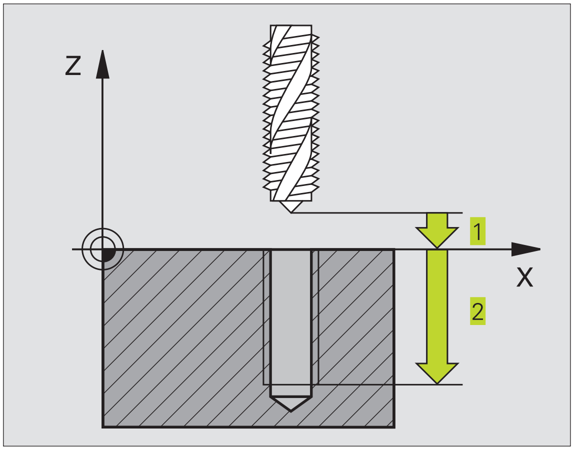
3: Displace tool in the positive main axis direction

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.

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.

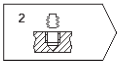
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

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.

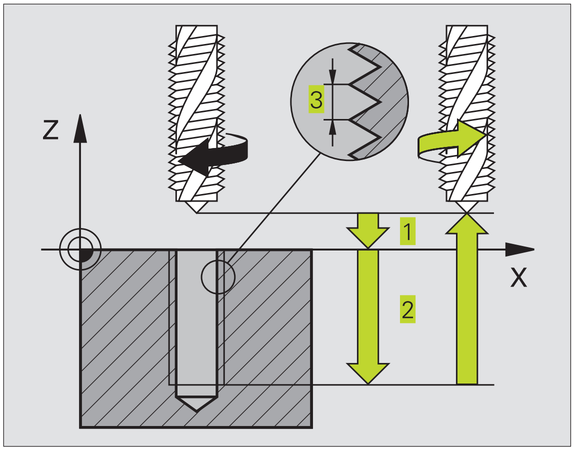


- Setup clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- Total hole depth 2 (thread length, incremental value): Distance between workpiece surface and end of thread
- Dwell time in seconds: Enter 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 x 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 (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.

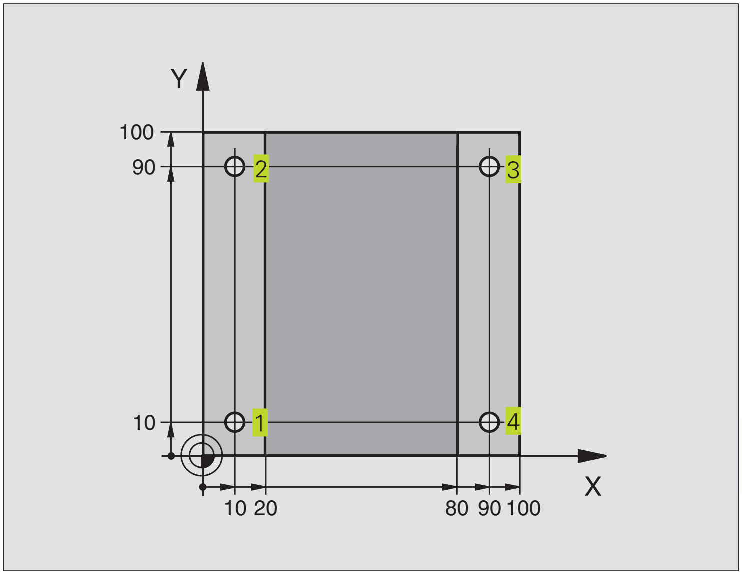
The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

17 RT

- 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: Drilling cycles

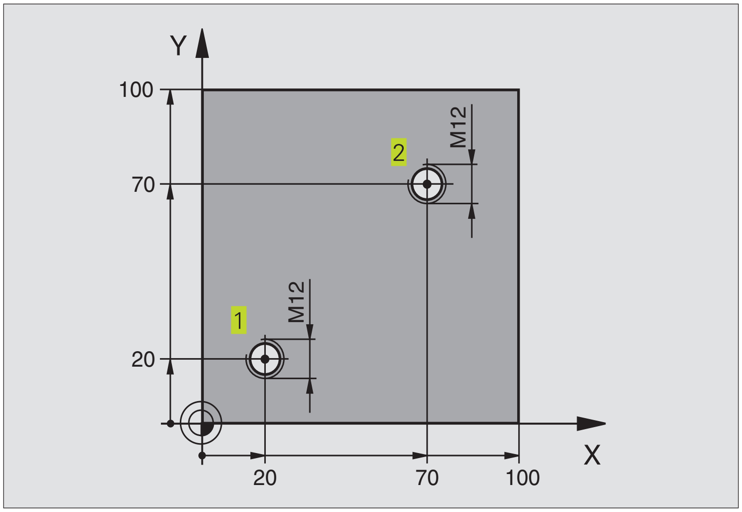


0 BEGIN PGM 200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+3	Define the tool
4 TOOL CALL 1 Z S4500	Tool call
5 L Z+250 R0 F MAX	Retract the tool
6 CYCL DEF 200 DRILLING	Define cycle
Q200=2 ;SET-UP CLEARANCE	Setup clearance
Q201=-15 ;DEPTH	Depth
Q206=250 ;FEED RATE FOR PLUNGING	Feed rate for drilling
Q202=5 ;PLUNGING DEPTH	Pecking
Q210=0 ;DWELL TIME AT TOP	Dwell time at top
Q203=-10 ;SURFACE COORDINATE	Surface coordinate
Q204=20 ;2ND SET-UP CLEARANCE	2nd set-up clearance
7 L X+10 Y+10 R0 F MAX M3	Approach hole 1, spindle ON
8 CYCL CALL	Call the cycle
9 L Y+90 R0 F MAX M99	Approach hole 2, call cycle
10 L X+90 R0 F MAX M99	Approach hole 3, call cycle
11 L Y+10 R0 F MAX M99	Approach hole 4, call cycle
12 L Z+250 R0 F MAX M2	Retract in the tool axis, end program
13 END PGM 200 MM	

Example: Drilling cycles

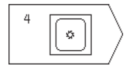
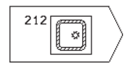
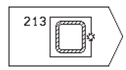
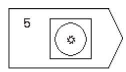
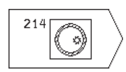
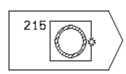
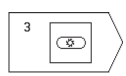
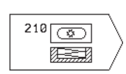
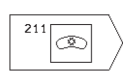
Program sequence

- 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 2 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+4.5`	Define the tool
`4 TOOL CALL 1 Z S100`	Tool call
`5 L Z+250 R0 FMAX`	Retract the tool
`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 in the tool axis, end program
`15 END PGM 2 MM`	

8.3 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	

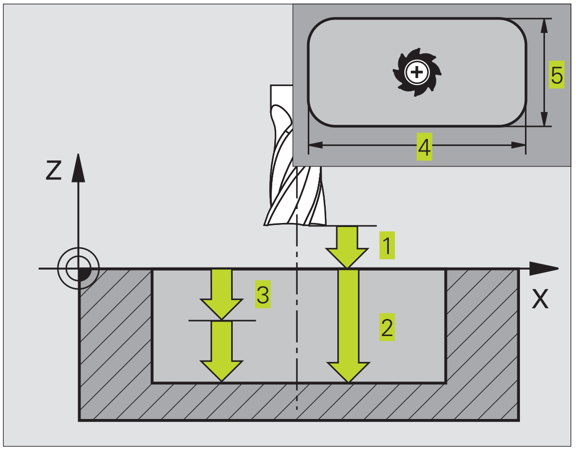
POCKET MILLING (Cycle 4)

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.



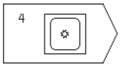
Before programming, note the following:

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.



- 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:
  n the plunging depth equals the depth
  n 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

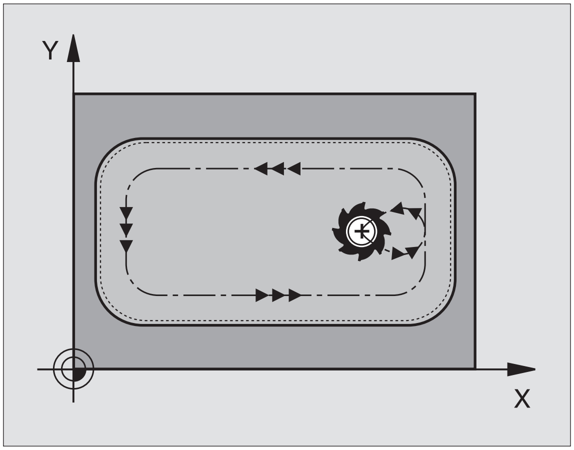
▶ DIRECTION OF THE MILLING PATH
DR **+** : climb milling with M3
DR **–** : up-cut milling with M3

▶ 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 x 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.

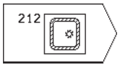
7 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or - if programmed - to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).

Before programming, note the following:

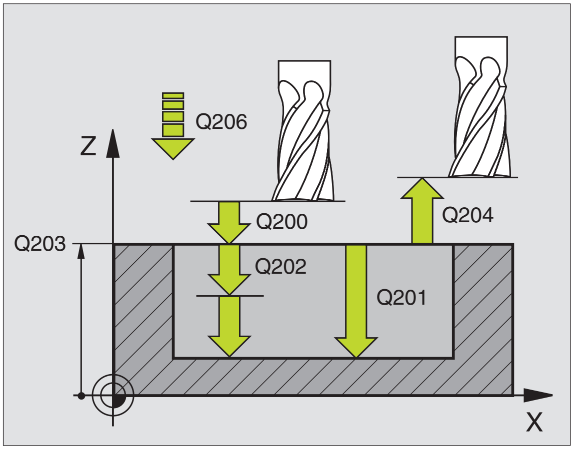
The algebraic sign for the depth parameter determines the working direction.

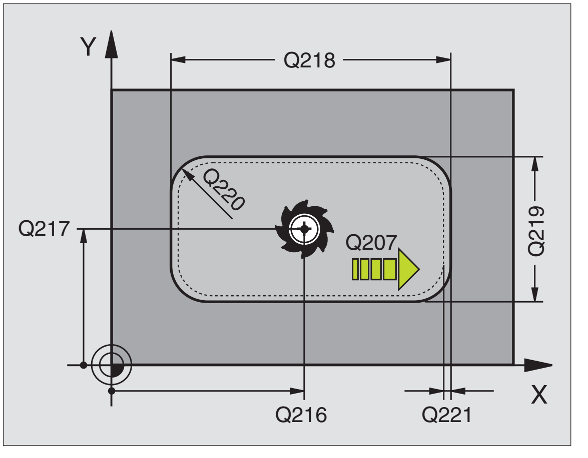
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

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 low value; if you have already cleared the pocket, 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 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.





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.

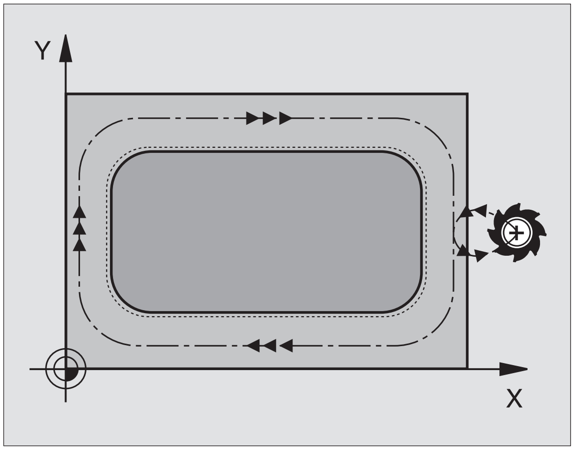
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

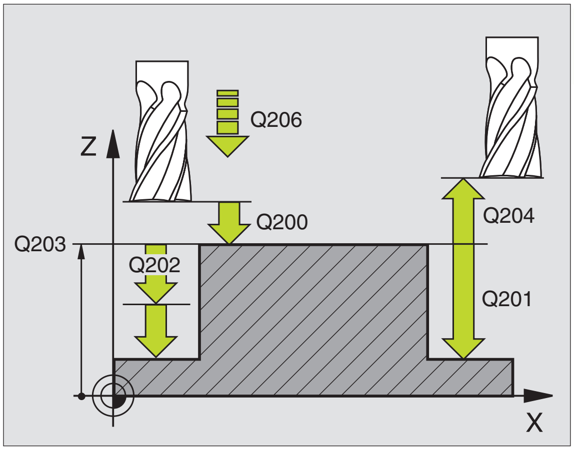
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

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:

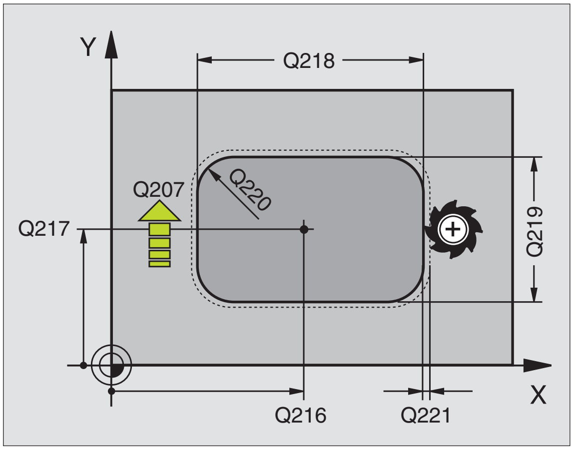
The algebraic sign for the depth parameter determines the working direction.

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.

213

- 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
- 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.



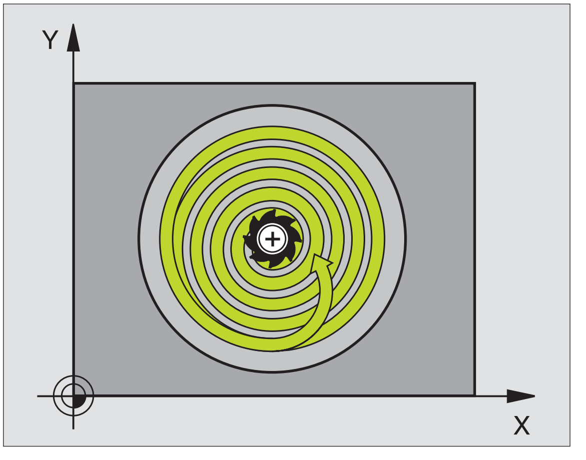
CIRCULAR POCKET MILLING (Cycle 5)

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.



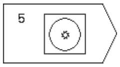
Before programming, note the following:

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

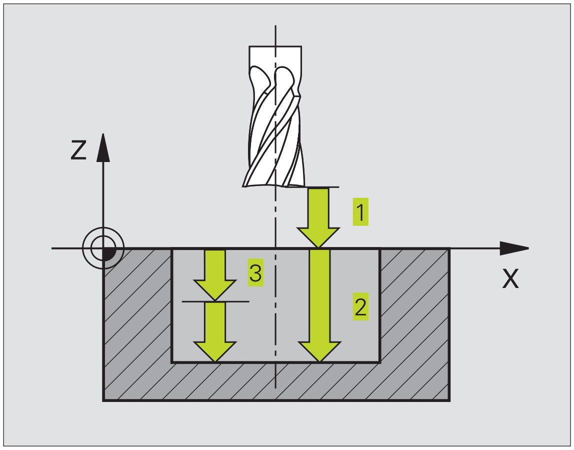
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

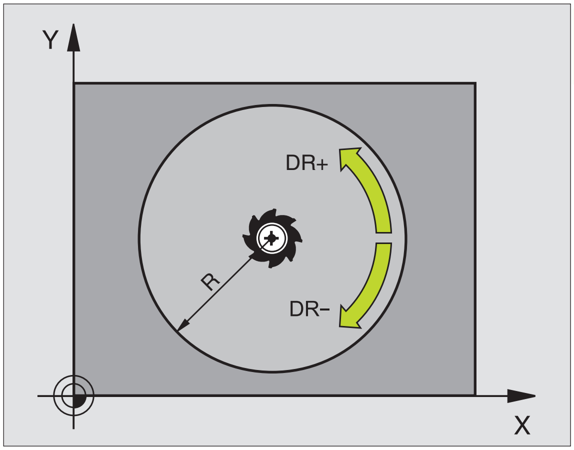
The algebraic sign for the depth parameter determines the working direction.

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

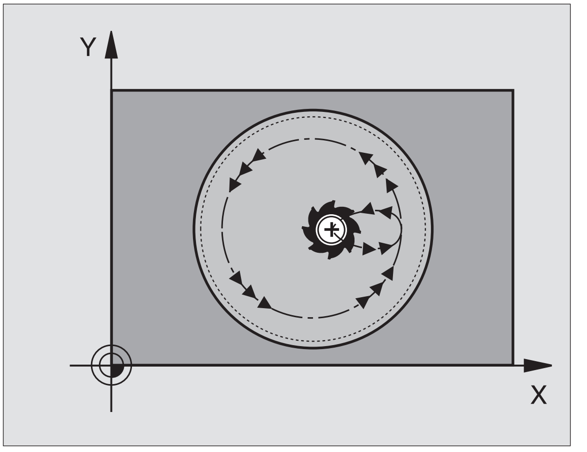


- Setup clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- Milling depth 2 (incremental value): Distance between workpiece surface and bottom of pocket
- Plunging depth 3 (incremental value): Infeed per cut. The tool will advance to the depth in one movement if:
  n the plunging depth equals the depth
  n the plunging depth is greater than the depth
- Feed rate for plunging: Traversing speed of the tool during penetration
- Circular radius: Radius of the circular pocket
- Feed rate F: Traversing speed of the tool in the working plane
- Direction of the milling path
  DR + : climb milling with M3
  DR – : up-cut milling with M3





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.

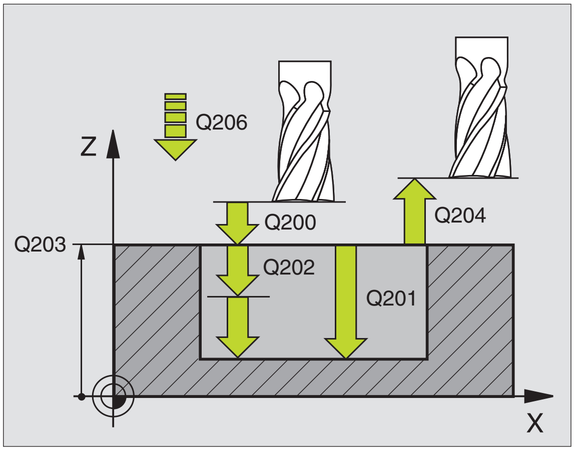
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

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.

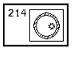
7 At the end of the cycle, the TNC retracts the tool in FMAX to set-up clearance, or - if programmed - to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).



Before programming, note the following:

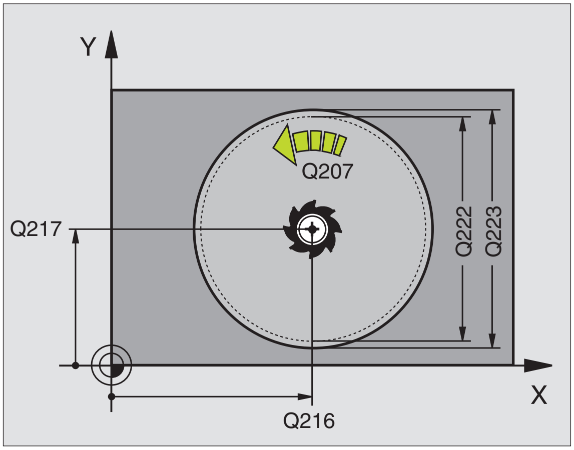
The algebraic sign for the depth parameter determines the working direction.

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 low value; if you have already cleared the stud, enter a higher feed rate.
- Plunging depth Q202 (incremental value): Infeed per cut
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.

- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the pocket in the 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.

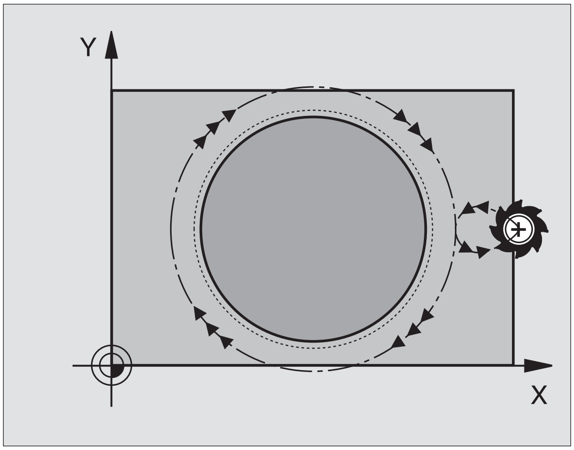
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

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.

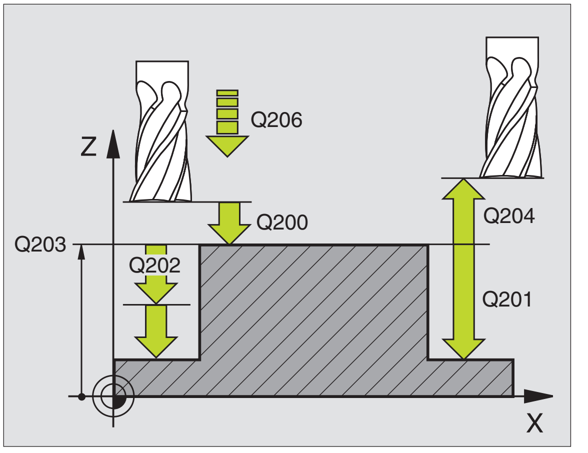
7 At the end of the cycle, the TNC retracts the tool in FMAX to set-up clearance, or — if programmed - to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).



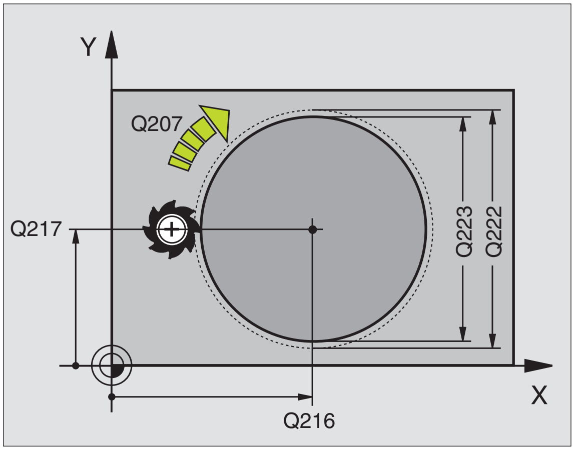
Before programming, note the following:

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

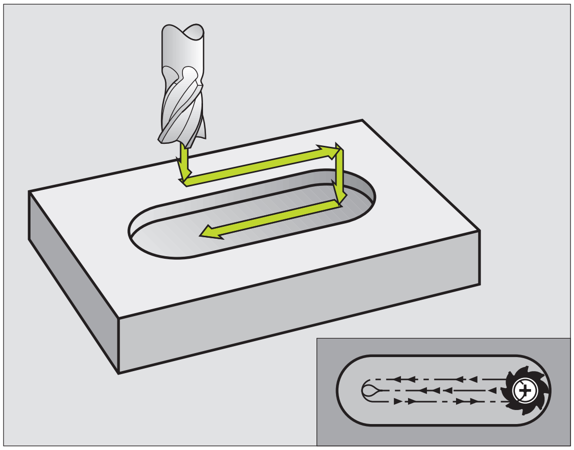
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



- 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.
- 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.



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.

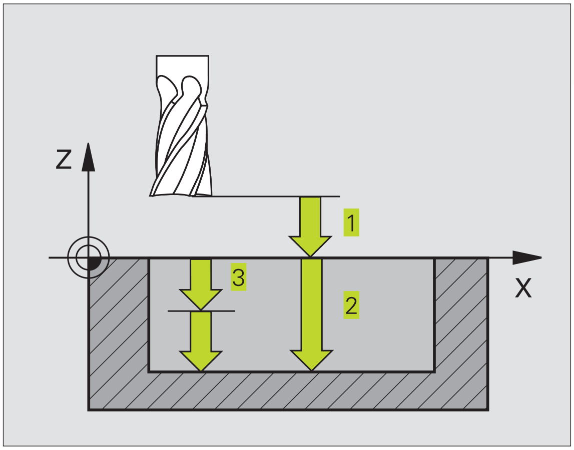
This process is repeated until the programmed milling depth is reached.

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).

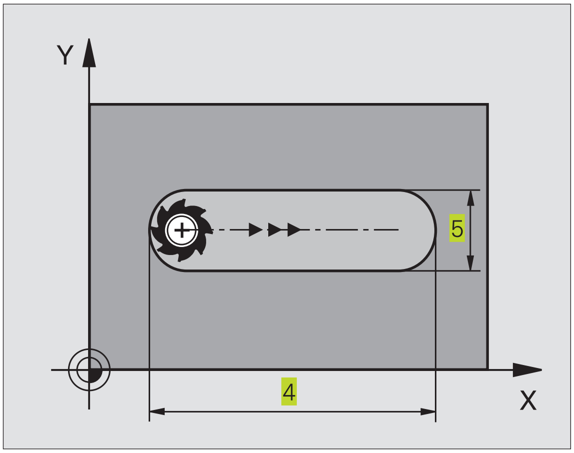
The algebraic sign for the depth parameter determines the working direction.

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.

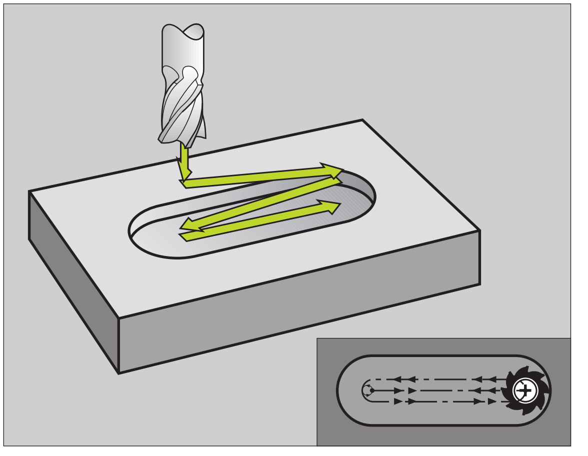
3

- 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 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 the feed rate for milling to the workpiece surface. From there, the cutter advances in the longitudinal direction of the slot — plunge-cutting obliquely into the material — until it reaches the center of the right circle.

3 The tool then moves back to the center of the left circle, again with oblique plunge-cutting. This process is repeated until the programmed milling depth is reached.

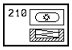
4 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.

Finishing process

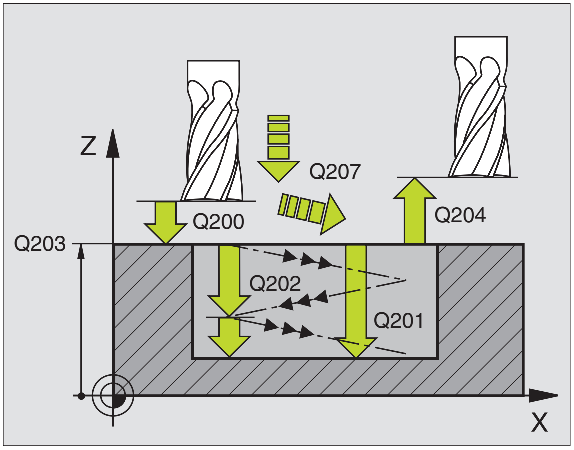
5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3).

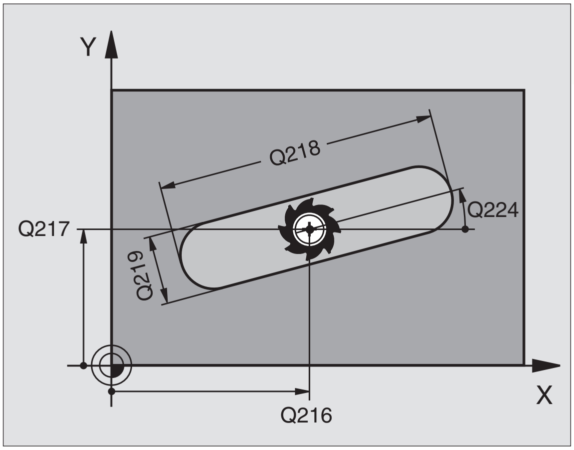
6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.

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.

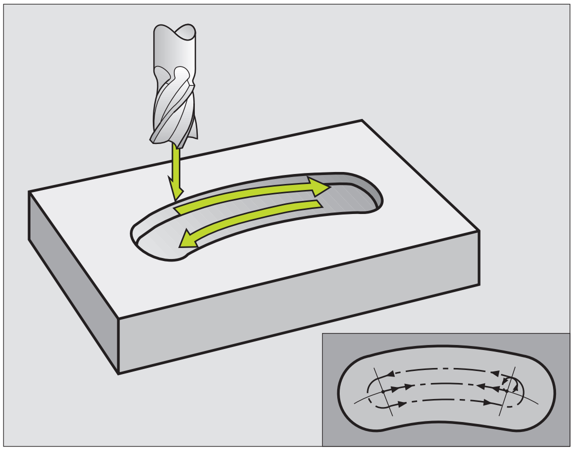


- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Plunging depth Q202 (incremental 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.





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 the milling feed rate to the workpiece surface. From there, the cutter advances — plunge-cutting obliquely into the material — to the other end of the slot.

3 The tool then moves at a downward angle back to the starting point, again with oblique plunge-cutting. This process (2 to 3) is repeated until the programmed milling depth is reached.

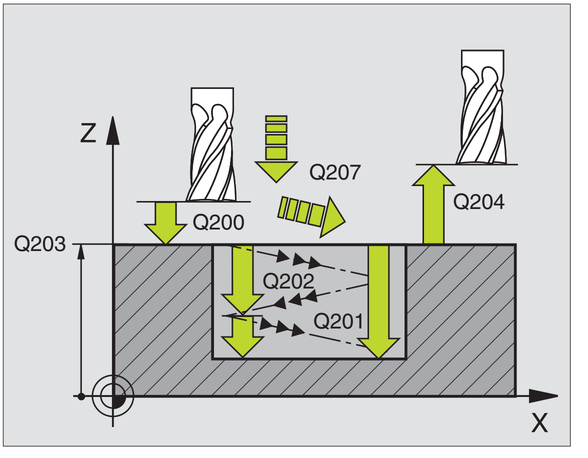
4 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot.

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:

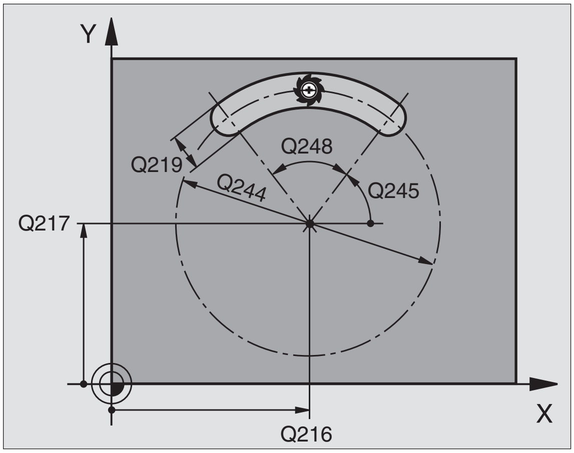
The algebraic sign for the depth parameter determines the working direction.

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.

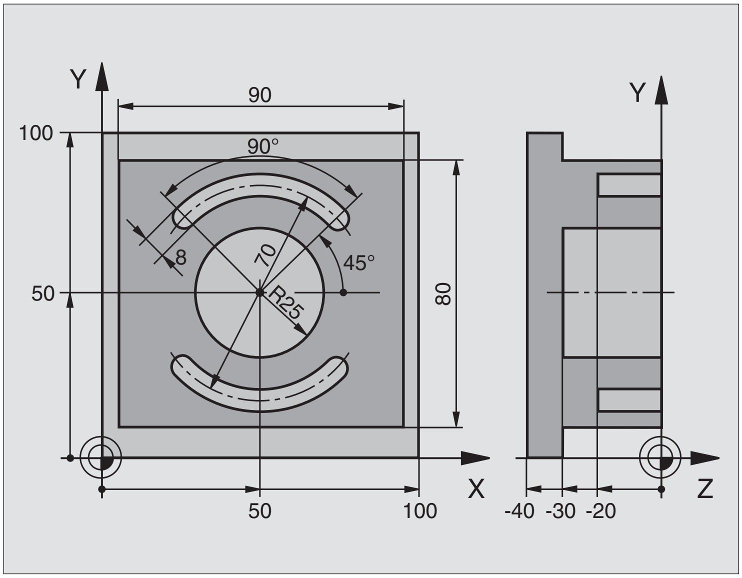
211

- 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: Milling pockets, studs and slots

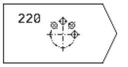
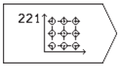


0 BEGIN PGM 210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+6	Define the tool for roughing/finishing
4 TOOL DEF 2 L+0 R+3	Define slotting mill
5 TOOL CALL 1 Z S3500	Call tool for roughing/finishing
6 L Z+250 R0 F MAX	Retract the tool
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 1ST AXIS	

8 CYCL CALL M3	Cycle call for stud
9 CYCL DEF 5.0 CIRCULAR POCKET	Define CIRCULAR POCKET MILLING cycle
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+	
15 L Z+2 R0 F MAX M99	Call CIRCULAR POCKET MILLING cycle
16 L Z+250 R0 F MAX M6	Tool change
17 TOOL CALL 2 Z S5000	Call slotting mill
18 CYCL DEF 211 CIRCULAR SLOT	Define cycle for slot 1
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 CYCL DEF 211 CIRCULAR SLOT	Cycle definition for slot 2
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=+225 ;START ANGLE	
Q248=90 ;ANGULAR LENGTH	
21 CYCL CALL	Call cycle for slot 2
22 L Z+250 R0 F MAX M2	Retract in the tool axis, end program
23 END PGM 210 MM	

8.4 Cycles for Machining Hole Patterns

The TNC provides two cycles for machining hole patterns:

Cycle	Soft key
220 CIRCULAR PATTERN	
221 LINEAR PATTERN	

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)

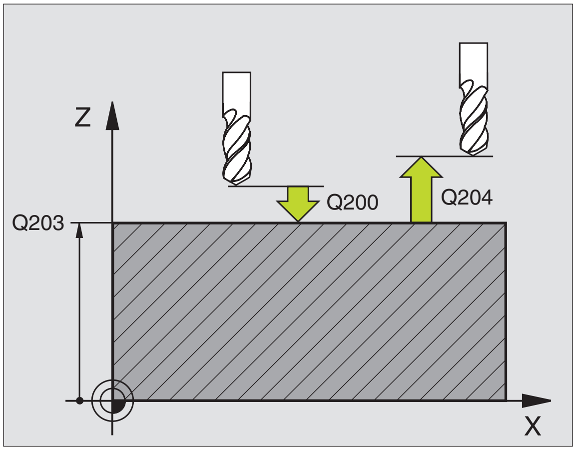
1 At rapid traverse, the TNC moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:
- Move to 2nd setup clearance (tool axis)
- Approach starting point in the machining plane
- Move to setup 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).

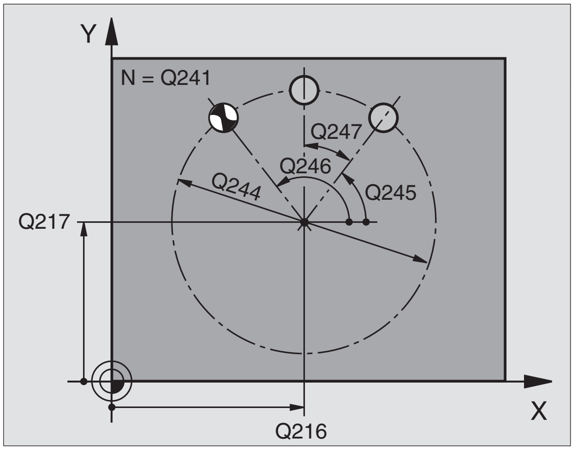
4 This process (1 to 3) is repeated until all machining operations have been executed.



Before programming, note the following:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

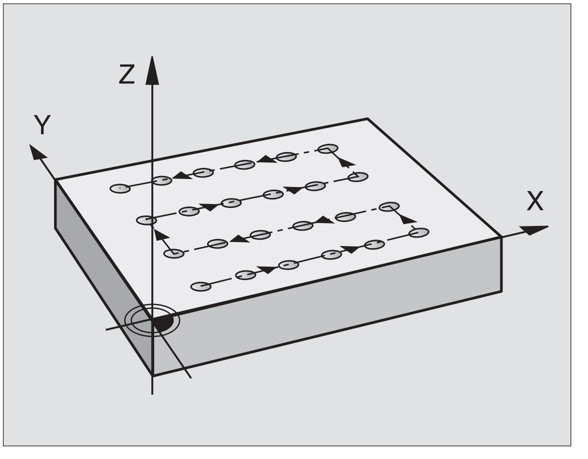
If you combine Cycle 220 with one of the fixed cycles 200 to 204 and 212 to 215, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 220 will be effective for the selected fixed cycle.



- 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. 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.

- Stepping angle Q247 (incremental value): Angle between two machining operations on a pitch circle. If you enter a stepping angle of 0, the TNC will calculate the stepping angle from the starting and stopping angles. If you enter a value other than 0, the TNC will not take the stopping angle into account. The sign for the stepping angle determines the working direction (– = clockwise).
- 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 setup clearance (tool axis)
- Approach starting point in the machining plane
- Move to setup 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.

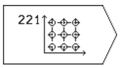
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

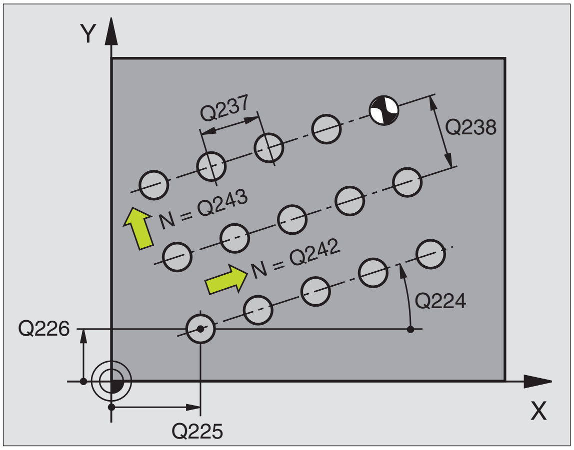
6 From this position, the tool approaches the starting point for the next machining operation in the negative main axis direction.

7 This process (5 to 6) is repeated until all machining operations in the second line have been executed.

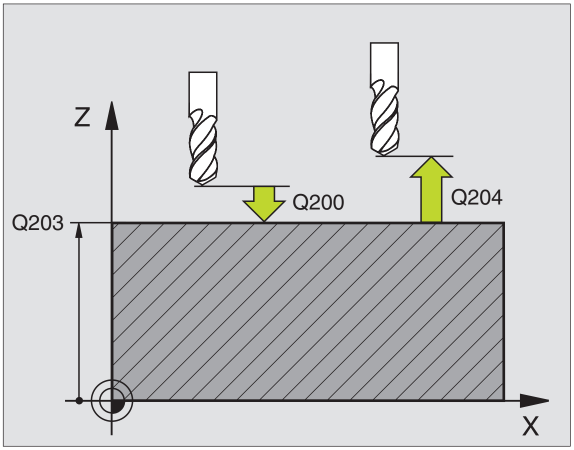
8 The tool then moves to the starting point of the next line.

9 All subsequent lines are processed in a reciprocating movement.

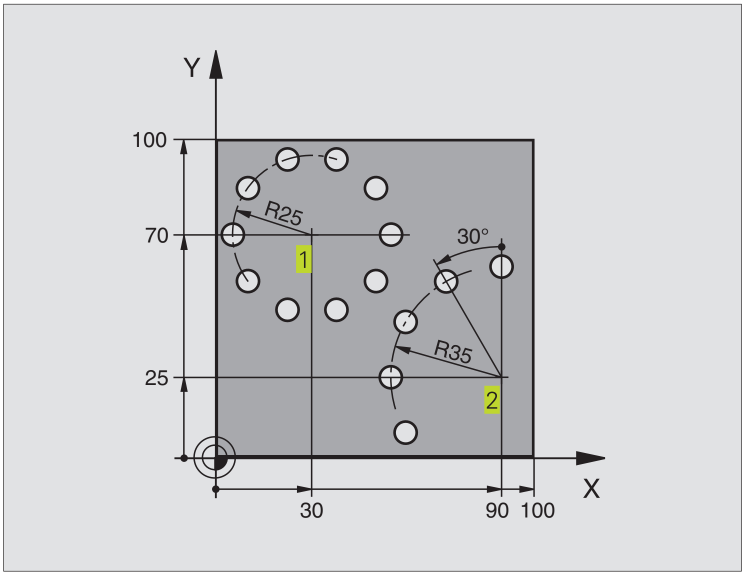




- Starting point 1st axis Q225 (absolute value): Coordinate of the starting point in the main axis of the working plane
- Starting point 2nd axis Q226 (absolute value): Coordinate of the starting point in the secondary axis of the working plane
- Spacing in 1st axis Q237 (incremental value): Spacing 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: Circular hole patterns



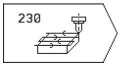
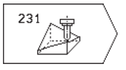
0 BEGIN PGM 3589M	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+3	Define the tool
4 TOOL CALL 1 Z S3500	Tool call
5 L Z+250 R0 F MAX M3	Retract the tool
6 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	Setup clearance
Q201=-15 ;DEPTH	Depth
Q206=250 ;FEED RATE FOR PLUNGING	Feed rate for drilling
Q202=4 ;PLUNGING DEPTH	Plunging depth
Q210=0 ;DWELL TIME	Dwell time at top
Q203=+0 ;SURFACE COORDINATE	Surface coordinate
Q204=0 ;2. SET-UP CLEARANCE	2nd 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 F MAX M2`	Retract in the tool axis, end program
`10 END PGM 3589 MM`	

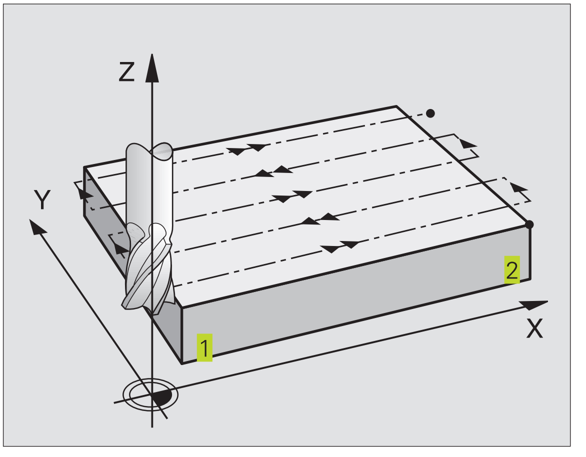
8.5 Cycles for multipass milling

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)

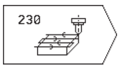


1. From the current position, the TNC positions the tool in rapid traverse in the working plane to the starting position. 1 During this movement, the TNC also offsets the tool by its radius to the left and upward.
2. The tool then moves in FMAX in the tool axis to set-up clearance. From there it approaches 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. 2 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.

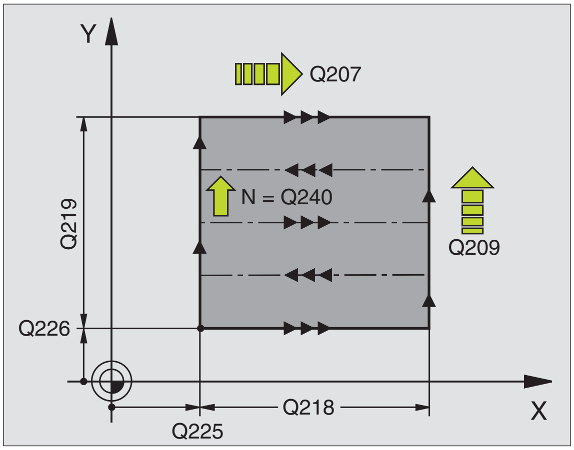
Before programming, note the following:

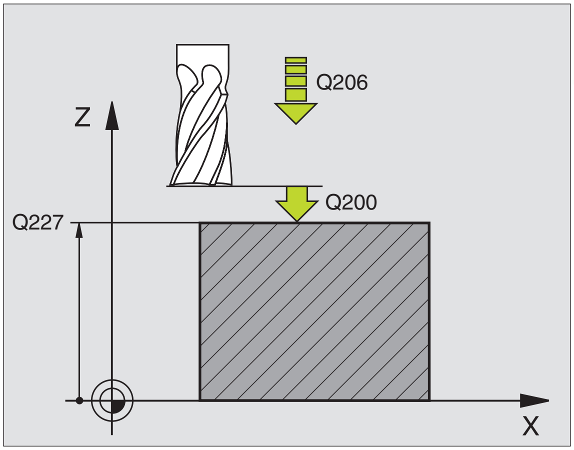
From the current position, the TNC positions the tool at the starting point 1, first in the working plane and then in the tool axis.

Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

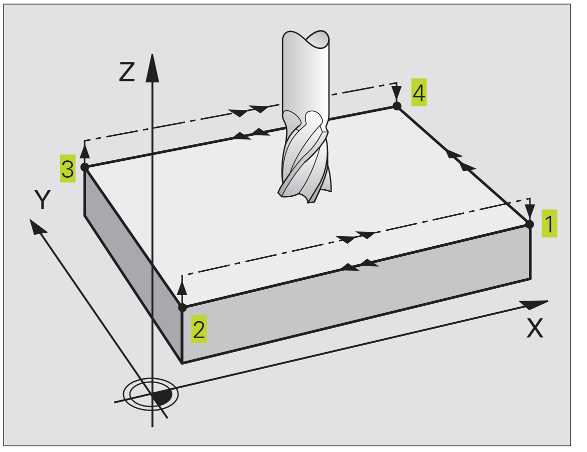


- Starting point in 1st axis Q225 (absolute value): Min. point coordinate of the surface to be multipass-milled in the main axis of the working plane
- Starting point in 2nd axis Q226 (absolute value): Min. point coordinate of the surface to be multipass-milled in the secondary axis of the working plane
- 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.





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.

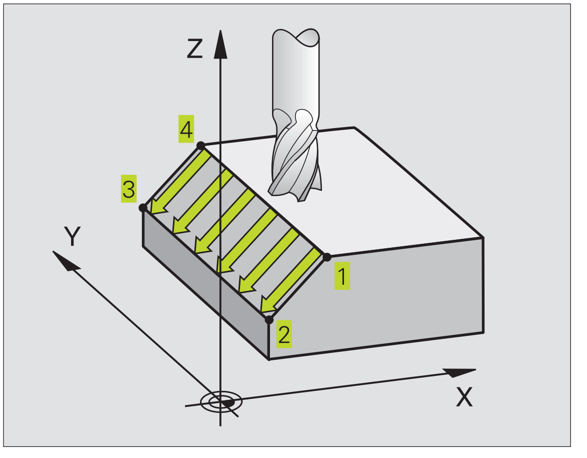
4 At the starting position 1 the TNC moves the tool back to the the last traversed Z value.

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.

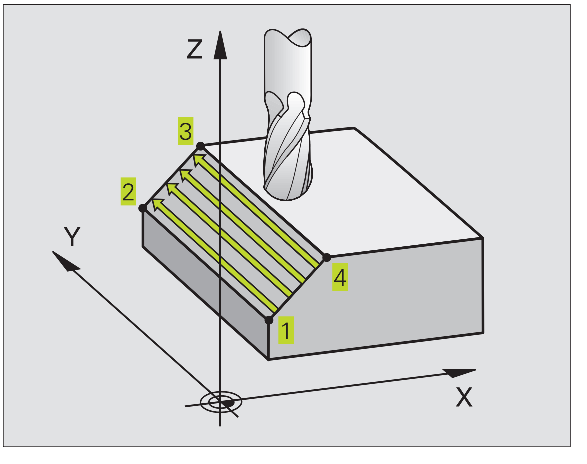
8 At the end of the cycle, the tool is positioned above the highest programmed point in the tool axis, offset by the tool diameter.



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 point 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

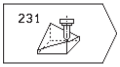
- When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) perpendicular to the direction of the steeper inclination. See figure at lower right.

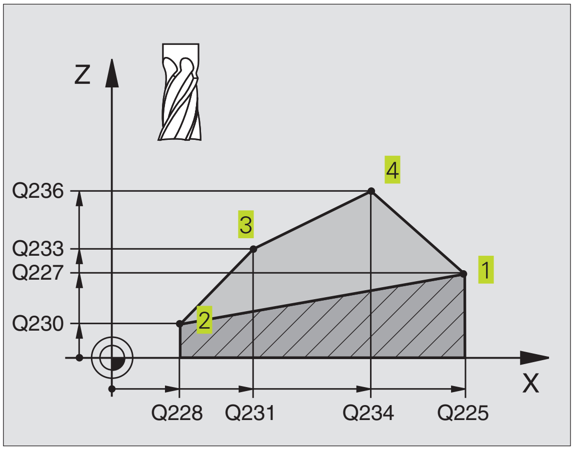
Before programming, note the following:

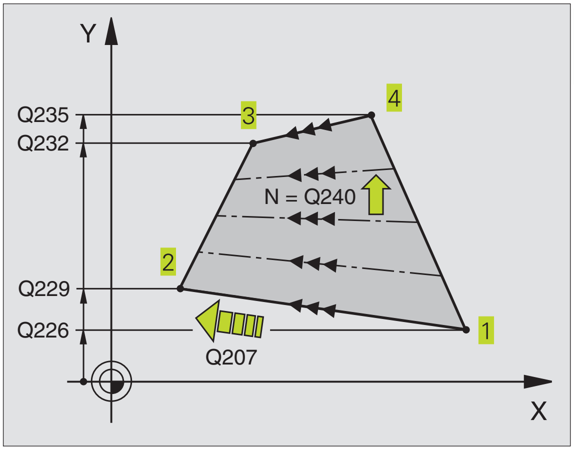
From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1. 1. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

The TNC moves the tool with radius compensation R0 to the programmed positions.

If required, use a center-cut end mill (ISO 1641).

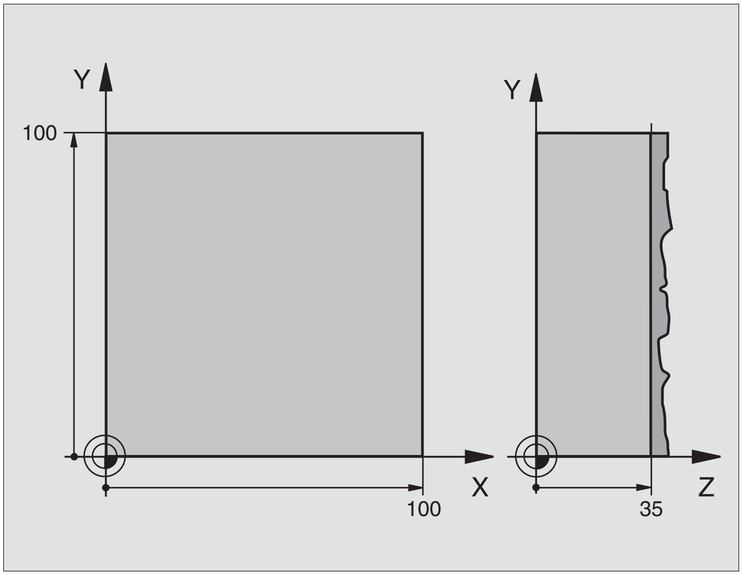






- Starting point in 1st axis Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the main axis of the working plane
- Starting point in 2nd axis Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the secondary axis of the working 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 when milling the first pass. The TNC calculates the feed rate for all subsequent passes dependent of the stepover factor of the tool (offset less than tool radius = higher feed rate, high stepover factor = lower feed rate)

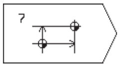
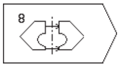
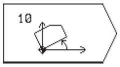
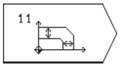
Example: Multipass milling



`0 BEGIN PGM 230 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z+0`	Define the workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+40`	
`3 TOOL DEF 1 L+0 R+5`	Define the tool
`4 TOOL CALL 1 Z S3500`	Tool call
`5 L Z+250 R0 F MAX`	Retract the tool
`6 CYCL DEF 230 MULTIPASS MILLNG`	Cycle definition: MULTIPASS MILLING
`Q225=+0 ;STARTNG PNT 1ST AXIS`	Starting point for X axis
`Q226=+0 ;STARTNG PNT 2ND AXIS`	Starting point for Y axis
`Q227=+35 ;STARTNG PNT 3RD AXIS`	Starting point for Z axis
`Q218=100 ;FIRST SIDE LENGTH`	1st side length
`Q219=100 ;SECOND SIDE LENGTH`	2nd side length
`Q240=25 ;NUMBER OF CUTS`	Number of cuts
`Q206=250 ;FEED RATE FOR PLUNGING`	Feed rate for plunging
`Q207=400 ;FEED RATE FOR MILLNG`	Feed rate for milling
`Q209=150 ;STEPOVER FEED RATE`	Feed rate for cross pecking
`Q200=2 ;SET-UP CLEARANCE`	Setup clearance
`7 L X-25 Y+0 R0 F MAX M3`	Pre-position near the starting point
`8 CYCL CALL`	Call the cycle
`9 L Z+250 R0 F MAX M2`	Retract in the tool axis, end program
`10 END PGM 230 MM`	

8.6 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	
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	

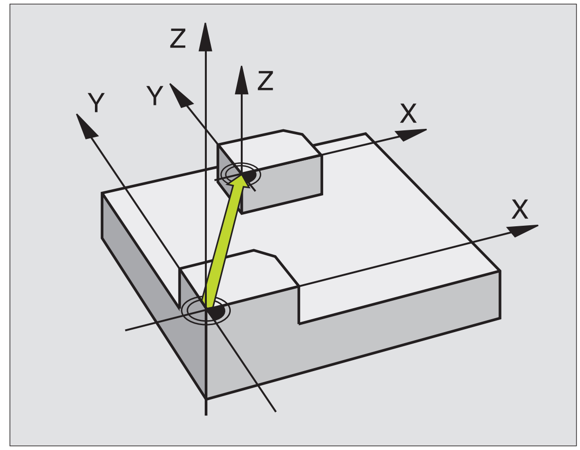
Effect of coordinate transformations
A coordinate transformation becomes effective as soon as it is defined — it is not called. It remains in effect until it is changed or canceled.

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

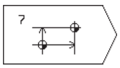
DATUM SHIFT (Cycle 7)

A datum shift allows machining operations to be repeated at various locations on the workpiece.



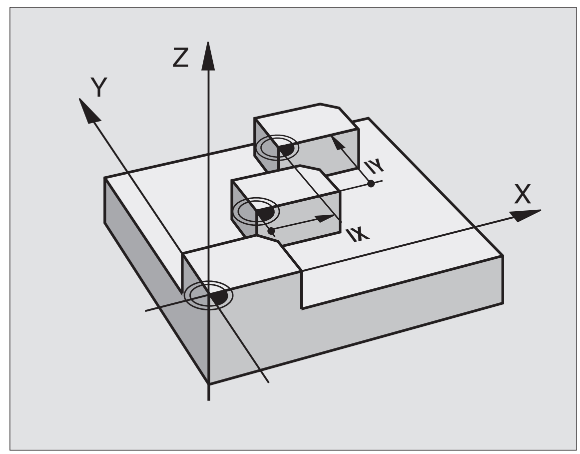
Function

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 (2nd soft-key row) to reference the programmed datum to the machine datum. In this case the TNC indicates the first cycle block with REF



Cancellation

A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Status Displays

If datums are referenced to the machine datum, then

- The actual position values are referenced to the active (shifted) datum.
- The datum shown in the additional status display is referenced to the machine datum, whereby the TNC accounts for 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.

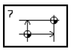
Remember that the datum numbers shift whenever you insert lines in an existing datum table (edit part program if necessary).

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:

PGM CALL

▶ 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.

PGM MGT

▶ 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.

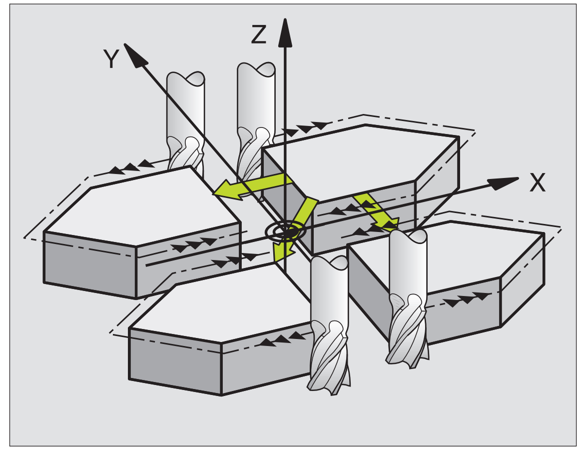
To leave a datum table

▶ 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	PAGE ⇧
Go to the previous page	PAGE ⇩
Move one word to the right	WORD ⇨
Move one word to the left	WORD ⇦
Confirm current position, e.g. for the Z-axis	ACT.POS. Z
Enter the number of lines to be inserted	INSERT N LINES
Delete and temporarily store a line	DELETE LINE
Insert a new line or the line last deleted	INSERT LINE
Go to the beginning of the table	BEGIN ⇧
Go to the end of the table	END ⇩

MIRROR IMAGE (Cycle 8)

The TNC can machine the mirror image of a contour in the working plane. See figure at upper right.



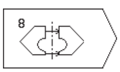
Function

The MIRROR IMAGE cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

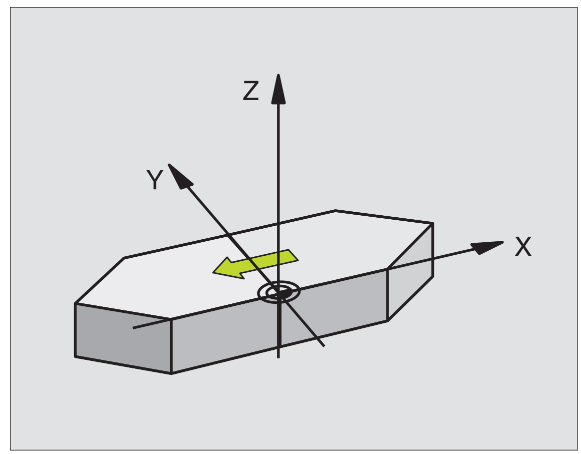
- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

The result of the mirror image depends on the location of the datum

- If the datum lies on the contour to be mirrored, the element simply flips over — 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.

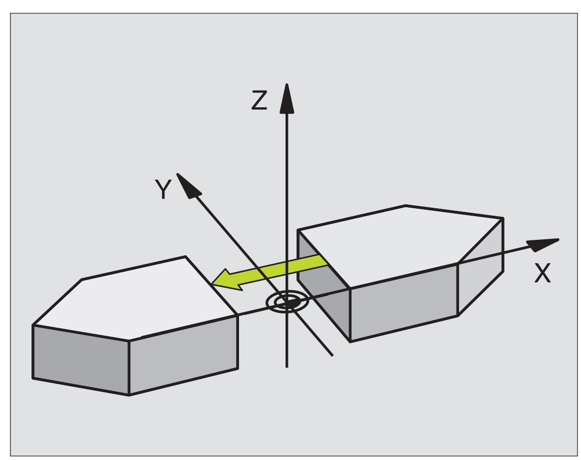


▶ Mirror image: Enter the axis you wish to mirror. The tool axis cannot be mirrored.



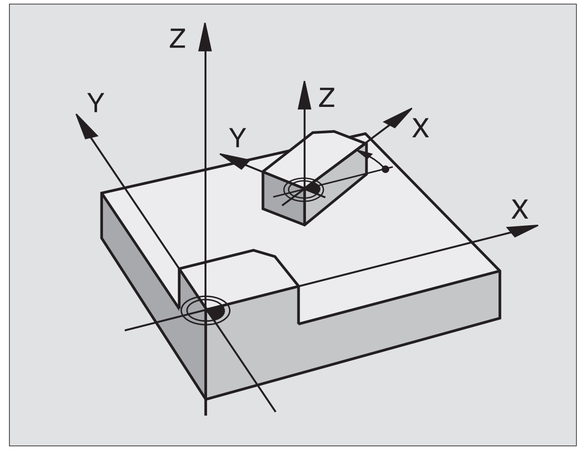
Cancellation

Program the MIRROR IMAGE cycle again without entering an axis.



ROTATION (Cycle 10)

The TNC can rotate the coordinate system about the active datum in the working plane within a program.



Function
The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

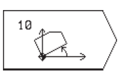
Reference axis for the rotation angle:

- X/Y plane   X axis
- Y/Z plane   Y axis
- Z/X plane   Spindle axis

Before programming, note the following:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

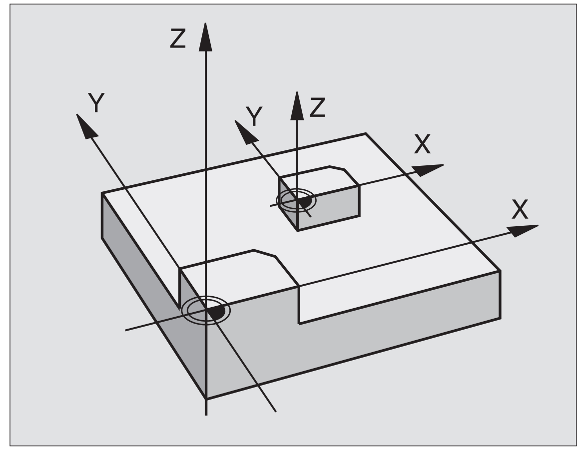
After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.



▶ Rotation: Enter the rotation angle in degrees (°). Input range: –360° to +360° (absolute or incremental).

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.

Function

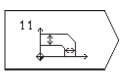
The scaling factor becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

Scaling factor

- 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)

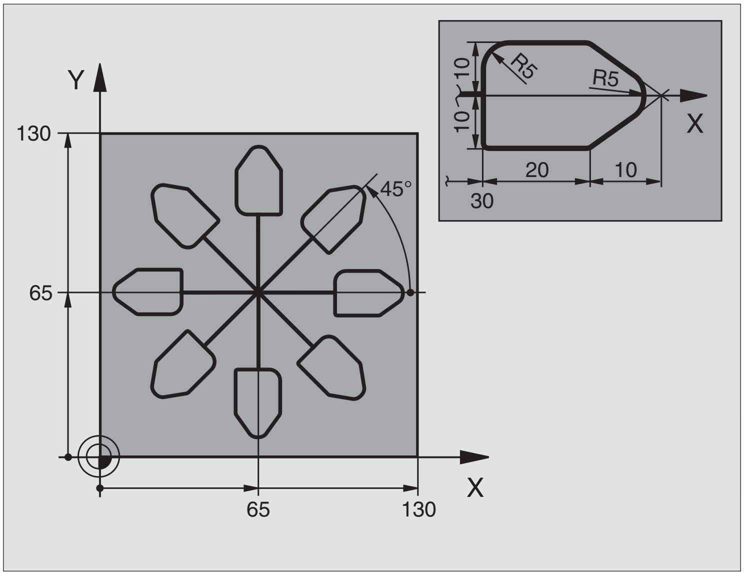
Cancellation

Program the SCALING FACTOR cycle once again with a scaling factor of 1.

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")

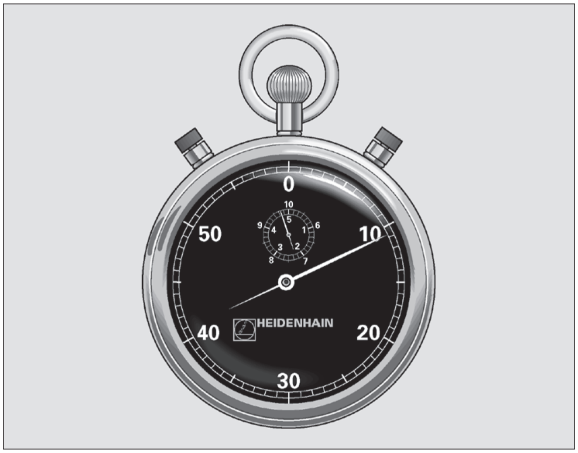


`0 BEGIN PGM 11 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`2 BLK FORM 0.2 X+130 Y+130 Z+0`	
`3 TOOL DEF 1 L+0 R+1`	Define the tool
`4 TOOL CALL 1 Z S4500`	Tool call
`5 L Z+250 R0 F MAX`	Retract the tool
`6 CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center
`7 CYCL DEF 7.1 X+65`	
`8 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 6/6`	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 F MAX M2`	Retract in the tool axis, end program

21 LBL 1	Subprogram 1:
22 L X+0 Y+0 R0 F MAX	Define milling operation
23 L Z+2 R0 F MAX 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 11 MM	

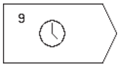
8.7 Special Cycles

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.

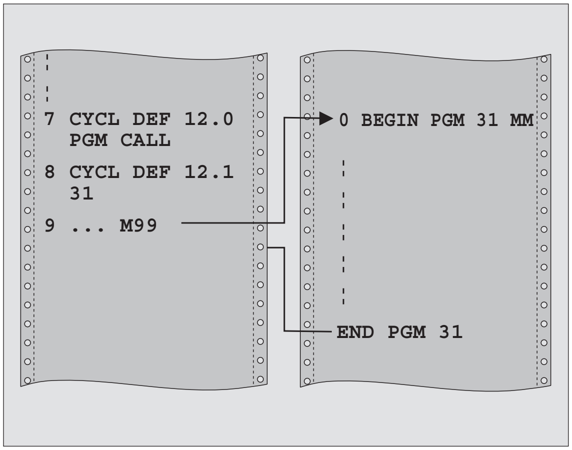
Function
Cycle 9 becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



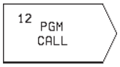
▶ Dwell time in seconds: Enter the dwell time in seconds

Input range: 0 to 30 000 seconds (approx. 8.3 hours) in increments of 0.001 seconds

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: Number of the program to be called up

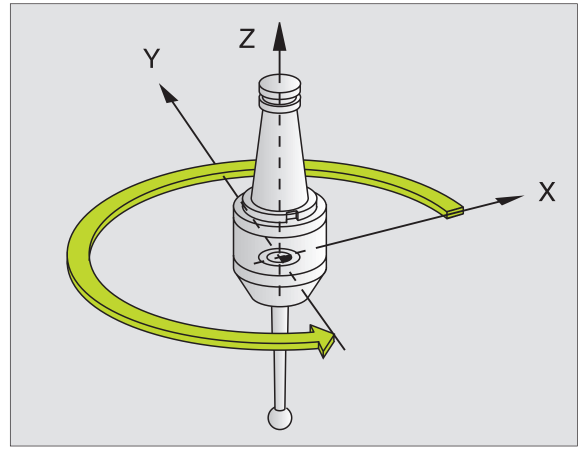
The program is called with
- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (modally)

Example: Program call
A callable program 50 is to be called into a program via a cycle call.

Example NC blocks

`55 CYCL DEF 12.0 PGM CALL`	Definition:
`56 CYCL DEF 12.1 PGM 50`	Program 50 is a cycle”
`57 L X+20 Y+50 FMAX M99`	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 4th axis and rotate it to a given angular position.

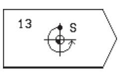
Oriented spindle stops are required for

- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

Function

The angle of orientation defined in the cycle is positioned to by entering M19.

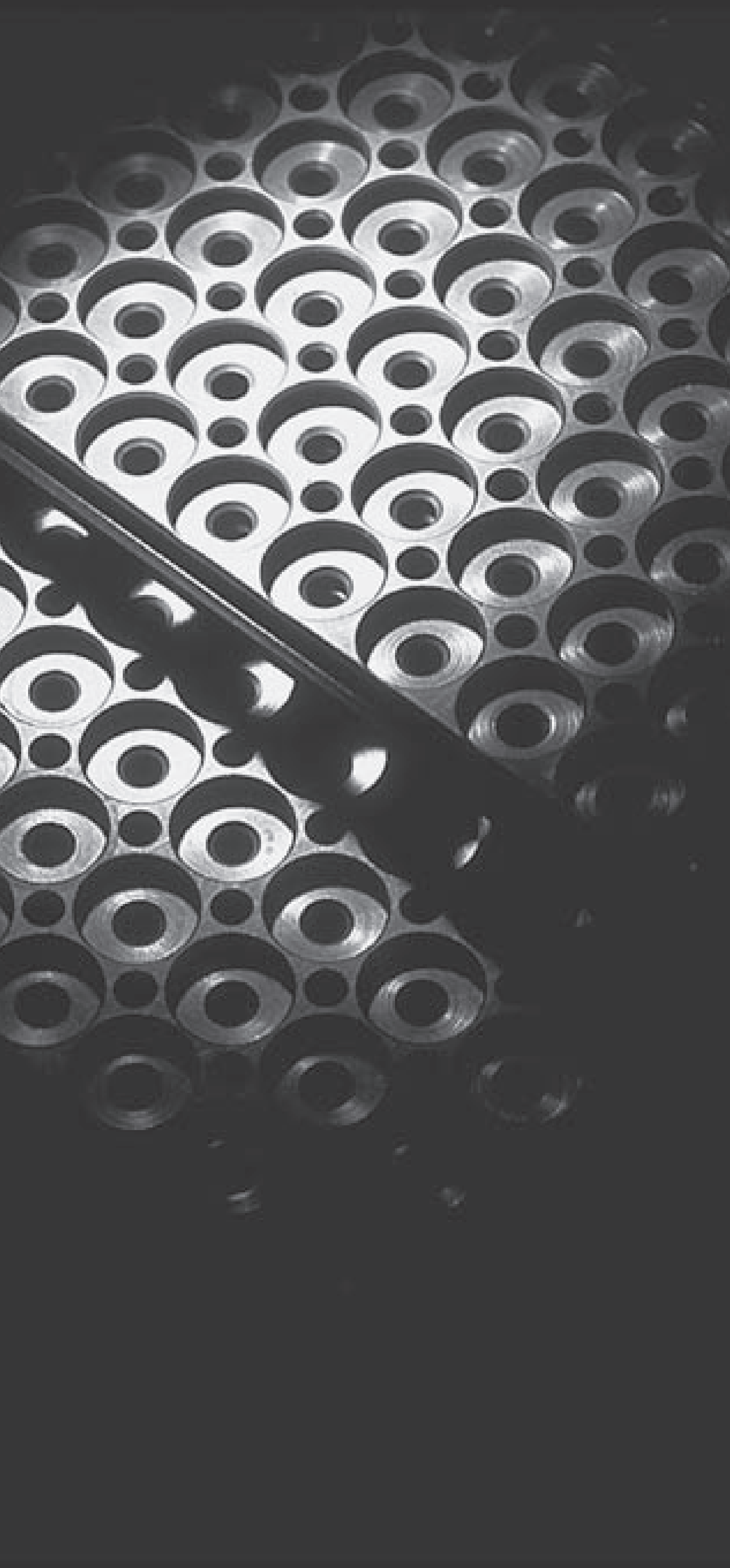
If you program M19 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set in a machine parameter (see your machine manual).



▶ Angle of orientation: Enter the angle according to the reference axis of the working plane.

Input range0 to 360°

Input resolution 0.1°



9

Programming:

Subprograms and Program Section Repeats

9.1 Labeling Subprograms and Program Section Repeats

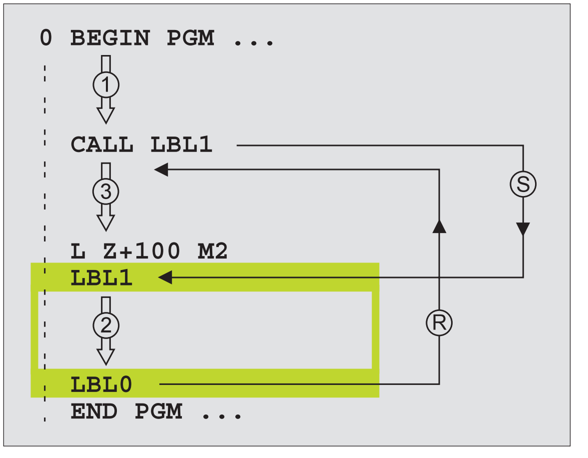
Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as desired.

Labels
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.

Programming notes
- A main program can contain up to 254 subprograms.
- You can call subprograms in any sequence and as often as desired.
- A subprogram cannot call itself.
- Write subprograms at the end of the main program (behind the block with M2 or M30).
- If subprograms are located before the block with M02 or M30, they will be executed at least once even if they are not called.

Programming a subprogram

- To mark the beginning, press the LBL SET key and enter a label number.
- Enter the subprogram.
- To mark the end, press the LBL SET key and enter the label number "0".

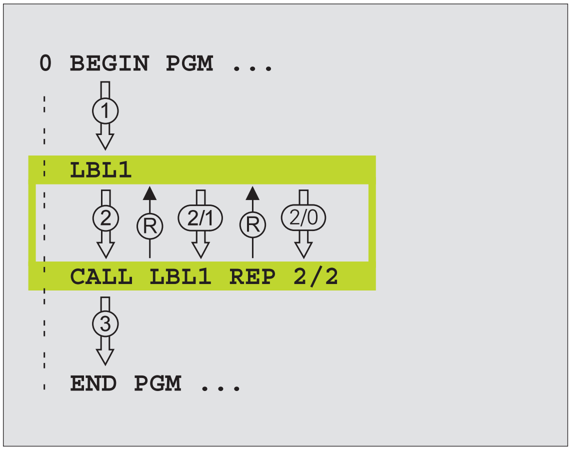
Calling a subprogram

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

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

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 labeled program section, i.e. to the block with CALL LBL /REP.

2 Then the program section between the called LBL and the label call is repeated the number of times entered after REP.

3 The TNC then resumes the part program after the last repetition.

Programming notes

- You can repeat a program section up to 65 534 times in succession.
- The number behind the slash after REP indicates the number of repetitions remaining to be run.
- 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

LBL SET

▶ 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

LBL CALL

▶ 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 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
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example NC blocks

0 BEGIN PGM 15 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 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 15 MM	

Program execution

1st step: Main program 15 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 15.

5th step: Main program 15 is executed from block 18 to block 35. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

`0 BEGIN PGM 16 MM`	
`...`	
`15  LBL 1`	Beginning of program section repeat 1
`...`	
`20  LBL 2`	Beginning of program section repeat 2
`...`	
`27  CALL LBL 2 REP 2/2`	The program section between this block and LBL 2
`...`	(block 20) is repeated twice.
`35  CALL LBL 1 REP 1/1`	The program section between this block and LBL 1
`...`	(block 15) is repeated once.
`50 END PGM 16 MM`	

Program execution

1st step: Main program 16 is executed up to block 27.

2nd step: Program section between block 27 and block 20 is repeated twice.

3rd step: Main program 16 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 16 is executed from block 36 to block 50. End of program.

Repeating a subprogram

Example NC blocks

0 BEGIN PGM 17 MM	
...	
10 LBL 1	Beginning of the program section repeat
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2/2	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 17 MM	

Program execution

1st step: Main program 17 is executed up to block 11.

2nd step: Subprogram 2 is called and executed.

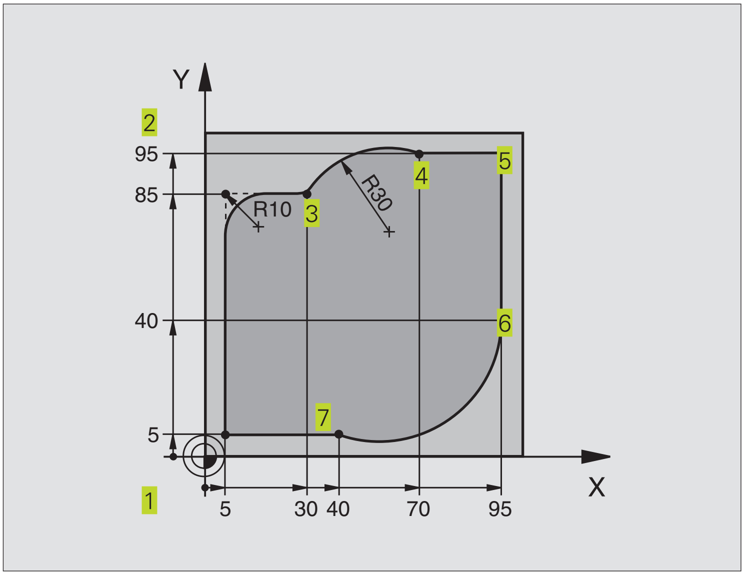
3rd step: Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.

4th step: Main program 17 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

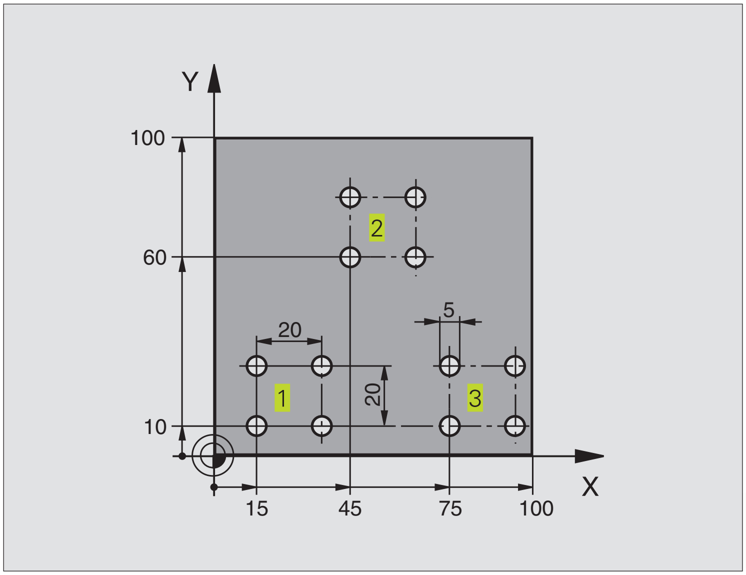


`0 BEGIN PGM 95 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+10`	Define the tool
`4 TOOL CALL 1 Z S4000`	Tool call
`5 L Z+250 R0 F MAX`	Retract the tool
`6 L X-20 Y-20 R0 F MAX`	Pre-position in the working plane
`7 L Z0 R0 F2000 M3`	Pre-position in the spindle axis
`8 LBL 1`	Set label for program section repeat
`9 L IZ-4 r0 F2000`	Infeed depth in incremental values (in the open)
`10 L X+5 Y+5 RL F300`	Approach contour
`11 RND R2`	
`12 L Y+85`	Point 2: first straight line for corner 2
`13 RND R10 F150`	Insert radius with R = 10 mm, feed rate: 150 mm/min
`14 L X+30`	Move to point 3
`15 CR X+70 Y+95 R+30 DR-`	Move to point 4
`16 L X+95`	Move to point 5
`17 L Y+40`	Move to point 6
`18 CT X+40 Y+5`	Move to point 7
`19 L X+5`	Move to last contour point 1
`20 RND R2`	
`21 L X-20 Y-20 R0 F1000`	Depart contour
`22 CALL LBL 1 REP 4/4`	Return jump to LBL 1; section is repeated a total of 4 times
`23 L Z+250 R0 F MAX M2`	Retract in the tool axis, end program
`24 END PGM 95 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



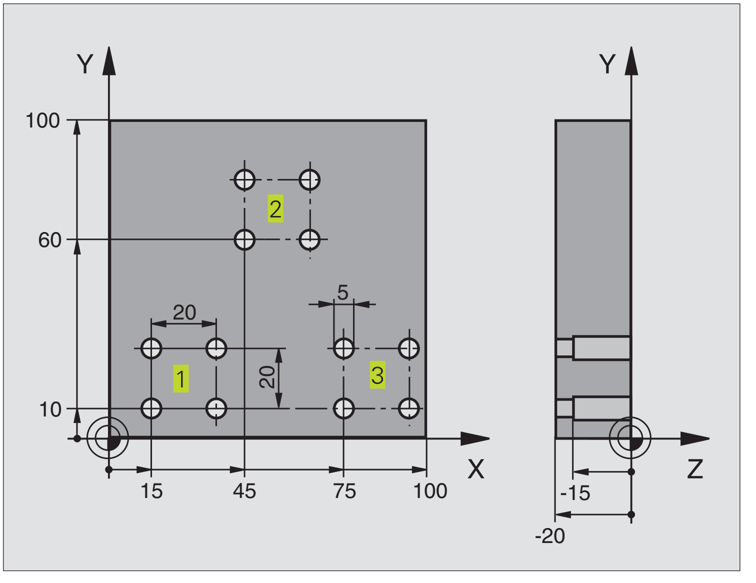
0 BEGIN PGM UP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+2.5	Define the tool
4 TOOL CALL 1 Z S5000	Tool call
5 L Z+250 R0 F MAX	Retract the tool
6 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=250 ;FEED RATE 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 F MAX 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 F MAX M99`	Move to 2nd hole, call cycle
`17 L IY+20 R0 F MAX M99`	Move to 3rd hole, call cycle
`18 L IX-20 R0 F MAX 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

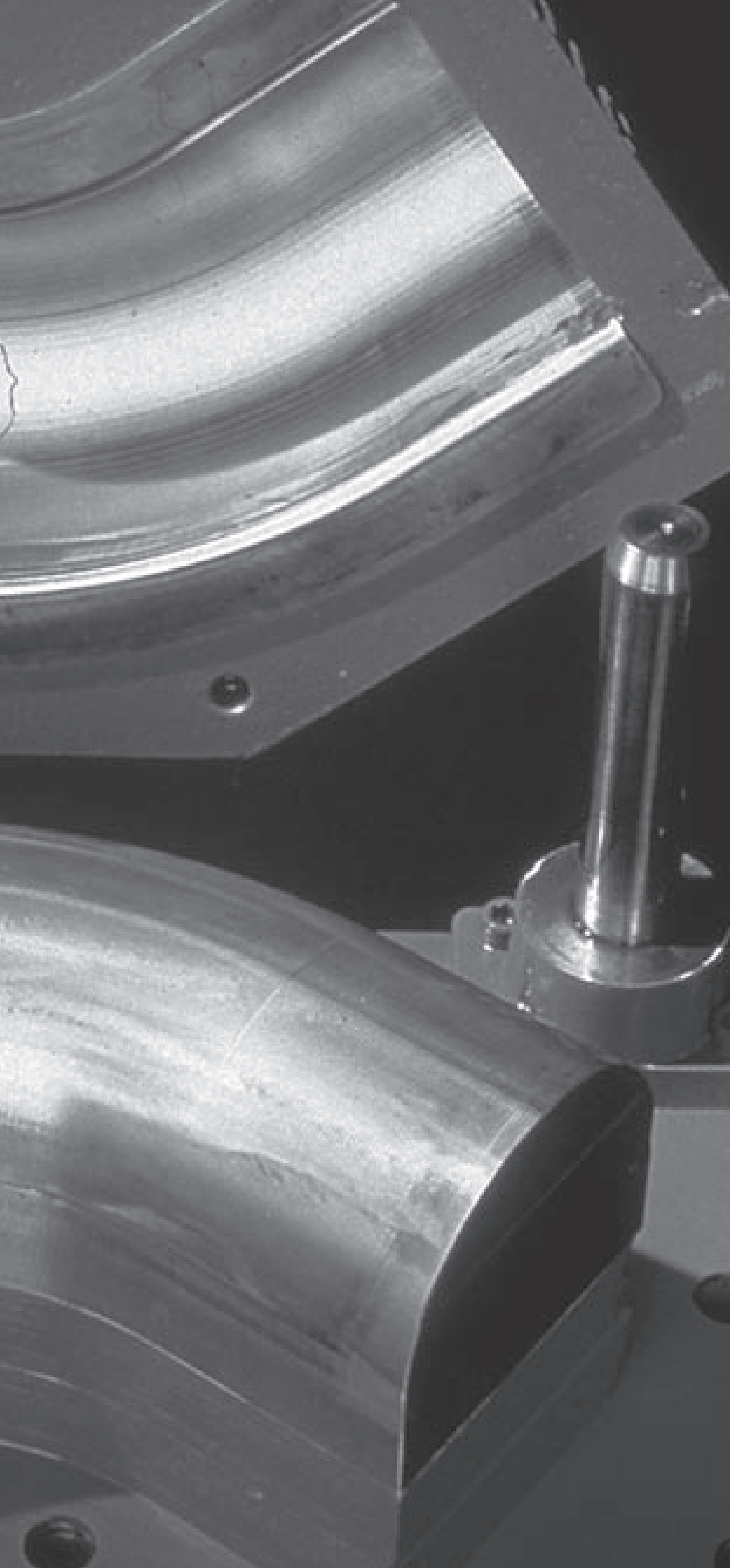
Program sequence

- 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 TOOL DEF 1 L+0 R+4`	Define tool: center drill
`4 TOOL DEF 2 L+0 R+3`	Tool definition: drill
`5 TOOL DEF 3 L+0 R+3.5`	Define tool: reamer
`6 TOOL CALL 1 Z S5000`	Call tool: center drill
`7 L Z+250 R0 F MAX`	Retract the tool

`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 F MAX 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 F MAX M99`	Move to 2nd hole, call cycle
`31 L IY+20 R0 F MAX M99`	Move to 3rd hole, call cycle
`32 L IX-20 R0 F MAX M99`	Move to 4th hole, call cycle
`33 LBL 0`	End of subprogram 2
`34 END PGM UP2 MM`	

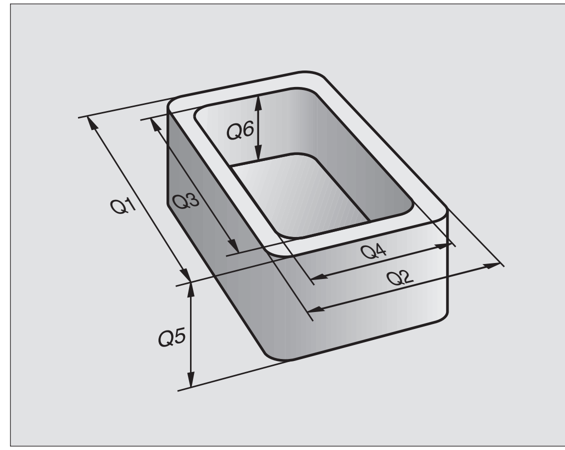


10

Programming:

Q Parameters

10.1 Principle and Overview



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

Q parameters can represent information such as:

- Coordinate values
- Feed rates
- 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 y ou call OEM cycles, these parameters are only effective locally (depending on MP7251).	Q0 to Q99
Parameters for special TNC functions	Q100 to Q150
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 PARAMETER FUNCTIONS soft key. The TNC then displays the following soft keys:

Function group	Soft key
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHMETIC
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

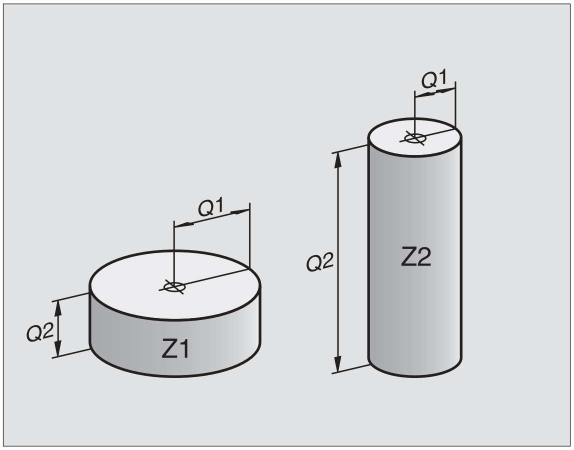
```
15 FN0: Q10 = 25            ASSIGN:
...                         Q10 contains the value 25
25 L X +Q10                 Means L  X +25
```

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 Operations

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

▶ To select the Q parameter function, press the PARAMETER FUNCTIONS soft key. The Q parameter functions are displayed in a soft-key row.

▶ To select the mathematical functions: Press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Function	Soft key
FN0: ASSIGN Example: FN0: Q5 = +60 Assigns a numerical value.	FN0 X = Y
FN1: ADDITION Example: FN1: Q1 = –Q2 + –5 Calculates and assigns the sum of two values.	FN1 X + Y
FN2: SUBTRACTION Example: FN2: Q1 = +10 – +5 Calculates and assigns the difference of two values.	FN2 X - Y
FN3: MULTIPLICATION Example: FN3: Q2 = +3 * +3 Calculates and assigns the product of two values.	FN3 X * Y
FN4: DIVISION e.g. FN4: Q4 = +8 DIV +Q2 Calculates and assigns the quotient of two values **Not permitted:** division by 0	FN4 X / Y
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	FN5 SQRT

At the right of the ”=” character you can enter:

- 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

DATUM TABLE	To select Q parameter functions: Press the PARAMETER FUNCTIONS soft key.
BASIC ARITHMETIC	To select the mathematical functions: Press the BASIC ARITHMETIC soft key.
FN0 X = Y	To select the Q parameter function ASSIGN, press the FN0 X = Y soft key.

Parameter number for result?

5 ENT	Enter a parameter number, for example 5.

1st value or parameter ?

10 END	Assign a value to Q5, for example 10.

DATUM TABLE	To select Q parameter functions: Press the PARAMETER FUNCTIONS soft key.
BASIC ARITHMETIC	To select the mathematical functions: Press the BASIC ARITHMETIC soft key.
FN3 X ∗ Y	To select the Q parameter function MULTIPLICATION, press the FN3 X ∗ Y soft key.

Parameter number for result?

12 ENT	Enter a Q parameter number, for example 12.

1st value or parameter ?

Q5 ENT	Enter Q5 for the first value.

Multiplier?

7 END	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 a are defined by the following equations:

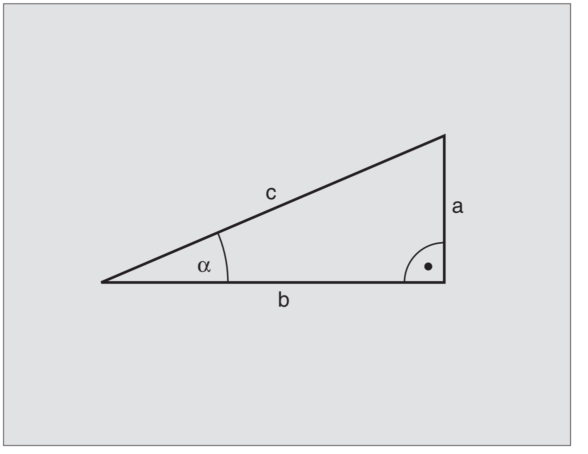
Sine: $\sin\alpha = a / c$

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 a
- 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:

$a = 10$ mm

$b = 10$ mm

$\alpha = \arctan(a / b) = \arctan 1 = 45°$

Furthermore:

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

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

Programming trigonometric functions
Press the TRIGONOMETRY soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table 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

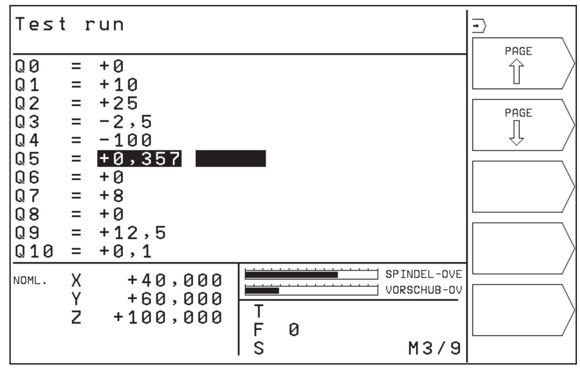
Programming If-Then decisions
Press the JUMP soft key to call the if-then conditions. The TNC then displays the following soft keys:

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.	FN9 IF X EQ Y GOTO
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.	FN10 IF X NE Y GOTO
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.	FN11 IF X GT Y GOTO
FN12: IF LESS THAN, 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.	FN12 IF X LT Y GOTO

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 PARAMETER TABLE soft 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.

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=

FN14: ERROR
Display 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. The program must then be restarted. The error numbers and the associated texts are listed in the table at right.

Example 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

FN15: 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 "13.4 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
```

FN18: SYS-DATUM READ
Read 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
Functions for	2	–	M91 system (see section "7.3 Miscellaneous Coordinate Data)."
Functions for	3	–	M92 system (see section "7.3 Miscellaneous 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.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
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	+
Subtraction Example: Q25 = Q7 – Q108	–
Multiplication Example: Q12 = 5 * Q5	*
Division Example: Q25 = Q1 / Q2	/
Open parentheses Example: Q12 = Q1 * (Q2 + Q3)	(
Close parentheses Example: Q12 = Q1 * (Q2 + Q3)	)
Square Example: Q15 = SQ 5	SQ
Square root Example: Q22 = SQRT 25	SQRT
Sine of an angle Example: Q44 = SIN 45	SIN
Cosine of an angle Example: Q45 = COS 45	COS
Tangent of an angle Example: Q46 = TAN 45	TAN

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, 2.7183^n 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
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: $10^2 = 100$
2nd step: $3^3 = 27$
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.

DATUM TABLE	To select Q parameter functions: Press the PARAMETER FUNCTIONS soft key.
FORMULA	To select formula entry: Press the Q key and the FORMULA soft key.

Parameter number for result?

25 ENT	Enter the parameter number.
▷ ATAN	Shift the soft-key row and select the arc tangent function.
◁ (	Shift the soft-key row and open parentheses.
Q	Enter Q parameter number 12.
/	Select division.
Q	Enter Q parameter number 13.
) END	Close parentheses and conclude formula entry.

Example 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

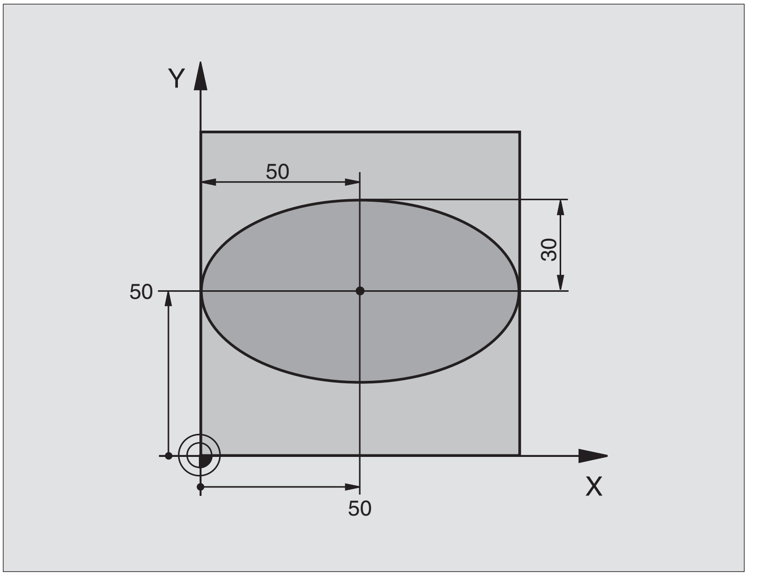
Active radius compensation	Parameter value
R0	Q123 = 0
RL	Q123 = 1
RR	Q123 = 2
R+	Q123 = 3
R–	Q123 = 4

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 workpiece blank
`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`	Tool call
`17 L Z+250 R0 FMAX`	Retract the tool
`18 CALL LBL 10`	Call machining operation
`19 L Z+100 R0 FMAX M2`	Retract in the tool axis, end program

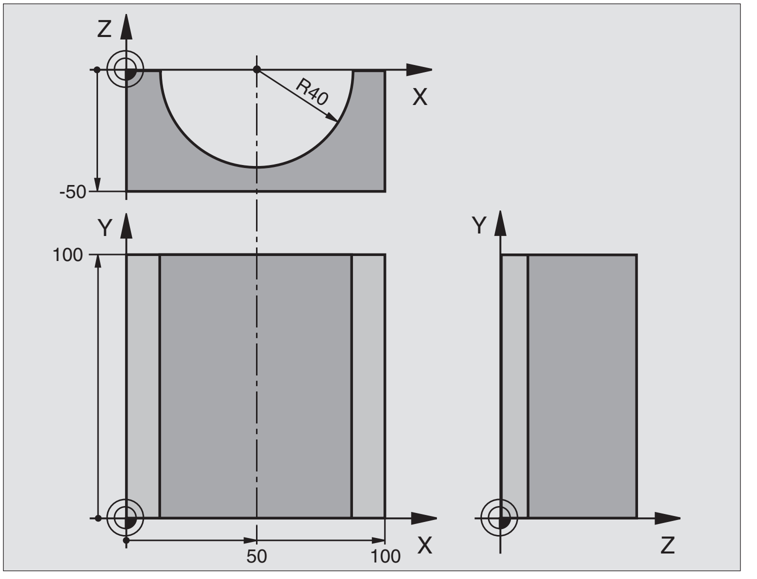
20 LBL 10	Subprogram 10: Machining operation
21 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of ellipse
22 CYCL DEF 7.1 X+Q1	
23 CYCL DEF 7.2 Y+Q2	
24 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
25 CYCL DEF 10.1 ROT+Q8	
26 Q35 = (Q6 - Q5) / Q7	Calculate angle increment
27 Q36 = 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

- Program functions only with a spherical cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The machining direction can be altered by changing the entries for the starting and end angles in space:

  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is compensated automatically.



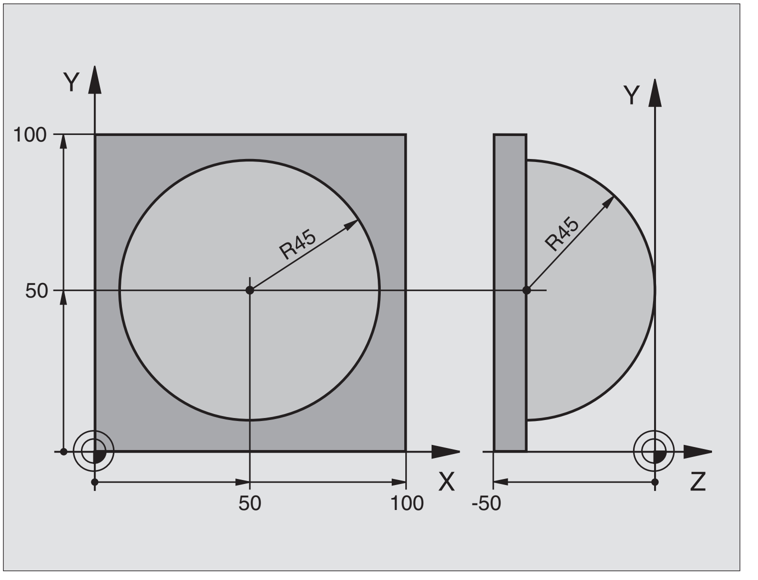
`0 BEGIN PGM CYLIN MM`	
`1 FN 0: Q1 = +50`	Center in X axis
`2 FN 0: Q2 = +0`	Center in Y axis
`3 FN 0: Q3 = +0`	Center in Z axis
`4 FN 0: Q4 = +90`	Starting angle in space (Z/X plane)
`5 FN 0: Q5 = +270`	End angle in space (Z/X plane)
`6 FN 0: Q6 = +40`	Radius of the cylinder
`7 FN 0: Q7 = +100`	Length 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 workpiece blank
`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`	Tool call
`17 L Z+250 R0 FMAX`	Retract the tool
`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 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 workpiece blank
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	Tool call
17 L Z+250 R0 FMAX	Retract the tool
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 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.

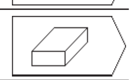
The TNC will not show a graphic if

- the current program has no valid blank form definition
- no program is selected

> A graphic simulation is not possible for program sections or programs in which rotary axis movements are defined. In this case, the TNC will display an error message.

Overview of display modes

After you have pressed the PGM TEST soft key in the operating mode Program Run, the TNC displays the following soft keys:

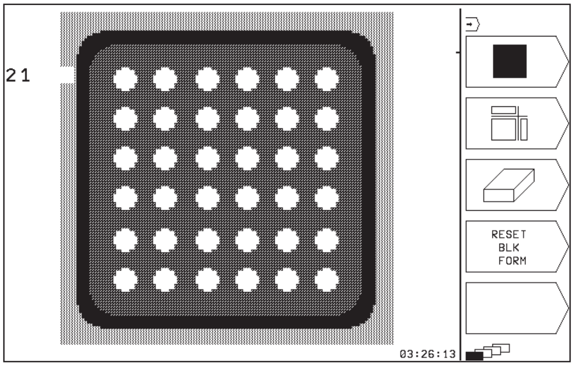
Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Plan view

▶ Press the soft key for plan view.

The deeper the surface, the darker the shade.

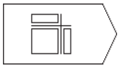
Plan view is the fastest of the three graphic display modes.



Projection in 3 planes

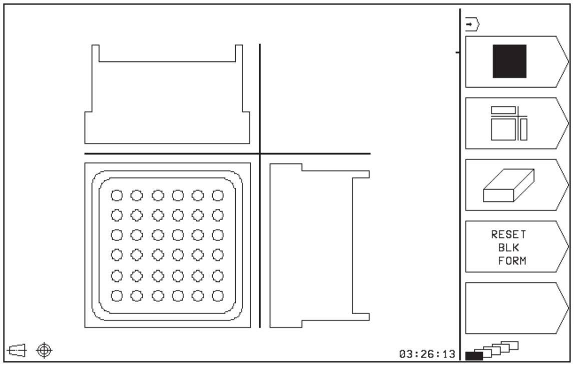
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

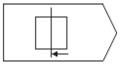
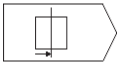
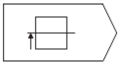
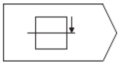
In addition, you can shift the sectional planes with the corresponding soft keys:



▶ 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 left or to the right	 
Shift the horizontal sectional plane upwards or downwards	 

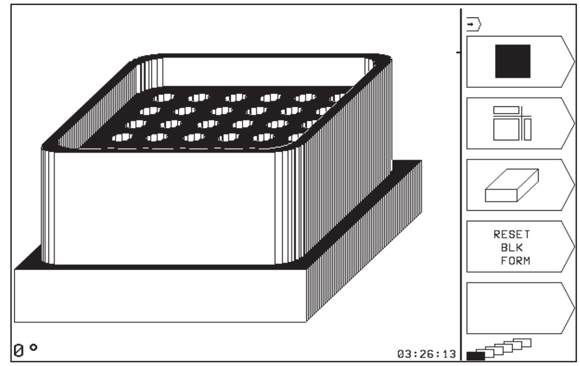
The positions of the sectional planes are visible during shifting.

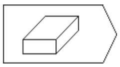
3-D view

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.

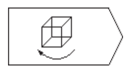
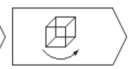
In the TEST RUN mode of operation you can isolate details for magnification (see “Magnifying details”).



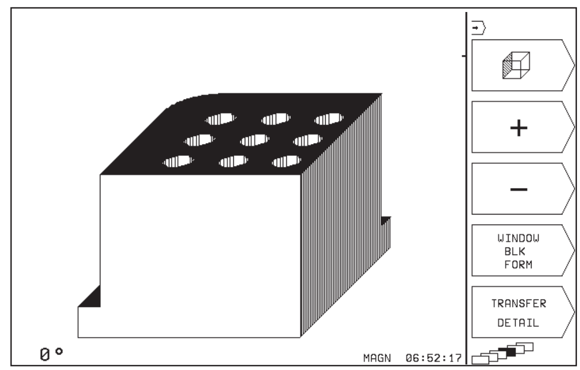


▶ Press the soft key for 3-D view.

To rotate the 3-D view
Shift the soft-key row until the following soft keys appear:

Function	Soft keys
Rotate the workpiece in 90° steps about the vertical axis	 

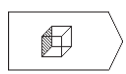
Magnifying details



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	TRANSFER DETAIL

To change the detail magnification:
The soft keys are listed in the table above.

- ▶ Interrupt the graphic simulation, if necessary.
- ▶ Select the workpiece surface with the corresponding soft key (see table).
- ▶ To reduce or magnify the blank form, press the minus or plus soft key, respectively.
- ▶ To select the isolated detail, press the TRANSFER DETAIL soft key
- ▶ Restart the test run or program run.

Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.

Function	Soft key
Restore workpiece blank to the detail magnification in which it was last shown	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM	WINDOW BLK FORM

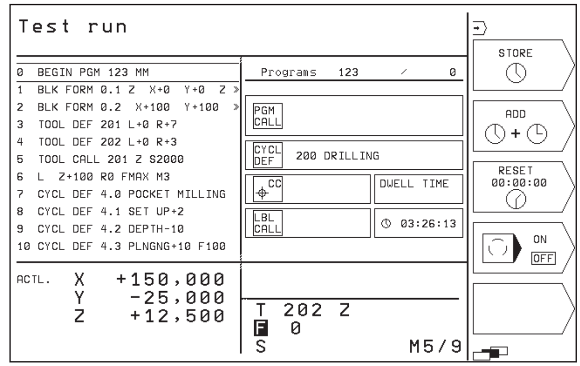
The WINDOW BLK FORM soft key will return the blank form to its original shape or size, even if a detail has been isolated and not yet magnified with TRANSFER DETAIL.

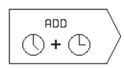
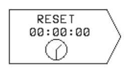
Measuring the machining time

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	
Display the sum of stored time and displayed time	
Clear displayed time	

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:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

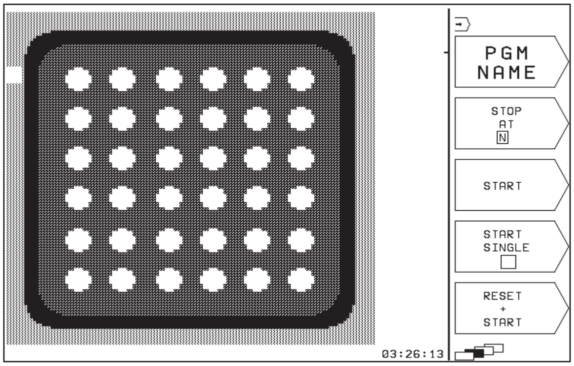
- Blockwise test run
- Interrupt test at any block
- Functions for graphic simulation
- Additional status display

Running a program test

- ▶ Select the PROGRAM RUN operating mode

PGM TEST

- ▶ Select the TEST RUN mode of operation.
- ▶ Call the file manager with the PGM NAME key and select the file you wish to test, or
- ▶ Go to the program beginning: Select line "0" with the GOTO key and confirm you entry with the ENT key.



The TNC displays the following soft keys (1st or 2nd soft-key row):

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

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.

- ▶ Go to the beginning of program in the Test Run mode of operation.
- ▶ To do a test run up to a certain block, press the STOP AT N soft key.

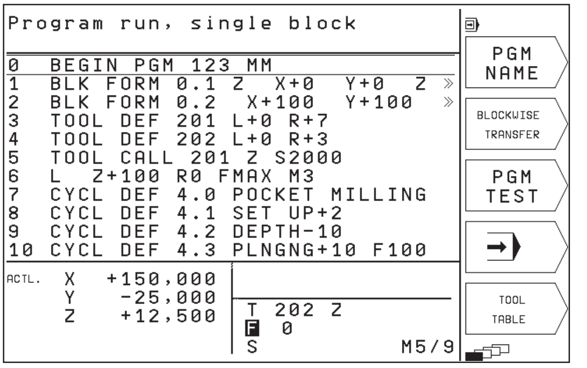
STOP AT N

- ▶ 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 have called with Cycle 12 PGM CALL, enter the number of the program containing the block with the selected block number.
- ▶ Repetitions: If the block number 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.

11.3 Program Run

In the Program Run operating mode the TNC executes a part program either in single block or continuously.

Function	Soft key
Program Run, Single Block (default setting)	
Program Run, Full Sequence	



Im the Program Run, Single Block mode you must start each block separately by pressing the NC START button.

Im the Program Run, Full Sequence mode the TNC executes a part program continuously to its end or up to a program stop.

The following TNC functions can be used in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Additional status display

Running a part program

Preparation

1 Clamp the workpiece to the machine table.

2 Datum setting

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 machining program with the NC start button

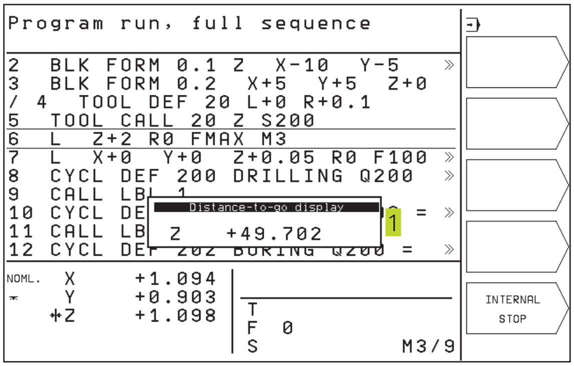
Program Run, Single Block

- Start each block of the part program individually with the NC START button.

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 (1 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.

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 functions M0, M1 (see „11.5 Optional Program Run Interruption“), M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

Interruption with NC STOP button

- ▶ Press the NC 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 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.

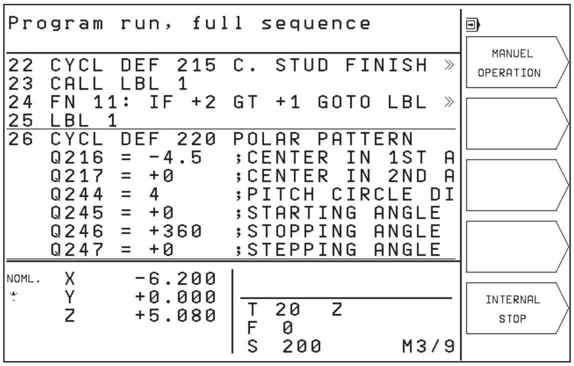
Example:
Retracting the spindle after tool breakage

▶ Interrupting 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.



Resuming program run after an interruption

> If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.

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 NC START button
You can resume program run by pressing the NC START button if the program was interrupted in one of the following ways:

- NC STOP button was pressed
- A programmed interruption
- The EMERGENCY STOP button was pressed (machine-dependent function)

> If you interrupted program run with the STOP soft key, you can select another block with the GOTO key and resume machining there.
>
> If you select 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*

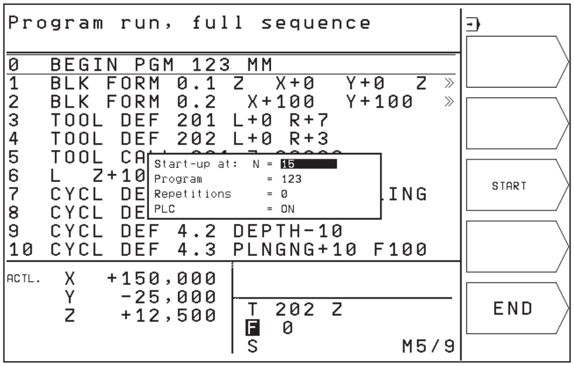
▶ Switch off the TNC and the machine.

▶ 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)



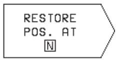
With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point.

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).

- 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:



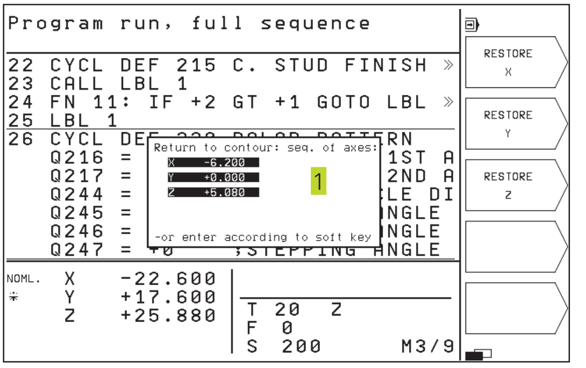
- Start-up at N: Enter the block number N at which the block scan should end.
- Program: Enter the name of the program containing block N.
- Repetitions: If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.
- PLC ON/OFF: To account for 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.
- 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.

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.



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 section 13.4 "Setting the External Data Interface").

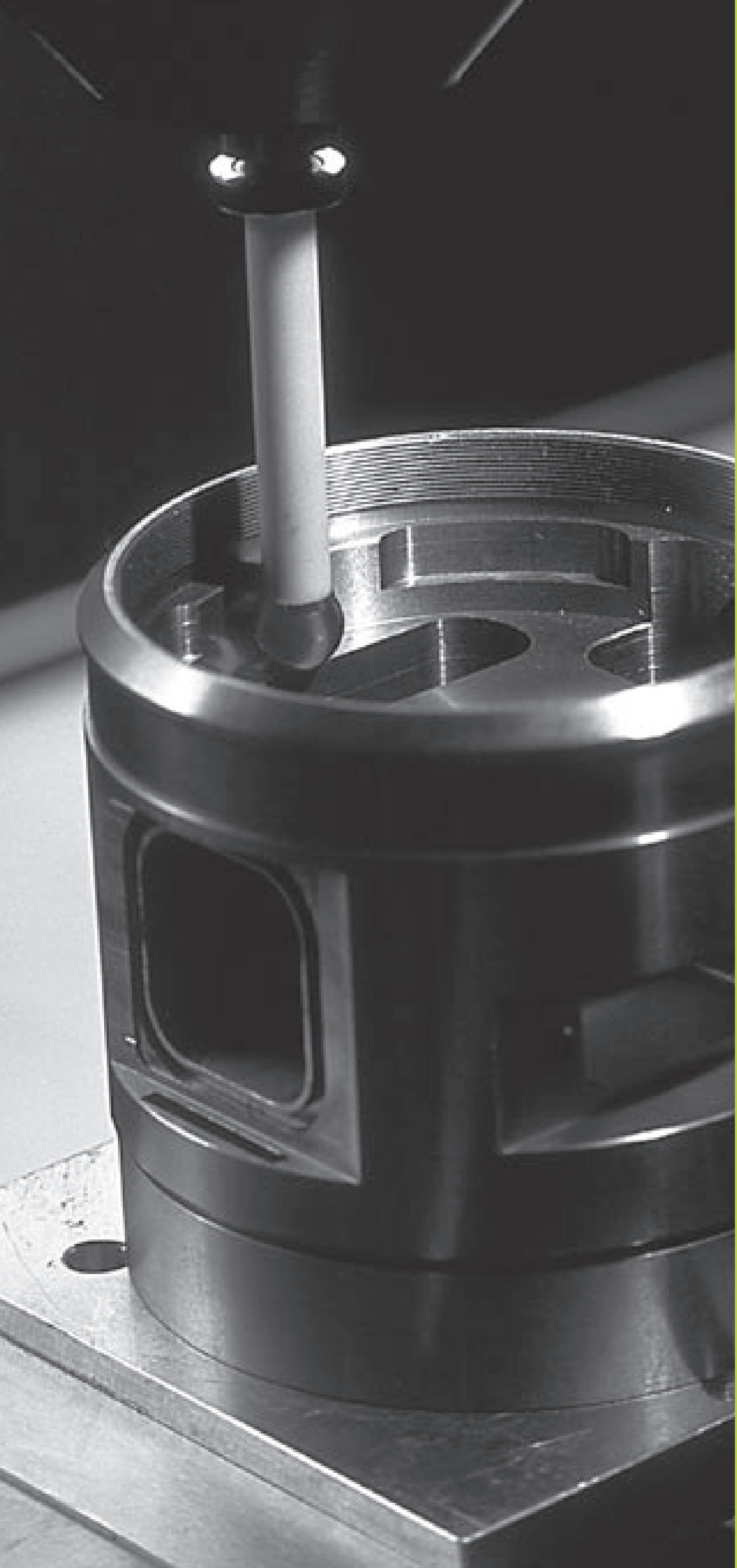
- ▶ Select the Program Run, Full Sequence mode or the Program Run, Single Block mode.
- ▶ Begin blockwise transfer: Press the BLOCKWISE TRANSFER soft key.
- ▶ Enter the program name and confirm your entry with the ENT key. The TNC reads in the selected program via data interface
- ▶ Start the part program 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 Program Run Interruption

The TNC optionally interrupts the program or test run at blocks containing M01:

▶ 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

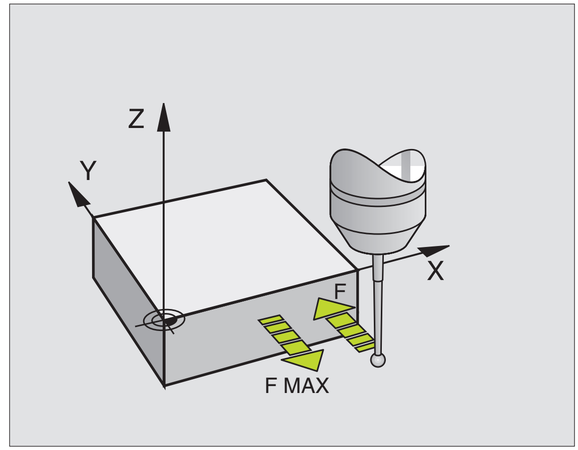


12

3-D Touch Probes

12.1 Touch Probe Cycles in the Manual Operation Mode

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.



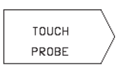
After you press the NC START button, the touch probe approaches the workpiece in a paraxial movement in the selected probe function. The machine tool builder sets the probe feed rate (see figure at right). When the probe contacts the workpiece, it

- transmits a signal to the TNC: the coordinates of the probed position are stored,
- stops moving, and
- returns to its starting position in rapid traverse.

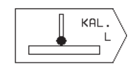
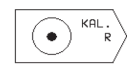
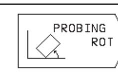
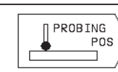
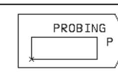
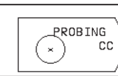
If the stylus is not deflected within a defined distance, the TNC displays an error message (MP 6130).

Select the touch probe function

▶ Select the Manual Operation mode.



▶ To choose the touch probe functions, press the TOUCH PROBE soft key (2nd soft-key row). The TNC displays additional soft keys — see table at right.

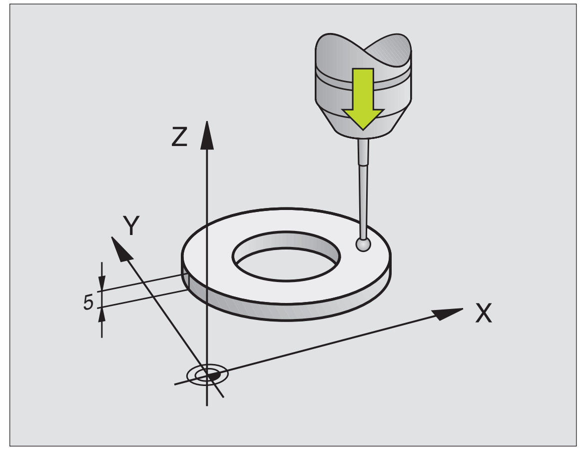
Function	Soft key
Calibrate the effective length (2nd soft-key row)	
Calibrate the effective radius (2nd soft-key row)	
Basic rotation	
Datum setting	
Set the datum at a corner	
Set the datum at a circle center	

Calibrating a touch trigger probe

The touch probe must be calibrated:

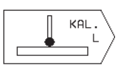
- during commissioning
- when the stylus breaks
- when the stylus is changed
- when the probe feed rate is changed
- in case of irregularities such as those resulting from thermal changes in the machine

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.

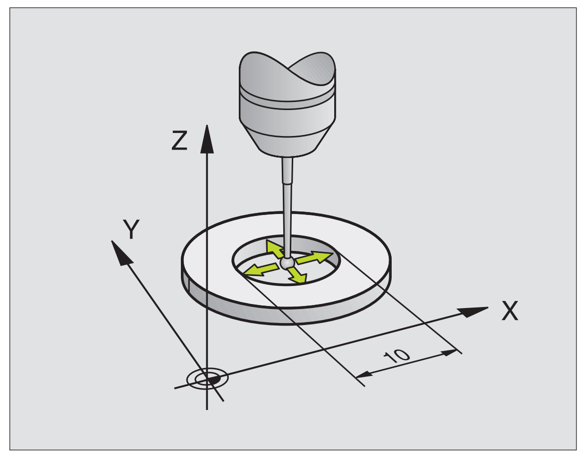
▶ Datum: Enter the height of the ring gauge.

▶ 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, press the NC START button



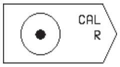
Calibrating the effective radius and compensating center misalignment

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.

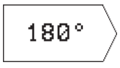
For this operation the TNC rotates the 3-D touch probe by 180°. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in the machine parameter 6160.

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.



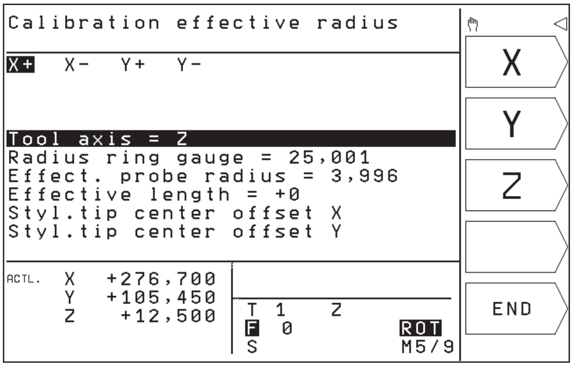
- To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL R soft key.
- Select the tool axis and enter the radius of the ring gauge.
- To start probing: 4 x NC START button. The touch probe contacts a position on the bore in each axis direction 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 start probing: 4 x NC START button. 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.

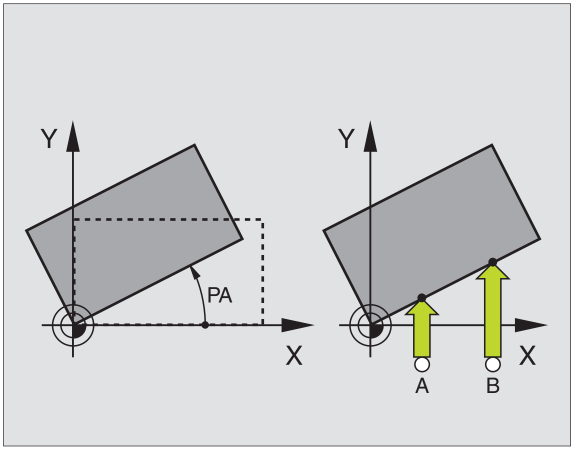


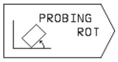
Compensating workpiece misalignment

The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

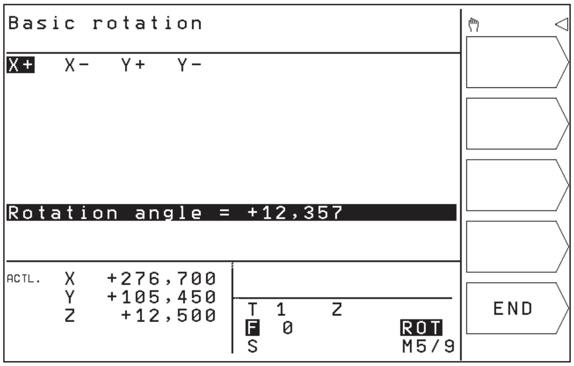
For this purpose, the TNC sets the rotation angle to the desired angle with respect to the reference axis in the working plane. See figure at lower right.

> 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 probing function by pressing 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 NC START button
- Position the ball tip at a starting position near the second touch point.
- To probe the workpiece, press the NC START button



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 (STATUS POS.) .

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:

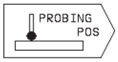
- Select the probing function by pressing the PROBING ROT soft key.
- Enter a rotation angle of zero and confirm with the ENT key.
- To terminate the probe function, press the END key.

12.2 Setting the Datum with a 3-D Touch Probe

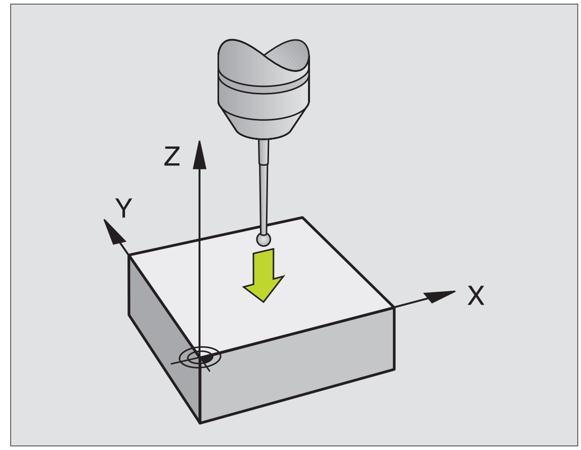
The following functions are available for setting the datum on an aligned workpiece:

- 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

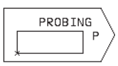
To set the datum in any axis (see figure at upper right)



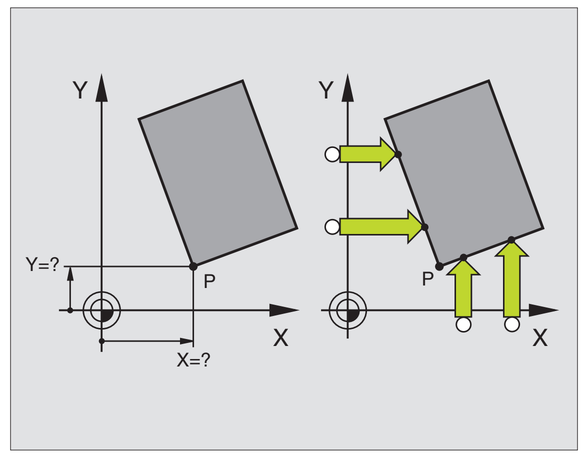
- To select the touch probe function: Press the PROBING POS soft key.
- Move the touch probe to a starting position near the touch point.
- Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–. Selection is made with the arrow keys.
- To probe the workpiece, press the NC START button
- Datum: Enter the nominal coordinate and confirm you entry with ENT.



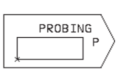
Corner as datum — using points that were already probed for a basic rotation (see figure at center right)



- Select the touch probe function: Press the PROBING P soft key
- TOUCH POINTS OF BASIC ROTATION?: Press YES 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 NC START button
- Position the touch probe near the second touch point on the same side.
- To probe the workpiece, press the NC 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.



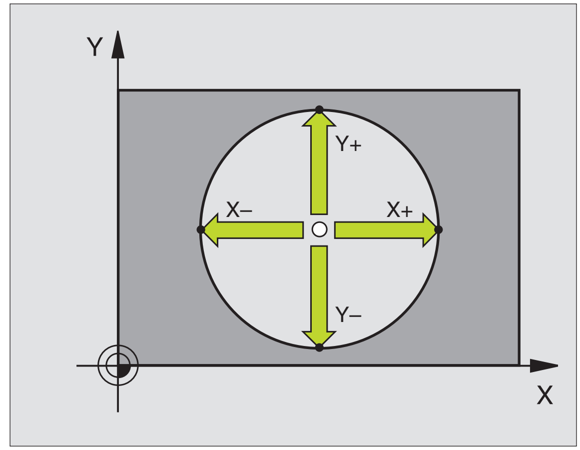
Corner as datum – without using points that were already probed for a basic rotation



- Select the touch probe function: Press the PROBING P soft key
- Touch points of basic rotation?: Press NO 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 coordinates of the datum 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.

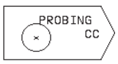


Inside circle

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

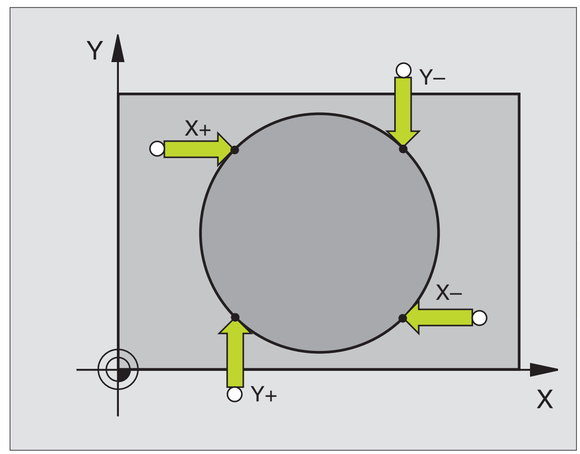
▶ To probe the workpiece, press the NC START button four times The touch probe touches four points on the inside of the circle.

▶ 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 NC START button

▶ Repeat the probing process for the remaining three points.See figure at center right.

▶ Enter the coordinates of the datum and confirm your entry with ENT.

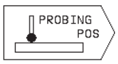
After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

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:



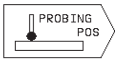
- ▶ To select the touch probe function: Press the PROBING POS soft key.
- ▶ Move the touch probe to a starting position near the touch point.
- ▶ Select the probe direction and axis of the coordinate. Use the arrow keys for selection.
- ▶ To start probing, press the NC START button

The TNC shows the coordinates of the touch point as datum.

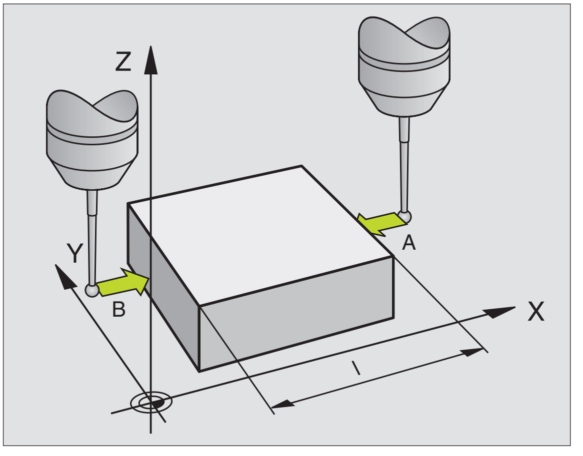
Finding the coordinates of a corner in the working plane

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



- ▶ To 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 key.
- ▶ To probe the workpiece, press the NC 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 again: 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 an arrow key: Same axis but opposite direction as for A.
- To probe the workpiece, press the NC 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:

- To 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 also 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.

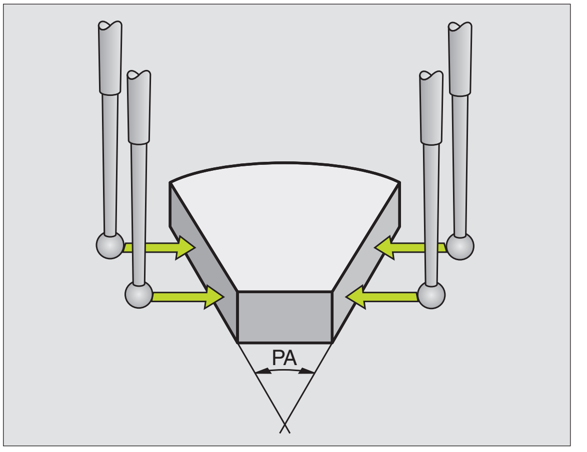
The measured angle is displayed as a value of maximum 90°.

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:

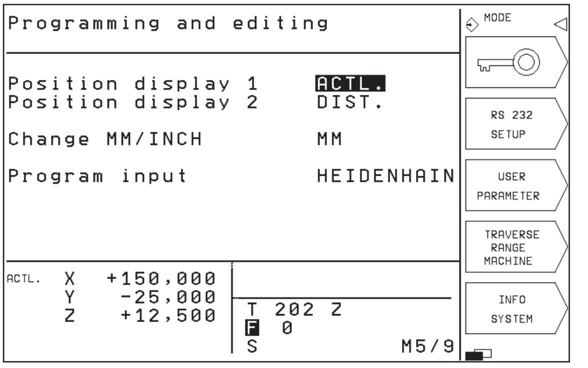
- Select the probing function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation for the first side (see "Compensating workpiece misalignment").
- Probe the second side 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.



13

MOD Functions

13.1 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.

MOD

▶ 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)
- 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:
▶ To exit the MOD function, press the END key.

13.2 System Information

You can use the soft key INFO SYSTEM to display the following information:

- Free program memory
- NC software number
- PLC software number

are displayed on the TNC screen after the functions have been selected.

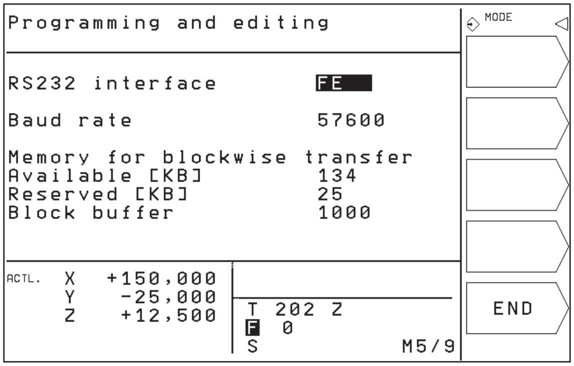
13.3 Entering the Code Number

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
Select user parameters	123
Cancel file protection	86357
Operating hours counter for: CONTROL ON PROGRAM RUN SPINDLE ON	857282

13.4 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 Punchers, PC without TNC.EXE	EXT1, EXT2
PC with HEIDENHAIN software for data transfer - TNCremo	FE
No data transfer; e.g. working without a connected 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 an individual baud rate. When you select baud rate using an arrow key, the TNC recalls the value that was last stored for this operating mode.

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

StartingTNCremo underWindows
Windows 3.1, 3.11, NT:

- Double-click on the icon in the program group HEIDENHAIN Applications

Windows 95:

- 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 310 and TNCremo
Ensure that:

- The TNC 310 is connected to the correct serial port on your PC
- 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.

EndTNCremo
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.

13.5 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 USER PARAMETERS. Your machine manual provides more detailed information.

13.6 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.

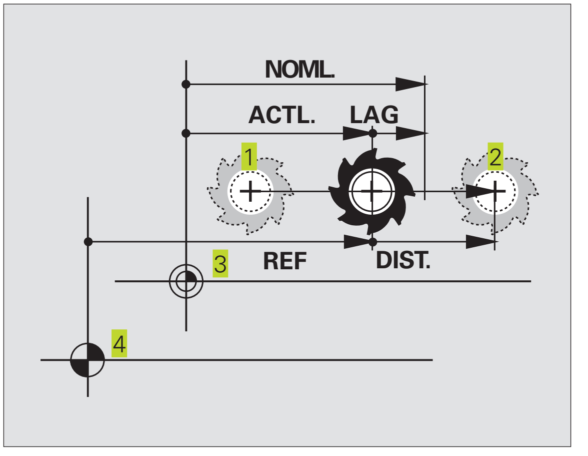
The figure at right shows the different tool positions:

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.

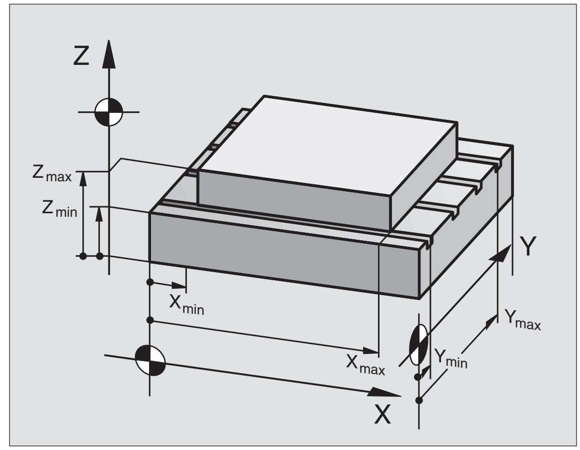
13.7 Unit of Measurement

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/INCH 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.

13.8 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.

TRAVERSE RANGE MACHINE

- Enter the limits for axis traverse: Press the TRAVERSE RANGE MACHINE soft key and enter the values that you wrote down as limits in the corresponding axes. Confirm your entry 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 (2nd soft-key row), after you have activated the MOD function.

In addition to the axis traverse limits, you can also define the position of the workpiece datum referenced to the machine datum.

To save any values you have changed, press the ENT key.

13.9 Running the HELP File

The HELP function is not available on every machine. Refer to your machine tool builder for more information.

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: Press the MOD key
- To select the HELP function: Press 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

14

Tables and Overviews

14.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements.

Some examples of user parameters are:

- Dialog language
- Interface behavior
- Traversing speeds
- Sequence of machining
- Effect of overrides

Input possibilities for machine parameters

Enter the machine parameters as decimal numbers

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign.

Selecting general user parameters

General user parameters are selected with code number 123 in the MOD functions.

> The MOD functions also include machine-specific user parameters.

External data transfer

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**
	RTS always active: **+0** RTS only active if data transfer has been started: **+256**
	Send EOT after ETX: **+0** Do not send EOT after ETX: **+512**

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**

3-D Touch Probes

Probing feed rate for triggering touch probes	**MP6120** **80** to **3000** [mm/min]
Maximum traverse to first probe point	**MP6130** **0.001** to **30 000** [mm]
Safety clearance to probing point during automatic measurement	**MP6140** **0.001** to **30 000** [mm]
Rapid traverse for triggering touch probes	**MP6150** **1** to **30 000** [mm/min]
Measure center misalignment of the stylus when calibrating a triggering touch probe	**MP6160** No 180° rotation of the 3-D touch probe during calibration: **0** M function for 180° rotation of the touch probe during calibration: **1** to **88**

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**
Dialog language	**MP7230** German: **0** English: **1**
Configure tool tables	**MP7260** Inactive: **0** Number of tools per tool table: **1** to **254**

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	**MP7290.0** 0.1 mm or 0.1°: **0** 0.05 mm or 0.05°: **1** 0.01 mm or 0.01°: **2** 0.005 mm or 0.005°: **3** 0.001 mm or 0.001°: **4**
Display step for the Y axis	**MP7290.1** see MP 7290.0
Display step for the Z axis	**MP7290.2** see MP 7290.0
Display step for the IVth axis	**MP7290.3** see MP 7290.0
Reset status display, Q parameters and tool data	**MP7300** Do not cancel Q parameters and status display: **+0** Q parameters and status display with M02, M30, END PGM: **+1** After a power interruption do not activate tool data that was last active: **+0** After a power interruption re-activate tool data that was last active: **+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**

Machining and program run

Cycle 17: Oriented spindle stop at beginning of cycle

MP7160
Spindle orientation: **0**
No spindle orientation: **1**

Effect of Cycle 11 SCALING FACTOR

MP7410
SCALING FACTOR effective in 3 axes: **0**
SCALING FACTOR effective in the working plane only: **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**
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**

Angle of tool-path directional change up to which the feed rate will remain constant

(applies for radius-compensated inside corners and corners with R0)
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

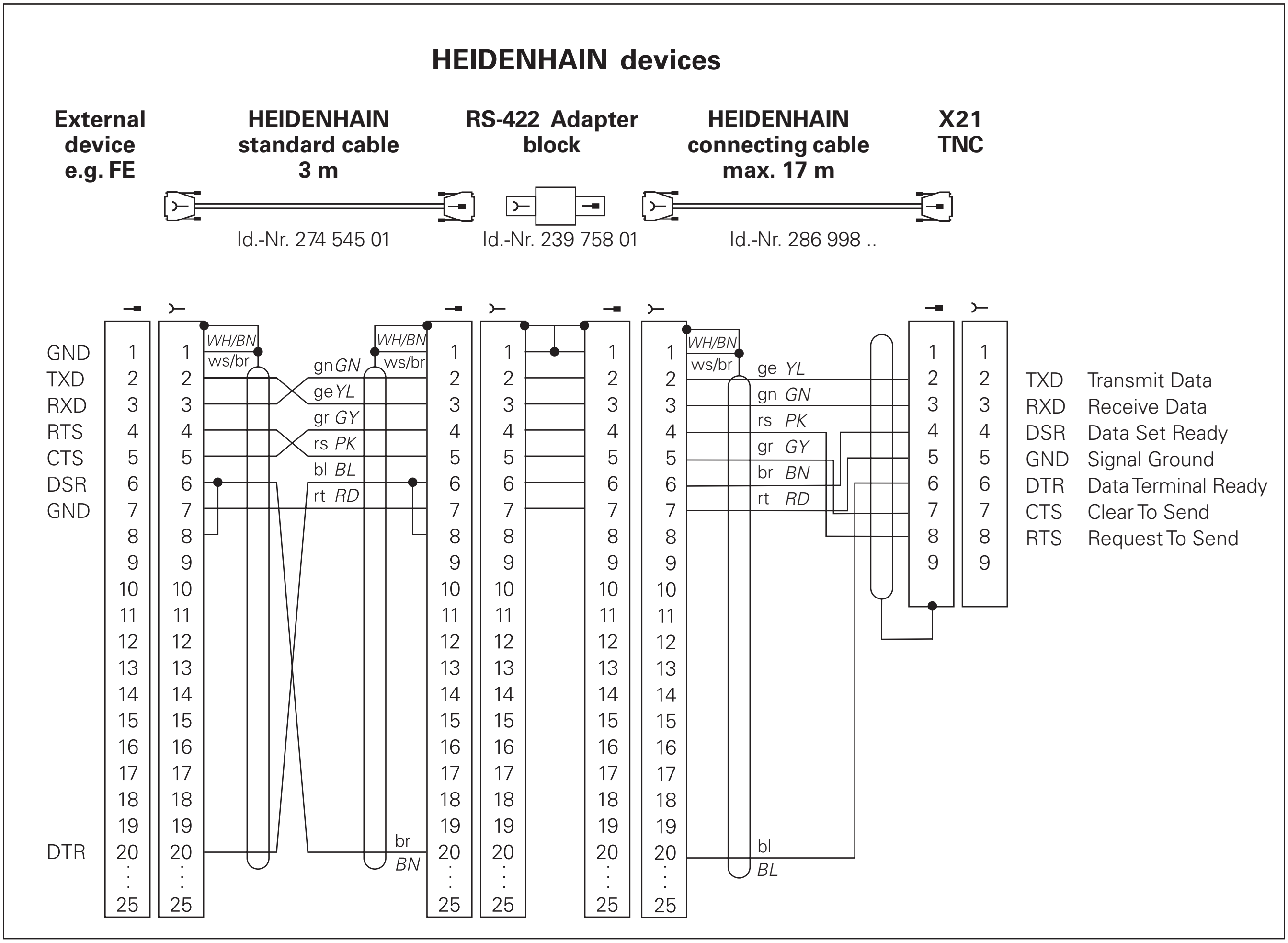
Handwheel type

MP7640
Machine without handwheel: **0**
HR 330 with additional keys _ the handwheel keys for traverse direction and rapid traverse are evaluated by the NC: **1**
HR 130 without additional keys: **2**
HR 330 with additional keys _ the handwheel keys for traverse direction and rapid traverse are evaluated by the PLC: **3**
HR 332 with twelve additional keys: **4**
Multi-axis handwheel with additional keys: **5**
HR 410 with miscellaneous functions: **6**

14.2 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 above shows the connector pin layout on the adapter block.

14.3 Technical Information

TNC features

Description	Contouring control for machines with: control of 4 axes without spindle control or control of 3 axes and with spindle control
Components	Compact control with integrated flat-panel display (192 mm x 120 mm, 640 x 400 Pixel) and integrated machine control buttons
Data interface	■ RS-232-C / V.24
Simultaneous axis control for contour elements	■ Straight lines: up to 3 axes ■ Circles: up to 2 axes ■ Helices: 3 axes
Background programming	One part program can be edited while the TNC runs another program
Graphics	■ Programming graphics ■ Test run graphics
File types	■ HEIDENHAIN conversational programming ■ Tool table
Program memory	■ Battery back-up for approx. 6000 NC blocks (depending on length of block), 128 KB ■ Up to 64 files possible
Tool definitions	Up to 254 tools in the program or in the tool table
Programming support	■ Functions for approaching and departing the contour ■ HELP function

Programmable functions

Contour elements	■ Straight line ■ Chamfer ■ Circular arc ■ Circle center ■ Circle radius ■ Tangentially connecting circle ■ Corner rounding ■ Straight lines and circular arcs for contour approach and departure
Program jumps	■ Subprograms ■ Program section repeats
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring back boring, tapping with a floating tap holder ■ Roughing and finishing rectangular and circular pockets ■ Cycles for milling linear and circular slots ■ Linear and circular hole patterns ■ Cycles for multipass milling of flat and irregular surfaces
Coordinate transformations	■ Datum shift ■ Mirroring ■ Rotation ■ Scaling
3-D touch probe applications	■ Touch probe functions for setting the datum

TNC Specifications

Block processing time	40 ms/block
Control loop cycle time	Contouring interpolation: 6 ms
Data transfer rate	Maximum 115 200 baud
Ambient temperature	■ In operation: 0°C to +45°C ■ Storage: –30°C to +70°C
Traverse range	Maximum 30 m (1181 inches)
Traversing speed	Maximum 30 m/min (1181 inches/min)
Spindle speed	Maximum 30 000 rpm
Input range	■ Minimum 1µm (0.0001 inches) or 0.001° ■ Maximum 30 000 mm (1 181 inches) or 30 000°

14.4 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 program entry impossible	Erase some old files to make room for new ones.
Entry value incorrect	■ Enter a correct label number ■ Note the input limits
Ext. in-/output not ready	■ 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	■ Pre-position the 3-D touch probe to a position nearer the model
Arithmetical error	You have calculated with non-permissible values ■ Define values within the range limits ■ Choose probe positions for the 3-D touch probe that are farther apart
Path offset wrongly ended	Do not cancel tool radius compensation in a block with a circular path
Path offset wrongly started	■ 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 incomplete	■ Define the cycles with all data in the proper sequence ■ Do not call the coordinate transformation cycles ■ Define the cycle before calling it ■ Enter a pecking depth other than 0.
BLK FORM definition incorrect	■ Program the MIN and MAX points according to the instructions. ■ Choose a ratio of sides that is less than 200:1.
Plane wrongly defined	■ 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	■ 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.
Gross positioning error	The TNC monitors positions and movements. If the actual position deviates excessively from the nominal position, this blinking error message is displayed. You must press the END key for a few seconds to acknowledge the error message (system reset).
No editing of running program	A program cannot be edited while it is being executed.
Circle end pos. incorrect	■ Enter complete information for connecting arc. ■ Enter end points that lie on the circular path
Circle center undefined	■ Define a circle center with the CC key. n 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	■ Enter a smaller tool radius ■ Enter a tool axis for simulation that is the same as the axis in the BLK FORM.
Radius comp. undefined	Radius compensation RR or RL is only possible when tool radius is not equal to 0
Rounding not permitted	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.
Probe system not ready	■ Check whether the touch probe is ready for operation
Program start undefined	■ 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	■ Enter the feed rate for the positioning block. n Enter FMAX in each block.
Tool radius too large	Enter a tool radius ■ that lies within the given limits ■ permits the contour elements to be calculated and machined
Angle reference missing	■ Complete your definition of the arc and its end points. ■ If you enter polar coordinates, define the polar angle correctly.
Excessive subprogramming	■ 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.

14.5 Exchanging the Buffer Battery

A buffer battery supplies the TNC with current to prevent the data in RAM memory from being lost when the TNC is switched off.

If the TNC displays the error message Exchange buffer battery, then you must replace the batteries. The batteries are located in the control housing (refer also to your Machine Manual). 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...203
- compensating center misalignment...203

3-D view...188

A

Accessories...11
Actual position transfer...59
Angle functions...164

B

Back boring...103
Blank form definition...36
Block buffer...214
Blocks
- copying...38
- deleting...38
- editing...38
- inserting...38

Blockwise transfer...199
Boring...100
Buffer battery, exchanging...232

C

Cable for data interface...226
Chamfer...69
Circle center CC...71
Circular hole pattern...127
Circular path...71, 72, 73, 79, 80
Circular pocket
- finishing...116
- roughing...114

Circular stud finishing...117
Code number...213
Code numbers...213
Compatibility...2
Compensating workpiece misalignment...204
Constant contouring speed: M90...89

C

Contour approach...60
Contour departure...60
Conversational dialog...37
Conversational format...37
Coordinate transformation
- overview...137

Corner rounding...74
Cycle
- calling...95
- defining...94
- groups...94

Cylinder...181

D

Data interface
- pin Layout...226
- setting...213

Data transfer speed...213
Data transfer software...214
Datum selection...30
Datum setting
- with 3-D touch probe...205
  - circle center as datum...207
  - corner as datum...206
  - in any axis...206
- without a 3-D touch probe...19

Datum shift...138
- with datum tables...138

Display HELP file...218
Distance-to-go mode...193
DNC mode...199
Drilling...97, 98, 101
Dwell time...145

E

Ellipse...179
Error messages, 229
- output of...167

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/out files...33
- renaming files...32

File status...31
Full circle...71

G

Graphic simulation...189
Graphics
- Detail enlargement...188
- during programming...39
- during test run...186
- Views...186

H

Helical interpolation...81
Helix...81
HELP files
- running...218

HELP function...41
Hole patterns
- circular pattern...127
- linear...128
- overview...126

I

Interrupting machining...194

J

Jog positioning...17

K

Keyboard...4

M

M functions. *See* Miscellaneous functions

Machine axes, moving the

with incremental jog positioning...17

with the axis-direction keys...15

with the electronic handwheel...16

Machine parameters

for 3-D touch probes...222

for external data transfer...221

Machine-referenced coordinates: M91/M92...87

Machining time...190

Main axes...27

Measuring workpieces...208

Mid-program startup...197

Mirror image...140

Miscellaneous functions

entering...86

for contouring behavior...89

for coordinate data...87

for program run control...87

for rotary axes...92

MOD functions

changing...212

exiting...212

selecting...212

Multipass milling...132

N

Nesting...151

Non-controlled axes in the NC program...193

O

Open contours: M98...91

Operating modes...4

Optional program run stop...200

P

Parameter programming. *See* Q parameter programming

Parentheses calculation...173

Part families...161

Path contours

Cartesian coordinates...68

circular path around circle center...71

circular path with defined radius...72

circular path with tangential connection...73

straight line...69

polar coordinates...78

circular path around pole...79

circular path with tangential connection...80

straight line...79

Path functions

fundamentals...57

circles and circular arcs...58

pre-positioning...58

Pecking...97

Pin Layout...226

Plan view...187

Polar coordinates

fundamentals...28

setting the pole...28

Position display, selecting...216

Positioning

with manual data input...22

Positioning with MDI...5, 22

POSITIP mode...193

Probing cycles...202

Program

editing...38

opening...35

structure...34

Program call via cycle...145

P

Program management. *See* File management

Program name *See* File management: File name

Program Run

executing...192

interrupting...194

moving the machine axes during an interruption...195

resuming after an interruption...195, 196

start program at any block...197

Program Section Repeats

calling...150

operating sequence...149

programming notes...149

programming...150

Programming graphics...39

Q

Q parameters

checking...166

preassigned...176, 177

transferring values to PLC...172

Q-parameter programming

additional functions...167

formula, entering...173

if/then decisions...165

mathematics, basic operations...162

programming notes...160

trigonometry...164

R

Radius compensation...51

corners, machining...54

entering...53

inside corners..54

outside corners...54

R

Rapid traverse...44
Reaming...99
Rectangular pocket
 finishing...111
 roughing...110
Reference system...27
Returning to the contour...198
Rotary axis
 reducing the display...92
Rotation...141
Round slot milling...122
RS232-C/V.24 setup...213
Ruled surface...134

S

Scaling factor...142
Screen layout 3
Secondary axes...27
Slot milling
 with reciprocating plunge cut...120
Slot milling...120, 122
Small contour steps: M97..90
Software number...212
Sphere...183
Spindle orientation...146
Spindle speed
 changing...18
 entering...18
Status display
 additional...8
 general...7
Straight line...69, 79
Subprogram
 calling...149
 operating sequence...148
 programming notes...148
 programming...149
Switch-on...14
System data, reading...169
System information...212

T

Tapping
 rigid...106
 with a floating tap holder...105
Teach in...59
Technical specifications...227
Test run
 execution...191
 overview...190
 up to a certain block...191
TNC 310...2
TNCremo...214
Tool change...49
Tool compensation
 length...51
 radius...51
Tool data
 calling...49
 delta values...46
 entering into tables...47
 entering into the program...46
Tool length...45
Tool movements
 entering...59
 overview...68
 programming...37
Tool number...45
Tool radius...46
Tool table
 available input data...47
 editing functions...48, 50
 editing...47
 leaving...47
 selecting...47
Traverse range limits...217
Traversing the reference points...14
Trigonometry...164

U

Unit of measure, selecting...35
Unit of measure...216
Universal drilling...101
User parameters
 general...220
  for 3-D touch probes...222
  for electronic handwheels...225
  for external data transfer...221
  for machining and program run...224
  for TNC displays, TNC editor...222
 machine specific...216

V

View in 3 planes...187
Visual display unit...3

W

Workpiece positions
 absolute...29
 incremental...29
 relative...29

M	Effect of M function	Effective at block - start	end	page
M00	Stop program run/spindle STOP/coolant OFF		■	**87**
M01	Optional Program Run Interruption		■	**200**
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	**87**
M03	Spindle ON clockwise	■		
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	**87**
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	**87**
M08	Coolant ON	■		
M09	Coolant OFF		■	**87**
M13	Spindle ON clockwise/coolant ON	■		
M14	Spindle ON counterclockwise/Coolant ON	■		**87**
M30	Same function as M02		■	**87**
M89	Vacant miscellaneous function **or**	■		
	Cycle call, modally effective (depending on machine parameter)		■	**95**
M90	Only in lag mode: Constant contouring speed at corners		■	**89**
M91	Within the positioning block: Coordinates are referenced to machine datum		■	**87**
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		**87**
M93	Within the positioning block: Coordinates are referenced to the current tool position		■	
M94	Reduce display of rotary axis to value under 360°	■		**92**
M97	Machine small contour steps		■	**90**
M98	Machine open contours completely		■	**91**
M99	Blockwise cycle call		■	**95**

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
E-Mail: service@heidenhain.de
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (711) 952803-0
E-Mail: service.hsf@heidenhain.de

www.heidenhain.de

Ve 00
331 645-22 · 4/2003 · pdf · Subject to change without notice