



**HEIDENHAIN**

**Pilot**

**TNC 426B**  
**TNC 430**

**NC-Software**  
**280 476-xx**  
**280 477-xx**





8/2000

# The Pilot

... is your concise programming guide for the HEIDENHAIN TNC 426 and TNC 430 contouring controls. For more comprehensive information on programming and operating, refer to the TNC User's Manual. There you will find complete information on:

- Q-parameter programming
- The central tool file
- 3-D tool compensation
- Tool measurement

Certain symbols are used in the Pilot to denote specific types of information:

	Important note
	WARNING: danger for the user or the machine!
	The TNC and the machine tool must be prepared by the machine tool builder to perform these functions!
	Chapter in User's Manual where you will find more detailed information on the current topic.

The information in this Pilot applies to TNCs with the following software numbers:

Control	NC Software Number
TNC 426, TNC 430	280 476-xx
TNC 426*, TNC 430*	280 477-xx

\*) Export version

# Contents

Fundamentals .....	<b>4</b>
Contour Approach and Departure .....	<b>13</b>
Path Functions .....	<b>18</b>
FK Free Contour Programming .....	<b>25</b>
Subprograms and Program Section Repeats .....	<b>33</b>
Working with Cycles .....	<b>36</b>
Cycles for Machining Holes and Threads .....	<b>39</b>
Pockets, Studs, and Slots .....	<b>56</b>
Point Patterns .....	<b>65</b>
SL Cycles .....	<b>67</b>
Multipass Milling .....	<b>75</b>
Coordinate Transformation Cycles .....	<b>78</b>
Special Cycles .....	<b>85</b>
Digitizing 3-D Surfaces .....	<b>88</b>
Graphics and Status Displays .....	<b>94</b>
ISO Programming .....	<b>97</b>
Miscellaneous Functions M .....	<b>103</b>

# Fundamentals

## Programs/Files



See "Programming, File Management"

The TNC keeps its programs, tables and texts in files.  
A file designation consists of two components:

THREAD2.H

### File name

Maximum length:  
16 characters

### File type

see table at right

## Files in the TNC

## File type

### Programs

- in HEIDENHAIN format
- in ISO format

**.H**  
**.I**

### Tables for

- Tools
- Datums
- Pallets
- Cutting data
- Positions

**.T**  
**.D**  
**.P**  
**.CDT**  
**.PNT**

### Texts as

- ASCII files

**.A**

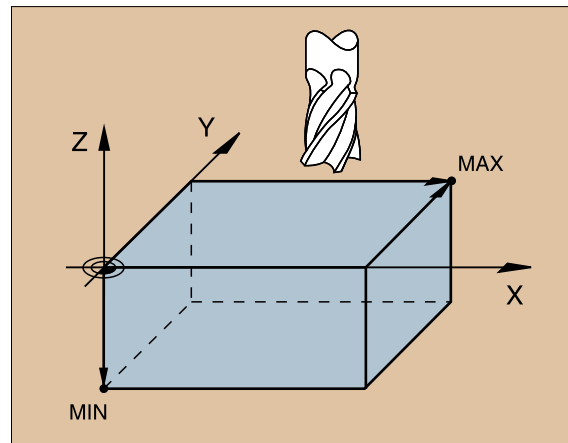
## Creating a New Part Program

PGM  
MGT

- ▶ Select the directory in which the program is stored
- ▶ Enter a new file name with file type
- ▶ Select unit of measure for dimensions (mm or inches)
- ▶ Define the blank form (BLK) for graphics:
  - ▶ Enter the spindle axis
  - ▶ Enter coordinates of the MIN point:  
the smallest X, Y and Z coordinates
  - ▶ Enter coordinates of the MAX point:  
the greatest X, Y and Z coordinates

**1 BLK FORM 0.1 Z X+0 Y+0 Z-50**

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



## Choosing the Screen Layout



See "Introduction, the TNC 426, TNC 430"



► Show soft keys for setting the screen layout

Mode of operation	Screen contents	
Manual operation	Positions	POSITION
Electronic handwheel	Positions at left	POSITION + STATUS
	Status at right	
Positioning with manual data input	Program	PGM
	Program at left	PGM + STATUS
	Status at right	
Program run, full sequence	Program	PGM
Program run, single block test run	Program at left	PGM + SECTS
	Program structure at right	
	Program at left	PGM + STATUS
	Status at right	
	Program at left	PGM + GRAPHICS
	Graphics at right	
	Graphics	GRAPHICS

Continued ►

Manual operation Programming and editing

ACTL.

+X

+6.278

+Y

+0.809

+Z

-95.962

+B

-2.887

+C

+357.479

DIST.

+X

+293.723

+Y

+24.191

+Z

+645.962

+B

+30002.773

+C

+29642.036

Basic rotation

+0.0000

M 5/9

S 0.034

T 0

Z S 150

F 0

0% S-IST 7:48

1% S-MOM LIMIT 1

M

S

F

TOUCH PROBE

DATUM SET

INCRE-  
MENT  
OFF / ON

3D ROT

TOOL TABLE

▲ Positions at left, status at right  
▼ Program at left, graphics at right

Manual operation Programming and editing

0 BEGIN PGM 3516 MM

1 BLK FORM 0.1 Z X-90 Y-90 Z-40

2 BLK FORM 0.2 X+90 Y+90 Z+0

3 TOOL CALL 1 Z S1400

4 L Z+50 R0 F MAX

5 CALL LBL 1

6 L Z+100 R0 F MAX M2

7 LBL 1

8 L X+0 Y+80 RL F250

9 FPOL X+0 Y+0

10 FC DR- R80 CCK+0 CCY+0

11 FCT DR- R7.5

12 FCT DR+ R90 CCK+69.282 CCY-40

13 FSELECT 2 ;

14 FCT DR+ R10 PDX+0 PDY+0 D20

BEGIN

END

PAGE

PAGE

FIND

START

START SINGLE

RESET + START

Mode of operation	Screen contents
Programming and editing	<div>Program</div> <div>PGM</div>
	<div>Program at left</div> <div>Program structure at right</div> <div>PGM + SECTS</div>
	<div>Program at left</div> <div>Programming graphics at right</div> <div>PGM + GRAPHICS</div>

Manual operation	Programming and editing					
1 BLK FORM 0.1 Z X+0 Y+0 Z-40 2 BLK FORM 0.2 X+100 Y+100 Z+0 3 * - Make hole pattern ID 27943KL1 4 TOOL CALL 1 Z S4500 5 CYCL DEF 262 THREAD MILLING 0335+10 \$NOMINAL DIAMETER 0239+1.5 \$THREAD PITCH 0201+18 \$DEPTH OF THREAD 0355+0 \$THREADS PER STEP 0253+750 \$F PRE-POSITIONING 0351+1 \$CLIMB OR UP-CUT 0200+2 \$SET-UP CLEARANCE 0203+0 \$SURFACE COORDINATE 0204+50 \$2ND SET-UP CLEARANCE 0207+500 \$FEED RATE FOR MILLING	BEGIN PGM 1GB - Make hole pattern ID 27943KL1 - Parameter definition - Make pocket - Rough out - Finishing - Make hole pattern - Center drill - Pecking - Tapping END PGM 1GB					
BEGIN ↑	END ↓	PAGE ↑	PAGE ↓	FIND		CHANGE WINDOW ↕

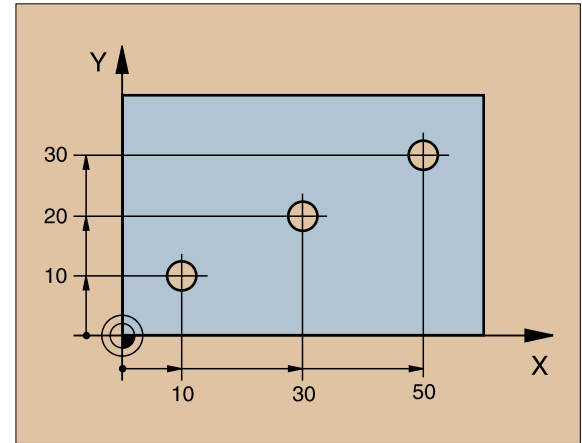
▲ Program at left, program structure at right

## Absolute Cartesian Coordinates

The dimensions are measured from the current datum.  
The tool moves **to** the absolute coordinates.

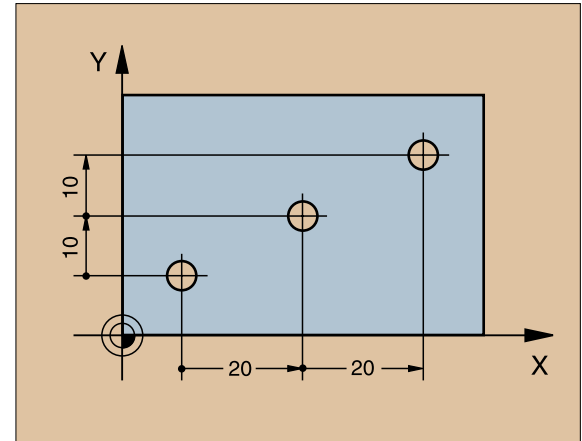
### Programmable axes in an NC block

Linear motion: 5 axes  
Circular motion: 2 linear axes in a plane or  
3 linear axes with cycle 19 WORKING PLANE



## Incremental Cartesian Coordinates

The dimensions are measured from the last programmed position of the tool.  
The tool moves **by** the incremental coordinates.



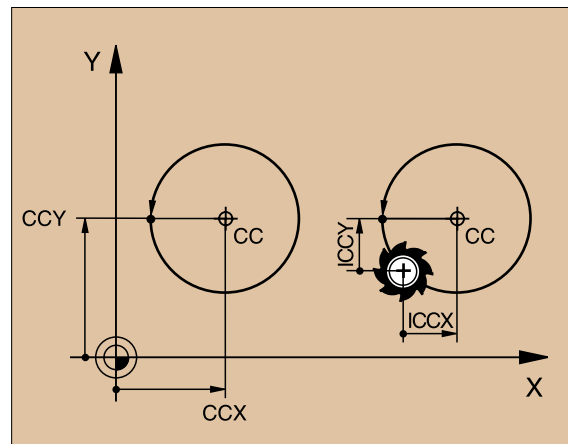
## Circle Center and Pole: CC

The circle center (CC) must be entered to program circular tool movements with the path function C (see page 21). CC is also needed to define the pole for polar coordinates.

CC is entered in Cartesian coordinates\*.

An absolutely defined circle center or pole is always measured from the workpiece datum.

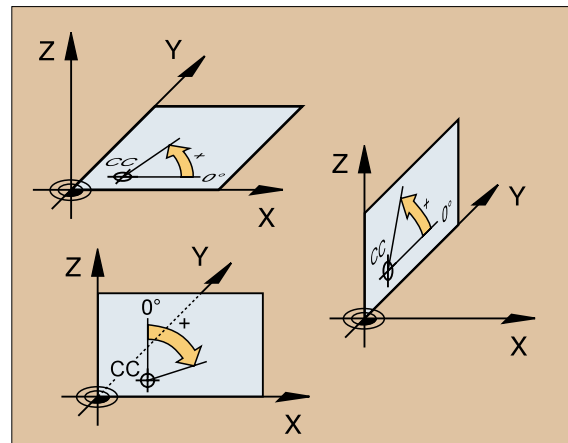
An incrementally defined circle center or pole is always measured from the last programmed position of the workpiece.



## Angle Reference Axis

Angles – such as a polar coordinate angle PA or an angle of rotation ROT – are measured from the angle reference axis.

Working plane	Ref. axis and 0° direction
X/Y	X
Y/Z	Y
Z/X	Z



\*Circle center in polar coordinates: See FK programming

## Polar Coordinates

Dimensions in polar coordinates are referenced to the pole (CC).

A position in the working plane is defined by

- Polar coordinate radius PR = Distance of the position from the pole
- Polar coordinate angle PA = Angle from the angle reference axis to the straight line CC – PR

### Incremental dimensions

Incremental dimensions in polar coordinates are measured from the last programmed position.

### Programming polar coordinates



► Select the path function



► Press the P key

► Answer the dialog prompts

## Defining Tools

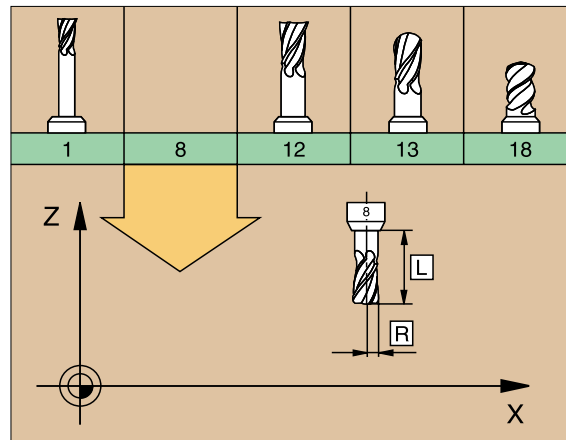
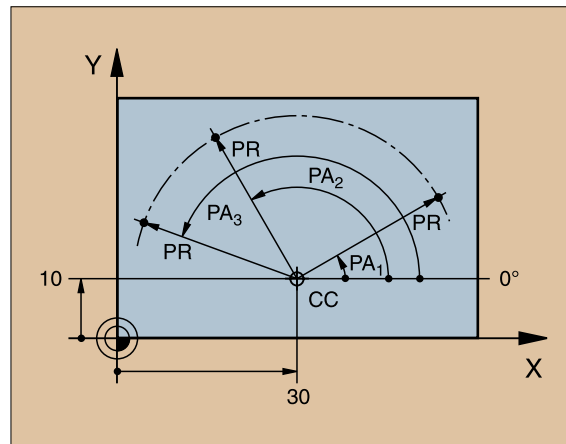
### Tool data

Every tool is designated by a tool number between 1 and 254 or, if you are using tool tables, by a tool name.

### Entering tool data

You can enter the tool data (length L and radius R)

- in a tool table (centrally, Program TOOL.T)
- or
- within the part program in TOOL DEF blocks (locally)





**TOOL  
DEF**

- ▶ Tool number
- ▶ Tool length L
- ▶ Tool radius R

- ▶ Program the tool length as its difference  $\Delta L$  to the zero tool:  
 $\Delta L > 0$ : The tool is longer than the zero tool  
 $\Delta L < 0$ : The tool is shorter than the zero tool
- ▶ With a tool presetter you can measure the actual tool length, then program that length.

**Calling the tool data**

**TOOL  
CALL**

- ▶ Tool number or name
- ▶ Working spindle axis: tool axis
- ▶ Spindle speed S
- ▶ Feed rate
- ▶ Tool length override DL (e.g. to compensate wear)
- ▶ Tool radius override DR (e.g. to compensate wear)

3 TOOL DEF 6 L+7.5 R+3

4 TOOL CALL 6 Z S2000 F650 DL+1 DR+0.5

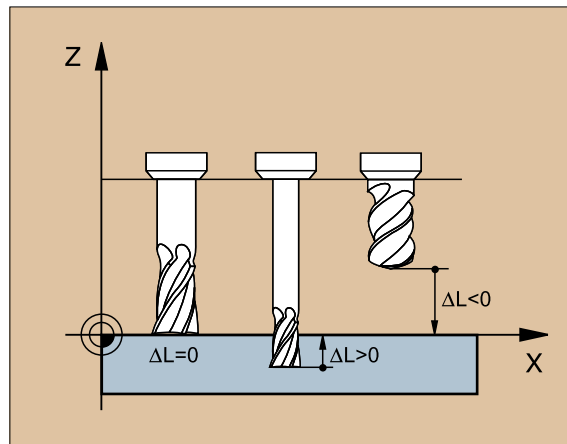
5 L Z+100 R0 FMAX

6 L X-10 Y-10 R0 FMAX M6

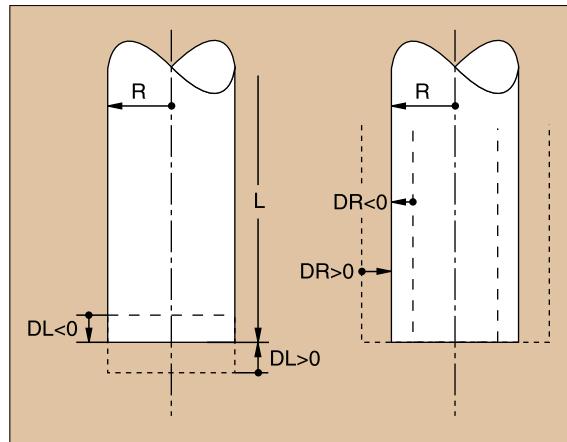
**Tool change**



- Beware of tool collision when moving to the tool change position!
- The direction of spindle rotation is defined by M function:  
M3: Clockwise  
M4: Counterclockwise
- The maximum permissible override for tool radius or length is  $\pm 99.999$  mm!



▼ Oversizes on an end mill



## Tool Compensation

The TNC compensates the length  $L$  and radius  $R$  of the tool during machining.

### Length compensation

**Beginning** of effect:

- Tool movement in the spindle axis

**End** of effect:

- Tool exchange or tool with the length  $L=0$

### Radius compensation

**Beginning** of effect:

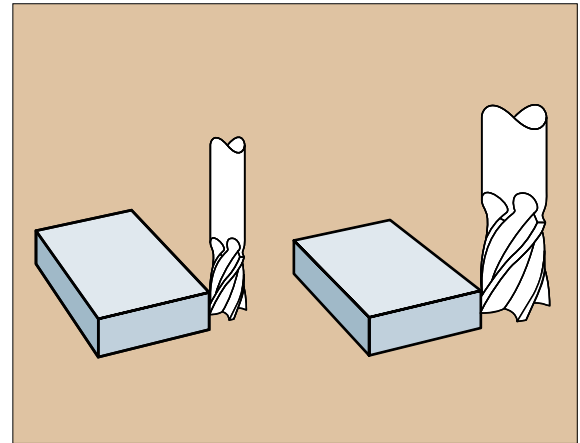
- Tool movement in the working plane with  $RR$  or  $RL$

**End** of effect:

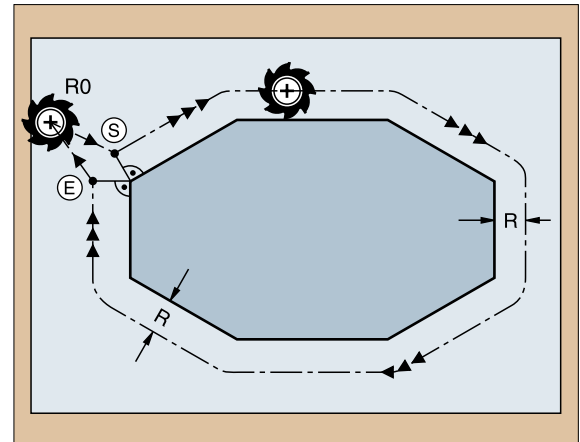
- Execution of a positioning block with  $R0$

Working **without radius compensation** (e.g. drilling):

- Tool movement with  $R0$



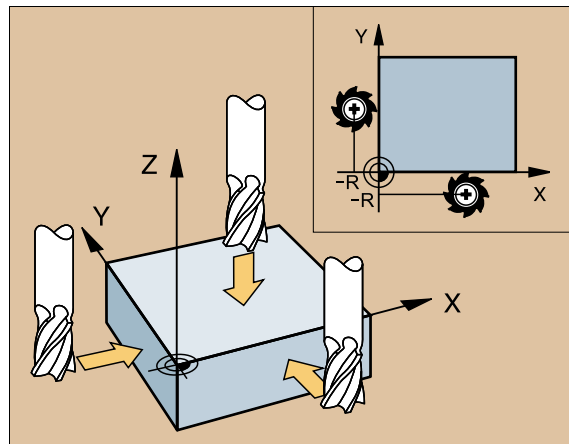
▼ (S) = Start; (E) = End



## Datum Setting without a 3-D Touch Probe

During datum setting you set the TNC display to the coordinates of a known position on the workpiece:

- ▶ Insert a zero tool with known radius
- ▶ Select the manual operation or electronic handwheel mode
- ▶ Touch the reference surface in the tool axis with the tool and enter its length
- ▶ Touch the reference surface in the working plane with the tool and enter the position of the tool center

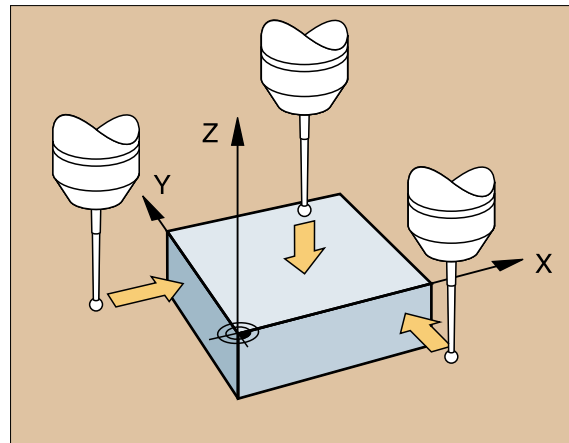


## Setup and Measurement with 3-D Touch Probes

A HEIDENHAIN 3-D touch probe enables you to setup the machine very quickly, simply and precisely.

Besides the probing functions for workpiece setup on the Manual and Electronic Handwheel modes, the Program Run modes provide a series of measuring cycles (see also the User's Manual for Touch Probe Cycles):

- Measuring cycles for measuring and compensating workpiece misalignment
- Measuring cycles for automatic datum setting
- Measuring cycles for automatic workpiece measurement with tolerance checking and automatic tool compensation



# Contour Approach and Departure

## Starting point $P_S$

$P_S$  lies outside of the contour and must be approached without radius compensation.

## Auxiliary point $P_H$

$P_H$  lies outside of the contour and is calculated by the TNC.



The tool moves from the starting point  $P_S$  to the auxiliary point  $P_H$  at the feed rate last programmed feed rate!

## First contour point $P_A$ and last contour point $P_E$

The first contour point  $P_A$  is programmed in the APPR (approach) block. The last contour point is programmed as usual.

## End point $P_N$

$P_N$  lies outside of the contour and results from the DEP (departure) block.  $P_N$  is automatically approached with R0.

## Path Functions for Approach and Departure



► Press the soft key with the desired path function:



Straight line with tangential connection



Straight line perpendicular to the contour point



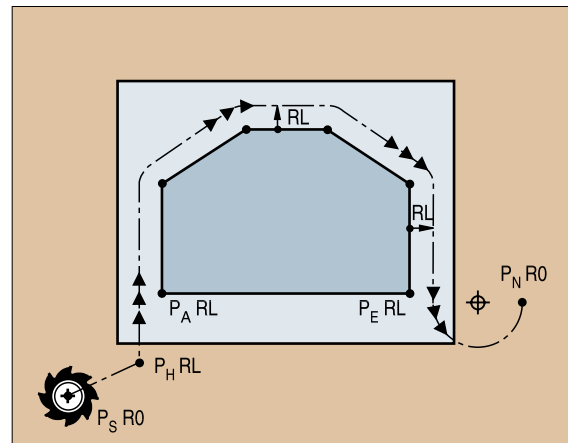
Circular arc with tangential connection



Straight line segment tangentially connected to the contour through an arc



- Program a radius compensation in the APPR block!
- DEP blocks set the radius compensation to 0!



## Approaching on a Straight Line with Tangential Connection

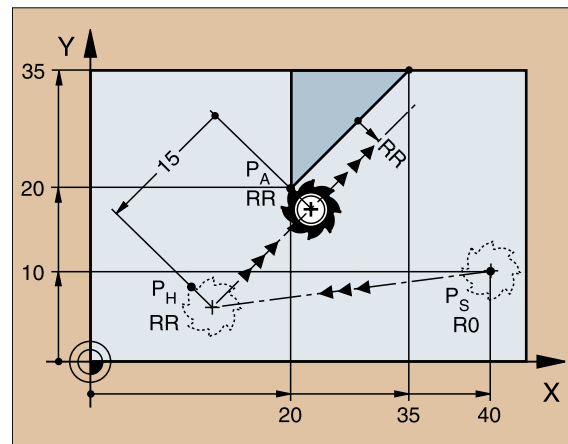


- Coordinates for the first contour point  $P_A$
- Distance Len (length) from  $P_H$  to  $P_A$   
Enter a length Len > 0
- Tool radius compensation RR/RL

7 L X+40 Y+10 R0 FMAX M3

8 APPR LT X+20 Y+20 LEN 15 RR F100

9 L X+35 Y+35



## Approaching on a Straight Line Perpendicular to the First Contour Element

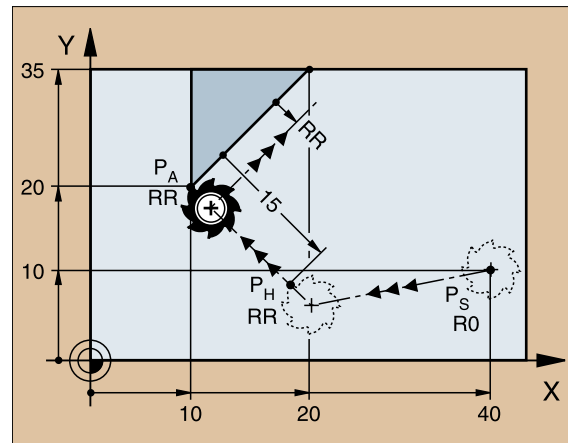


- Coordinates for the first contour point  $P_A$
- Distance Len (length) from  $P_H$  to  $P_A$   
Enter a length Len > 0
- Tool radius compensation RR/RL

7 L X+40 Y+10 R0 FMAX M3

8 APPR LN X+10 Y+20 LEN 15 RR F100

9 L X+20 Y+35



## Approaching Tangentially on an Arc

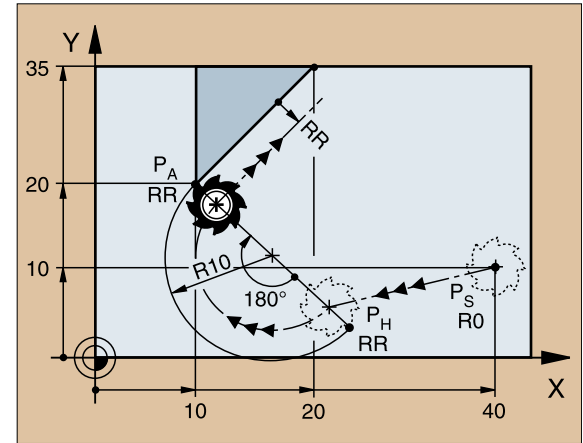


- Coordinates for the first contour point  $P_A$
- Radius  $R$   
Enter a radius  $R > 0$
- Circle center angle (CCA)  
Enter a CCA  $> 0$
- Tool radius compensation  $RR/RL$

```
7 L X+40 Y+10 R0 FMAX M3
```

```
8 APPR CT X+10 Y+20 CCA 180 R10 RR F100
```

```
9 L X+20 Y+35
```



## Approaching Tangentially on an Arc and a Straight Line

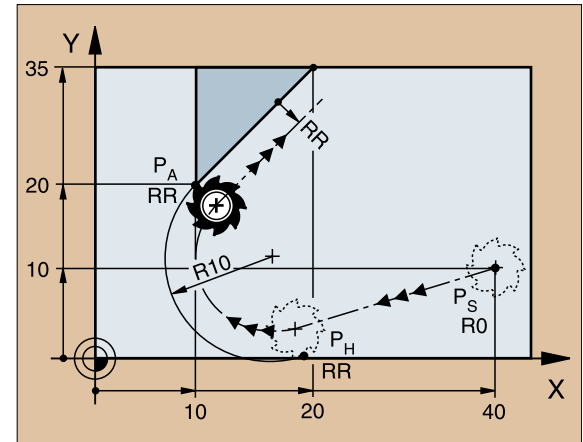


- Coordinates for the first contour point  $P_A$
- Radius  $R$   
Enter a radius  $R > 0$
- Tool radius compensation  $RR/RL$

```
7 L X+40 Y+10 R0 FMAX M3
```

```
8 APPR LCT X+10 Y+20 R10 RR F100
```

```
9 L X+20 Y+35
```



## Departing Tangentially on a Straight Line

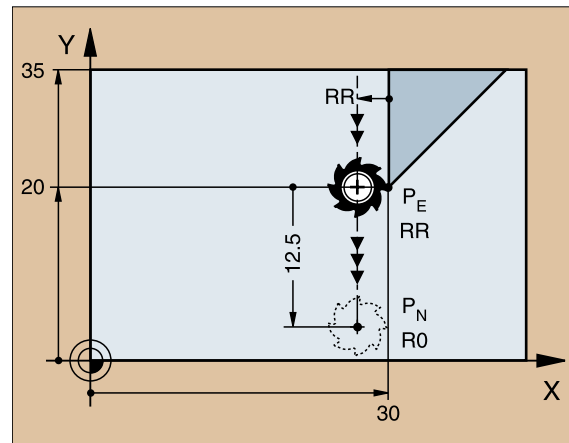


► Distance Len (length) from  $P_E$  to  $P_N$   
Enter a length Len > 0

23 L X+30 Y+35 RR F100

24 L Y+20 RR F100

25 DEP LT LEN 12.5 F100 M2



## Departing on a Straight Line Perpendicular to the Last Contour Element

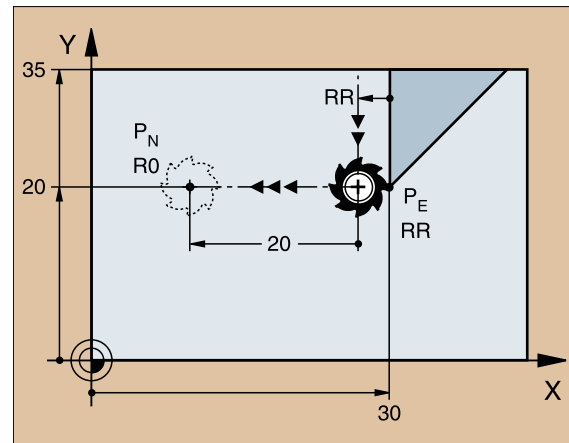


► Distance Len (length) from  $P_E$  to  $P_N$   
Enter a length Len > 0

23 L X+30 Y+35 RR F100

24 L Y+20 RR F100

25 DEP LN LEN+20 F100 M2

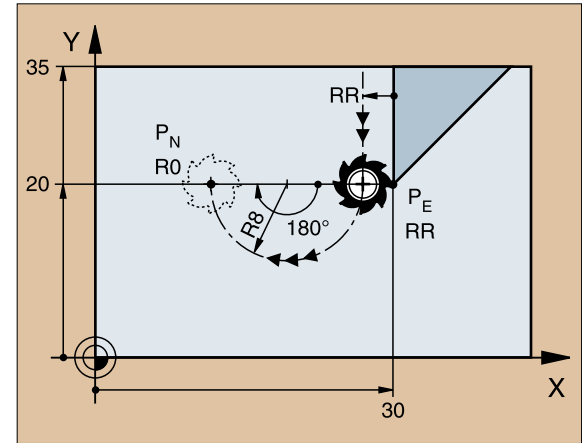


## Departing Tangentially on an Arc



- ▶ Radius R  
Enter a radius  $R > 0$
- ▶ Circle center angle (CCA)

```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F10
25 DEP CT CCA 180 R+8 F100 M2
```

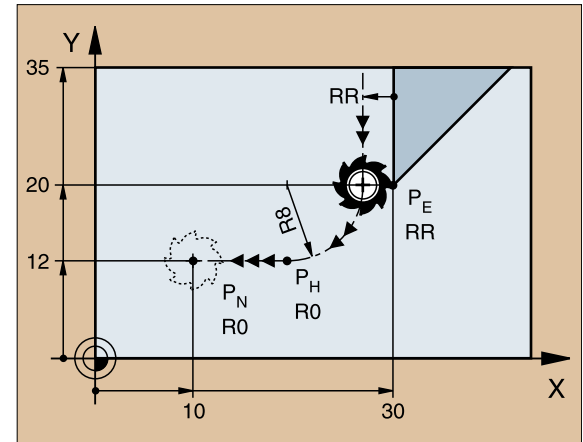


## Departing on an Arc Tangentially Connecting the Contour and a Straight Line



- ▶ Coordinates of the end point  $P_N$
- ▶ Radius R  
Enter a radius  $R > 0$

```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F100
25 DEP LCT X+10 Y+12 R8 F100 M2
```





# Path Functions for Positioning Blocks



See „Programming: Programming contours“.

## Programming the Direction of Traverse

Regardless of whether the tool or the workpiece is actually moving, you always program as if the tool is moving and the workpiece is stationary.

## Entering the Target Positions

Target positions can be entered in Cartesian or polar coordinates – either as absolute or incremental values, or with both absolute and incremental values in the same block.

## Entries in the Positioning Block

A complete positioning block contains the following data:

- Path function
- Coordinates of the contour element end points (target position)
- Radius compensation RR/RL/R0
- Feed rate F
- Miscellaneous function M



Before you execute a part program, always pre-position the tool to prevent the possibility of damaging the tool or workpiece!

## Path functions

### Straight line



Page 19

### Chamfer between two straight lines



Page 20

### Corner rounding



Page 20

### Circle center or pole for polar coordinates



Page 21

### Circular path around the circle center CC



Page 21

### Circular path with known radius



Page 22

### Circular path with tangential connection to previous contour



Page 23

### FK Free Contour Programming



Page 25

## Straight Line



- ▶ Coordinates of the straight line end point
- ▶ Tool radius compensation RR/RL/R0
- ▶ Feed rate F
- ▶ Miscellaneous function M

With Cartesian coordinates:

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

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

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

With polar coordinates:

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

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

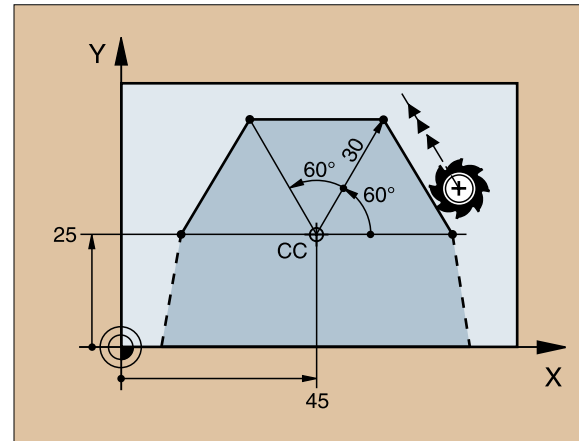
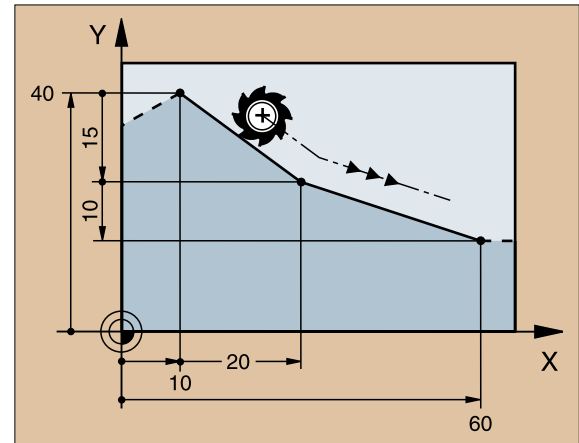
```
14 LP PA+60
```

```
15 LP IPA+60
```

```
16 LP PA+180
```



- You must first define the pole CC before you can program polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!



## Inserting a Chamfer Between Two Straight Lines



- ▶ Chamfer side length
- ▶ Feed rate F for the chamfer

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

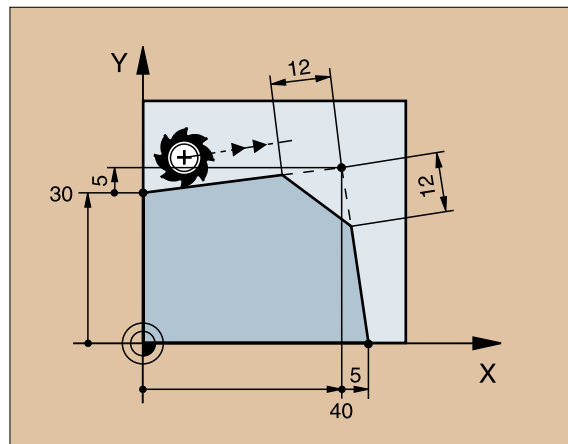
```
8 L X+40 IY+5
```

```
9 CHF 12 F250
```

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



- You cannot start a contour with a CHF block!
- The radius compensation before and after the CHF block must be the same!
- An inside chamfer must be large enough to accommodate the current tool!



## Corner Rounding

The beginning and end of the arc extend tangentially from the previous and subsequent contour elements.



- ▶ Radius R of the circular arc
- ▶ Feed rate F for corner rounding

```
5 L X+10 Y+40 RL F300 M3
```

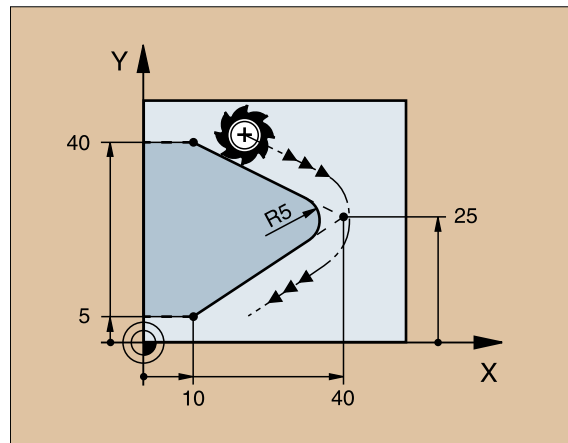
```
6 L X+40 Y+25
```

```
7 RND R5 F100
```

```
8 L X+10 Y+5
```



- An inside arc must be large enough to accommodate the current tool!



## Circular Path Around the Circle Center CC



► Coordinates of the circle center CC



► Coordinates of the arc end point

► Direction of rotation DR

C and CP enable you to program a complete circle in one block.

With cartesian coordinates:

**5 CC X+25 Y+25**

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

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

With polar coordinates:

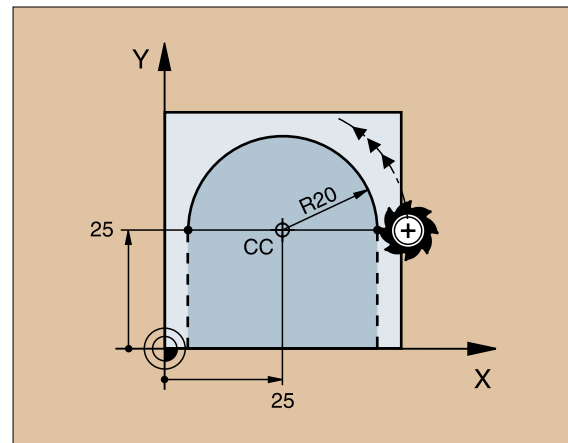
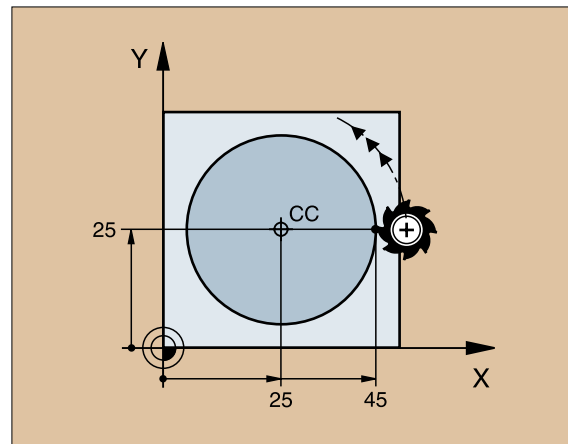
**18 CC X+25 Y+25**

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

**20 CP PA+180 DR+**



- Define the pole CC before programming polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!
- The arc end point can be defined only with the polar coordinate angle (PA)!



## Circular Path with Known Radius (CR)



- Coordinates of the arc end point
- Radius R
- If the central angle  $ZW > 180$ , R is negative.
- If the central angle  $ZW < 180$ , R is positive.
- Direction of rotation DR

10 L X+40 Y+40 RL F200 M3 Arc starting point

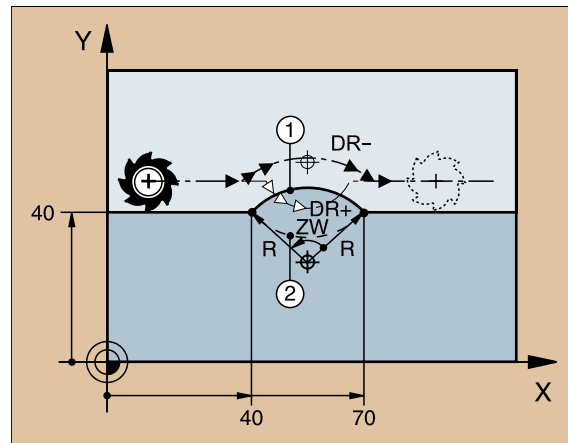
11 CR X+70 Y+40 R+20 DR- Arc ① or

11 CR X+70 Y+40 R+20 DR+ Arc ②

10 L X+40 Y+40 RL F200 M3 Arc starting point

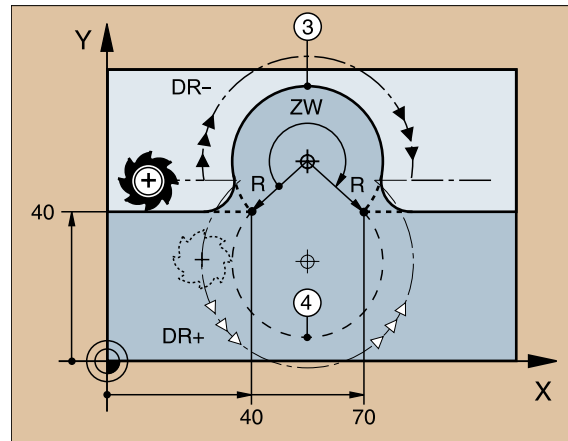
11 CR X+70 Y+40 R-20 DR- Arc ③ or

11 CR X+70 Y+40 R-20 DR+ Arc ④



▲ Arcs ① and ②

▼ Arcs ③ and ④



## Circular Path CT with Tangential Connection



- Coordinates of the arc end point
- Radius compensation RR/RL/R0
- Feed rate F
- Miscellaneous function M

With cartesian coordinates:

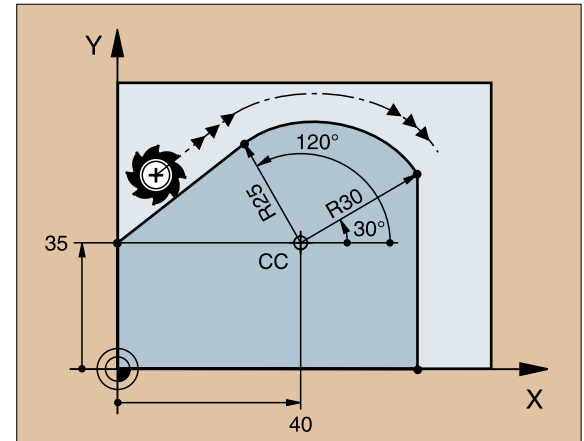
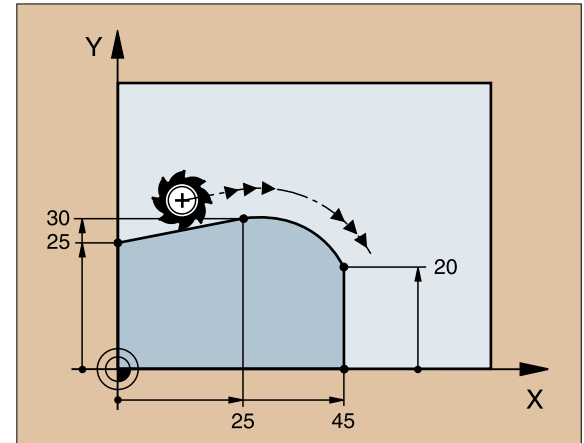
```
5 L X+0 Y+25 RL F250 M3
6 L X+25 Y+30
7 CT X+45 Y+20
8 L Y+0
```

With polar coordinates:

```
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
```



- Define the pole CC before programming polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!



## Helix (Only in Polar Coordinates)

### Calculations (upward milling direction)

Path revolutions:  $n$  = Thread revolutions + overrun at start and end of thread

Total height:  $h$  = Pitch  $P$  x path revolutions  $n$

Incr. coord. angle:  $IPA$  = Path revolutions  $n$  x  $360^\circ$

Start angle:  $PA$  = Angle at start of thread + angle for overrun

Start coordinate:  $Z$  = Pitch  $P$  x (thread revolutions + thread overrun at start of thread)

### Shape of helix

Internal thread	Work direction	Direction	Radius comp.
Right-hand	Z+	DR+	RL
Left-hand	Z+	DR-	RR
Right-hand	Z-	DR-	RR
Left-hand	Z-	DR+	RL

### External thread

Right-hand	Z+	DR+	RR
Left-hand	Z+	DR-	RL
Right-hand	Z-	DR-	RL
Left-hand	Z-	DR+	RR

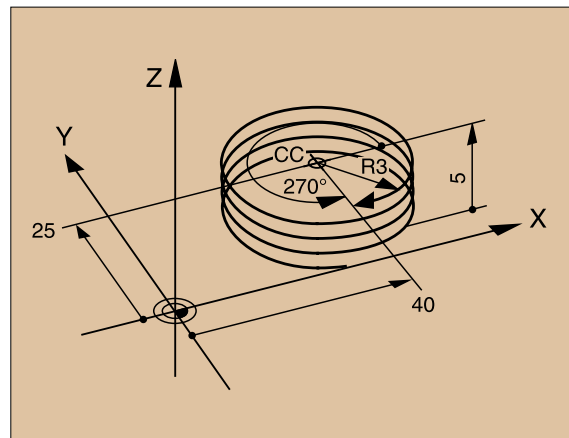
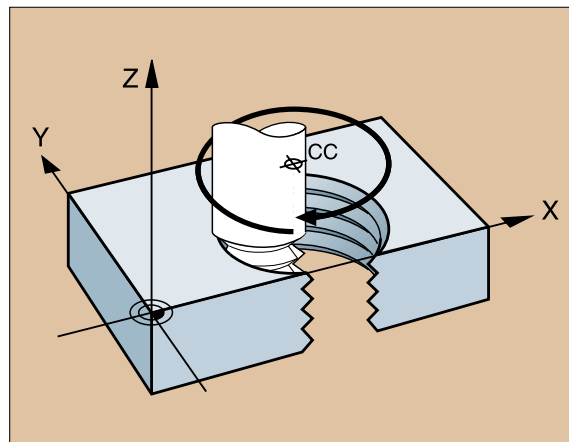
### M6 x 1 mm thread with 5 revolutions:

12 CC X+40 Y+25

13 L Z+0 F100 M3

14 LP PR+3 PA+270 RL

15 CP IPA-1800 IZ+5 DR- RL F50



# FK Free Contour Programming



See “Programming Tool Movements – FK Free Contour Programming”

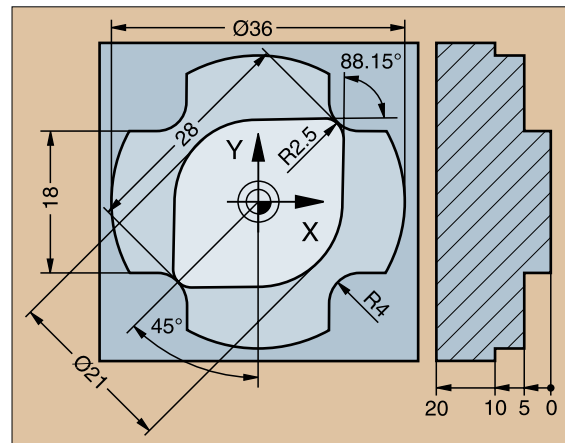
If the end point coordinates are not given in the workpiece drawing or if the drawing gives dimensions that cannot be entered with the gray path function keys, you can still program the part by using the “FK Free Contour Programming.”

## Possible data on a contour element:

- Known coordinates of the end point
- Auxiliary points on the contour element
- Auxiliary points near the contour element
- A reference to another contour element
- Directional data (angle) / position data
- Data regarding the course of the contour

## To use FK programming properly:

- All contour elements must lie in the working plane.
- Enter all available data on each contour element.
- If a program contains both FK and conventional blocks, the FK contour must be fully defined before you can return to conventional programming.



▲ These dimensions can be programmed with FK



## Working with the Interactive Graphics



Select the PGM+GRAPHICS screen layout!

The interactive graphics show the contour as you are programming it. If the data you enter can apply to more than one solution, the following soft keys will appear:

SHOW SOLUTION	To show the possible solutions
SELECT SOLUTION	To enter the displayed solution in the part program
END SELECT	To enter data for subsequent contour elements
START SINGLE <input type="checkbox"/>	To graphically display the next programmed block

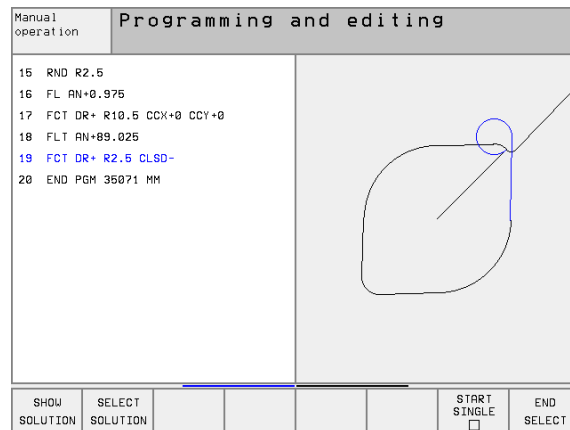
### Standard colors of the interactive graphics

Fully defined contour element

The displayed element is one of a limited number of possible solutions

The element is one of an infinite number of solutions

Contour element from a subprogram



## Initiating the FK Dialog



Initiate the FK dialog

Straight Circular



Contour element without tangential connection



Contour element with tangential connection



Pole for FK programming

## End Point Coordinates X, Y or PA, PR



Cartesian coordinates X and Y



Polar coordinates referenced to FPOL

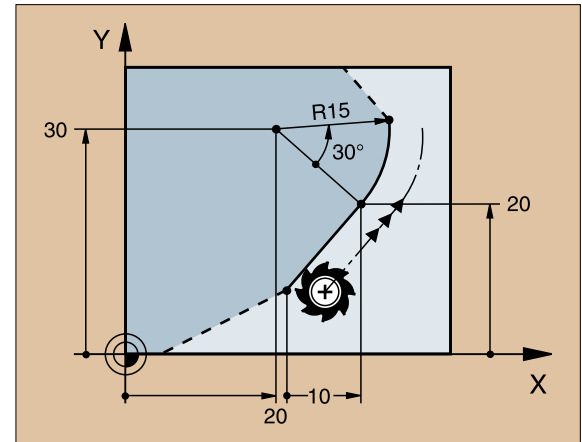


Incremental input

7 FPOL X+20 Y+30

8 FL IX+10 Y+20 RR F100

9 FCT PR+15 IPA+30 DR+ R15



**I** Incremental input

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

## Direction and Length of the Contour Element

### Data on a straight line



Gradient angle of a straight line



Length of a straight line

### Data on a circular path



Gradient angle of the entry tangent



Length of an arc chord

```
27 FLT X+25 LEN 12.5 AN+35 RL F200
```

```
28 FC DR+ R6 LEN 10 AN-45
```

```
29 FCT DR- R15 LEN 15
```

### Identifying a closed contour



Beginning: CLSD+

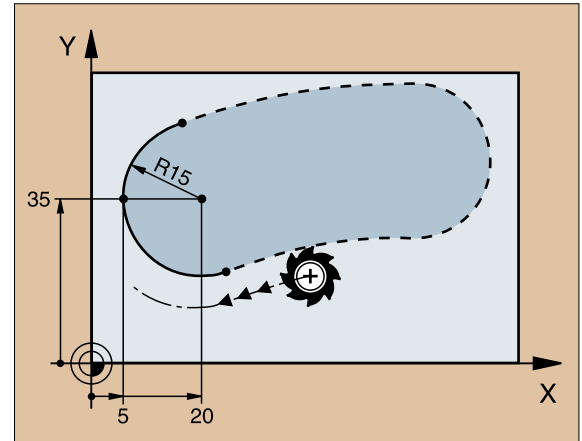
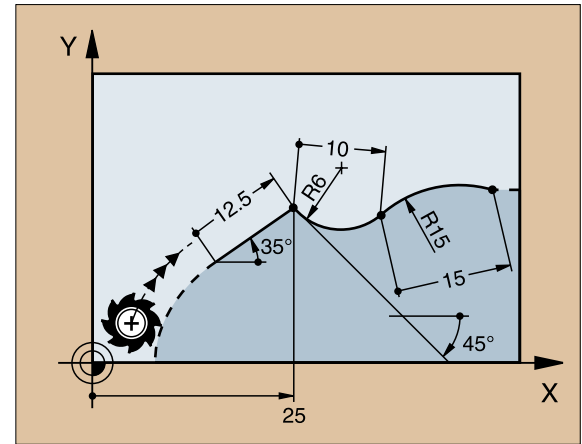
End: CLSD-

```
12 L X+5 Y+35 RL F500 M3
```

```
13 FC DR- R15 CLSD+ CCX+20 CCY+35
```

```
...
```

```
17 FCT DR- R+15 CLSD-
```



## Values Relative to Block N: Entering Coordinates

RX[N]

RY[N]

Cartesian coordinates relative to block N

RPR[N]

RPA[N]

Polar coordinates relative to block N



- Relative data must be entered incrementally!
- CC can also be programmed in relative values!

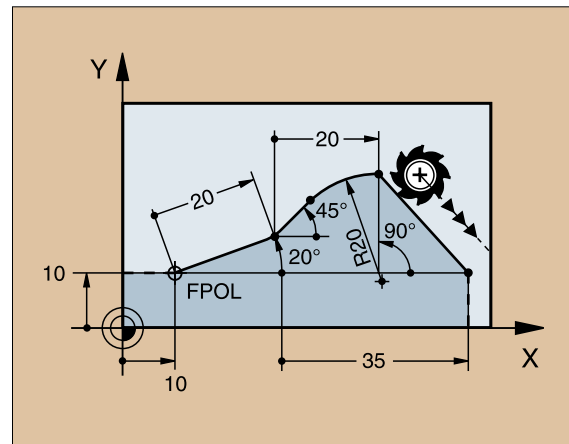
12 FPOL X+10 Y+10

13 FL PR+20 PA+20

14 FL AN+45

15 FCT IX+20 DR- R20 CCA+90 RX 13

16 FL IPR+35 PA+0 RPR 13



## Values Relative to Block N: Direction and Distance of the Contour Element

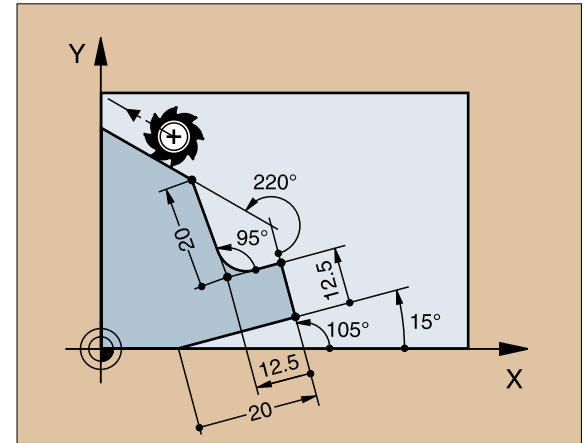
- RAN  $\angle$  Gradient angle
- PAR  $\angle$  Parallel to a straight contour element  
Parallel to the entry tangent of an arc
- DP  $\angle$  Distance from a parallel element



Always enter relative values incrementally!

```

17 FL LEN 20 AN+15
18 FL AN+105
19 FL LEN 12.5 PAR 17 DP 12.5
20 FSELECT 2
21 FL LEN 20 IAN+95
22 FL IAN+220 RAN 18
    
```



## Values Relative to Block N: Circle Center CC

RCCX[N]

RCCY[N]

Cartesian coordinates of a circle center relative to block N

RCCPR[N]

RCCPPR[N]

Polar coordinates of the circle center relative to block N



Always enter relative data as incremental values!

12 FL X+10 Y+10 RL

13 FL ...

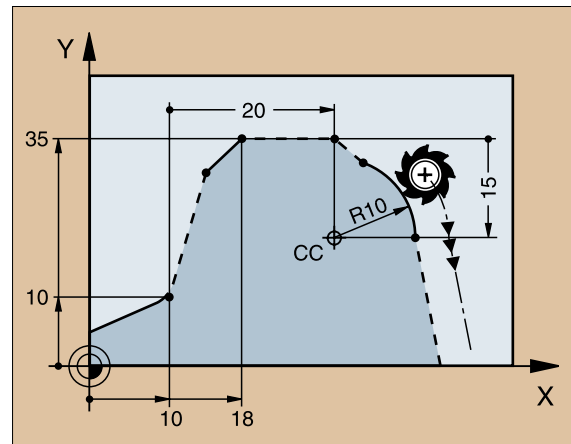
14 FL X+18 Y+35

15 FL ...

16 FL ...

17 FC DR- R10 CCA+0 ICCX+20 ICCY-15

RCCX12 RCCY14



# 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 needed.

## Working with Subprograms

- ① The main program runs up to the subprogram call CALL LBL1.
- ② The subprogram—labeled with LBL1—runs through to its end LBL0.
- ③ The main program resumes.

It's good practice to place subprograms after the main program end (M2).



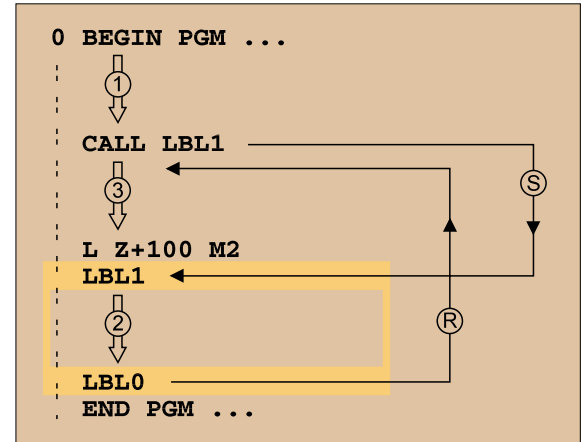
- Answer the dialog prompt REP with the NOENT key!
- You cannot call LBL0!

## Working with Program Section Repeats

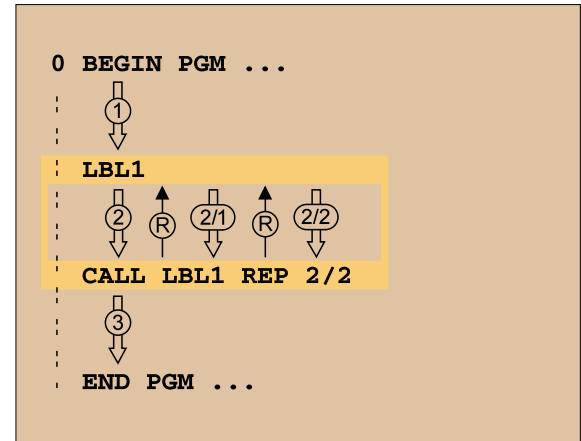
- ① The main program runs up to the call for a section repeat CALL LBL1 REP2/2.
- ② The program section between LBL1 and CALL LBL1 REP2/2 is repeated the number of times indicated with REP.
- ③ After the last repetition the main program resumes.



Altogether, the program section is run once more than the number of programmed repeats!



◆ S = Jump; R = Return jump



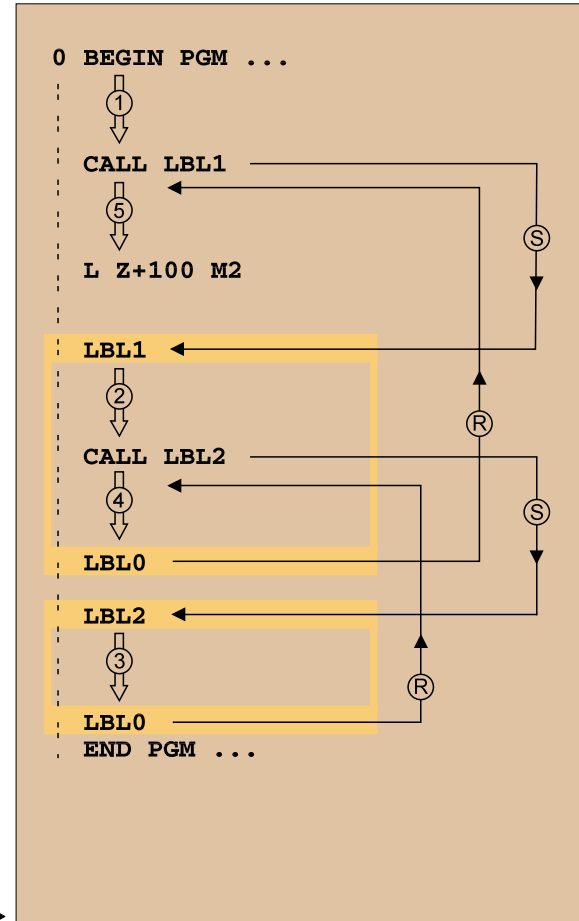


## Subprogram Nesting: A Subprogram within a Subprogram

- ① The main program runs up to the first subprogram call CALL LBL1.
- ② Subprogram 1 runs up to the second subprogram call CALL LBL2.
- ③ Subprogram 2 runs to its end.
- ④ Subprogram 1 resumes and runs to its end.
- ⑤ The main program resumes.



- A subprogram cannot call itself!
- Subprograms can be nested up to a maximum depth of 8 levels!

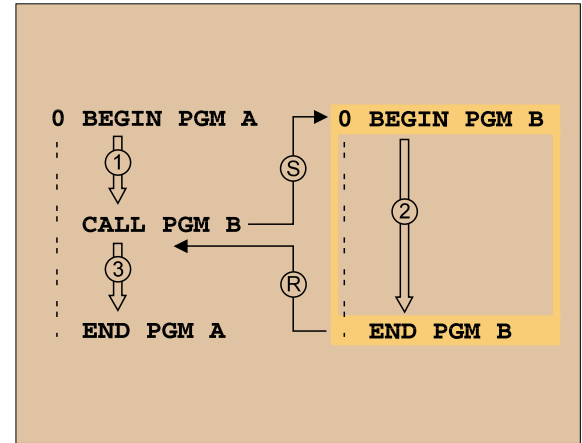


## Any Program as a Subprogram

- ① The calling program A runs up to the program call CALL PGM B.
- ② The called program B runs through to its end.
- ③ The calling program A resumes.



The **called** program must not end with M2 or M30!



▲ (S) = Jump; (R) = Return jump

# Working with Cycles

Certain frequently needed machining sequences are stored in the TNC as cycles. Coordinate transformations and some special functions are also available as cycles.



- In a cycle, positioning data entered in the tool axis are always incremental, even without the I key!
- The algebraic sign of the cycle parameter depth determines the working direction!

Example

```
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP 2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG 10
...
```

Feed rates are entered in mm/min, the dwell time in seconds.

## Defining cycles

**CYCL  
DEF**

► Select the Cycle Overview:

DRILLING

► Select the cycle group



► Select the cycle

## Cycles for Machining Holes and Threads

1	PECKING	Page 39
200	DRILLING	Page 40
201	REAMING	Page 41
202	BORING	Page 42
203	UNIVERSAL DRILLING	Page 43
204	COUNTERBORE BACK	Page 44
205	UNIVERSAL PECKING	Page 45
208	BORE MILLING	Page 46
2	TAPPING	Page 47
206	TAPPING NEW	Page 48
17	RIGID TAPPING	Page 48
207	RIGID TAPPING NEW	Page 49
18	THREAD CUTTING	Page 49
209	TAPPING W/ CHIP BRKG	Page 50
262	THREAD MILLING	Page 51
263	THREAD MLLNG/CNTSNKG	Page 52
264	THREAD DRILLNG/MLLNG	Page 53
265	HEL. THREAD DRLG/MLG	Page 54
267	OUTSIDE THREAD MLLNG	Page 55

Continued on next page ►

**Pockets, Studs, and Slots**

4	POCKET MILLING	Page 56
212	POCKET FINISHING	Page 57
213	STUD FINISHING	Page 58
5	CIRCULAR POCKET MILLING	Page 59
214	CIRCULAR POCKET FINISHING	Page 60
215	CIRCULAR STUD FINISHING	Page 61
3	SLOT MILLING	Page 62
210	SLOT WITH RECIP. PLUNGE	Page 63
211	CIRCULAR SLOT	Page 64

**Point Patterns**

220	CIRCULAR PATTERN	Page 65
221	LINEAR PATTERN	Page 66

**SL Cycles**

14	CONTOUR GEOMETRY	Page 67
20	CONTOUR DATA	Page 68
21	PILOT DRILLING	Page 69
22	ROUGH-OUT	Page 69
23	FLOOR FINISHING	Page 70
24	SIDE FINISHING	Page 70
25	CONTOUR TRAIN	Page 71
27	CYLINDER SURFACE	Page 72
28	CYLINDER SURFACE SLOT	Page 73

**Multipass Milling**

30	RUN DIGITIZED DATA	Page 74
230	MULTIPASS MILLING	Page 75
231	RULED SURFACE	Page 76

**Cycles for Coordinate Transformations**

7	DATUM SHIFT	Page 78
247	DATUM SETTING	Page 79
8	MIRROR IMAGE	Page 80
10	ROTATION	Page 81
19	WORKING PLANE	Page 82
11	SCALING FACTOR	Page 83
26	AXIS-SPECIFIC SCALING	Page 84

**Spezial Cycles**

9	DWELL TIME	Page 85
12	PGM CALL	Page 85
13	ORIENTED SPINDLE STOP	Page 86
32	TOLERANCE	Page 87

## Graphic Support During Cycle Programming

As you create a program, the TNC provides you with graphic illustrations of the input parameters.

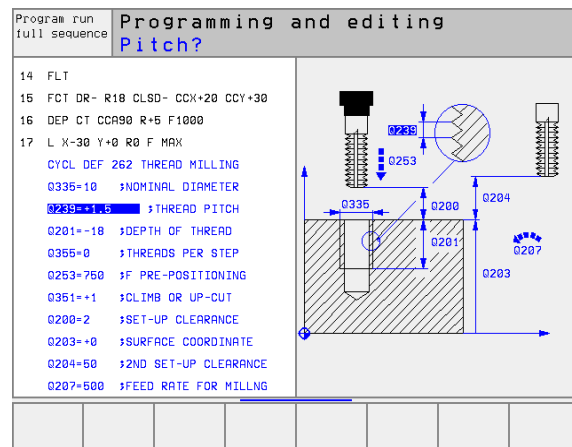
### Calling a Cycle

The following cycles are effective as soon as they are defined:

- Cycles for coordinate transformations
- DWELL TIME cycle
- The SL cycles CONTOUR GEOMETRY and CONTOUR DATA
- Point patterns
- TOLERANCE cycle

All other cycles go into effect when they are called through

- CYCL CALL: effective for one block
- CYCL CALL PAT: used non-modally in connection with point tables
- M99: effective for one block
- M89: effective until canceled (depends on machine parameter settings)



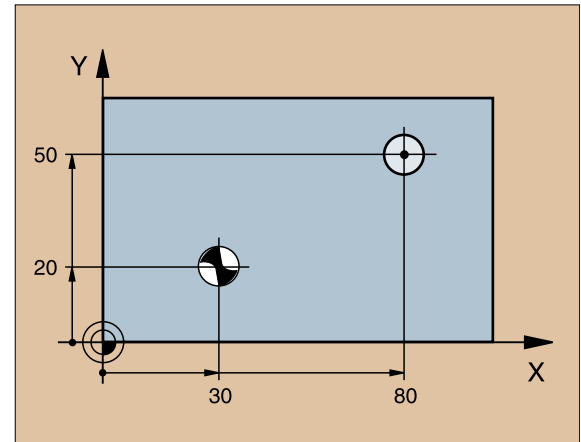
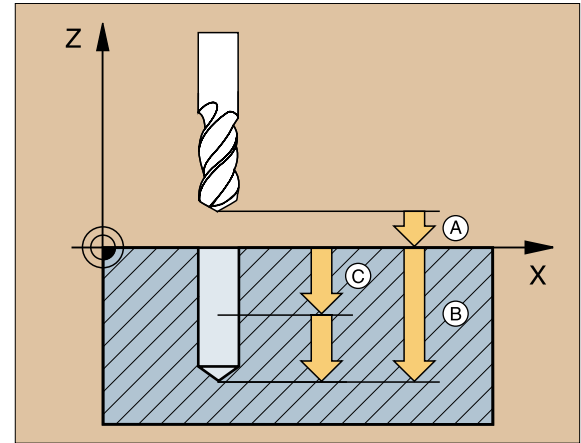
# Cycles for Machining Holes and Threads

## PECKING (1)

- ▶ CYCL DEF: Select Cycle 1 PECKING
  - ▶ Set-up clearance: (A)
  - ▶ Total hole depth (distance from the workpiece surface to the bottom of the hole): (B)
  - ▶ Pecking depth: (C)
  - ▶ Dwell time in seconds
  - ▶ Feed rate F

If the total hole depth is greater than or equal to the pecking depth, the tool drills the entire hole in one plunge.

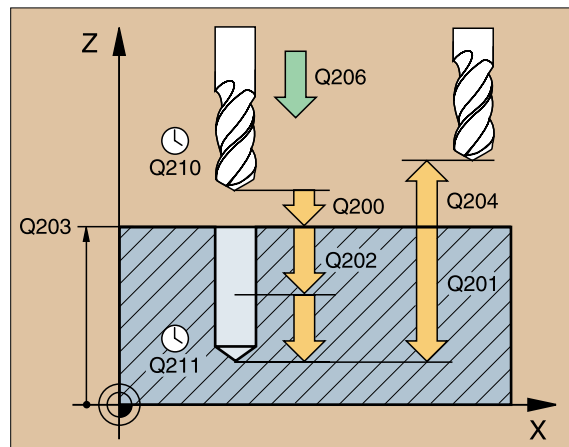
```
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP +2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG +7.5
10 CYCL DEF 1.4 DWELL 1
11 CYCL DEF 1.5 F80
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 L Z+2 FMAX M99
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```



## DRILLING (200)

- ▶ CYCL DEF: Select Cycle 200 DRILLING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Dwell time at top: Q210
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Dwell time at depth: Q211

The TNC automatically pre-positions the tool in the tool axis. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.



### 11 CYCL DEF 200 DRILLING

Q200 = 2 ;SET-UP CLEARANCE

Q201 = -15 ;DEPTH

Q206 = 250 ;FEED RATE FOR PLUNGING

Q202 = 5 ;PLUNGING DEPTH

Q210 = 0 ;DWELL TIME AT TOP

Q203 = +0 ;SURFACE COORDINATE

Q204 = 100 ;2ND SET-UP CLEARANCE

Q211 = 0.1 ;DWELL TIME AT DEPTH

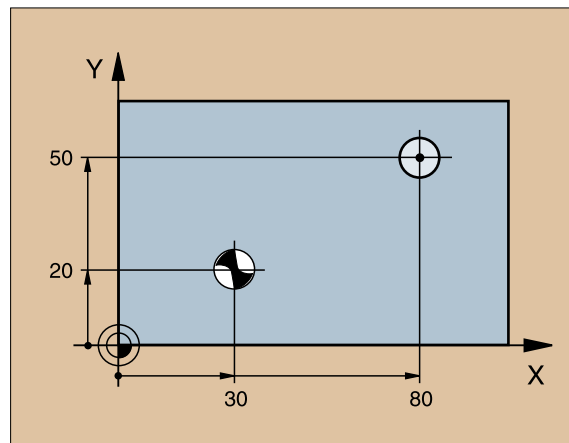
12 L Z+100 R0 FMAX M6

13 L X+30 Y+20 FMAX M3

14 CYCL CALL

15 L X+80 Y+50 FMAX M99

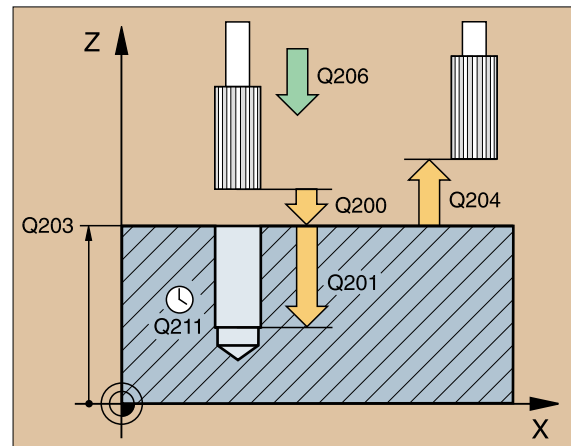
16 L Z+100 FMAX M2



## REAMING (201)

- ▶ CYCL DEF: Select Cycle 201 REAMING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Dwell time at depth: Q211
  - ▶ Retraction feed rate: Q208
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204

The TNC automatically pre-positions the tool in the tool axis.



### 11 CYCL DEF 201 REAMING

Q200 = 2 ;SET-UP CLEARANCE

Q201 = -15 ;DEPTH

Q206 = 100 ;FEED RATE FOR PLNGNG

Q211 = 0.5 ;DWELL TIME AT DEPTH

Q208 = 250 ;RETRACTION FEED RATE

Q203 = +0 ;SURFACE COORDINATE

Q204 = 100 ;2ND SET-UP CLEARANCE

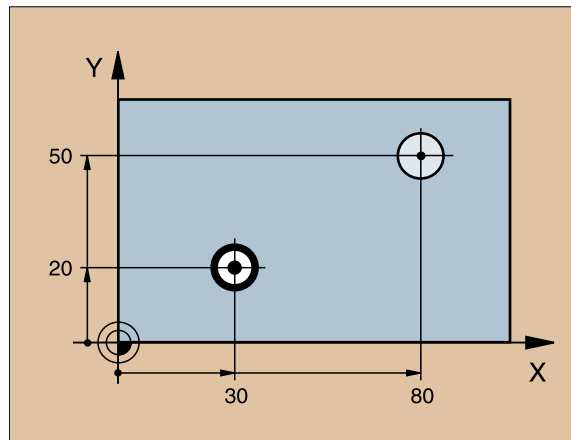
12 L Z+100 R0 FMAX M6

13 L X+30 Y+20 FMAX M3

14 CYCL CALL

15 L X+80 Y+50 FMAX M99

16 L Z+100 FMAX M2





## BORING (202)



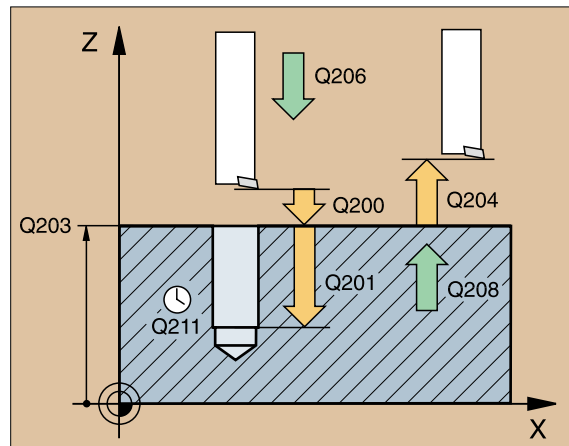
- The machine and TNC must be prepared for the BORING cycle by the machine tool builder!
- This cycle requires a position-controlled spindle!



Danger of collision! Choose a disengaging direction that moves the tool away from the wall of the hole.

- CYCL DEF: Select Cycle 202 BORING
  - Set-up clearance: Q200
  - Depth — Distance between workpiece surface and bottom of hole: Q201
  - Feed rate for plunging: Q206
  - Dwell time at depth: Q211
  - Retraction feed rate: Q208
  - Surface coordinate: Q203
  - 2nd set-up clearance: Q204
  - Disengaging directn (0/1/2/3/4) at bottom of hole: Q214
  - Angle for oriented spindle stop: Q336

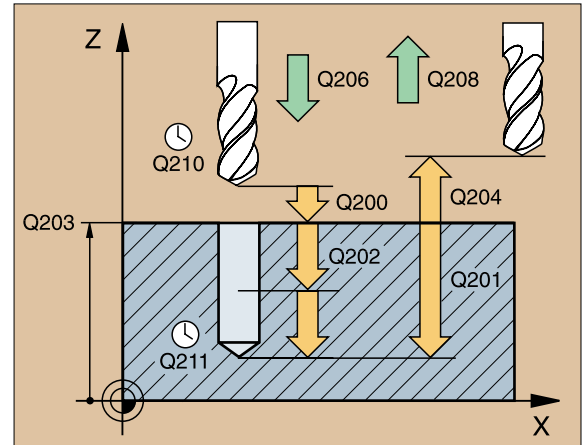
The TNC automatically pre-positions the tool in the tool axis.



## UNIVERSAL DRILLING (203)

- ▶ CYCL DEF: Select Cycle 203 UNIVERSAL DRILLING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Dwell time at top: Q210
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Decrement after each pecking depth: Q212
  - ▶ Nr of breaks – Number of chip breaks before retraction: Q213
  - ▶ Min. pecking depth if a decrement has been entered: Q205
  - ▶ Dwell time at depth: Q211
  - ▶ Retraction feed rate: Q208
  - ▶ Retract dist. for chip breaking: Q256

The TNC automatically pre-positions the tool in the tool axis. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.



## COUNTERBORE BACK (204)



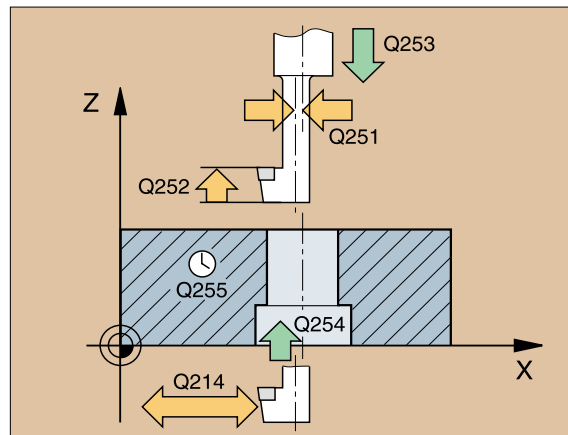
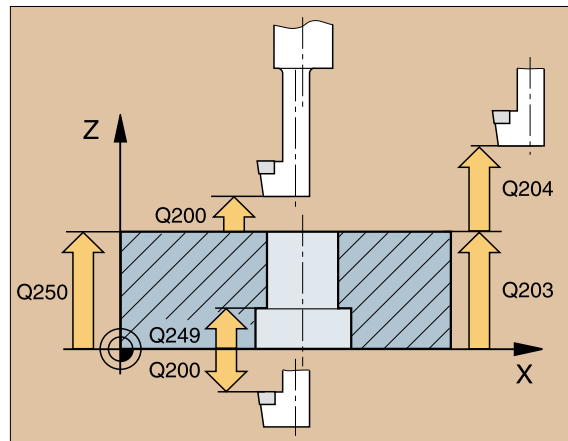
- The machine and TNC must be prepared for the COUNTERBORE BACK cycle by the machine tool builder!
- This cycle requires a position-controlled spindle!



- Danger of collision! Select the disengaging direction that gets the tool clear of the counterbore floor!
- Use this cycle only with a reverse boring bar!

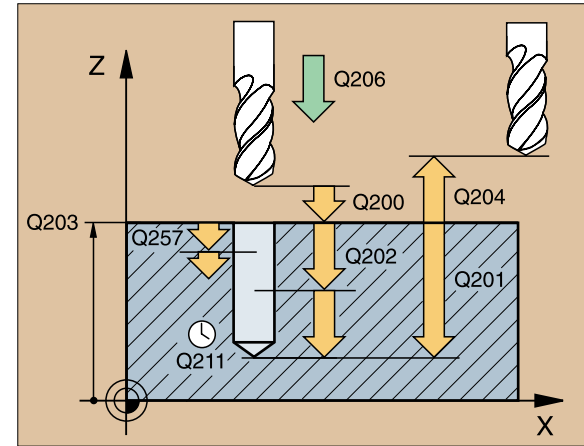
### ► CYCL DEF: Select Cycle 204 COUNTERBORE BACK

- Set-up clearance: Q200
- Depth of counterbore: Q249
- Material thickness: Q250
- Tool edge off-center distance: Q251
- Tool edge height: Q252
- Feed rate for pre-positioning: Q253
- Feed rate for counterboring: Q254
- Dwell time at counterbore floor: Q255
- Workpiece surface coordinate: Q203
- 2nd set-up clearance: Q204
- Disengaging direction (0/1/2/3/4): Q214
- Angle for oriented spindle stop: Q336



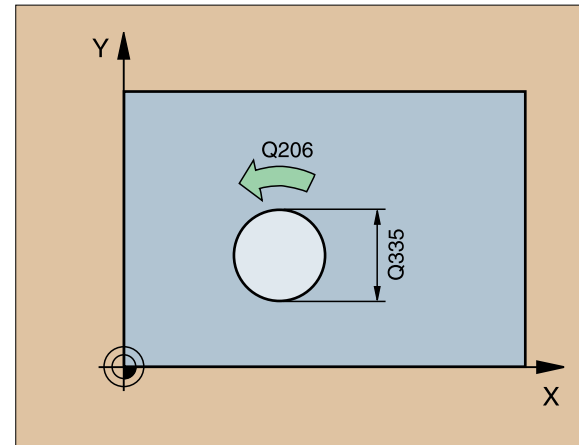
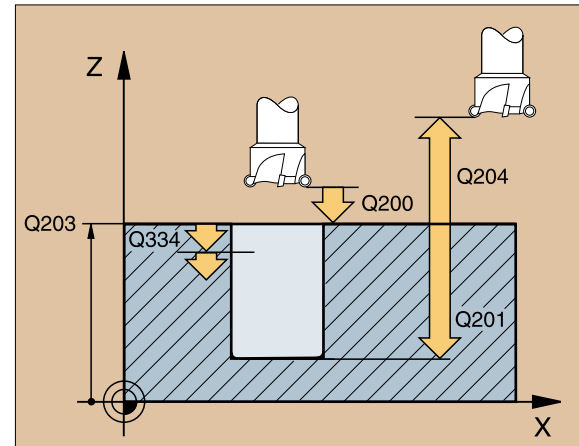
## UNIVERSAL PECKING (205)

- ▶ CYCL DEF: Select Cycle 205 UNIVERSAL PECKING
  - ▶ Set-up clearance: Q200
  - ▶ Depth: Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Decrement after each pecking depth: Q212
  - ▶ Minimum pecking depth if decrement value entered: Q205
  - ▶ Upper advanced stop distance: Q258
  - ▶ Lower advanced stop distance: Q259
  - ▶ Infeed depth for chip breaking: Q257
  - ▶ Retract dist. for chip breaking: Q256
  - ▶ Dwell time at bottom: Q211



## BORE MILLING (208)

- ▶ Pre-position to the center of the hole with R0
- ▶ CYCL DEF: Select Cycle 208 BORE MILLING
  - ▶ Set-up clearance: Q200
  - ▶ Depth: Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Infeed per helix: Q334
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Nominal diameter of hole: Q335
  - ▶ Pilot-drilled diameter: Q342



## TAPPING (2) with Floating Tap Holder

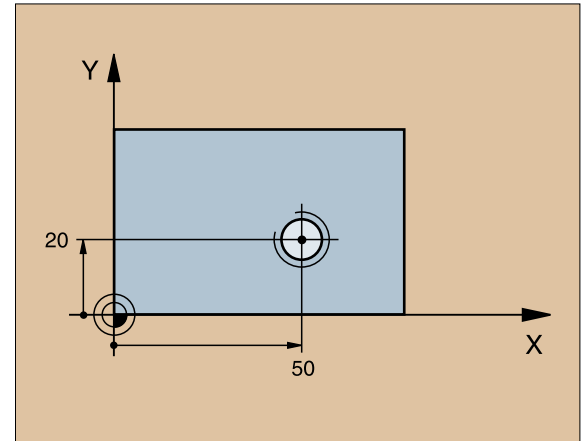
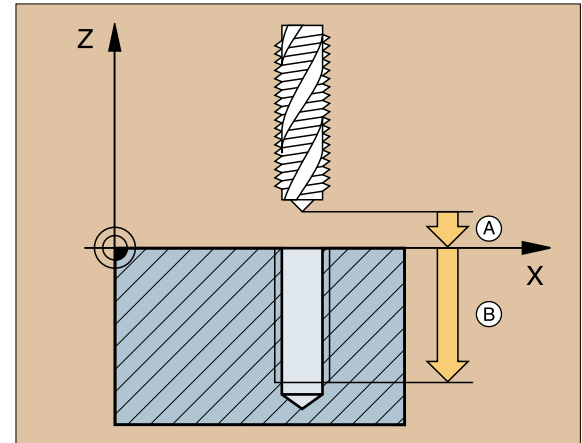
- ▶ Insert the floating tap holder
- ▶ CYCL DEF: Select cycle 2 TAPPING
  - ▶ Set-up clearance: (A)
  - ▶ Total hole depth (thread length = distance between the workpiece surface and the end of the thread): (B)
  - ▶ Dwell time in seconds (a value between 0 and 0.5 seconds)
  - ▶ Feed rate F = Spindle speed S x thread pitch P



For tapping right-hand threads, actuate the spindle with M3, for left-hand threads use M4!

```

25 CYCL DEF 2.0 TAPPING
26 CYCL DEF 2.1 SET UP 3
27 CYCL DEF 2.2 DEPTH -20
28 CYCL DEF 2.3 DWELL 0.4
29 CYCL DEF 2.4 F100
30 L Z+100 R0 FMAX M6
31 L X+50 Y+20 FMAX M3
32 L Z+3 FMAX M99
    
```

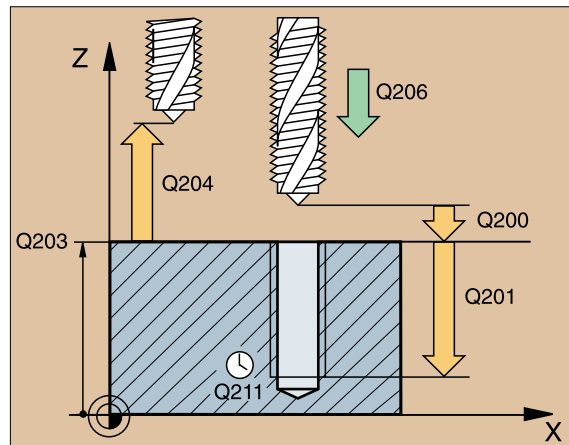


## TAPPING NEW (206) with Floating Tap Holder

- ▶ Insert the floating tap holder
- ▶ CYCL DEF: Select Cycle 206 TAPPING NEW
  - ▶ Set-up clearance: Q200
  - ▶ Depth: thread length = distance between the workpiece surface and the end of the thread: Q201
  - ▶ Feed rate F = spindle speed S x thread pitch P: Q206
  - ▶ Dwell time at bottom (enter a value between 0 and 0.5 seconds): Q211
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204



For tapping right-hand threads, actuate the spindle with M3, for left-hand threads use M4!

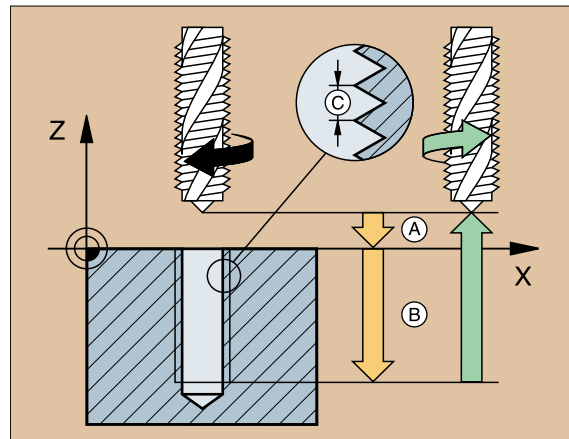


## RIGID TAPPING (17) without Floating Tap Holder



- Machine and TNC must be prepared by the machine tool builder to perform rigid tapping!
- In rigid tapping, the spindle speed is synchronized with the tool axis feed rate!

- ▶ CYCL DEF: Select cycle 17 RIGID TAPPING
  - ▶ Set-up clearance: (A)
  - ▶ Tapping depth (distance between workpiece surface and end of thread): (B)
  - ▶ Pitch: (C)
- The algebraic sign determines the direction of the thread:
  - Right-hand thread: +
  - Left-hand thread: -



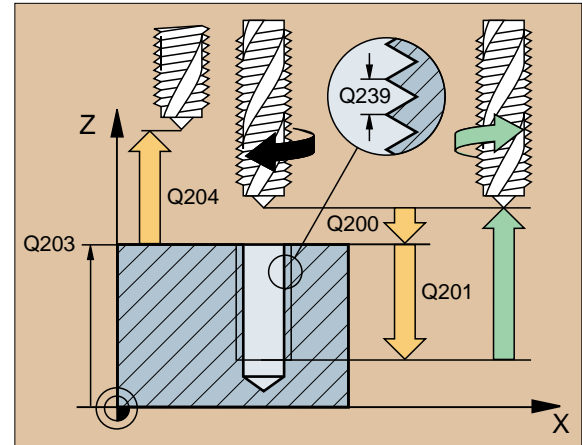
## RIGID TAPPING NEW (207) without Floating Tap Holder



- Machine and TNC must be prepared by the machine tool builder to perform rigid tapping!
- Rigid tapping is carried out with a controlled spindle!

### ► CYCL DEF: Select Cycle 207 RIGID TAPPING NEW

- Set-up clearance: Q200
- Depth: thread length = distance between workpiece surface and end of thread: Q201
- Pitch: Q239
  - The algebraic sign determines the direction of the thread:
    - Right-hand thread: +
    - Left-hand thread: -
- Workpiece surface coordinate: Q203
- 2nd set-up clearance: Q204



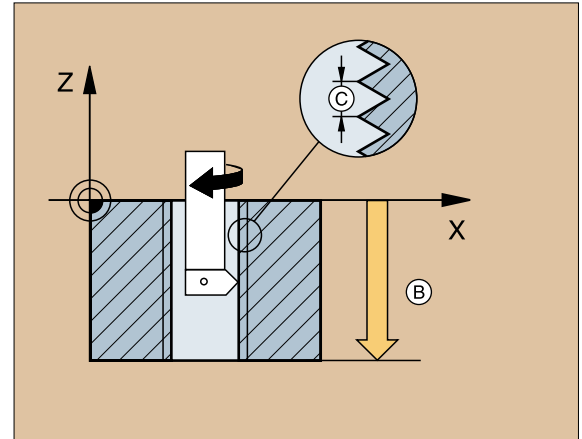
## THREAD CUTTING (18)



- The machine and TNC must be prepared by the machine tool builder for THREAD CUTTING!
- The spindle speed is synchronized with the tool axis feed rate!

### ► CYCL DEF: Select cycle 18 THREAD CUTTING

- Depth (distance between workpiece surface and end of thread): (B)
- Pitch: (C)
  - The algebraic sign:
    - Right-hand thread: +
    - Left-hand thread: -



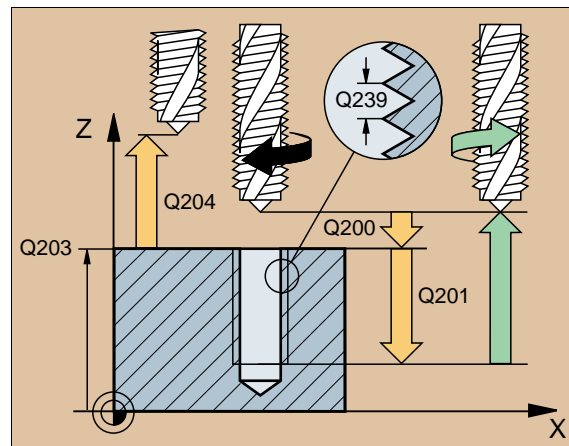


## TAPPING WITH CHIP BREAKING (209)



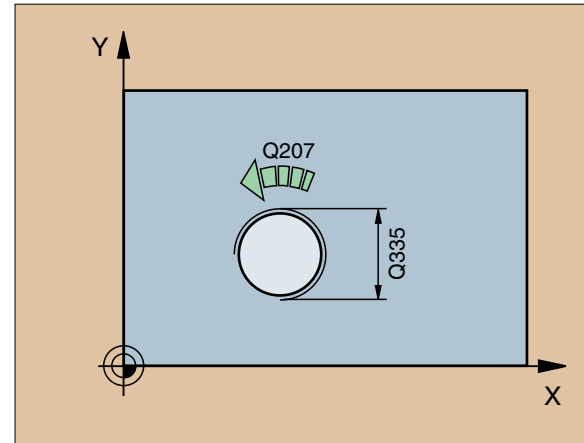
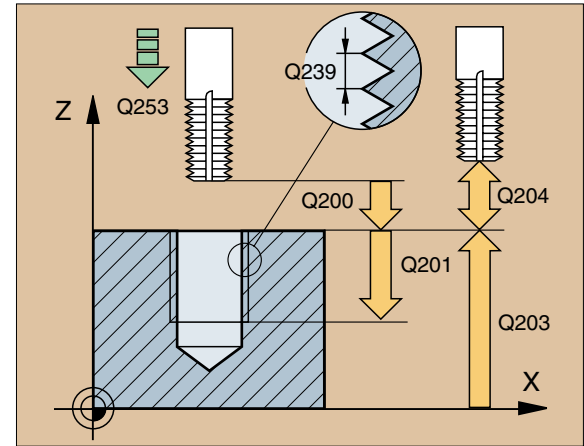
- The machine and TNC must be prepared for the TAPPING WITH CHIP BREAKING cycle by the machine tool builder!
- This cycle requires a position-controlled spindle!

- CYCL DEF: Select Cycle 209 TAPPING W/ CHIP BRKG .
  - Set-up clearance: Q200
  - Thread depth: Thread length = Distance between workpiece surface and thread termination: Q201
  - Thread pitch: Q239  
The algebraic sign determines the direction of the thread:
    - Right-hand thread: +
    - Left-hand thread: -
  - Coordinate of top of workpiece: Q203
  - 2nd set-up clearance: Q204
  - Infeed depth for chip breaking: Q257
  - Retraction distance for chip breaking: Q256
  - Angle for spindle orientation: Q336



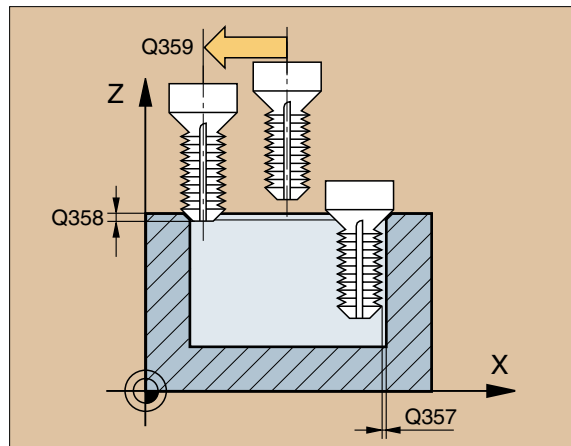
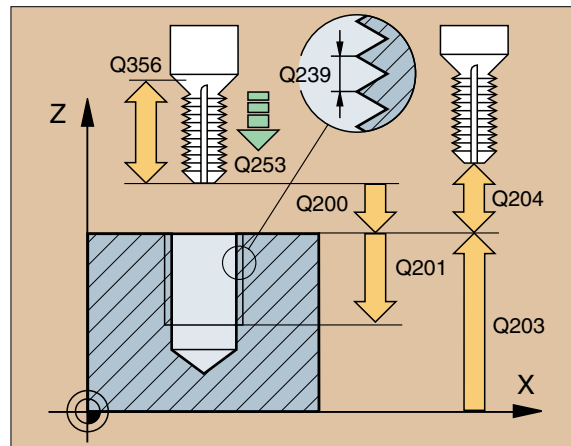
## THREAD MILLING (262)

- ▶ Pre-position above the hole center with R0
- ▶ CYCL DEF: Select Cycle 262 THREAD MILLING
  - ▶ Nominal diameter of the thread: Q335
  - ▶ Thread pitch: Q239
  - ▶ The algebraic sign determines the thread direction:
    - Right-hand thread: +
    - Left-hand thread: -
  - ▶ Thread depth: Distance from top of workpiece to thread termination: Q201
  - ▶ Number of threads per step: Q355
  - ▶ Feed rate for pre-positioning: Q253
  - ▶ Type of milling: Q351
    - Climb: +1
    - Up-cut: -1
  - ▶ Set-up clearance: Q200
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Feed rate for milling: Q207



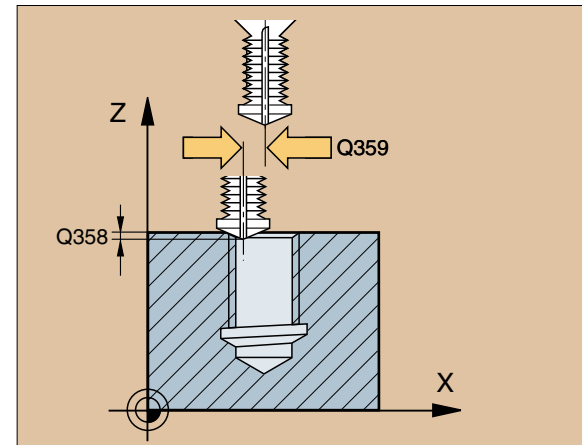
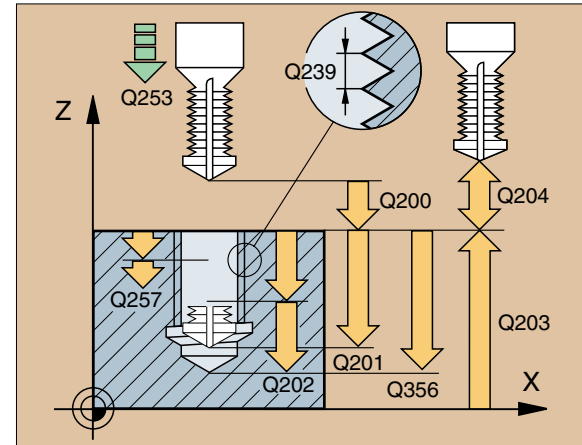
## THREAD MILLING AND COUNTERSINKING (263)

- ▶ Pre-position above the hole center with R0
- ▶ CYCL DEF: Select Cycle 263 THREAD MILLING AND COUNTERSINKING
  - ▶ Nominal diameter of thread: Q335
  - ▶ Thread pitch: Q239
    - The algebraic sign determines the direction of the thread:
    - Right-hand thread: +
    - Left-hand thread: -
  - ▶ Thread depth: Distance from top of workpiece to thread termination: Q201
  - ▶ Countersinking depth: Distance from workpiece surface to bottom of hole: Q356
  - ▶ Feed rate for pre-positioning: Q253
  - ▶ Type of milling: Q351
    - Climb: +1
    - Up-cut: -1
  - ▶ Set-up clearance: Q200
  - ▶ Lateral set-up clearance: Q357
  - ▶ Sinking depth at front: Q358
  - ▶ Countersinking offset at front: Q359
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Feed rate for counterboring: Q254
  - ▶ Feed rate for milling: Q207



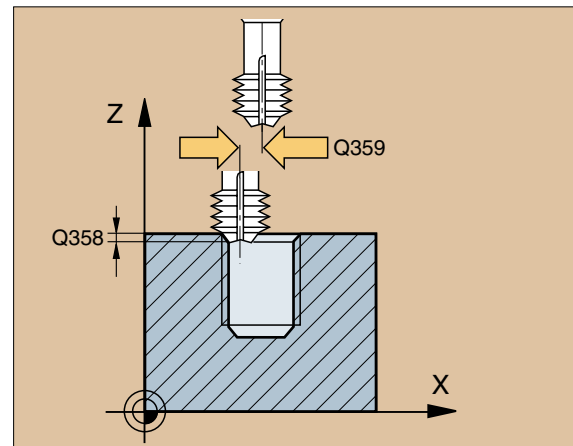
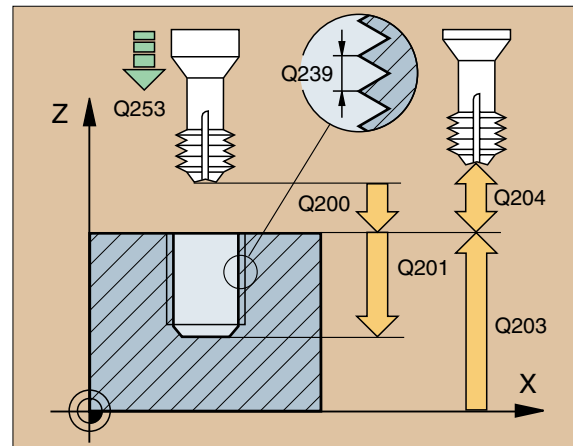
## THREAD DRILLING AND MILLING (264)

- ▶ Pre-position over the hole center with R0
- ▶ CYCL DEF: Select Cycle 264 THREAD DRLLNG/MLLNG
  - ▶ Nominal diameter of thread: Q335
  - ▶ Thread pitch: Q239
    - The algebraic sign determines the thread direction:
      - Right-hand thread: +
      - Left-hand thread: -
  - ▶ Thread depth: Distance from top of workpiece to thread termination: Q201
  - ▶ Hole depth: Distance from top of workpiece to bottom of hole: Q201
  - ▶ Feed rate for pre-positioning: Q253
  - ▶ Type of milling: Q351
    - Climb: +1
    - Up-cut: -1
  - ▶ Plunging depth: Q202
  - ▶ Upper advanced stop distance: Q258
  - ▶ Infeed depth for chip breaking: Q257
  - ▶ Retraction distance for chip breaking: Q256
  - ▶ Dwell time at bottom: Q211
  - ▶ Sinking depth at front: Q358
  - ▶ Countersinking offset at front: Q359
  - ▶ Set-up clearance: Q200
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Feed rate for plunging: Q206
  - ▶ Feed rate for milling: Q207



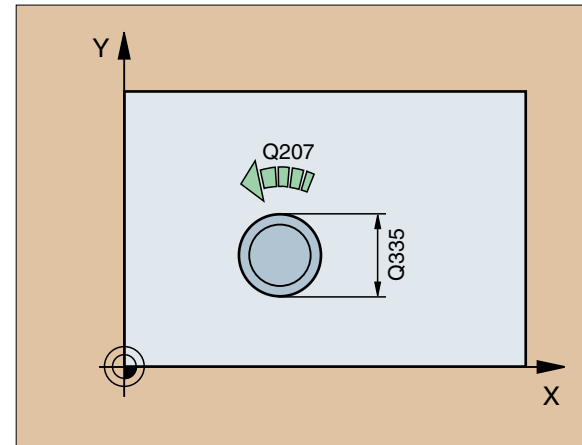
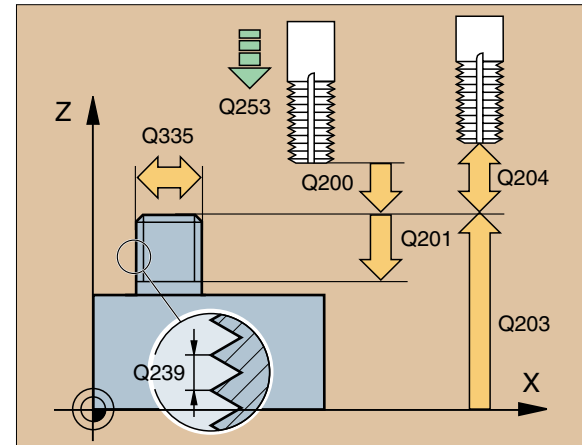
## HELICAL THREAD DRILLING AND MILLING (265)

- ▶ Pre-position over the hole center with R0
- ▶ CYCL DEF: Select Cycle 265 HEL. THREAD DRLG/MLG
  - ▶ Nominal diameter of the thread: Q335
  - ▶ Thread pitch: Q239
  - The algebraic sign determines the thread direction:
    - Right-hand thread: +
    - Left-hand thread: -
  - ▶ Thread depth: Distance from top of workpiece to thread termination: Q201
  - ▶ Feed rate for pre-positioning: Q253
  - ▶ Sinking depth at front: Q358
  - ▶ Countersinking offset at front: Q359
  - ▶ Countersink: Q360
  - ▶ Set-up clearance: Q200
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Feed rate for countersinking: Q254
  - ▶ Feed rate for milling: Q207



## OUTSIDE THREAD MILLING (267)

- ▶ Pre-position over the hole center with R0
- ▶ CYCL DEF: Select Cycle 267 OUTSIDE THREAD MILLING
  - ▶ Nominal diameter of thread: Q335
  - ▶ Thread pitch: Q239
    - The algebraic sign determines the thread direction:
      - Right-hand thread: +
      - Left-hand thread: -
  - ▶ Hole depth: Distance from top of workpiece to bottom of hole: Q201
  - ▶ Number of threads per step: Q355
  - ▶ Feed rate for pre-positioning: Q253
  - ▶ Type of milling: Q351
    - Climb: +1
    - Up-cut: -1
  - ▶ Set-up clearance: Q200
  - ▶ Sinking depth at front: Q358
  - ▶ Countersinking offset at front: Q359
  - ▶ Workpiece surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Feed rate for countersinking: Q254
  - ▶ Feed rate for milling: Q207



# Pockets, Studs, and Slots

## POCKET MILLING (4)



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

The tool begins milling in the positive axis direction of the longer side. In square pockets it moves in the positive Y direction.

- ▶ The tool must be pre-positioned over the center of the slot with tool radius compensation **R0**
- ▶ CYCL DEF: Select cycle 4 POCKET MILLING
  - ▶ Set-up clearance: (A)
  - ▶ Milling depth (depth of the pocket): (B)
  - ▶ Pecking depth: (C)
  - ▶ Feed rate for pecking
  - ▶ First side length (length of the pocket, parallel to the first main axis of the working plane): (D)
  - ▶ Second side length (width of pocket, sign always positive): (E)
  - ▶ Feed rate
  - ▶ Rotation clockwise: DR-  
Climb milling with M3: DR+  
Up-cut milling with M3: DR-
  - ▶ Rounding-off radius R (radius for the pocket corners)

12 CYCL DEF 4.0 POCKET MILLING

13 CYCL DEF 4.1 SET UP2

14 CYCL DEF 4.2 DEPTH-10

15 CYCL DEF 4.3 PECKG4 F80

16 CYCL DEF 4.4 X80

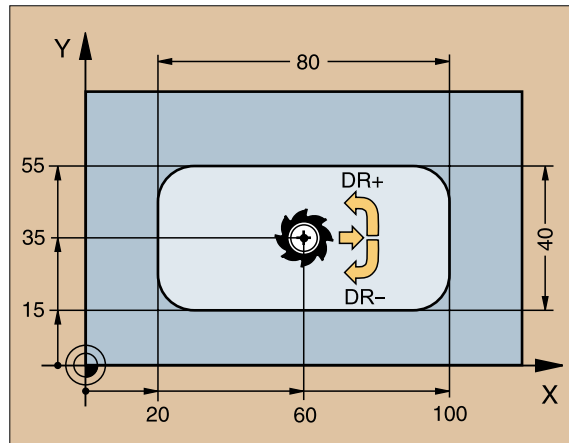
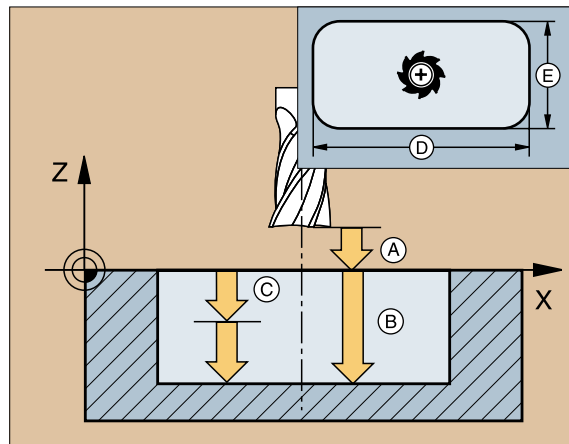
17 CYCL DEF 4.5 Y40

18 CYCL DEF 4.6 F100 DR+ RADIUS 10

19 L Z+100 R0 FMAX M6

20 L X+60 Y+35 FMAX M3

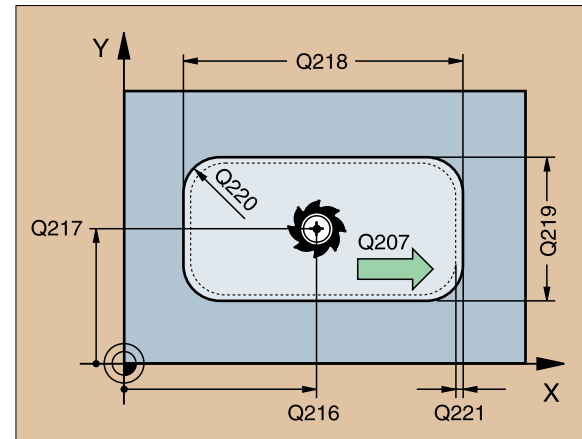
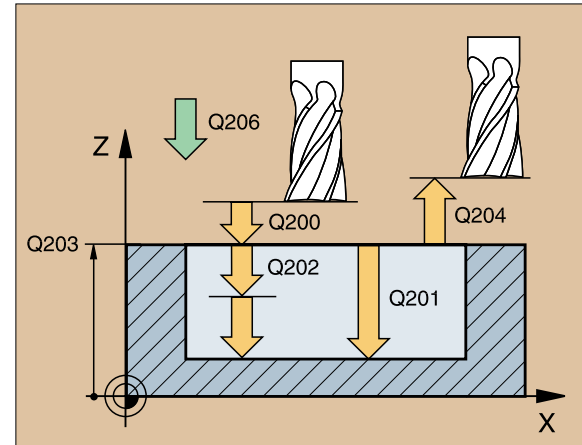
21 L Z+2 FMAX M99



## POCKET FINISHING (212)

- ▶ CYCL DEF: Select Cycle 212 POCKET FINISHING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Feed rate for milling: Q207
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Center in 1st axis: Q216
  - ▶ Center in 2nd axis: Q217
  - ▶ First side length: Q218
  - ▶ Second side length: Q219
  - ▶ Corner radius: Q220
  - ▶ Allowance in 1st axs: Q221

The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.

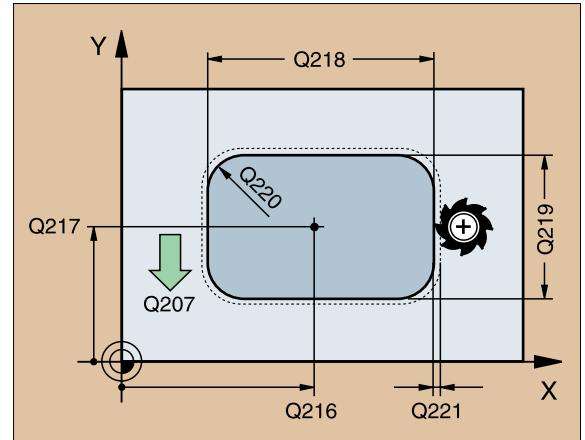
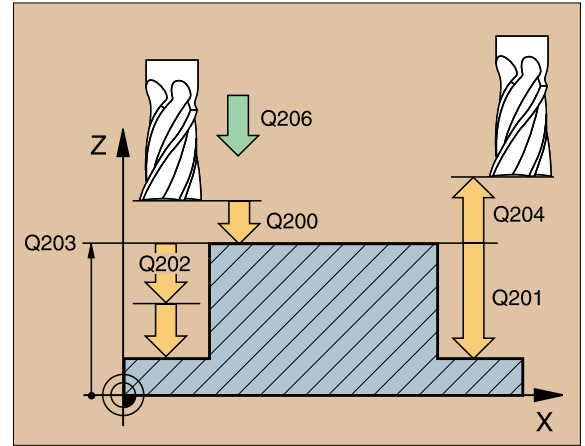




## STUD FINISHING (213)

- ▶ CYCL DEF: Select Cycle 213 STUD FINISHING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Feed rate for milling: Q207
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Center in 1st axis: Q216
  - ▶ Center in 2nd axis: Q217
  - ▶ First side length: Q218
  - ▶ Second side length: Q219
  - ▶ Corner radius: Q220
  - ▶ Corner radius: Q220
  - ▶ Allowance in 1st axis: Q221

The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.



## CIRCULAR POCKET MILLING (5)



This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at pocket center!

- ▶ The tool must be pre-positioned over the center of the slot with tool radius compensation **R0**
- ▶ CYCL DEF: Select cycle 5
  - ▶ Set-up clearance: (A)
  - ▶ Milling depth (depth of the pocket): (B)
  - ▶ Pecking depth: (C)
  - ▶ Feed rate for pecking
  - ▶ Circle radius R (radius of the pocket)
  - ▶ Feed rate
  - ▶ Rotation clockwise: DR-  
Climb milling with M3: DR+  
Up-cut milling with M3: DR-

17 CYCL DEF 5.0 CIRCULAR POCKET

18 CYCL DEF 5.1 SET UP 2

19 CYCL DEF 5.2 DEPTH -12

20 CYCL DEF 5.3 PECKG 6 F80

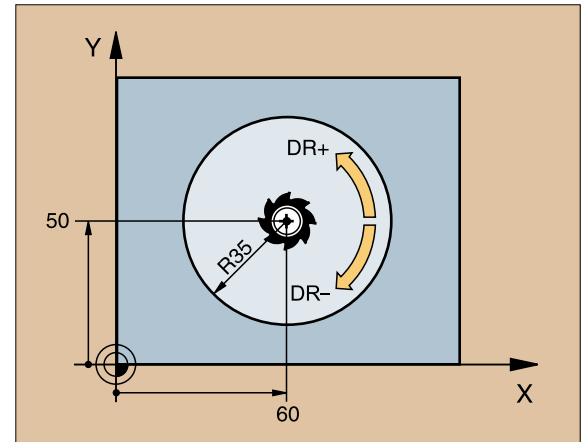
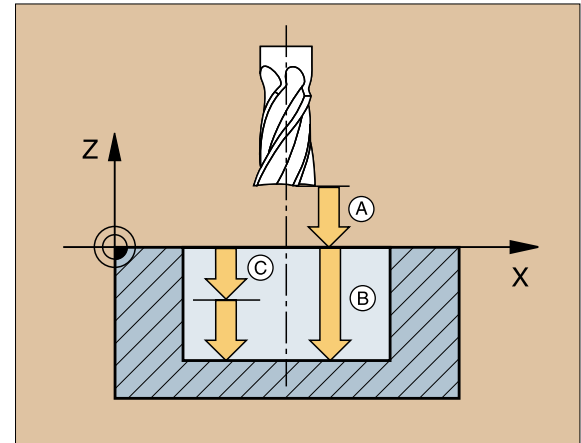
21 CYCL DEF 5.4 RADIUS 35

22 CYCL DEF 5.5 F100 DR+

23 L Z+100 R0 FMAX M6

24 L X+60 Y+50 FMAX M3

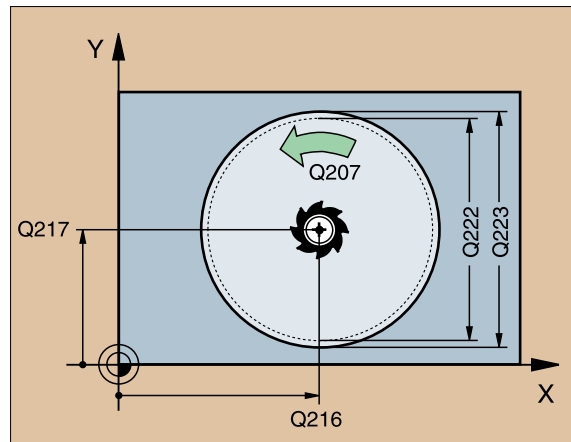
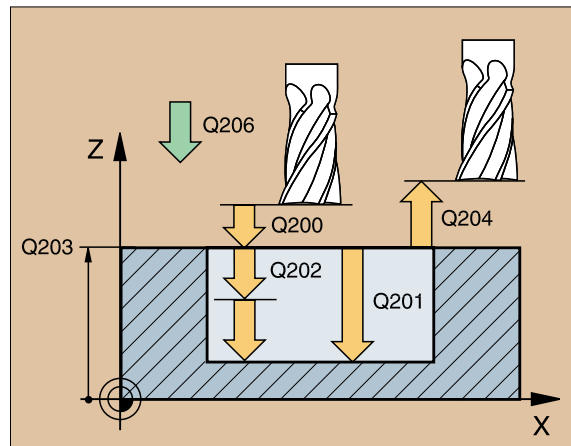
25 L Z+2 FMAX M99



## CIRCULAR POCKET FINISHING (214)

- ▶ CYCL DEF: Select Cycle 214 CIRCULAR POCKET FINISHING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Feed rate for milling: Q207
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Center in 1st axis: Q216
  - ▶ Center in 2nd axis: Q217
  - ▶ Workpiece blank dia.: Q222
  - ▶ Finished part dia.: Q223

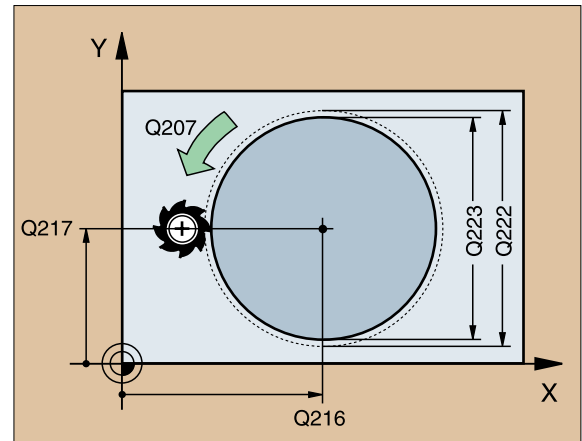
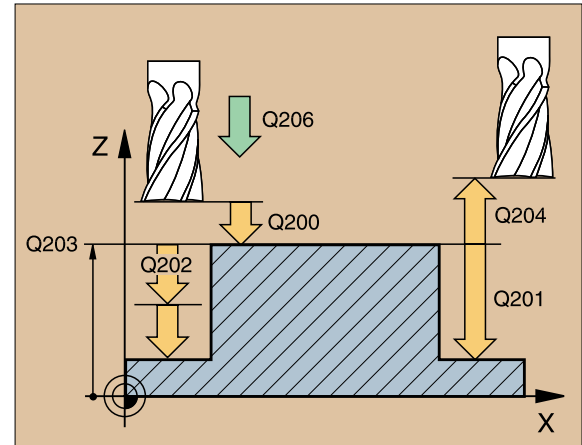
The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.



## CIRCULAR STUD FINISHING (215)

- ▶ CYCL DEF: Select Cycle 215 CIRCULAR STUD FINISHING
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for plunging: Q206
  - ▶ Pecking depth: Q202
  - ▶ Feed rate for milling: Q207
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Center in 1st axis: Q216
  - ▶ Center in 2nd axis: Q217
  - ▶ Workpiece blank dia.: Q222
  - ▶ Finished part dia.: Q223

The TNC automatically pre-positions the tool in the tool axis and in the working plane. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.



## SLOT MILLING (3)

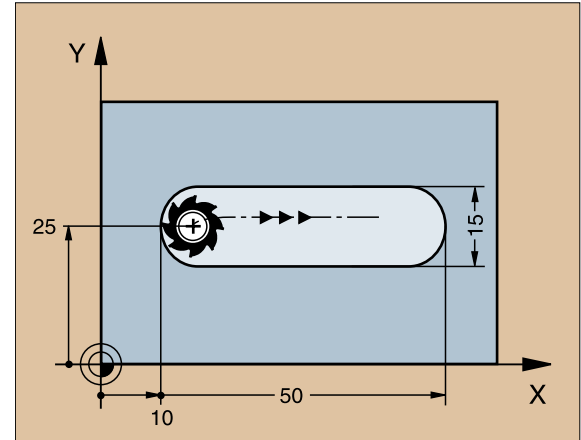
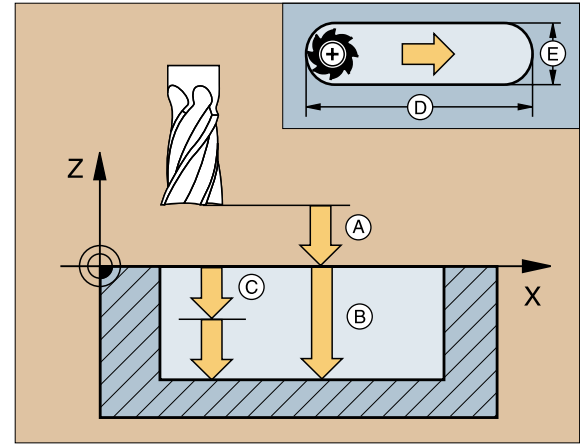


- This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at the starting point!
- The cutter diameter must be smaller than the slot width and larger than half the slot width!

- ▶ The tool must be pre-positioned over the midpoint of the slot and offset by the tool radius with tool radius compensation at **R0**
- ▶ CYCL DEF: Select cycle 3 SLOT MILLING
  - ▶ Set-up clearance: (A)
  - ▶ Milling depth (depth of the slot): (B)
  - ▶ Pecking depth: (C)
  - ▶ Feed rate for pecking (traverse velocity for plunging)
  - ▶ First side length ? (length of the slot): (D)  
The algebraic sign determines the first cutting direction
  - ▶ Second side length ? (width of the slot): (E)
  - ▶ Feed rate (for milling)

```

10 TOOL DEF 1 L+0 R+6
11 TOOL CALL 1 Z S1500
12 CYCL DEF 3.0 SLOT MILLING
13 CYCL DEF 3.1 SET UP 2
14 CYCL DEF 3.2 DEPTH -15
15 CYCL DEF 3.3 PECKG 5 F80
16 CYCL DEF 3.4 X50
17 CYCL DEF 3.5 Y15
18 CYCL DEF 3.6 F120
19 L Z+100 R0 FMAX M6
20 L X+16 Y+25 R0 FMAX M3
21 L Z+2 M99
  
```



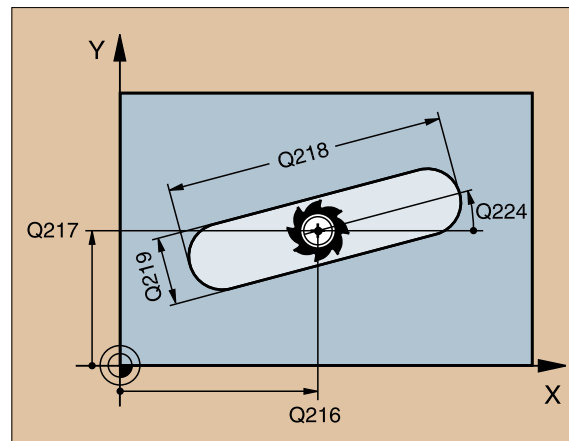
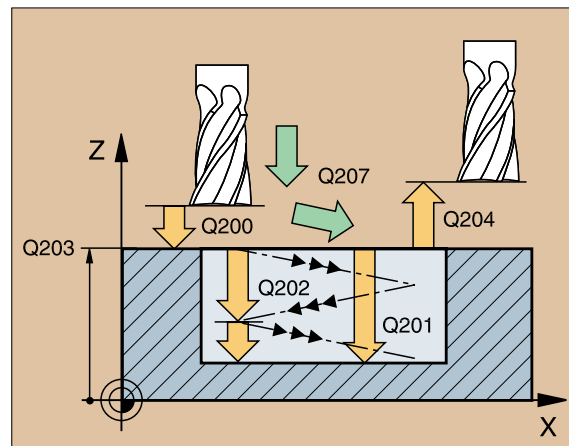
## SLOT WITH RECIPROCATING PLUNGE-CUT (210)



The cutter diameter must be no larger than the width of the slot, and no smaller than one third!

- ▶ CYCL DEF: Select Cycle 210 SLOT RECIP. PLNG
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for milling: Q207
  - ▶ Pecking depth: Q202
  - ▶ Machining operation (0/1/2) – 0 = roughing and finishing, 1 = roughing only, 2 = finishing only: Q215
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Center in 1st axis: Q216
  - ▶ Center in 2nd axis: Q217
  - ▶ First side length: Q218
  - ▶ Second side length: Q219
  - ▶ Angle of rotation (angle by with the slot is rotated): Q224
  - ▶ Infeed for finishing: Q338

The TNC automatically pre-positions the tool in the tool axis and in the working plane. During roughing the tool plunges obliquely into the metal in a back-and-forth motion between the ends of the slot. Pilot drilling is therefore unnecessary.



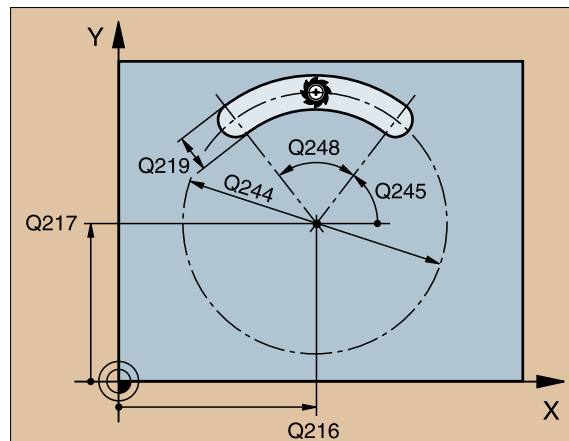
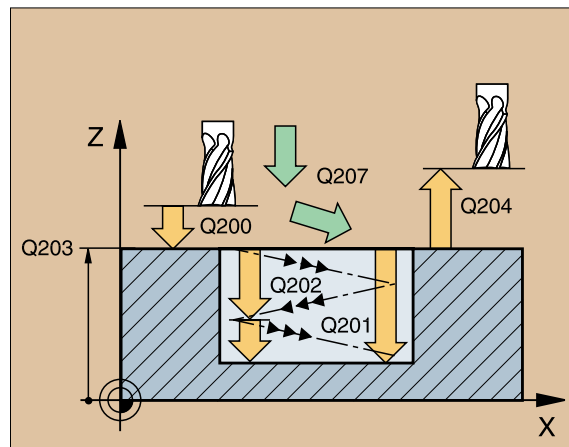
## CIRCULAR SLOT with reciprocating plunge (211)



The cutter diameter must be no larger than the width of the slot, and no smaller than one third!

- ▶ CYCL DEF: Select Cycle 211 CIRCULAR SLOT
  - ▶ Set-up clearance: Q200
  - ▶ Depth – Distance between workpiece surface and bottom of hole: Q201
  - ▶ Feed rate for milling: Q207
  - ▶ Pecking depth: Q202
  - ▶ Machining operation (0/1/2) – 0 = roughing and finishing, 1 = roughing only, 2 = finishing only: Q215
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204
  - ▶ Center in 1st axis: Q216
  - ▶ Center in 2nd axis: Q217
  - ▶ Pitch circular dia.: Q244
  - ▶ Second side length: Q219
  - ▶ Starting angle of the slot: Q245
  - ▶ Angular length of the slot: Q248
  - ▶ Infeed for finishing: Q338

The TNC automatically pre-positions the tool in the tool axis and in the working plane. During roughing the tool plunges obliquely into the metal in a back-and-forth helical motion between the ends of the slot. Pilot drilling is therefore unnecessary.



# Point Patterns

## CIRCULAR PATTERN (220)

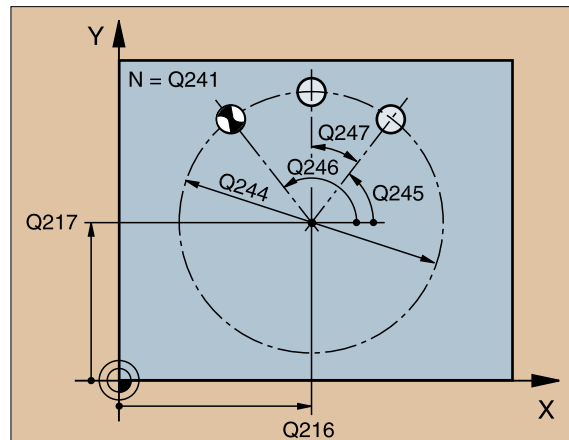
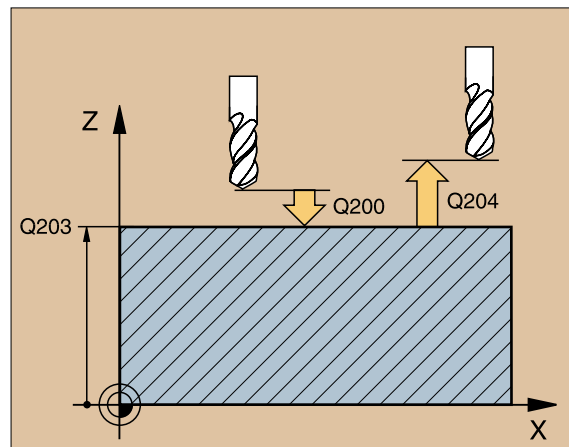
### ► CYCL DEF: Select Cycle 220 CIRCULAR PATTERN

- Center in 1st axis: Q216
- Center in 2nd axis: Q217
- Angle of rotation: Q244
- Starting angle: Q245
- Stopping angle: Q246
- Stepping angle: Q247
- Nr of repetitions: Q241
- Set-up clearance: Q200
- Surface coordinate: Q203
- 2nd set-up clearance: Q204
- Move to clearance height: Q301



- Cycle 220 POLAR PATTERN is effective immediately upon definition!
- Cycle 220 automatically calls the last defined fixed cycle!
- Cycle 220 can be combined with Cycles 1, 2, 3, 4, 5, 17, 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 212, 213, 214, 215, 262, 263, 264, 265, 267
- In combined cycles, the set-up clearance, surface coordinate and 2nd set-up-clearance are always taken from Cycle 220!

The TNC automatically pre-positions the tool in the tool axis and in the working plane.





## LINEAR PATTERN (221)

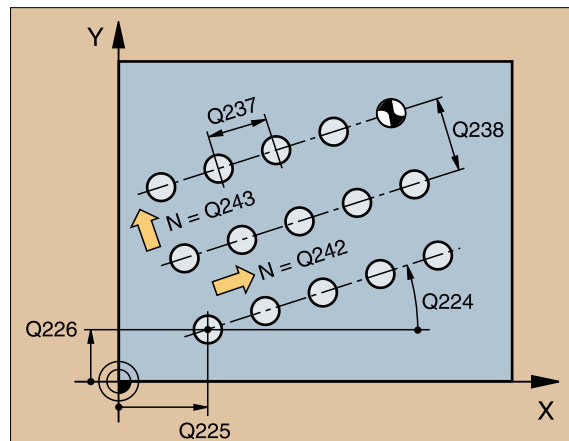
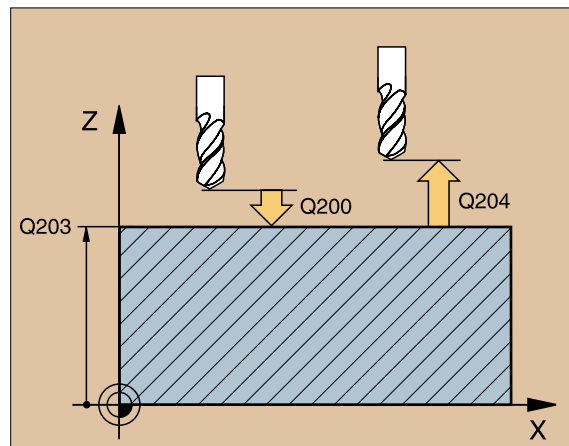
### ► CYCL DEF: Select Cycle 221 LINEAR PATTERN

- Startng pnt 1st axis: Q225
- Startng pnt 2nd axis: Q226
- Spacing in 1st axis: Q237
- Spacing in 2nd axis: Q238
- Number of columns: Q242
- Number of lines: Q243
- Angle of rotation: Q224
- Set-up clearance: Q200
- Surface coordinate: Q203
- 2nd set-up clearance: Q204
- Move to clearance height: Q301



- Cycle 221 LINEAR PATTERN is effective immediately upon definition!
- Cycle 221 automatically calls the last defined fixed cycle!
- Cycle 221 can be combined with Cycles 1, 2, 3, 4, 5, 17, 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 212, 213, 214, 215, 262, 263, 264, 265, 267
- In combined cycles, the set-up clearance, surface coordinate and 2nd set-up-clearance are always taken from Cycle 221!

The TNC automatically pre-positions the tool in the tool axis and in the working plane.



# SL Cycles

## General Information

SL cycles are useful when you wish to machine a contour consisting of several subcontours (up to 12 islands or pockets).

The subcontours are defined in subprograms.



When working with subcontours, always remember:

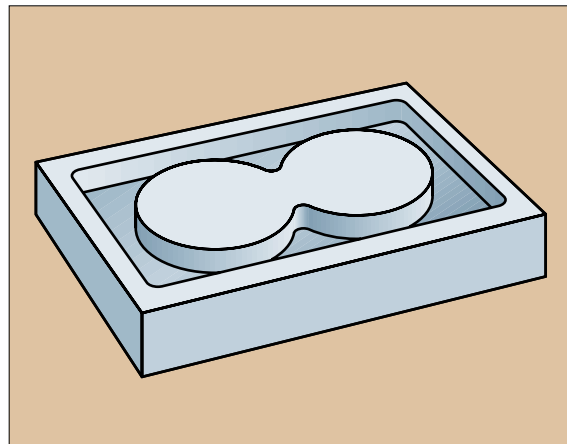
- For a **pocket** the tool machines an inside contour, for an **island** it is an outside contour!
- Tool **approach** and **departure** as well as **infeed in the tool axis cannot** be programmed in SL cycles!
- Each contour listed in Cycle 14 CONTOUR GEOMETRY must be a closed contour!
- There is a limit to the amount of memory an SL cycle can occupy! A maximum of 128 straight line blocks, for example, can be programmed in an SL cycle.



The contour for cycle 25 CONTOUR TRAIN must not be closed!



Make a graphic test run before actually machining a part. That way you can be sure that you defined the contour correctly!



## CONTOUR GEOMETRY (14)

In Cycle 14 CONTOUR GEOMETRY you list the subprograms that you wish to superimpose to make a complete closed contour.

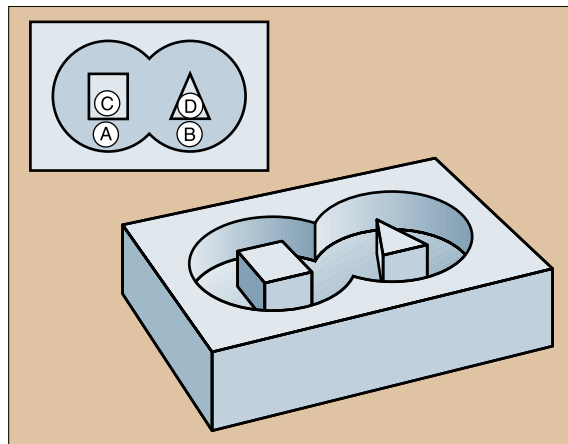
- ▶ CYCL DEF: Select Cycle 14 CONTOUR GEOMETRY
  - ▶ Label nubers for contour: List the LABEL numbers of the subprograms that you wish to superimpose to make a complete closed contour.



Cycle 14 CONTOUR GEOMETRY is effective immediately upon definition!

```

4 CYCL DEF 14.0 CONTOUR GEOM
5 CYCL DEF 14.1 CONTOUR LABEL 1/2/3
...
36 L Z+200 R0 FMAX M2
37 LBL1
38 L X+0 Y+10 RR
39 L X+20 Y+10
40 CC X+50 Y+50
...
45 LBL0
46 LBL2
...
58 LBL0
  
```



▲ (A) and (B) are pockets, (C) and (D) islands

## CONTOUR DATA (20)

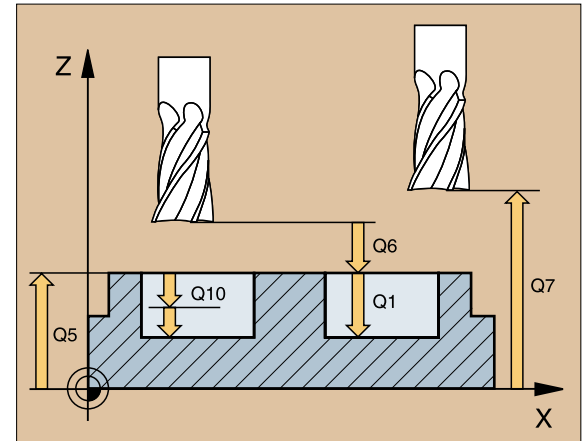
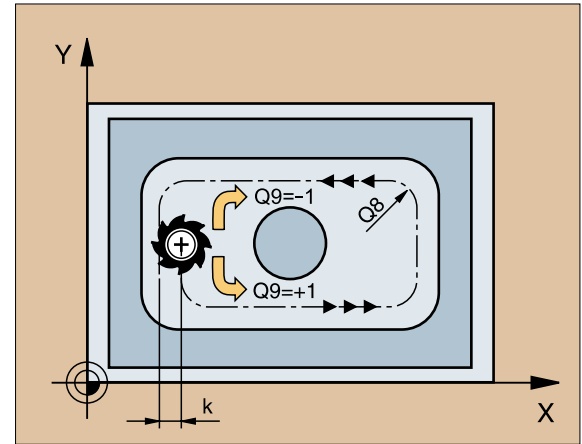
Cycle 20 CONTOUR DATA defines the machining information for cycles 21 to 24.

### ► CYCL DEF: Select Cycle 20 CONTOUR DATA

- Milling depth Q1:  
Distance from workpiece surface to pocket floor; incremental
- Path overlap factor Q2:  
 $Q2 \times \text{tool radius} = \text{stepover factor } k$
- Allowance for side Q3:  
Finishing allowance for the walls of the pocket or island
- Allowance for floor Q4:  
Finishing allowance for the pocket floor
- Workpiece surface coordinates Q5:  
Coordinate of the workpiece surface referenced to the current datum; absolute
- Set-up clearance Q6:  
Distance from the tool to the workpiece surface; incremental
- Clearance height Q7:  
Height at which the tool cannot collide with the workpiece; absolute
- Rounding radius Q8:  
Rounding radius of the tool at inside corners
- Direction of rotation Q9:  
  - Clockwise  $Q9 = -1$
  - Counter clockwise  $Q9 = +1$

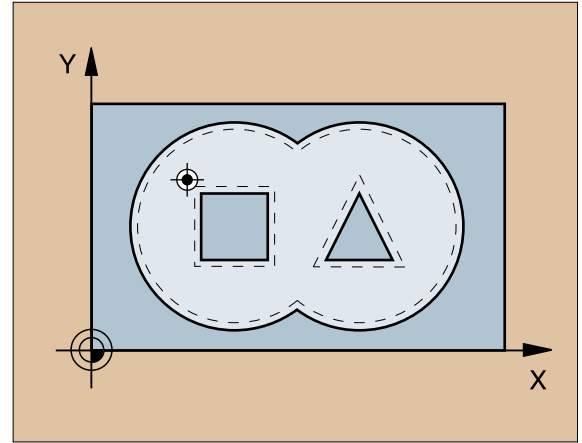


Cycle 20 CONTOUR DATA is effective immediately upon definition!



## PILOT DRILLING (21)

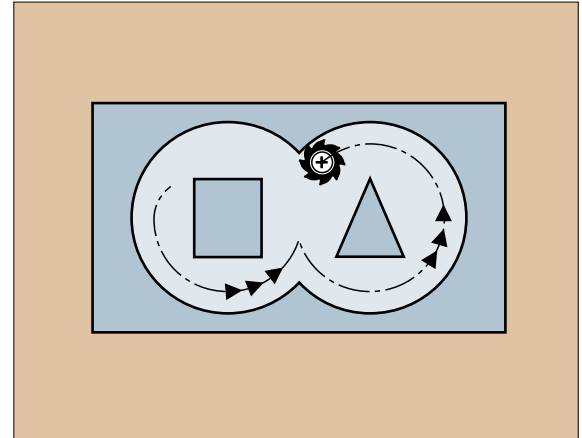
- ▶ CYCL DEF: Select Cycle 21 PILOT DRILLING
  - ▶ Pecking depth Q10; incremental
  - ▶ Feed rate for pecking Q11
  - ▶ Rough mill Q13: Number of the roughing tool



## ROUGH-OUT (22)

The tool moves parallel to the contour at every pecking depth.

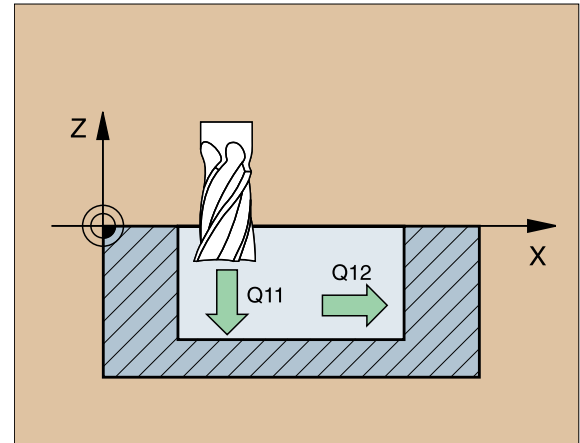
- ▶ CYCL DEF: Select Cycle 22 ROUGH-OUT
  - ▶ Pecking depth Q10; incremental
  - ▶ Feed rate for pecking Q11
  - ▶ Feed rate for milling Q12
  - ▶ Coarse roughing tool number Q18
  - ▶ Feed rate for reciprocation Q19



## FLOOR FINISHING (23)

During finishing, the surface is machined parallel to the contour and to the depth previously entered under ALLOWANCE FOR FLOOR.

- ▶ CYCL DEF: Select Cycle 23 FLOOR FINISHING
  - ▶ Feed rate for pecking Q11
  - ▶ Feed rate for milling Q12



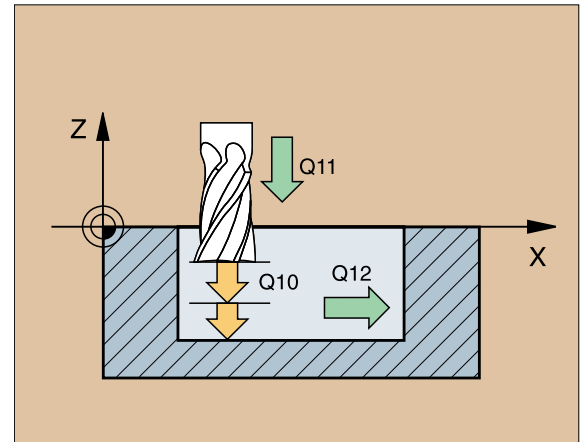
## SIDE FINISHING (24)

Finishing the individual contour elements

- ▶ CYCL DEF: Select Cycle 24 SIDE FINISHING
  - ▶ Direction of rotation? Clockwise = -1 Q9:
    - Clockwise Q9 = -1
    - Counterclockwise Q9 = +1
  - ▶ Pecking depth Q10; incremental
  - ▶ Feed rate for pecking Q11
  - ▶ Feed rate for milling Q12
  - ▶ Finishing allowance for side Q14: Allowance for finishing in several passes



- The sum of Q14 + finishing mill radius must be smaller than the sums Q3 (Cycle 20) + roughing tool radius!
- Call Cycle 22 ROUGH-OUT before calling Cycle 24!



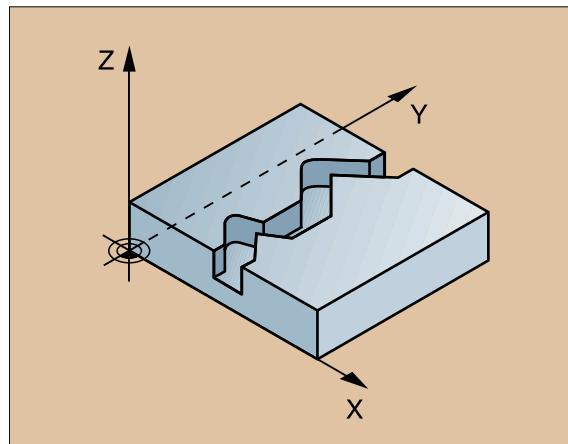
## CONTOUR TRAIN (25)

This cycle is for entering data for machining an open contour that has been defined in a contour subprogram.

- ▶ CYCL DEF: Select Cycle 25 CONTOUR TRAIN
  - ▶ Milling depth Q1; incremental
  - ▶ Allowance for side Q3:
    - Finishing allowance in the working plane
  - ▶ Workpiece surface coordinates Q5:
    - Coordinates referenced to the workpiece datum; absolute
  - ▶ Clearance height Q7:
    - Height at which the tool cannot collide with the workpiece; absolute
  - ▶ Pecking depth Q10; incremental
  - ▶ Feed rate for pecking Q11
  - ▶ Feed rate for milling Q12
  - ▶ Climb or up-cut ? Up-cut = -1 Q15
    - Climb milling: Q15 = +1
    - Up-cut milling: Q15 = -1
    - Alternately in reciprocating cuts: Q15 = 0



- Cycle 14 CONTOUR can have only one label number.
- A subprogram can hold no more than 128 line segments.



## CYLINDER SURFACE (27)



This cycle requires a center-cut end mill (ISO 1641)!

Cycle 27 CYLINDER SURFACE enables you to program a cylindrical contour in only two axes, as if in a plane. The TNC then rolls it onto a cylindrical surface.

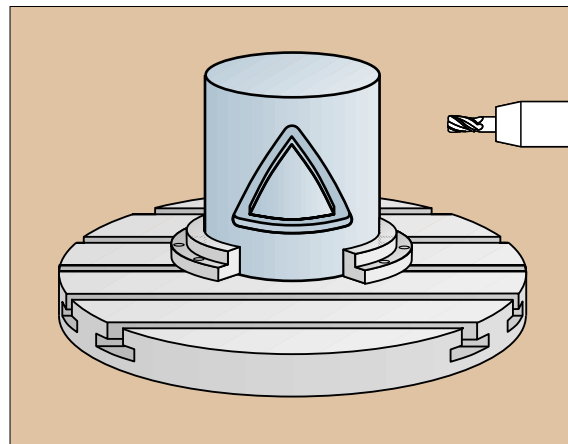
- ▶ Define a contour in a subprogram and list it in Cycle 14 CONTOUR GEOMETRY
- ▶ CYCL DEF: Select Cycle 27 CYLINDER SURFACE
  - ▶ Milling depth Q1
  - ▶ Finishing allowance for side Q3: Enter the finishing allowance (Either  $Q3 > 0$  or  $Q3 < 0$ )
  - ▶ Set-up clearance ? Q6: Distance from the tool to the workpiece
  - ▶ Plunging depth Q10
  - ▶ Feed rate for plunging Q11
  - ▶ Feed rate for milling Q12
  - ▶ Cylinder radius Q16: Radius of the cylinder
  - ▶ Dimension type? Deg=0 mm/inch=1 Q17: You can enter coordinates in the subprogram in degrees or millimeters



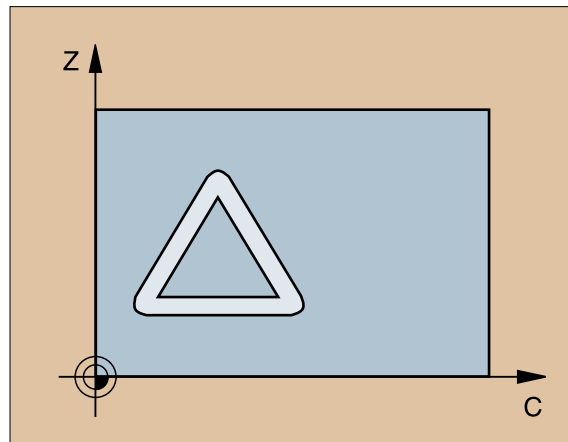
The machine and TNC must be prepared for the CYLINDER SURFACE cycle by the machine tool builder!



- The workpiece must be set up concentrically on the rotary table!
- The tool axis must be perpendicular to the axis of the rotary table!
- Cycle 14 CONTOUR GEOMETRY can have only one label number!
- A subprogram can hold no more than 128 line segments!



▼ The unrolled contour





## CYLINDER SURFACE (28)



This cycle requires a center-cut end mill (ISO 1641)!

Cycle 28 CYLINDER SURFACE enables you to program a slot in only two axes and then machine it on a cylindrical surface without distorting the angle of the slot walls.

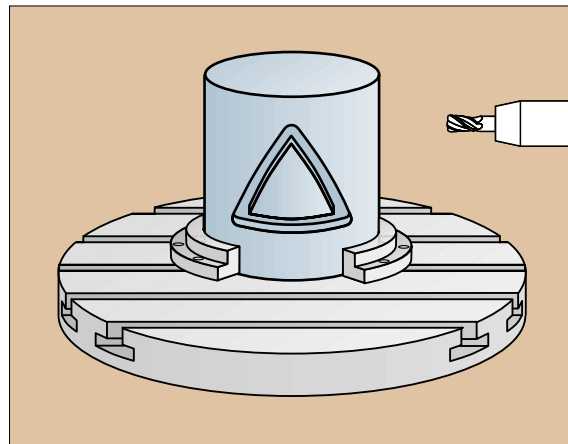
- ▶ Define a contour in a subprogram and list it in Cycle 14 CONTOUR GEOMETRY.
- ▶ CYCL DEF: Select Cycle 28 CYLINDER SURFACE
  - ▶ Milling depth Q1
  - ▶ Finishing allowance for side Q3: Enter the finishing allowance ( $Q3 > 0$  or  $Q3 < 0$ )
  - ▶ Set-up clearance Q6: Distance from the tool to the workpiece surface
  - ▶ Plunging depth Q10
  - ▶ Feed rate for plunging Q11
  - ▶ Feed rate for milling Q12
  - ▶ Cylinder radius Q16: Radius of the cylinder
  - ▶ Dimension type? Deg=0 mm/inch=1 Q17: Coordinates in the subprogram in degrees or millimeters
  - ▶ Slot width Q20



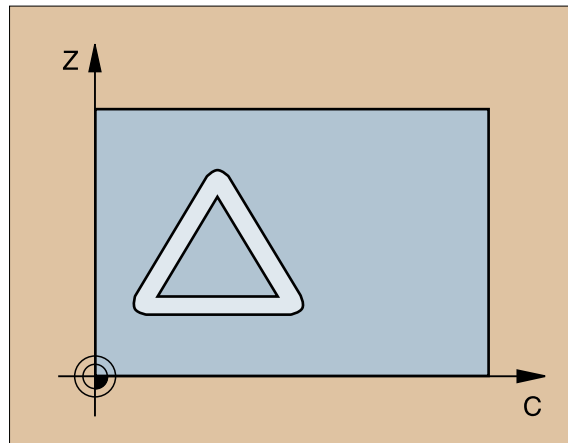
- The machine and TNC must be prepared for the CYLINDER SURFACE CYCLE by the machine tool builder!



- The workpiece must be set up concentrically on the table!
- The tool axis must be perpendicular to the rotary table axis!
- Cycle 14 CONTOUR GEOMETRY can have only one label number!
- A subprogram can hold no more than 128 line segments!



▼ The unrolled contour



# Multipass Milling

## RUN DIGITIZED DATA (30)



This cycle requires a center-cut end mill as per ISO 1641!

- ▶ CYCL DEF: Select Cycle 30 RUN DIGITIZED DATA
  - ▶ pgm name for digitized data
  - ▶ MIN. point range
  - ▶ MAX. point range
  - ▶ Set-up clearance: (A)
  - ▶ Pecking depth: (C)
  - ▶ Feed rate for pecking: (D)
  - ▶ Feed rate: (B)
  - ▶ Miscellaneous function M

7 CYCL DEF 30.0 RUN DIGITIZED DATA

8 CYCL DEF 30.1 PROGRAM1

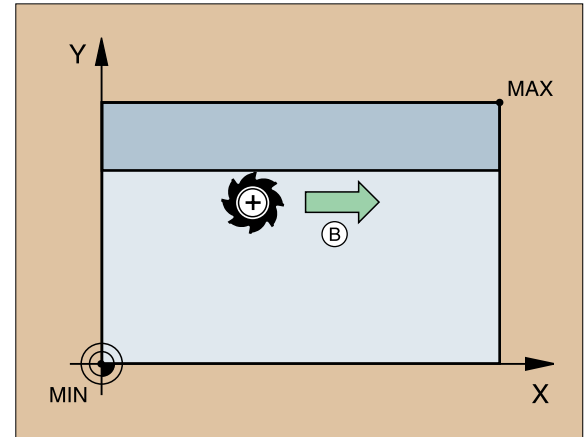
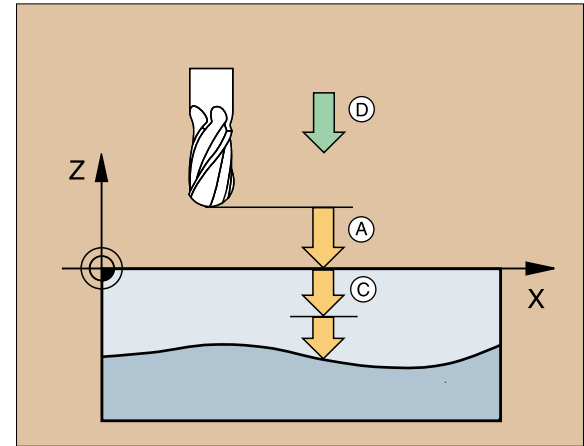
9 CYCL DEF 30.2 X+0 Y+0 Z-35

10 CYCL DEF 30.3 X+250 Y+125 Z+15

11 CYCL DEF 30.4 SET UP 2

12 CYCL DEF 30.5 PECKG 5 F125

13 CYCL DEF 30.6 F350 M112 T0.01 A+10



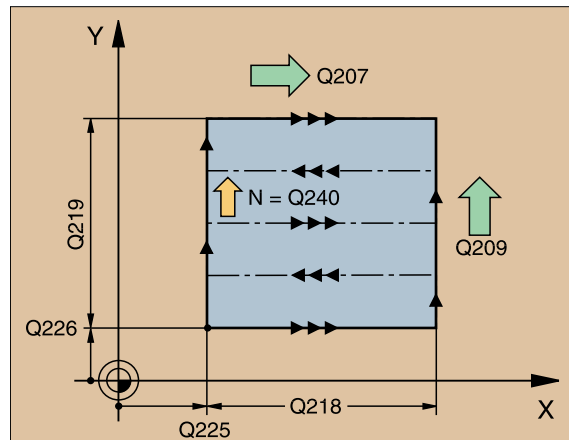
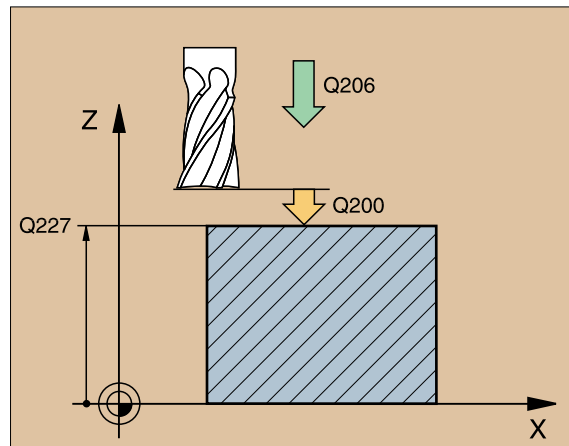
## MULTIPASS MILLING (230)



From the current position, the TNC positions the tool automatically at the starting point of the first machining operation, first in the working plane and then in the tool axis. Pre-position the tool in such a way that there is no danger of collision with the workpiece or fixtures.

### ► CYCL DEF: Select Cycle 230 MULTIPASS MILLING

- Starting point in 1st axis: Q225
- Starting point in 2nd axis: Q226
- Starting point in 3rd axis: Q227
- First side length: Q218
- Second side length: Q219
- Number of cuts: Q240
- Feed rate for plunging: Q206
- Feed rate for milling: Q207
- Stepmover feed rate: Q209
- Set-up clearance: Q200



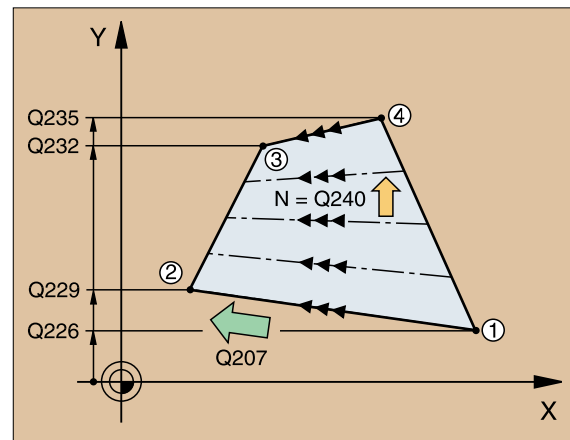
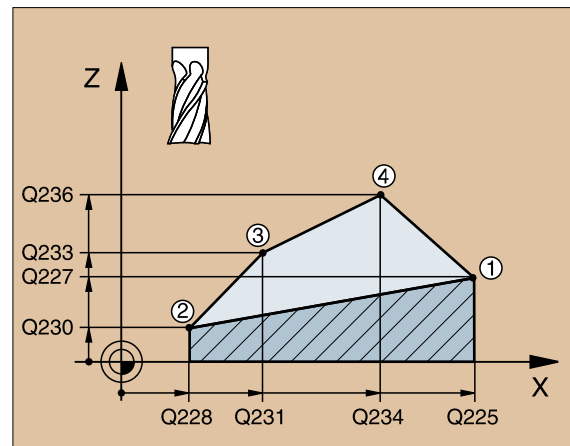
## RULED SURFACE (231)



Starting from the initial position, the TNC positions the tool at the starting point (point 1), first in the working plane and then in the tool axis.

### ► CYCL DEF: Select Cycle 231 RULED SURFACE

- Starting point in 1st axis: Q225
- Starting point in 2nd axis: Q226
- Starting point in 3rd axis: Q227
- 2nd point in 1st axis: Q228
- 2nd point in 2nd axis: Q229
- 2nd point in 3rd axis: Q230
- 3rd point in 1st axis: Q231
- 3rd point in 2nd axis: Q232
- 3rd point in 3rd axis: Q233
- 4th point in 1st axis: Q234
- 4th point in 2nd axis: Q235
- 4th point in 3rd axis: Q236
- Number of cuts: Q240
- Feed rate for milling: Q207

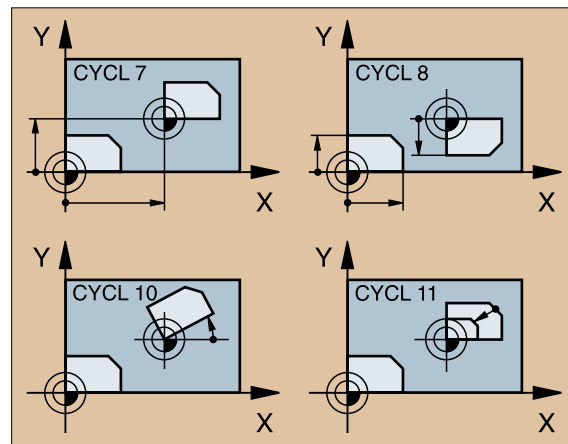


## Cycles for Coordinate Transformation

Cycles for coordinate transformation permit contours to be

• Shifted	Cycle 7 DATUM SHIFT
• Mirrored	Cycle 8 MIRRORIMAGE
• Rotated (in the plane)	Cycle 10 ROTATION
• Tilted out of the plane	Cycle 19 WORKING PLANE
• Enlarged or reduced	Cycle 11 SCALING
	Cycle 26 AXIS-SPECIFIC SCALING

Cycles for coordinate transformation are effective upon definition until they are reset or redefined. The original contour should be defined in a subprogram. Input values can be both absolute and incremental.



### DATUM SHIFT (7)

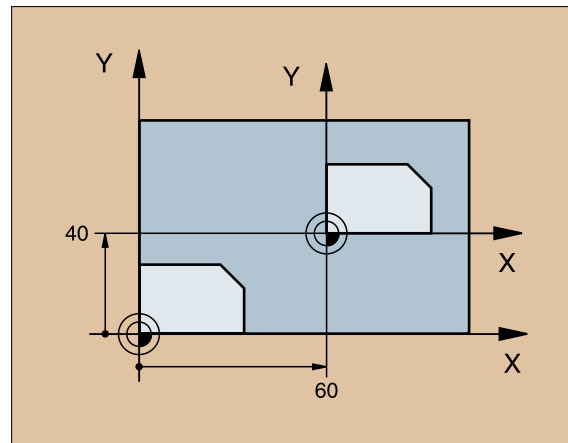
- CYCL DEF: Select Cycle 7 DATUM SHIFT
  - Enter the coordinates of the new datum or the number of the datum from the datum table.

To cancel a datum shift: Re-enter the cycle definition with the input value 0.

<b>9 CALL LBL1</b>	Call the part subprogram
<b>10 CYCL DEF 7.0 DATUM SHIFT</b>	
<b>11 CYCL DEF 7.1 X+60</b>	
<b>12 CYCL DEF 7.2 Y+40</b>	
<b>13 CALL LBL1</b>	Call the part subprogram



When combining transformations, the datum shift must be programmed before the other transformations!



## DATUM SETTING (247)

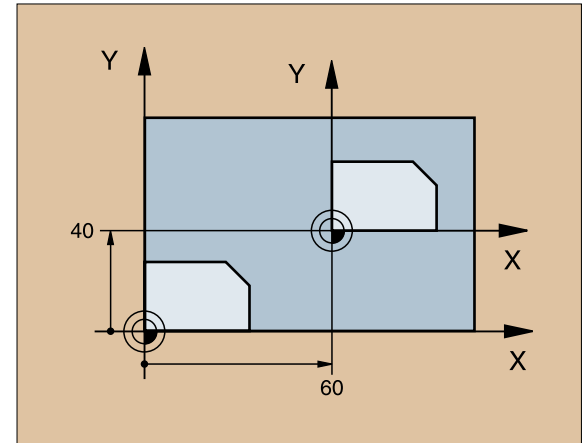
- CYCL DEF: Select Cycle 247 DATUM SETTING
  - Datum number: Enter the number from the active datum table containing the REF coordinates of the datum to be set.

### Reset

You can reactivate the datum that was last set in the Manual operating mode by entering the miscellaneous function M104.



- If required, activate the desired datum table with the NC block SEL TABLE.
- The TNC sets the datum only in the axes that are active in the datum table.
- Cycle 247 always interprets the values saved in the datum tables as coordinates relative to the machine datum. Machine parameter 7475 has no influence.



## MIRROR IMAGE (8)

- CYCL DEF: Select Cycle 8 MIRROR IMAGE
  - Enter the mirror image axis: Either X, Y, or both

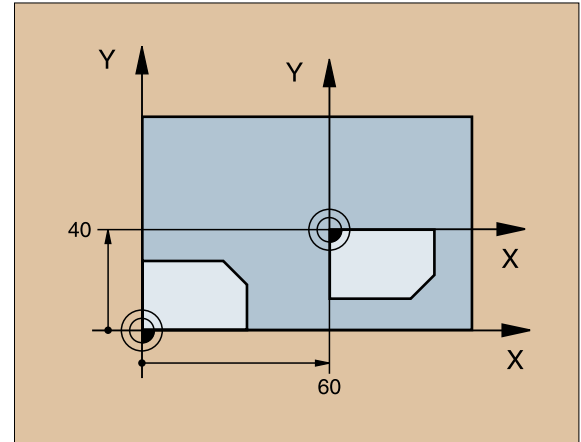
To reset the mirror image, re-enter the cycle definition with NO ENT.

```

15 CALL LBL1
16 CYCL DEF 7.0 DATUM SHIFT
17 CYCL DEF 7.1 X+60
18 CYCL DEF 7.2 Y+40
19 CYCL DEF 8.0 MIRROR IMAGE
20 CYCL DEF 8.1 Y
21 CALL LBL1
    
```



- The tool axis cannot be mirrored!
- The cycle always mirrors the original contour (in this example in subprogram LBL1)!



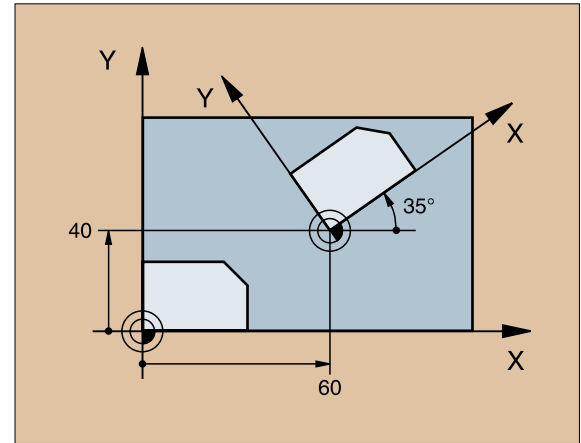
## Rotation (10)

- CYCL DEF: Select Cycle 10 ROTATION
  - Enter the rotation angle:
    - Input range  $-360^{\circ}$  to  $+360^{\circ}$
    - Reference axes for the rotation angle

Working plane	Reference axis and 0° direction
X/Y	X
Y/Z	Y
Z/X	Z

To reset a ROTATION, re-enter the cycle with the rotation angle 0.

```
12 CALL LBL1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 ROTATION
17 CYCL DEF 10.1 ROT+35
18 CALL LBL1
```

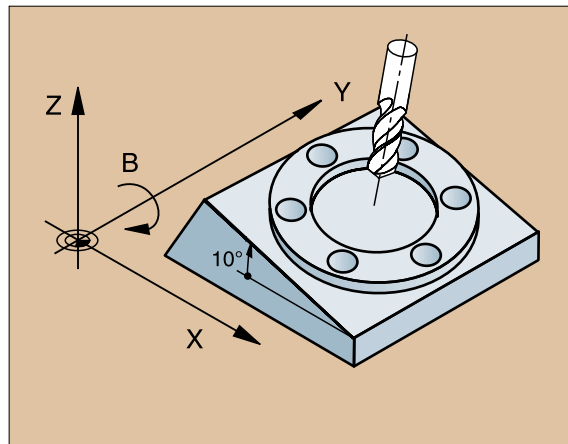




## WORKING PLANE (19)

Cycle 19 WORKING PLANE supports machining operations with a swivel head and/or tilting table.

- ▶ Call the tool
- ▶ Retract the tool in the tool axis (to prevent collision)
- ▶ If required, use an L block to position the rotary axes to the desired angle
- ▶ CYCL DEF: Select Cycle 19 WORKING PLANE
  - ▶ Enter the tilt angle of the corresponding axis or angle in space
  - ▶ If required, enter the feed rate of the rotary axes during automatic positioning
  - ▶ If required, enter the setup-clearance
- ▶ Activate compensation: move all the axes
- ▶ Program the contour as if the plane were not tilted



To cancel the WORKING PLANE cycle, re-enter the cycle definition with a 0° angle.



The machine and TNC must be prepared for the WORKING PLANE cycle by the machine tool builder!

```

4 TOOL CALL 1 Z S2500
5 L Z+350 R0 FMAX
6 L B+10 C+90 R0 FMAX
7 CYCL DEF 19.0 WORKING PLANE
8 CYCL DEF 19.1 B+10 C+90
9 L Z+200 R0 F1000
10 L X-50 Y-50 R0
  
```

## SCALING (11)

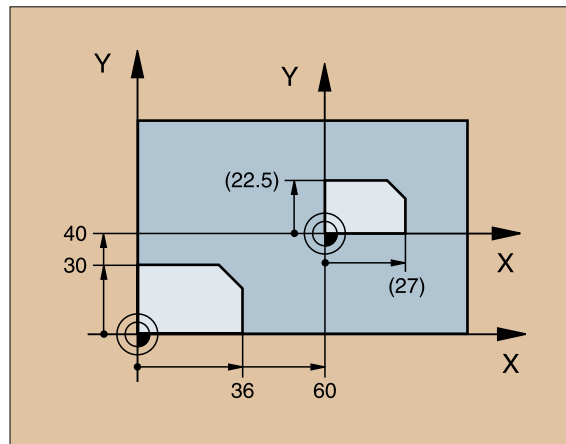
- CYCL DEF: Select Cycle 11 SCALING
  - Enter the scaling factor (SCL):
    - Input range 0.000001 to 99.999999:
      - To reduce the contour ...  $SCL < 1$
      - To enlarge the contour ...  $SCL > 1$

To cancel the SCALING, re-enter the cycle definition with SCL1.

```
11 CALL LBL1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL1
```



SCALING can be effective in the working plane only or in all three main axes (depending on machine parameter 7410)!



## AXIS-SPECIFIC SCALING (26)

- ▶ CYCL DEF: Select Cycle 20 AXIS-SPEC. SCALING
  - ▶ Axis and factor: Coordinate axes and factors for extending or compressing contour dimensions
  - ▶ Centerpoint coord. of extension: Center of the extension or compression

To cancel the AXIS-SPEC. SCALING, re-enter the cycle definition assigning the factor 1 to the affected axes.



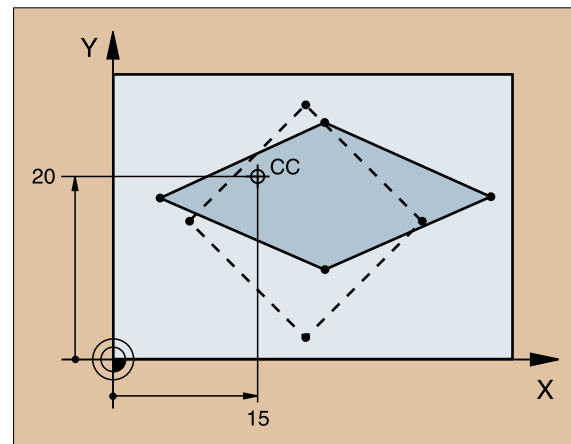
Coordinate axes sharing coordinates for arcs must be extended or compressed by the same scaling factor!

```
25 CALL LBL1
```

```
26 CYCL DEF 26.0 AXIS-SPEC. SCALING
```

```
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
```

```
28 CALL LBL1
```



# Special Cycles

## DWELL TIME (9)

The program run is interrupted for the duration of the DWELL TIME.

- ▶ CYCL DEF: Select cycle 9 DWELL TIME
  - ▶ Enter the dwell time in seconds

```
48 CYCL DEF 9.0 DWELL TIME
```

```
49 CYCL DEF 9.1 DWELL 0.5
```

## PGM CALL (12)

- ▶ CYCL DEF: Select cycle 12 PGM CALL
  - ▶ Enter the name of the program that you wish to call

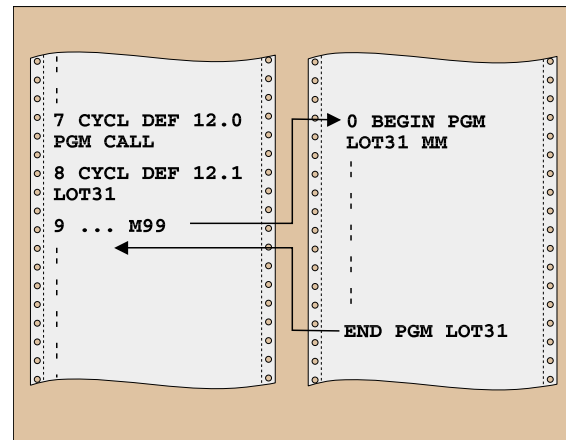
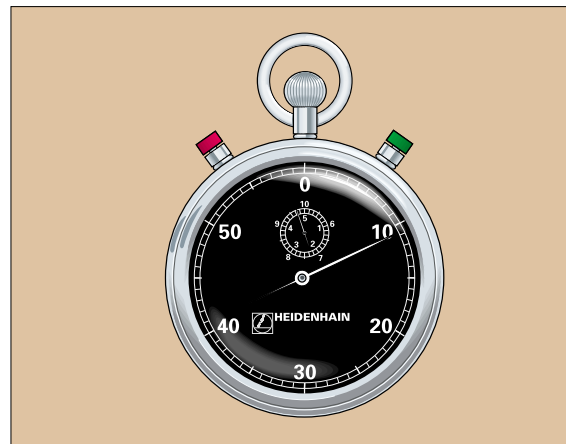


Cycle 12 PGM CALL must be called to become active!

```
7 CYCL DEF 12.0 PGM CALL
```

```
8 CYCL DEF 12.1 LOT31
```

```
9 L X+37.5 Y-12 R0 FMAX M99
```



## Spindle ORIENTATION

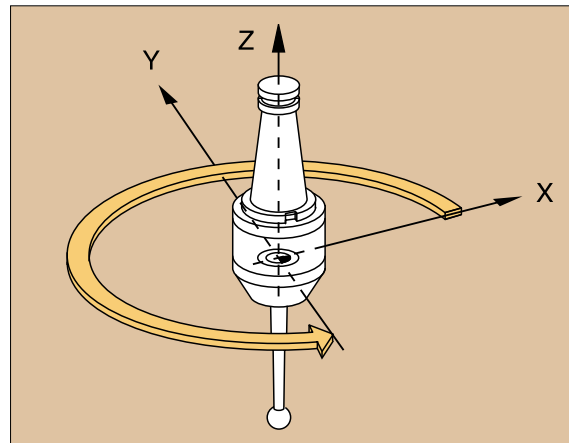
- ▶ CYCL DEF: Select cycle 13 ORIENTATION
  - ▶ Enter the orientation angle referenced to the angle reference axis of the working plane:
    - Input range 0 to 360°
    - Input resolution 0.1°
- ▶ Call the cycle with M19 or M20



The machine and TNC must be prepared for spindle ORIENTATION by the machine tool builder!

**12 CYCL DEF 13.0 ORIENTATION**

**13 CYCL DEF 13.1 ANGLE 90**



## TOLERANCE (32)



The machine and the TNC must be specially prepared for fast contour milling by the machine tool builder!

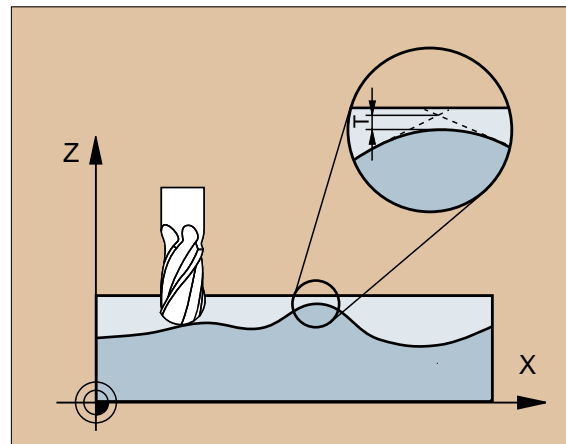


Cycle 32 TOLERANCE is effective as soon as it is defined in the part program!

The TNC automatically smooths the contour between any (compensated or uncompensated) contour elements. The tool therefore moves continuously on the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be run at the **fastest possible** speed and without "jerk".

A contour deviation results from the smoothing out. The size of this deviation (TOLERANCE VALUE) is set in a machine parameter by the machine manufacturer. You can change the pre-set tolerance value with Cycle 32 (see figure at top right).

- CYCL DEF: Select Cycle 32 TOLERANCE
  - Tolerance T: permissible contour deviation in mm



# Digitizing 3D Surfaces



The machine and TNC must be prepared for digitizing by the machine tool builder!

The TNC features the following cycles for digitizing with a measuring touch probe:

- Fix the scanning range: TCH PROBE 5 RANGE  
TCH PROBE 15 RANGE
- Digitize in reciprocating lines: TCH PROBE 16 MEANDER
- Digitize level by level: TCH PROBE 17 CONTOUR LINES
- Digitize in unidirectional lines: TCH PROBE 18 LINE

The digitizing cycles can be programmed only in plain language dialog. They can be programmed for the main axes X, Y and Z as well as for the rotary axes A, B and C.



- Digitizing is not possible while coordinate transformations or a basic rotation is active!
- Digitizing cycles need not be called. They are effective immediately upon definition!

## Selecting digitizing cycles



- Call an overview of touch probe functions



- Select digitizing cycles



- e.g. select Cycle 15

## Digitizing Cycle RANGE (5)

- ▶ Define the data transmission interface
- ▶ TOUCH PROBE: Select Cycle 5 RANGE
  - ▶ PGM name for digitized data: Enter a name for the NC program in which the digitized data should be stored.
  - ▶ Tch probe axis: Enter the axis of the touch probe
  - ▶ MIN. point range
  - ▶ MAX. point range
  - ▶ Clearance height: Height at which the stylus cannot collide with the model surface:  $Z_s$

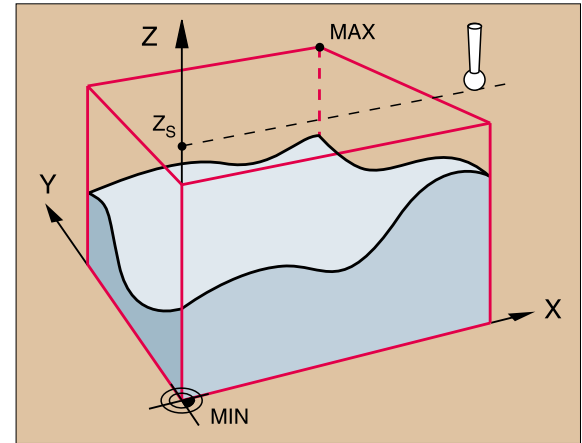
5 TCH PROBE 5.0 RANGE

6 TCH PROBE 5.1 PGM NAME: DIGI1

7 TCH PROBE 5.2 Z X+0 Y+0 Z+0

8 TCH PROBE 5.3 X+100 Y+100 Z+20

9 TCH PROBE 5.4 HEIGHT: +100





## Digitizing Cycle RANGE (15)

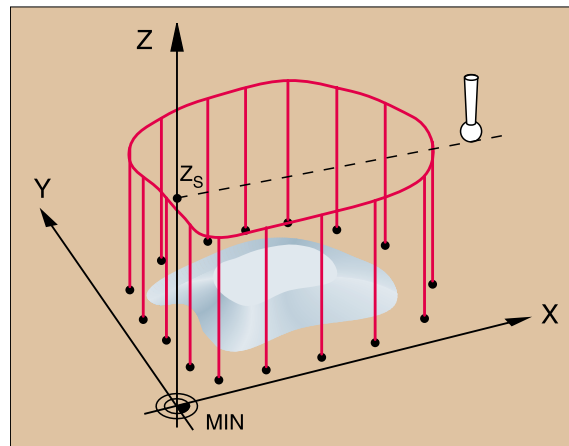
- ▶ Define the data transmission interface
- ▶ TOUCH PROBE: Select Cycle 15 RANGE
  - ▶ PGM name for digitized data: Enter a name for the NC program in which the digitized data should be stored.
  - ▶ Tch probe axis: Enter the axis of the touch probe
  - ▶ PGM name for range data: The name of the point table in which the range is defined
  - ▶ MIN point TCH PROBE axis: The minimum point in the touch probe axis
  - ▶ MAX point TCH PROBE axis: The maximum point in the touch probe axis
  - ▶ Clearance height: Height at which the stylus cannot collide with the model surface:  $Z_s$

5 TCH PROBE 15.0 RANGE

6 TCH PROBE 15.1 PGM DIGIT.: DATA

7 TCH PROBE 15.2 Z PGM RANGE: TAB1

8 TCH PROBE 15.3 MIN:+0 MAX:+35 HEIGHT:+125



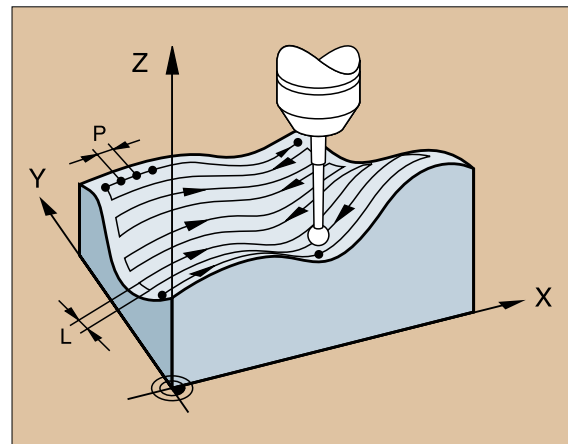
## Digitizing Cycle MEANDER (16)

Cycle 16 MEANDER is for digitizing a 3D contour in a series of **back-and-forth line movements**.

- ▶ Define Cycle 5 RANGE or 15 RANGE
- ▶ TOUCH PROBE: Select Cycle 16 MEANDER
  - ▶ Line direction: Coordinate axis in whose positive direction the probe moves after touching the first contour point
  - ▶ Scanning angle: Direction of touch probe traverse relative to the axis entered in line direction
  - ▶ Feed rate F: Maximum digitizing feed rate
  - ▶ Min. feed rate: Feed rate for scanning the first line
  - ▶ Feed rate reduction at edges: Distance at which the TNC begins to reduce the scanning feed rate before steep edges
  - ▶ Min. line spacing: Minimum distance moved forward to start the next line at steep surfaces
  - ▶ Line spacing: Max. distance moved forward to start the next line
  - ▶ Max. probe point interval
  - ▶ Tolerance value: The TNC suppresses the storage of probe points whose distance from a straight line defined by the last two stored points is less than the tolerance value.



- The line spacing and max. probe point interval cannot exceed 20 mm!
- Set a line direction that is as perpendicular as possible to steep surfaces!



▲ P: PP. INT = Probe point interval  
L: L. SPAC = Line spacing

**7 TCH PROBE 16.0 MEANDER**

**8 TCH PROBE 16.1 DIRECTN X ANGLE: +0**

**9 TCH PROBE 16.2 F1500 FMIN 500 MIN.L.SPAC:0.2**

**L.SPAC:0.5 PP.INT:0.5 TOL:0.1 DIST 0.5**

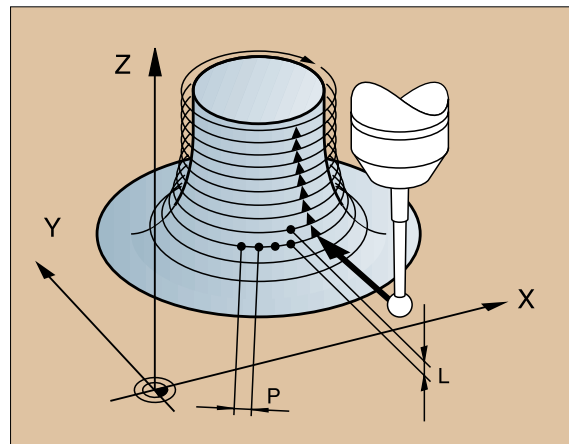
## Digitizing Cycle **CONTOUR LINES (17)**

Cycle 17 **CONTOUR LINES** enables you to digitize a 3D surface **level by level**.

- ▶ Define Cycle 5 RANGE or 15 RANGE
- ▶ **TOUCH PROBE:** Select Cycle 17 **CONTOUR LINES**
  - ▶ Time limit: If the touch probe has not orbited the model and returned to the first touch point within this time, the TNC will terminate the cycle. If you do not want a time limit, enter 0.
  - ▶ Starting point: Coordinates of the starting position
  - ▶ Axis and direction of approach: Coordinate axis and direction in which the probe approaches the model
  - ▶ Starting probe axis and direction: Coordinate axis and direction in which the probe begins scanning the model
  - ▶ Feed rate F: Maximum digitizing feed rate
  - ▶ Min. feed rate: Feed rate for scanning the first line
  - ▶ Feed rate reduction at edges: Distance at which the TNC begins to reduce the scanning feed rate before steep edges
  - ▶ Min. line spacing: Minimum height moved to start the next line at slightly inclined surfaces
  - ▶ Line spacing and direction: Maximum height moved to start the next contour line
  - ▶ Max. probe point interval
  - ▶ Tolerance value: The TNC suppresses the storage of probe points whose distance from a straight line defined by the last two stored points is less than the tolerance value.



The line spacing and max. probe point interval cannot exceed 20 mm!



▲ P: PP.INT = Probe point interval  
L: L.SPAC = Line spacing

10	TCH PROBE 17.0 CONTOUR LINES
11	TCH PROBE 17.1 TIME: 200 X+50 Y+0
12	TCH PROBE 17.2 ORDER Y+/X+
13	TCH PROBE 17.3 F1000 FMIN 400 MIN.L.SPAC:0.2
	L.SPAC:0.5 PP.INT:0.5 TOL:0.1 DIST 0.5

## Digitizing Cycle LINE (18)

Cycle 18 LINE is for digitizing a 3D surface in **lines in one direction**. It was developed mainly for digitizing with rotary axes.

- ▶ Define Cycle 5 RANGE or 15 RANGE
- ▶ TOUCH PROBE: Select Cycle 18 LINE
  - ▶ Line direction: Coordinate axis of the digitizing lines.
  - ▶ Scanning angle: Direction of touch probe traverse relative to the axis entered in line direction
  - ▶ Height for feed rate reduction: Coordinate in the tool axis at which at the start of each line the TNC switches from rapid traverse to the probing feed rate.
  - ▶ Feed rate F: Maximum digitizing feed rate
  - ▶ Min. feed rate: Feed rate for scanning the first line
  - ▶ Feed rate reduction at edges: Distance at which the TNC begins to reduce the scanning feed rate before steep edges
  - ▶ Min. line spacing: Minimum distance moved forward to start the next line at steep surfaces
  - ▶ Line spacing an direction: Maximum distance moved to start the next line
  - ▶ Max. probe point interval
  - ▶ Tolerance value: The TNC suppresses the storage of probe points whose distance from a straight line defined by the last two stored points is less than the tolerance value.



The line spacing and max. probe point interval cannot exceed 20 mm!

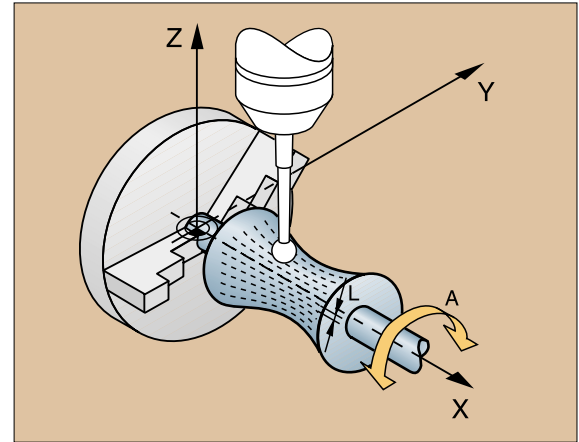
**10 TCH PROBE 18.0 LINE**

**11 TCH PROBE 18.1 DIRECTN X**

**ANGLE:+0 HEIGHT:+125**

**12 TCH PROBE 18.2 F1000 FMIN 400 MIN.L.SPAC:0.2**

**L.SPAC:0.5 PP.INT:0.5 TOL:0.1 DIST 0.5**



# Graphics and Status Displays



See "Graphics and Status Displays"

## Defining the Workpiece in the Graphic Window

The dialog prompt for the BLK-FORM appears automatically whenever you create a new part program.

- ▶ Create a new program or, if you are already in a program, press the soft key BLK FORM
  - ▶ Spindle axis
  - ▶ MIN and MAX point

The following is a selection of frequently needed functions.

## Interactive Programming Graphics



Select the PGM+GRAPHICS screen layout!

The TNC can generate a two-dimensional graphic of the contour while you are programming it:



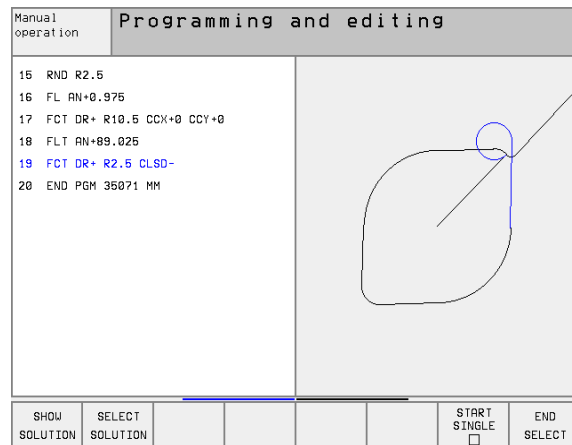
- ▶ Automatic graphic generation during programming



- ▶ Manually start graphic generation



- ▶ Generate interactive graphics blockwise



## Test Graphics and Program Run Graphics



Select the GRAPHICS or PGM+GRAPHICS screen layout!

In the test run and program run modes the TNC can graphically simulate the machining process. The following display types are available via soft key:



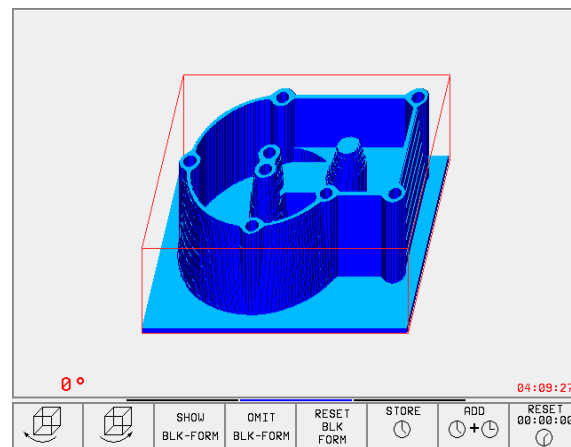
► Plan view



► Projection in three planes



► 3D view



## Status Displays



Select the PGM+STATUS or POSITION+STATUS screen layout!

In the program run modes a window in the lower part of the screen shows information on

- Tool position
- Feed rate
- Active miscellaneous functions

Further status information is available via soft key for display in an additional window:

STATUS PGM	► Program information
STATUS POS.	► Tool positions
STATUS TOOL	► Tool data
STATUS COORD. TRANSF.	► Coordinate transformations
STATUS TOOL PROBE	► Tool measurement
STATUS OF M FUNCT.	► Active miscellaneous functions M

Program run, single block						Programming and editing	
<pre> 0 BEGIN PGM FK1 MM 1 BLK FORM 0.1 Z X+0 Y+0 Z-20 2 BLK FORM 0.2 X+100 Y+100 Z+0 3 TOOL CALL 1 Z 4 L Z+250 R0 F MAX 5 L X-20 Y+30 R0 F MAX 6 L Z-10 R0 F1000 M3 7 APPR CT X+2 Y+30 CCR90 R+5 RL F250 8 FC DR- R10 CLSD+ CCK+20 CCY+30           </pre>						DIST. X +0.000 Y +0.000 Z +0.000 ↓ B +0.362 ↓ C +0.076	
						A +0.0000 B +100.0000 C +90.0000	
Basic rotation +0.0000							
0% S-IST 17:2 4% S-MGM LIMIT 1							
+0.224 Y +295.720 Z +24.447 +B -0.477+c +8.439 S 257.122							
ACTL.  T 5 Z S 1300 F 0 M 0/0							
STATUS PGM	STATUS POS.	STATUS TOOL	STATUS COORD. TRANSF.	STATUS TOOL PROBE	STATUS OF M FUNCT.		

# ISO Programming

## Programming Tool Movements with Cartesian Coordinates

<b>G00</b>	Linear motion in rapid traverse
<b>G01</b>	Linear motion
<b>G02</b>	Circular motion, clockwise
<b>G03</b>	Circular motion, counterclockwise
<b>G05</b>	Circular motion without directional data
<b>G06</b>	Circular movement with tangential contour connection
<b>G07*</b>	Paraxial positioning block

## Programming Tool Movements with Polar Coordinates

<b>G10</b>	Linear motion in rapid traverse
<b>G11</b>	Linear motion
<b>G12</b>	Circular motion, clockwise
<b>G13</b>	Circular motion, counterclockwise
<b>G15</b>	Circular motion without directional data
<b>G16</b>	Circular movement with tangential contour connection

\*) Effective blockwise

## Drilling Cycles

<b>G83</b>	Pecking
<b>G200</b>	Drilling
<b>G201</b>	Reaming
<b>G202</b>	Boring
<b>G203</b>	Universal boring
<b>G204</b>	Back boring
<b>G205</b>	Universal pecking
<b>G208</b>	Bore milling
<b>G84</b>	Tapping
<b>G206</b>	Tapping NEW
<b>G85</b>	Rigid tapping (controlled spindle)
<b>G207</b>	Rigid tapping (controlled spindle) NEW
<b>G86</b>	Thread cutting
<b>G209</b>	Tapping with chip breaking
<b>G262</b>	Thread milling
<b>G263</b>	Thread milling and countersinking
<b>G264</b>	Thread drilling and milling
<b>G265</b>	Helical thread drilling and milling
<b>G267</b>	Outside thread milling



### Pockets, Studs and Slots

- G75** Rectangular pocket milling, clockwise machining direction
- G76** Rectangular pocket milling, counterclockwise machining direction
- G212** Pocket milling
- G213** Stud milling
- G77** Circular pocket milling, clockwise machining direction
- G78** Circular pocket milling, counterclockwise machining direction
- G214** Circular pocket finishing
- G215** Circular stud finishing
- G74** Slot milling
- G210** Slot milling with reciprocating plunge
- G211** Circular slot

### Point Patterns

- G220** Circular point pattern
- G221** Linear point pattern

### SL Cycles, Group I

- G37** List of contour subprograms
- G56** Pilot drilling
- G57** Rough-out
- G58** Contour milling, clockwise
- G59** Contour milling, counterclockwise

### SL Cycles, Group II

- G37** List of contour subprograms
- G120** Contour data
- G121** Pilot drilling
- G122** Rough-out
- G123** Floor finishing
- G124** Side finishing
- G125** Contour train
- G127** Cylinder surface
- G128** Cylinder surface slot milling

### Multipass milling

- G60** Run digitized data
- G230** Multipass milling
- G231** Ruled surface

### Cycles for Coordinate Transformation

- G53** Datum shift from datum tables
- G54** Entering datum shift directly
- G247** Datum setting
- G28** Mirror image
- G73** Rotating the coordinate system
- G72** Scaling factor: enlarging/reducing contours
- G80** Working plane

**Special Cycles**

- G04\*** Dwell time  
**G36** Oriented spindle stop  
**G39** Designating a program as a cycle  
**G79\*** Cycle call

**Touch Probe Cycles**

- G55\*** Measure coordinate  
**G400\*** Basic rotation over 2 points  
**G401\*** Basic rotation over 2 holes  
**G402\*** Basic rotation over 2 studs  
**G403\*** Basic rotation over a rotary table  
**G404\*** Set basic rotation  
**G405\*** Basic rotation over rotary table, hole center

**Touch Probe Cycles**

- G410\*** Datum at center of rectangular pocket  
**G411\*** Datum at center of rectangular stud  
**G412\*** Datum at center of hole  
**G413\*** Datum at center of circular stud  
**G414\*** Datum at outside corner  
**G415\*** Datum at inside corner  
**G416\*** Datum at center of bolt hole circle  
**G417\*** Datum in touch probe axis  
**G418\*** Datum at center of 4 holes  
**G420\*** Measure angle  
**G421\*** Measure hole  
**G422\*** Measure circular stud  
**G423\*** Measure rectangular pocket  
**G424\*** Measure rectangular stud  
**G425\*** Measure slot width  
**G426\*** Measure ridge width  
**G427\*** Measure any coordinate  
**G430\*** Measure bolt hole circle  
**G431\*** Measure plane  
**G440\*** Thermal compensation  
**G480\*** Calibrate TT  
**G481\*** Measuring tool length  
**G482\*** Measuring tool length  
**G483\*** Measuring tool length and radius

\*) Effective blockwise

### Defining the Working Plane

- G17** X/Y working plane, tool axis Z  
**G18** Z/X working plane, tool axis Y  
**G19** Y/Z working plane, tool axis X  
**G20** Fourth axis is tool axis

### Chamfer, Rounding, Approach/Departure

- G24\*** Chamfer with side length R  
**G25\*** Corner rounding with radius R  
**G26\*** Tangential contour approach on an arc with radius R  
**G27\*** Tangential contour departure on an arc with radius R

### Tool Definition

- G99\*** Tool definition in the program with length L and radius R

### Tool Radius Compensation

- G40** No radius compensation  
**G41** Radius compensation to the left of the contour  
**G42** Radius compensation to the right of the contour  
**G43** Paraxial radius compensation: the path is lengthened  
**G44** Paraxial radius compensation: the path is shortened

### Dimensional Data

- G90** Absolute dimensions  
**G91** Incremental (chain) dimensions

### Unit of Measure (at Beginning of Program)

- G70** Inches  
**G71** Millimeters

### Blank Form Definition for Graphics

- G30** Setting the working plane, MIN point coordinates  
**G31** Dimensional data (with G90, G91), coordinates of the MAX point

### Other G functions

- G29** Define last nominal position value as pole  
**G38** Stopping the program run  
**G51\*** Calling the next tool (only with central tool file)  
**G98\*** Setting a label number

---

\*) Effective blockwise

**Q Parameter Functions**

- D00** Assign a value directly
  - D01** Calculate and assign the sum of two values
  - D02** Calculate and assign the difference of two values
  - D03** Calculate and assign the product of two values
  - D04** Calculate and assign the quotient of two values
  - D05** Calculate and assign the root from a value
  - D06** Calculate and assign the sine of an angle in degrees
  - D07** Calculate and assign the cosine of an angle in degrees
  - D08** Calculate and assign the square root of the sum of two squares (Pythagorean theorem)
  - D09** If equal, jump to the given label
  - D10** If not equal, jump to the given label
  - D11** If greater than, jump to the given label
  - D12** If less than, jump to the given label
  - D13** Find and assign an angle from the arc tangent of two sides or from the sine and cosine of an angle
  - D14** Output text to screen
  - D15** Output text or parameter contents through the data interface
  - D19** Transfer numerical values or Q parameters to the PLC
-

## Addresses

<b>%</b>	Program beginning
<b>A</b>	Swiveling axis around X
<b>B</b>	Swiveling axis around Y
<b>C</b>	Rotary axis around Z
<b>D</b>	Define Q-parameter functions
<b>E</b>	Tolerance for rounding arc with M112
<b>F</b>	Feed rate in mm/min in positioning blocks
<b>F</b>	Dwell time in seconds with G04
<b>F</b>	Scaling factor with G72
<b>G</b>	G functions (see list of G functions)
<b>H</b>	Polar coordinate angle
<b>H</b>	Angle of rotation with G73
<b>I</b>	X coordinate of the circle center or pole
<b>J</b>	Y coordinate of the circle center or pole
<b>K</b>	Z coordinate of the circle center or pole
<b>L</b>	Label number with G98
<b>L</b>	Jump to a label number
<b>L</b>	Tool length with G99
<b>M</b>	Miscellaneous function
<b>N</b>	Block number
<b>P</b>	Cycle parameter for fixed cycles
<b>P</b>	Value or Q parameter with Q parameter definitions
<b>Q</b>	Variable Q parameter

<b>R</b>	Polar coordinate radius with G10/G11/G12/ G13/G15/G16/
<b>R</b>	Circle radius with G02/G03/G05
<b>R</b>	Corner radius with G25/G26/G27
<b>R</b>	Chamfer length with G24
<b>R</b>	Tool radius with G99
<b>S</b>	Spindle speed in rpm
<b>S</b>	Angle for spindle orientation with G36
<b>T</b>	Tool number with G99
<b>T</b>	Tool call
<b>T</b>	Call next tool with G51
<b>U</b>	Parallel axis to X
<b>V</b>	Parallel axis to Y
<b>W</b>	Parallel axis to Z
<b>X</b>	X axis
<b>Y</b>	Y axis
<b>Z</b>	Z axis
<b>*</b>	Character for end of block

## Miscellaneous Functions M

<b>M00</b>	Stop program run/Stop spindle/Coolant off
<b>M01</b>	Optional program stop
<b>M02</b>	Stop program run/Stop spindle/Coolant off Jump back to block 1/Clear status display (depending on machine parameters)
<b>M03</b>	Spindle on clockwise
<b>M04</b>	Spindle on counterclockwise
<b>M05</b>	Optional stop
<b>M06</b>	Tool change/Stop program run (depending on machine parameters) Stop spindle
<b>M08</b>	Coolant on
<b>M09</b>	Coolant off
<b>M13</b>	Spindle on clockwise/Coolant on
<b>M14</b>	Spindle on counterclockwise/Coolant on
<b>M30</b>	Same function as M02
<b>M89</b>	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameters)
<b>M90</b>	Constant contour speed at corners (effective only in lag mode)
<b>M91</b>	Within the positioning block: Coordinates are referenced to the machine datum
<b>M92</b>	Within the positioning block: The coordinates are referenced to a position defined by the machine tool builder

<b>M93</b>	Reserved
<b>M94</b>	Reduce rotary axis display to a value below 360°
<b>M95</b>	Reserved
<b>M96</b>	Reserved
<b>M97</b>	Machine small contour steps
<b>M98</b>	Suspend tool path compensation
<b>M99</b>	Cycle call, effective blockwise
<b>M101</b>	Automatic tool change after tool lifetime expires
<b>M102</b>	Reset M101
<b>M103</b>	Reduce the feed rate during plunging to factor F
<b>M104</b>	Reactivate most recently defined datum
<b>M105</b>	Machine with second $k_v$ factor
<b>M106</b>	Machine with first $k_v$ factor
<b>M107</b>	See User's Manual
<b>M108</b>	Reset M107
<b>M109</b>	Constant contouring speed of tool cutting edge on arcs (increasing and decreasing the feed rate)
<b>M110</b>	Constant contouring speed of tool cutting edge on arcs (only decreasing the feed rate)
<b>M111</b>	Reset M109/M110
<b>M114</b>	Automatic compensation of machine geometry when working with tilting axes

---

<b>M115</b>	Reset M114
<b>M116</b>	Feed rate for rotary axes in mm/min
<b>M117</b>	Reset M116
<b>M118<sup>1)</sup></b>	Superimpose handwheel positioning during program run
<b>M120<sup>1)</sup></b>	LOOK AHEAD: Calculate the radius-compensated tool path ahead of time
<b>M126</b>	Permit zero crossover on 360° rotary axes
<b>M127</b>	Reset M126
<b>M128</b>	Retain position of tool tip when positioning tilting axes (TCPM) <sup>2)</sup>
<b>M129</b>	Reset M128
<b>M130<sup>1)</sup></b>	Within the positioning block: points are referenced to the non-tilted coordinate system
<b>M134</b>	Exact stop when positioning with rotary axes
<b>M135</b>	Reset M134
<b>M136</b>	Feed rate F in microns per spindle revolution
<b>M137</b>	Feed rate F in millimeters per minute

---



---

<b>M138</b>	Selection of tilted axes for M114, M128 and the tilted working plane cycle
<b>M140</b>	Retraction from the contour in the positive tool axis direction
<b>M141</b>	Suppress touch probe monitoring
<b>M142</b>	Delete modal program information
<b>M143</b>	Delete basic rotation
<b>M144</b>	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block
<b>M145</b>	Reset M144
<b>M200<sup>1)</sup></b>	Miscellaneous function for laser cutting machines
⋮	
<b>M204<sup>1)</sup></b>	See User's Manual

---

<sup>1)</sup> Only with conversational programming

<sup>2)</sup> TCPM: Tool Center Point Management


# HEIDENHAIN


---

## **DR. JOHANNES HEIDENHAIN GmbH**

Dr.-Johannes-Heidenhain-Straße 5

**83301 Traunreut, Germany**

 +49 (86 69) 31-0

 +49 (86 69) 50 61

e-mail: [info@heidenhain.de](mailto:info@heidenhain.de)

---

**Technical support**  +49 (86 69) 31-10 00

**Measuring systems**  +49 (86 69) 31-31 04

e-mail: [service.ms-support@heidenhain.de](mailto:service.ms-support@heidenhain.de)

**TNC support**  +49 (86 69) 31-31 01

e-mail: [service.nc-support@heidenhain.de](mailto:service.nc-support@heidenhain.de)

**NC programming**  +49 (86 69) 31-31 03

e-mail: [service.nc-pgm@heidenhain.de](mailto:service.nc-pgm@heidenhain.de)

**PLC programming**  +49 (86 69) 31-31 02

e-mail: [service.plc@heidenhain.de](mailto:service.plc@heidenhain.de)

**Lathe controls**  +49 (7 11) 95 28 03-0

e-mail: [service.hsf@heidenhain.de](mailto:service.hsf@heidenhain.de)

---

[www.heidenhain.de](http://www.heidenhain.de)