HEIDENHAIN

Pilot

TNC 410

NC-Software
286 060-xx

8/2000

The Pilot

... is your concise programming guide for the HEIDENHAIN TNC 410 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
- 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 the TNC 410 with the following software number:

Control	NC Software Number
TNC 410	286 060-xx

Contents

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	File type
Maximum length: 8 characters	see table at right

Files in the TNC	File type
Programs	
• in HEIDENHAIN format	.H
• in ISO format	.I
Table for	
• Tools	TOOL.T
• Tool pockets	TOOLP.TCH
• Datums	.D
• Points	.PNT

Creating a New Part Program

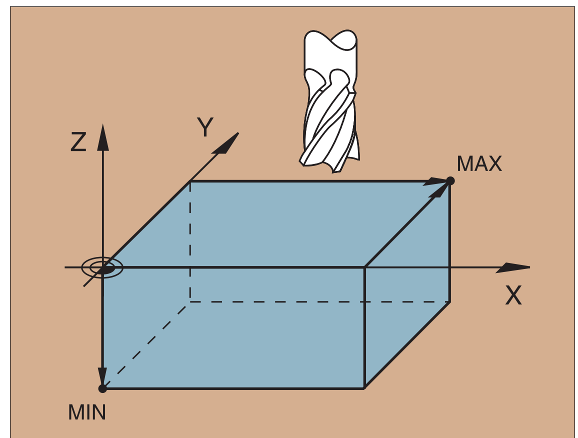
PGM MGT

- Enter new file name
- Select file type via soft key
- Select unit of measure for dimensions (mm or inches)

BLK FORM

- 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
```



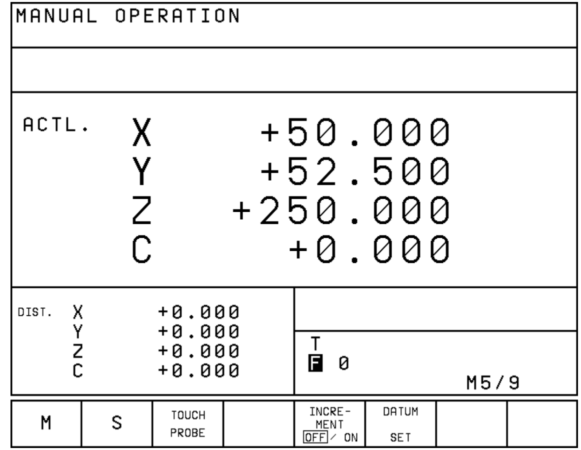
Choosing the screen layout

See "Introduction, the TNC 410"

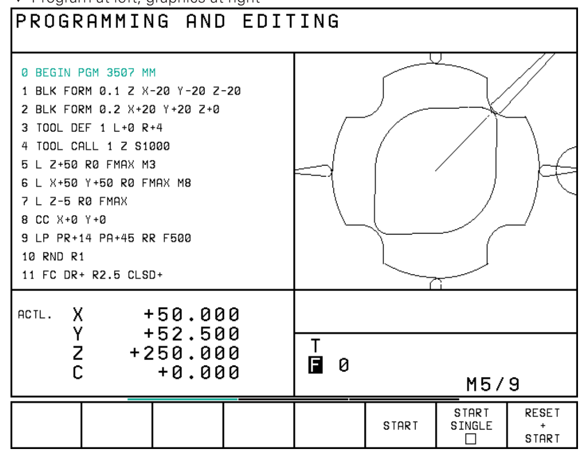
▶ Show soft keys for setting the screen layout

Mode of operation	Options	
Program run, full seq. Program run, single block Test run	Program	PGM
	Program at left Program information at right	PGM + PGM STATUS
	Program at left Additional position display at right	PGM + POS. STATUS
	Program at left Tool information at right	PGM + TOOL STATUS
	Program at left Active coordinate transformations at right	PGM + C.TRANS. STATUS
	Program at left Tool measurement information at right	PGM + T.PROBE STATUS

Continued ▶

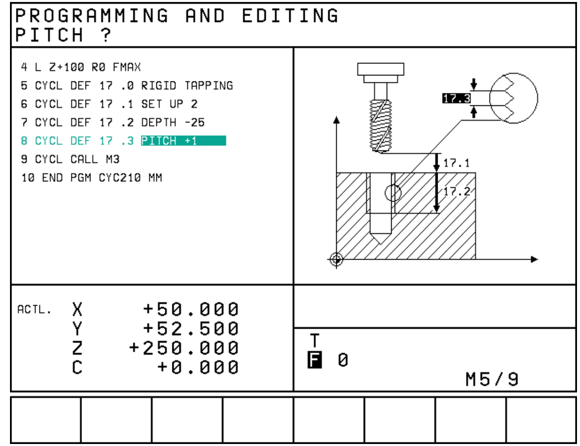


▲ Positions
▼ Program at left, graphics at right



Mode of operation	Options	
Programming and Editing	Program	PGM
	Programming graphics	GRAPHICS
	Program at left Programming graphics right	PGM + GRAPHICS
	Program at left Graphics illustrating input parameters at right	PGM + FIGURE

Mode of operation	Options	
Manuell operation Handwheel	Position	POSITION
	Position at left Program information at right	POSITION+ PGM STATUS
	Position at left Additional position display at right	POSITION+ POS.DISP. STATUS
	Position at left Tool information at right	POSITION+ TOOL STATUS
	Position at left Active coordinate transformations at right	POSITION+ C.TRANS. STATUS

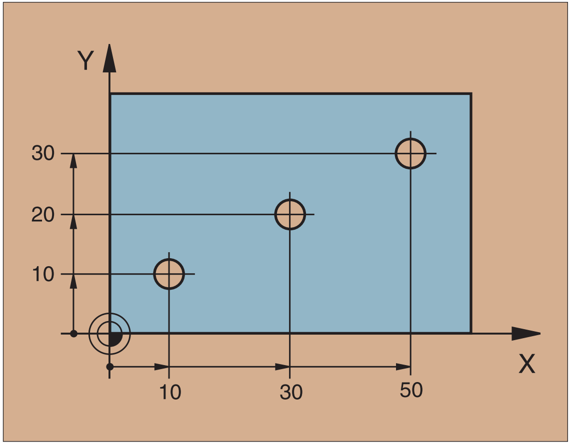


▲ Program at left, graphic support at right

Absolute Cartesian Coordinates

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

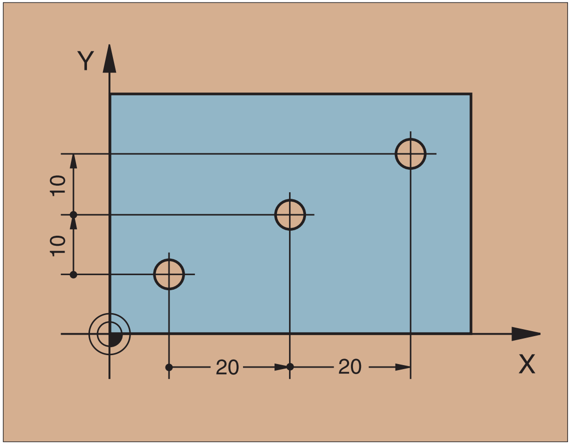
Programmable axes in an NC block
Linear motion: 5 axes
Circular motion: 2 linear axes in a plane or
3 linear axes with cycle 19 WORKING PLANE



Incremental Cartesian Coordinates

The dimensions are measured from the last programmed position of the tool.

The tool moves by the incremental coordinates.



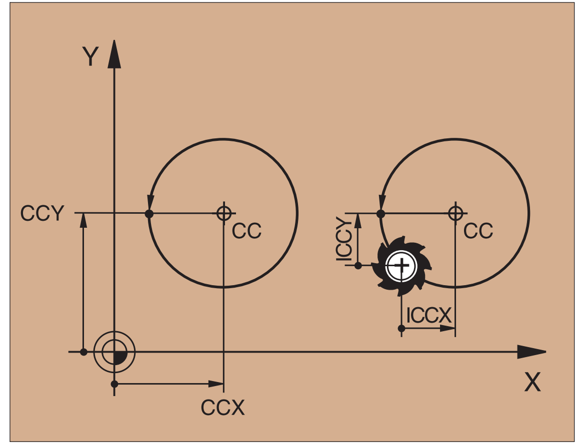
Circle Center and Pole: CC

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.

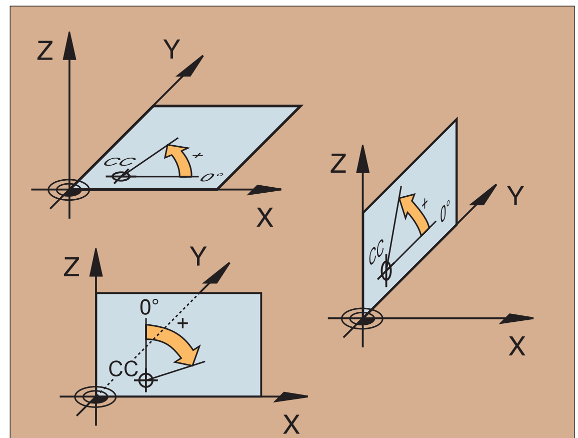
An incrementally defined circle center or pole is always measured from the last programmed position of the workpiece.



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

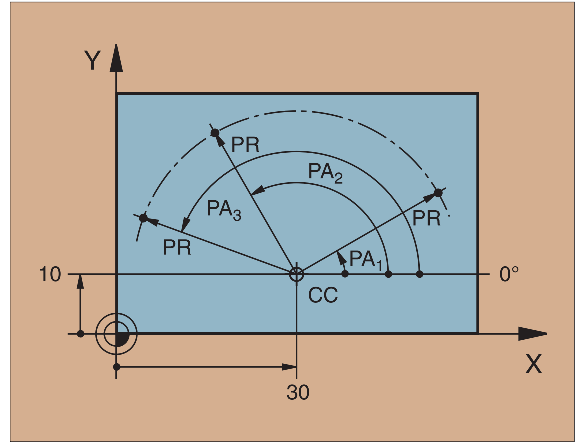
Incremental dimensions

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

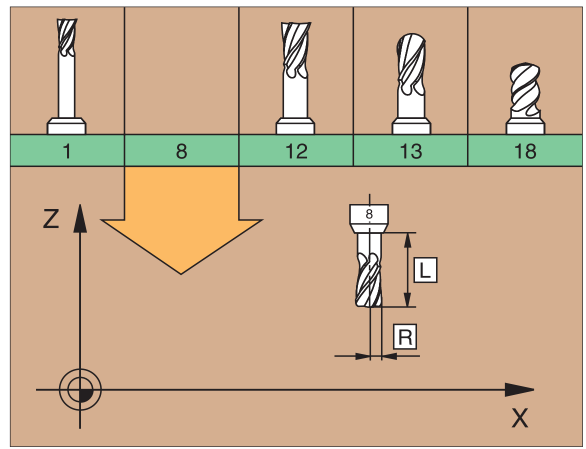
Tool data

Each tool is identified with a number between 1 and 254.

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)

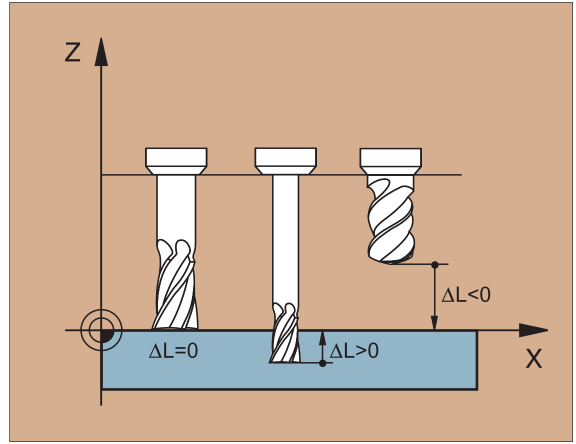


TOOL DEF

- Tool number
- Tool length L
- Tool radius R

- Program the tool length as its difference ΔL to the zero tool:
  ΔL>0: The tool is longer than the zero tool
  ΔL<0: The tool is shorter than the zero tool

- With a tool presetter you can measure the actual tool length, then program that length.



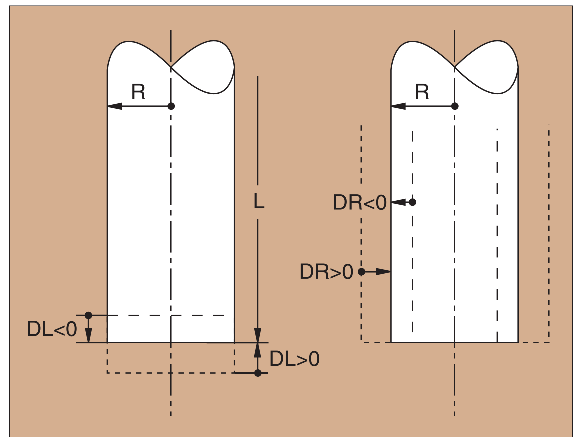
Calling the tool data

TOOL CALL

- Tool number
- Working spindle axis: tool axis
- Spindle speed S
- Tool length oversize DL (e.g. to compensate wear)
- Tool radius oversize DR (e.g. to compensate wear)

```
3 TOOL DEF 6 L+7.5 R+3
4 TOOL CALL 6 Z S2000 DL+1 DR+0.5
5 L Z+100 R0 FMAX
6 L X-10 Y-10 R0 FMAX M6
```

▼ Oversizes on an end mill



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 oversize for tool radius or length is ± 99.999 mm!

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:

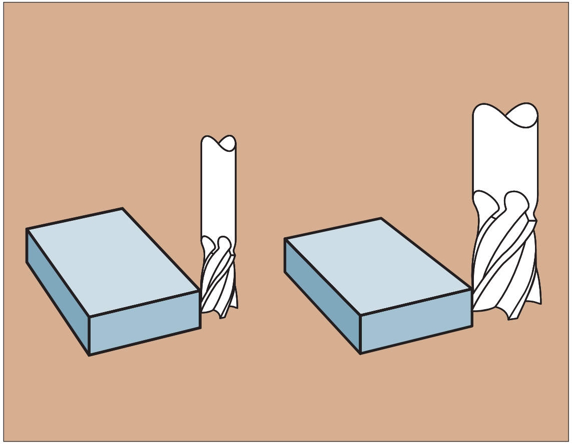
- Tool movement in the working plane with RR or RL

End of effect:

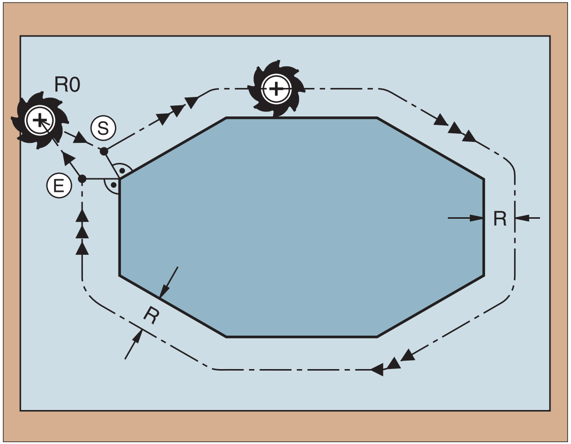
- Execution of a positioning block with R0

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

- Tool movement with R0



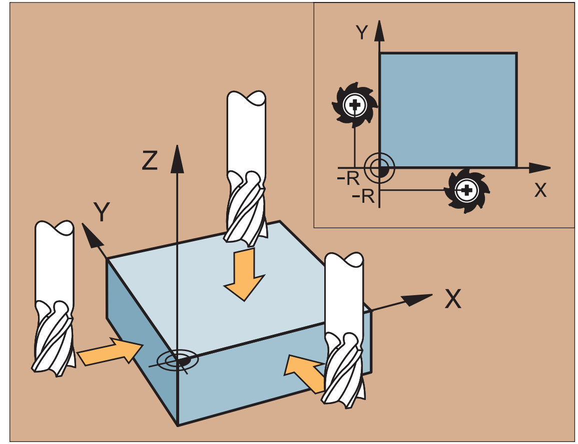
▼ S = Start; E = End



Datum Setting Without a 3D 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

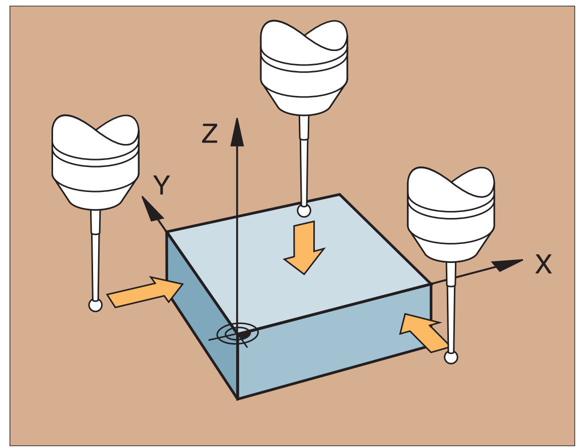


Datum Setting with a 3D Touch Probe

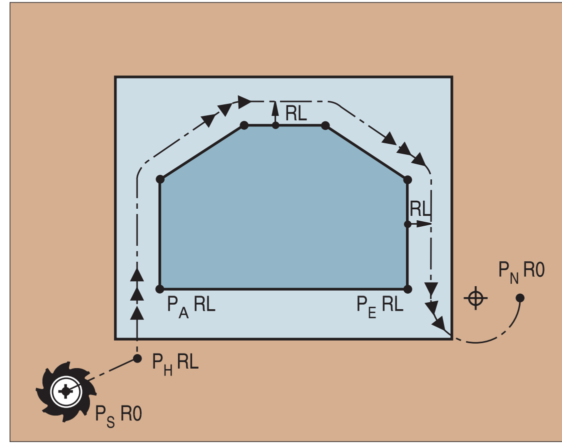
The fastest, simplest and most accurate way to set a datum is to use a HEIDENHAIN 3D touch probe.

The following probe functions are provided by the manual operation and electronic handwheel modes of operation:

PROBING ROT	Basic rotation
PROBING POS	Datum setting in one axis
PROBING P	Datum setting at a corner
PROBING CC	Datum setting at a circle center



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.

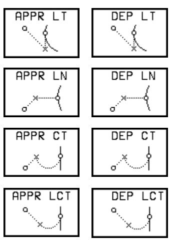
End point P_N

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

APPR DEP

▶ Press the soft key with the desired path function:



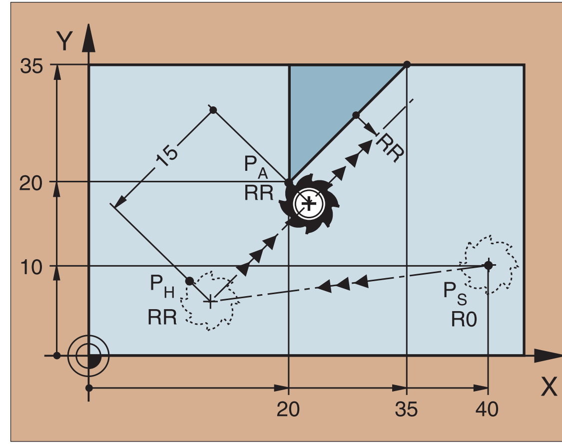
APPR LT / DEP LT	Straight line with tangential connection
APPR LN / DEP LN	Straight line perpendicular to the contour point
APPR CT / DEP CT	Circular arc with tangential connection
APPR LCT / DEP LCT	Straight line segment tangentially connected to the contour through an arc

- 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

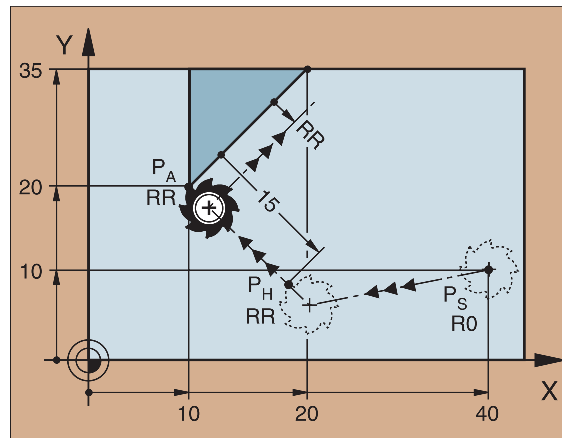
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LT X+20 Y+20 LEN 15 RR F100
9 L X+35 Y+35
```



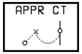
Approaching on a Straight Line Perpendicular to the First Contour Element

- Coordinates for the first contour point P_A
- Distance len (length) from P_H to P_A
  Enter a length Len > 0
- 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
```

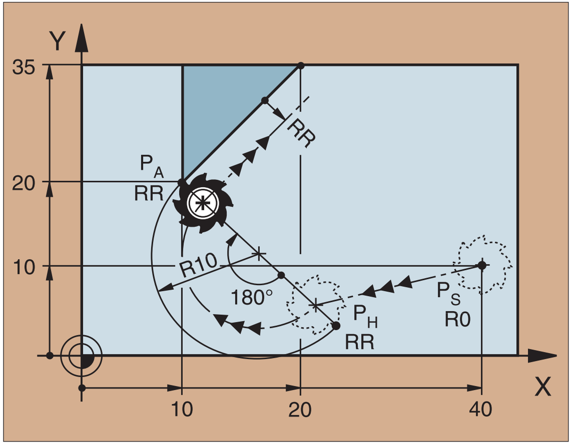


Approaching Tangentially on an Arc



- 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
- Tool radius compensation RR/RL

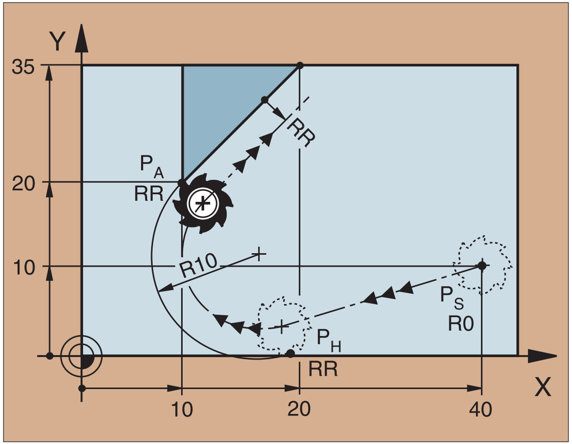
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR CT X+10 Y+20 CCA 180 R10 RR F100
9 L X+20 Y+35
```



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

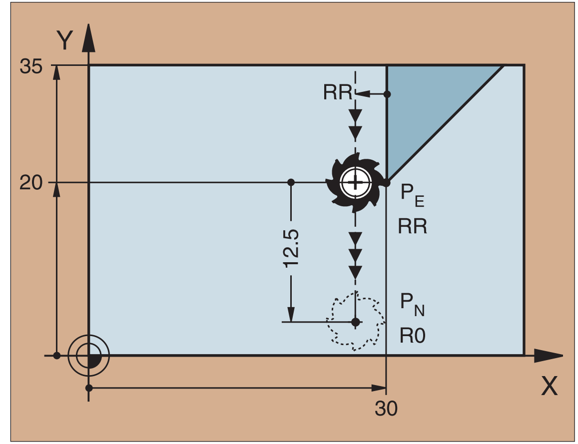
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LCT X+10 Y+20 R10 RR F100
9 L X+20 Y+35
```



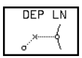
Departing Tangentially on a Straight Line

- 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
```

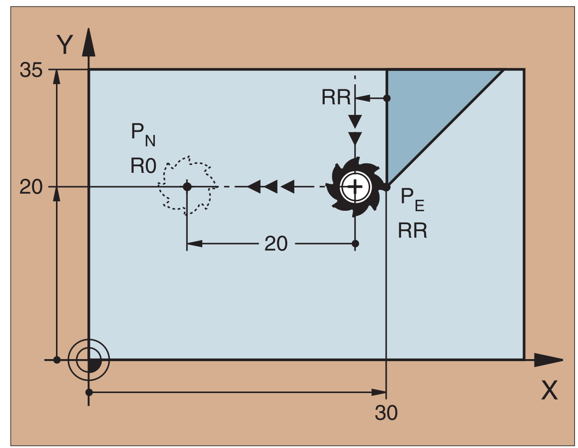


Departing on a Straight Line Perpendicular to the Last Contour Element

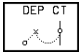


- Distance len (length) from P_E to P_N
  Enter a length LEN > 0

```
23 L X+30 Y+35 RR F100
24 L Y+20 RR F100
25 DEP LN LEN+20 F100 M2
```

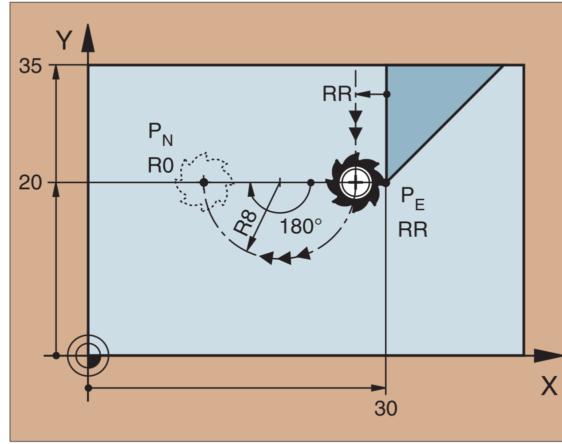


Departing Tangentially on an Arc

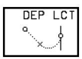


- Radius R
  Enter a radius R > 0
- Circle center angle (CCA)

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

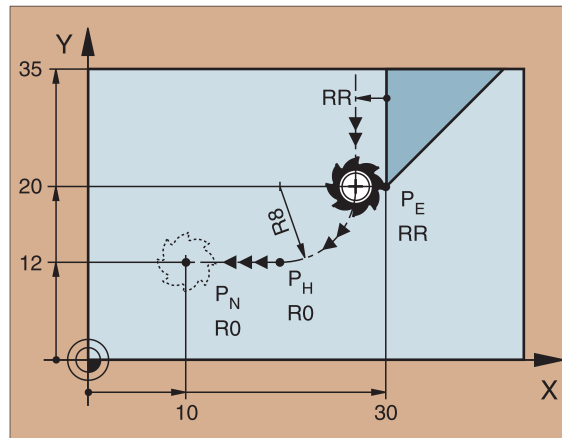


Departing on an Arc Tangentially Connecting the Contour and a Straight Line



- 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:

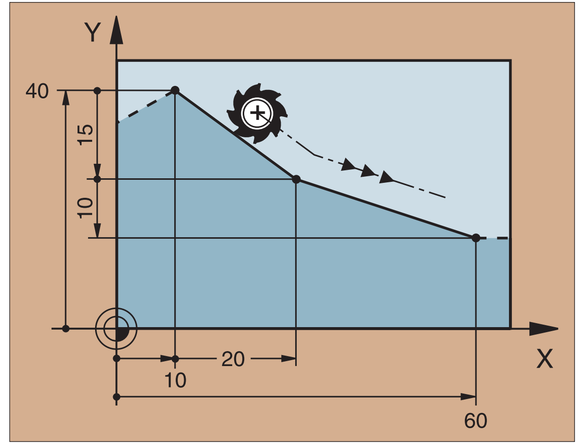
- Path function
- Coordinates of the contour element end points (target position)
- Radius compensation RR/RL/R0
- Feed rate F
- Miscellaneous function M

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

Path functions	Key	
Straight line	L	Page 19
Chamfer between two straight lines	CHF	Page 20
Corner rounding	RND	Page 20
Circle center or pole for polar coordinates	CC	Page 21
Circular path around the circle center CC	C	Page 21
Circular path with known radius	CR	Page 22
Circular path with tangential connection to previous contour	CT	Page 23
Free contour programming	FK	Page 25

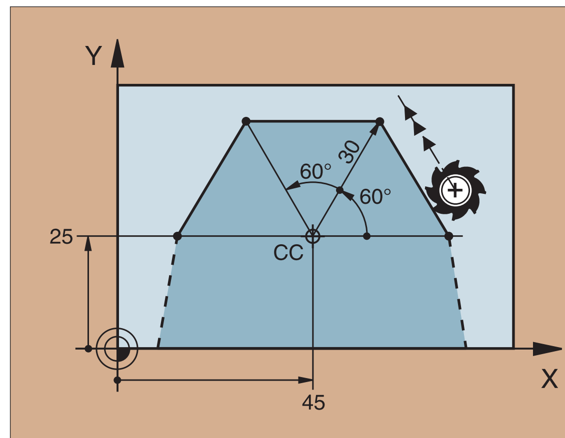
Straight Line

- 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!

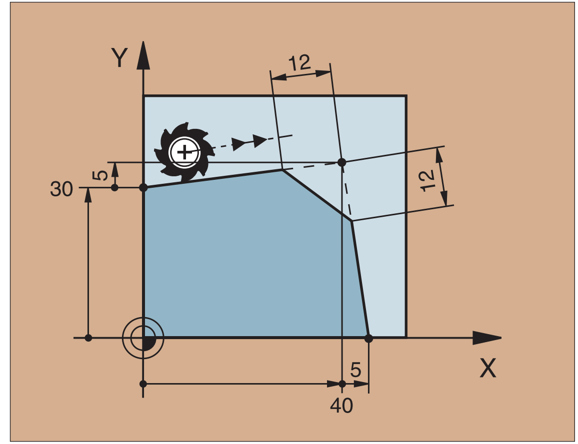
Inserting a Chamfer Between Two Straight Lines

CHF

▶ Chamfer Side Length

```
7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12
10 L IX+5 Y+0
```

- You cannot start a contour with a CHF block!
- 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

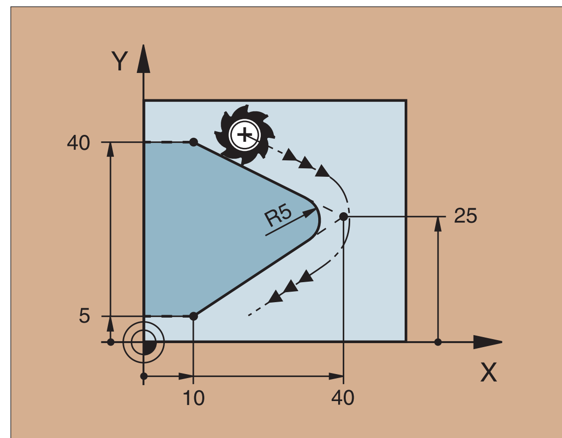
The beginning and end of the arc extend tangentially from the previous and subsequent contour elements.

RND

▶ 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

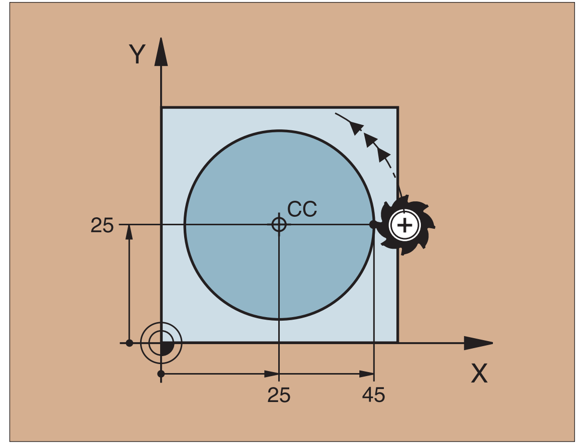
CC ▶ Coordinates of the circle center CC

C ▶ Coordinates of the arc end point
▶ Direction of rotation DR

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

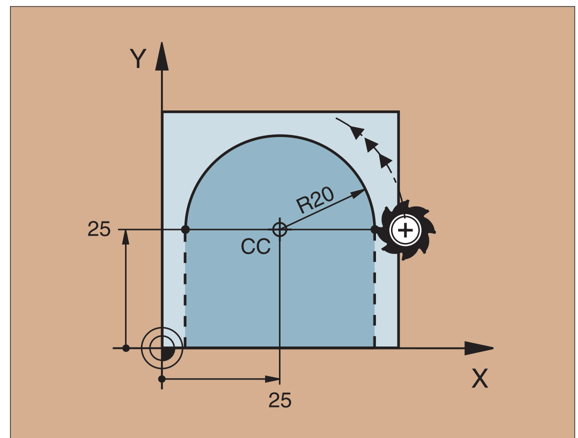
With Cartesian coordinates:

```
5 CC X+25 Y+25
6 L X+45 Y+25 RR F200 M3
7 C X+45 Y+25 DR+
```



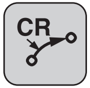
With polar coordinates:

```
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)!

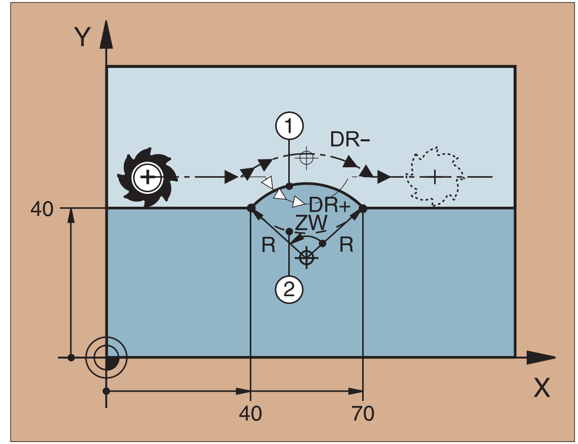
Circular Path with Known Radius (CR)



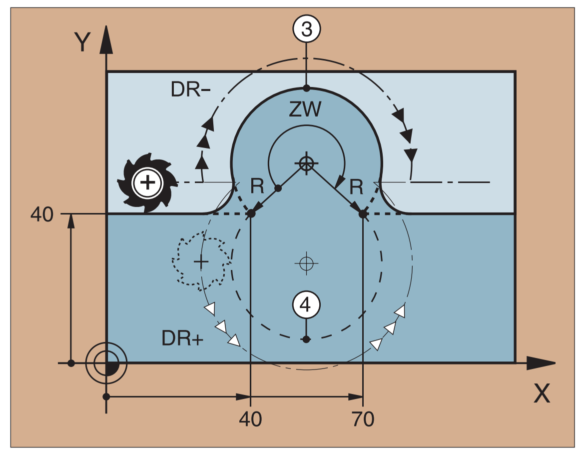
- Coordinates of the arc end point
- Radius R
  If the central angle ZW > 180, R is negative.
  If the central angle ZW < 180, R is positive.
- Direction of rotation DR

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

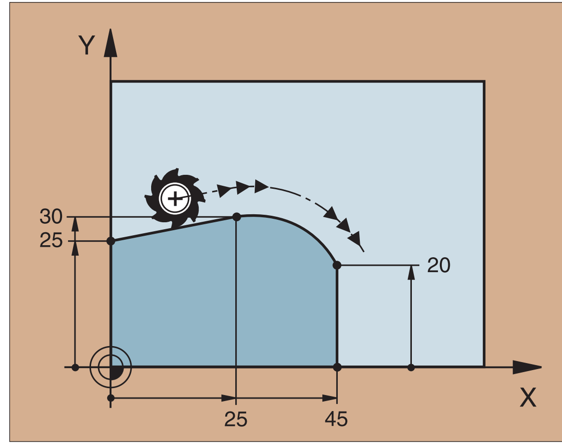


Circular Path CT with Tangential Connection

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

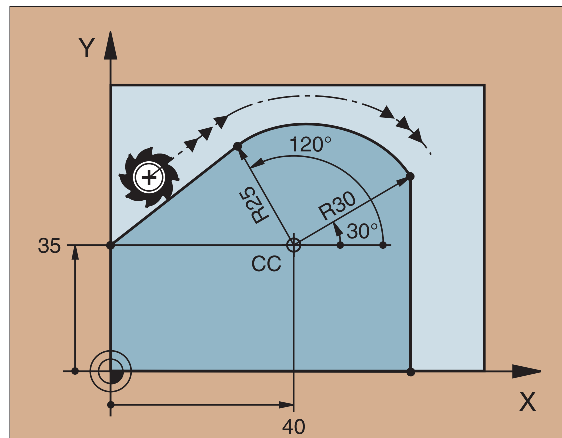
With Cartesian coordinates:

```
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)

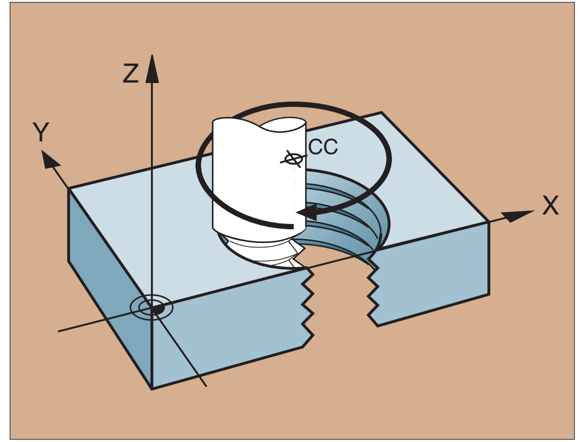
Path revolutions:	n	=	Thread revolutions + overrun at start and end of thread
Total height:	h	=	Pitch P x path revolutions n
Incr. coord. angle:	IPA	=	Path revolutions n x 360°
Start angle:	PA	=	Angle at start of thread + angle for overrun
Start coordinate:	Z	=	Pitch P x (thread revolutions + thread overrun at start of thread)

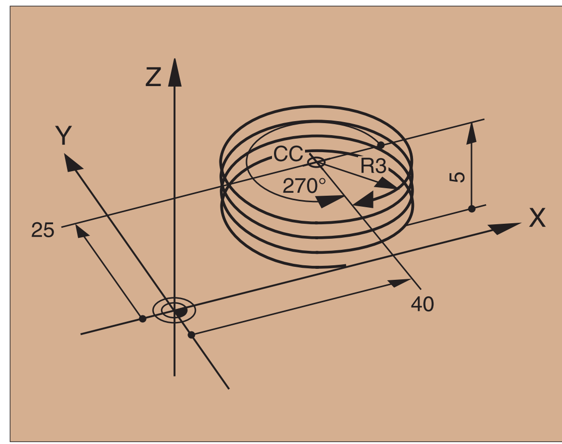
Shape of helix

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"

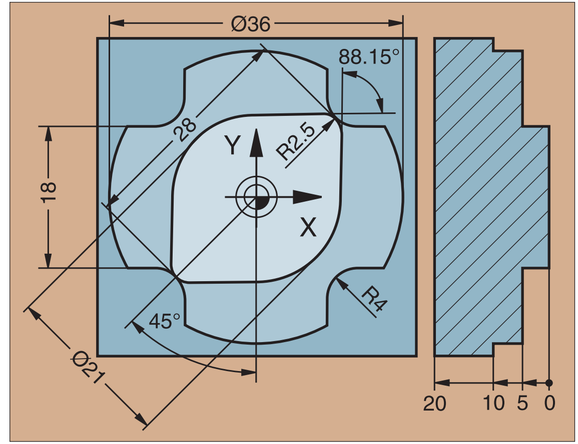
If the end point coordinates are not given in the workpiece drawing or if the drawing gives dimensions that cannot be entered with the gray path function keys, you can still program the part by using the "FK Free Contour Programming."

Possible data on a contour element:

- Known coordinates of the end point
- Auxiliary point on the contour element
- Auxiliary point near the 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
FSELECT	To enter the displayed solution in the part program
END SELECT	To enter data for subsequent contour elements
START SINGLE	To graphically display the next programmed block

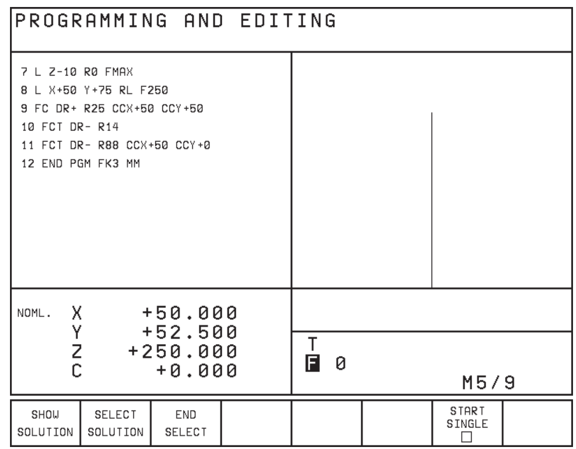
Standard colors of the interactive graphics

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

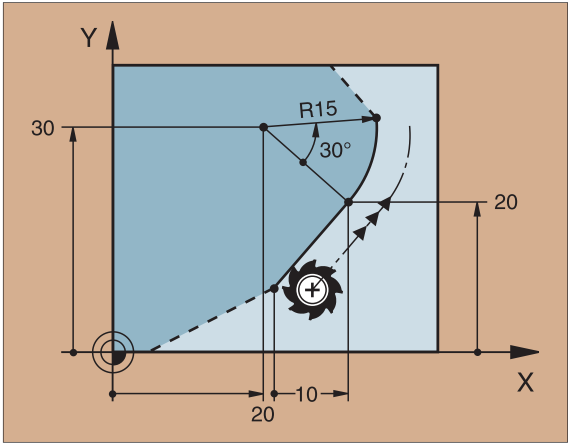
Straight Circular

FL	FC	Contour element without tangential connection
FLT	FCT	Contour element with tangential connection
FPOL		Pole for FK programming

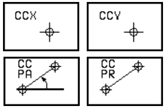
End Point Coordinates X, Y or PA, PR

X	Y	Cartesian coordinates X and Y
PA	PR	Polar coordinates referenced to FPOL
I		Incremental input

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



Circle Center (CC) in an FC/FCT block

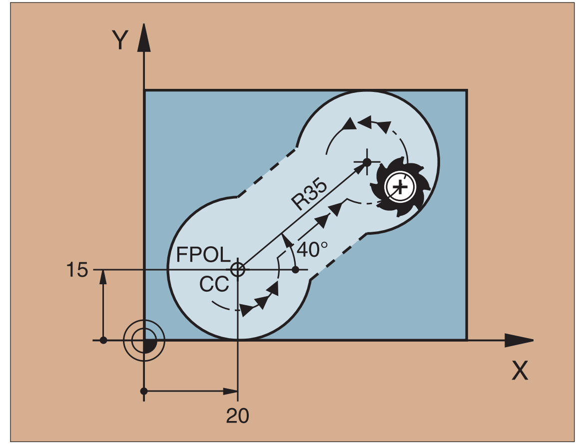


Cartesian coordinates of the circle center

Polar coordinates of the circle center referenced to FPOL

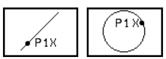
Incremental input

```
10 FC CCX+20 CCY+15 DR+ R15
11 FPOL X+20 Y+15
...
13 FC DR+ R15 CCPR+35 CCPA+40
```

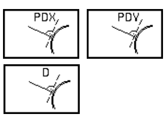


Auxiliary Point

... P1 on a contour



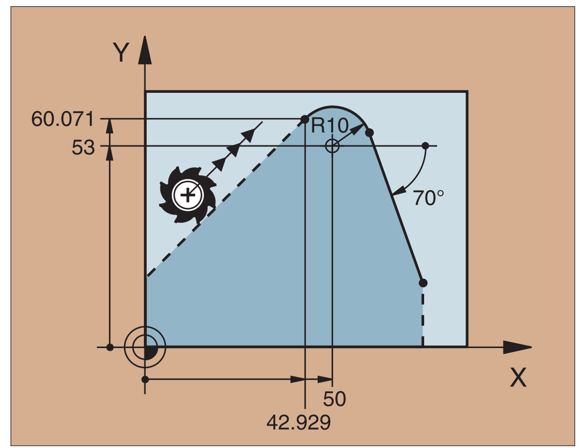
... PD next to a contour



Coordinates of the auxiliary points

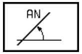
Perpendicular distance

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

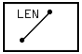


Direction and Length of the Contour Element

Data on a straight line

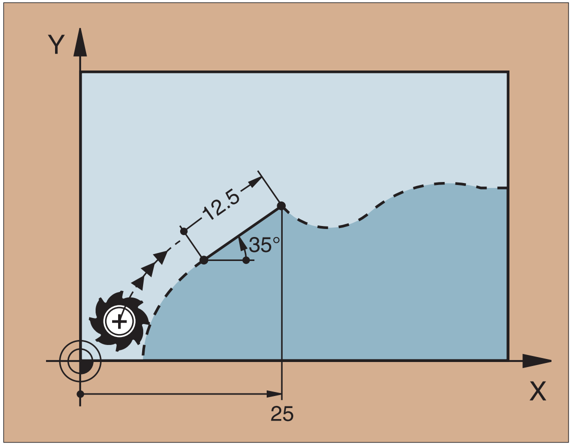


Gradient angle of a straight line

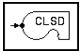


Length of a straight line

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

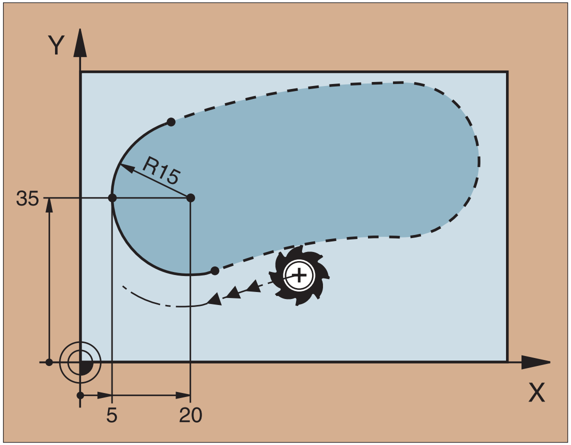


Identifying a closed contour



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-
```

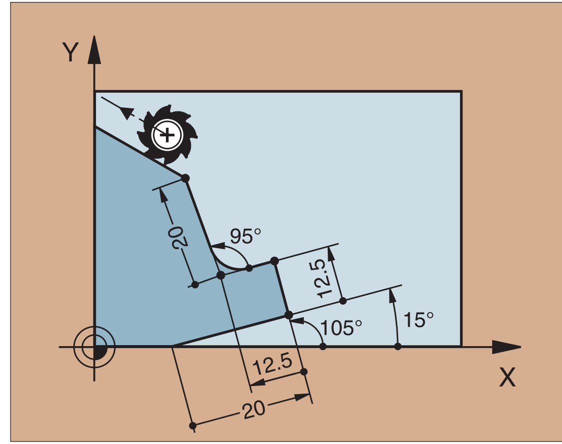


Values Relative to Block N: Distance of the Contour Element

PAR N	Parallel to a straight contour element Parallel to the entry tangent of an arc
DP	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
```



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.

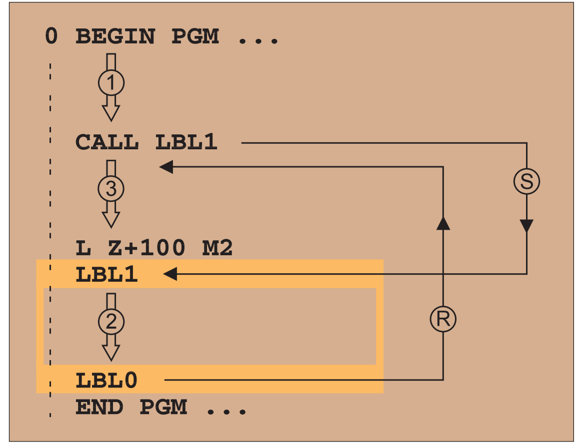
It's good practice to place subprograms after the main program end (M2).

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

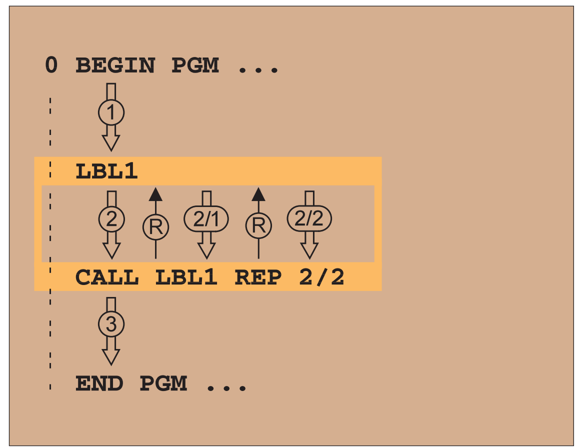
Working with Program Section Repeats

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

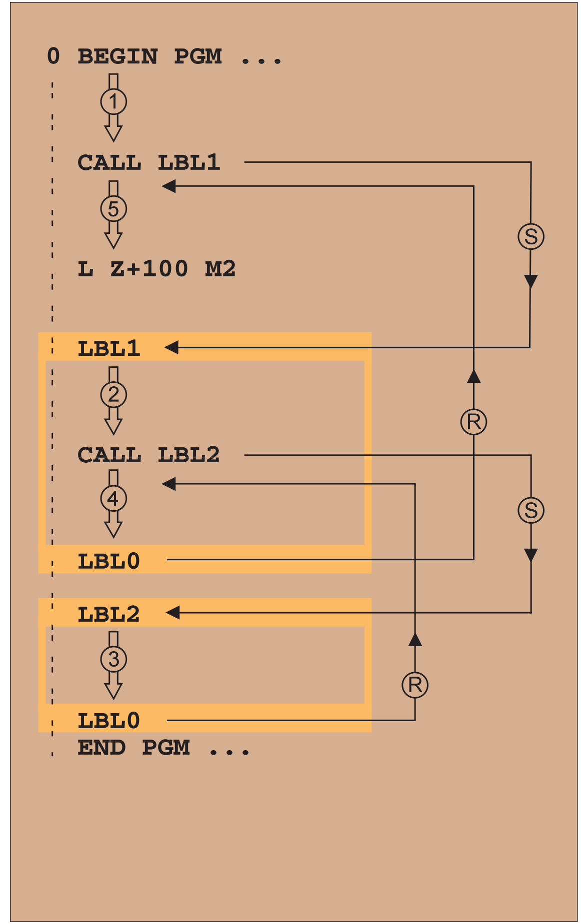


Subprogram Nesting: A Subprogram within a Subprogram

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!

S = Jump; R = Return jump ▶



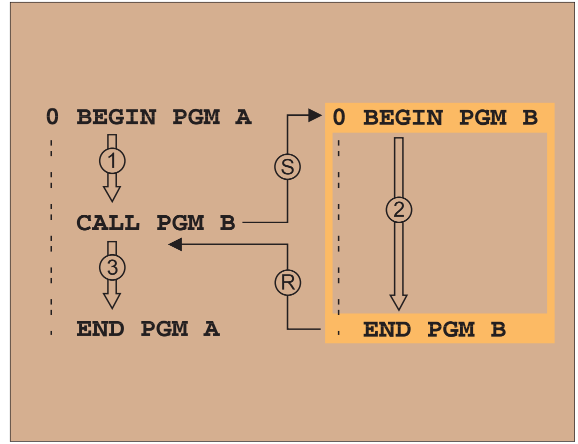
Any Program as a Subprogram

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.

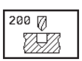
Defining cycles

CYCL DEF

▶ Select the desired cycle:

DRILLING

▶ Select the cycle group

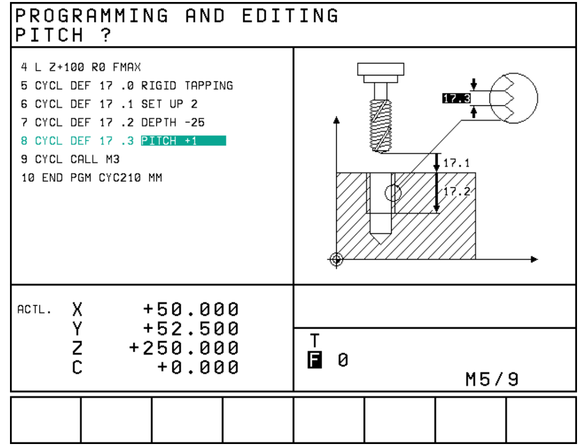


▶ Select the cycle

Continued on next page ▶

Graphic Support During Cycle Programming

Select the PGM+FIGURE screen layout!



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 cycle CONTOUR GEOMETRY
- Point patterns

All other cycles go into effect when they are called through

- CYCL CALL: effective for one block
- M99: effective for one block
- M89: effective until canceled (depends on machine parameter settings)

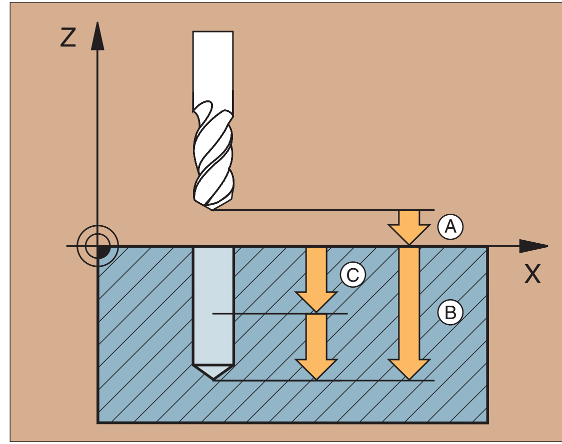
All machining cycles can also be called up in conjunction with point tables. For this, use the function CYCL CALL PAT (see User's Manual)

Drilling Cycles

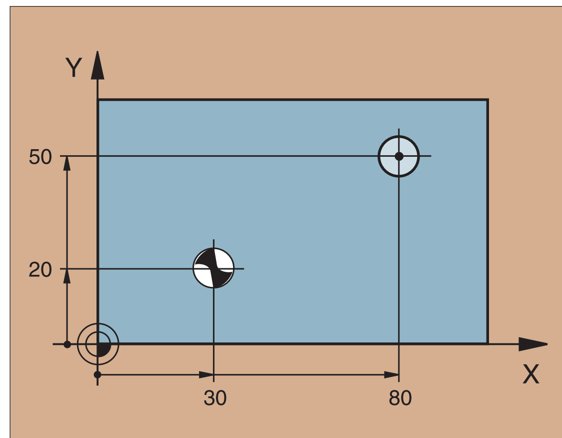
PECKING (1)

- 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.



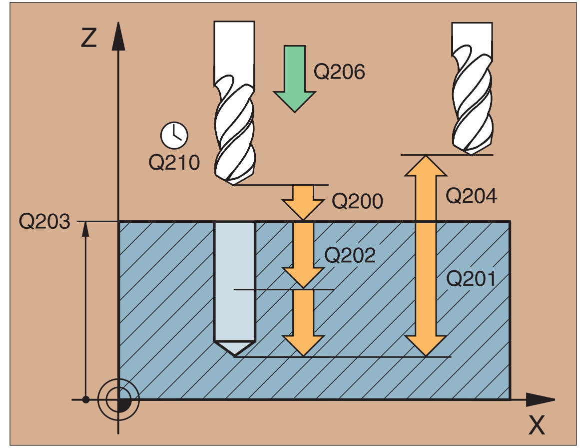

```
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP +2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG +7.5
10 CYCL DEF 1.4 DWELL 1
11 CYCL DEF 1.5 F80
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 L Z+2 FMAX M99
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```



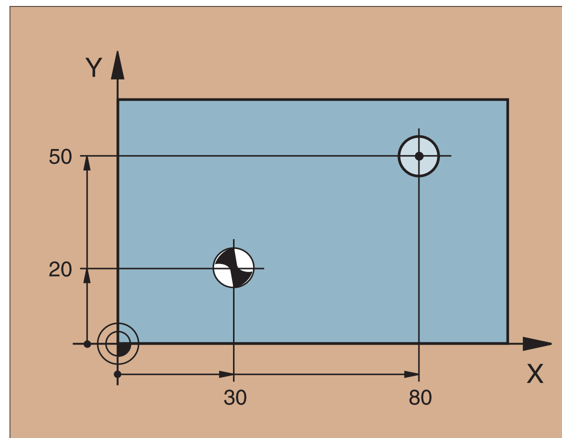
DRILLING (200)

- 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

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.



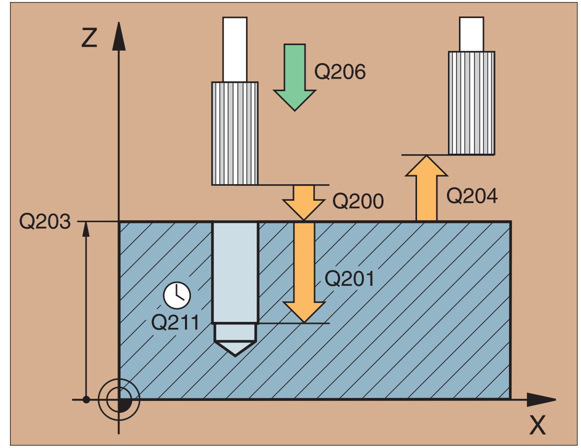

```
11 CYCL DEF 200 DRILLING
   Q200 = 2       ;SET-UP CLEARANCE
   Q201 = -15     ;DEPTH
   Q206 = 250     ;FEED RATE FOR PLUNGING
   Q202 = 5       ;PECKING DEPTH
   Q210 = 0       ;DWELL TIME AT TOP
   Q203 = +0      ;SURFACE COORDINATE
   Q204 = 100     ;2ND SET-UP CLEARANCE
12 L Z+100 R0 FMAX M6
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
```



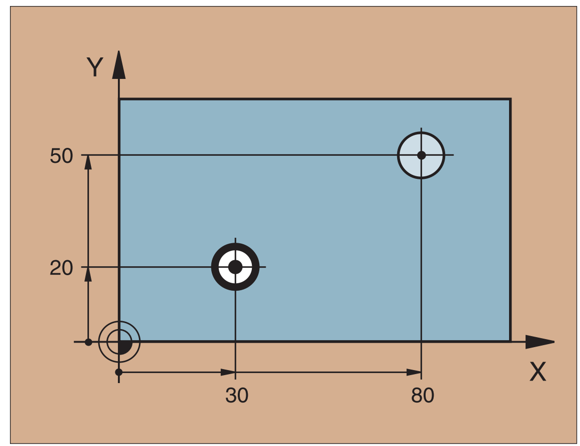
REAMING (201)

- 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 PLUNGING
   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
```



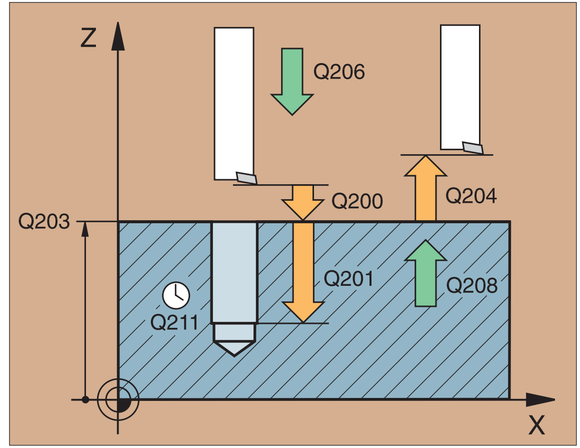
BORING (202)

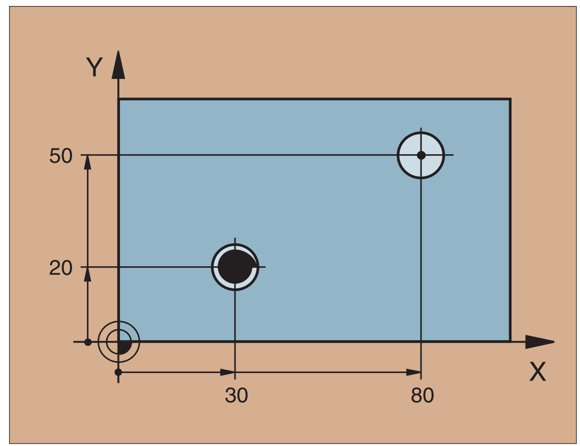
> ⚠ Danger of collision! Choose a disengaging direction that moves the tool away from the wall of the hole.

- 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 direction (0/1/2/3/4) at bottom of hole: Q214

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

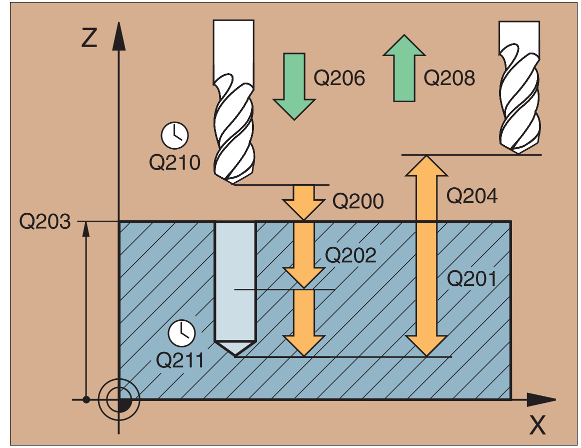
```
11 CYCL DEF 202 BORING
   Q200 = 2       ;SET-UP CLEARANCE
   Q201 = -15     ;DEPTH
   Q206 = 100     ;FEED RATE FOR PLUNGING
   Q211 = 0.5     ;DWELL TIME AT DEPTH
   Q208 = 250     ;RETRACTION FEED RATE
   Q203 = +0      ;SURFACE COORDINATE
   Q204 = 100     ;2ND SET-UP CLEARANCE
   Q214 = 1Di     ;DISENGAGING DIRECTION
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
```





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



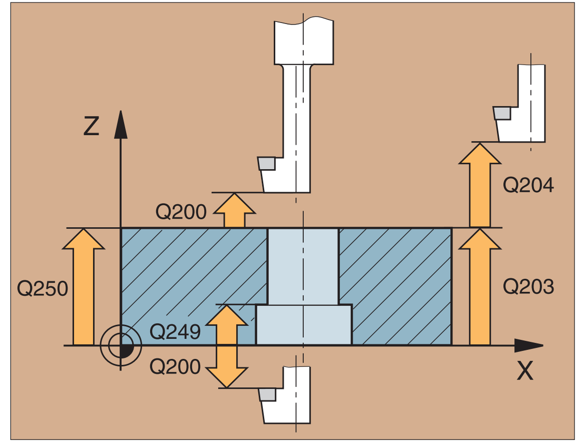
The TNC automatically pre-positions the tool in the tool axis. If the depth is greater than or equal to the pecking depth, the tool drills to the depth in one plunge.

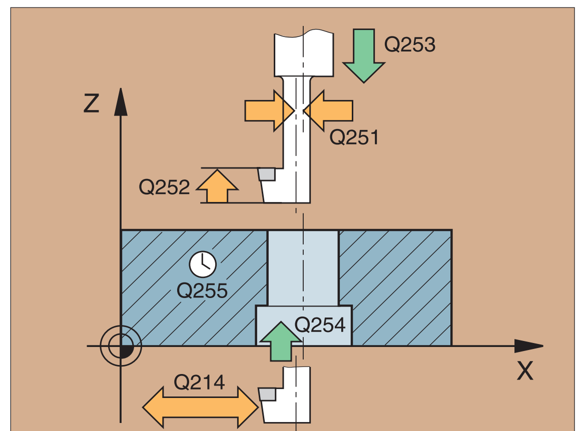
COUNTERBORE BACK (204)

- 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

> ⚠
> - 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!

```
11 CYCL DEF 204 COUNTERBORE BACK
   Q200 = 2      ;SET-UP CLEARANCE
   Q249 = +5     ;DEPTH OF COUNTERBORE
   Q250 = 20     ;MATERIAL THICKNESS
   Q251 = 3.5    ;OFF-CENTER DISTANCE
   Q252 = 15     ;TOOL EDGE HEIGHT
   Q253 = 750    ;F PRE-POSITIONING
   Q254 = 200    ;F COUNTERBORING
   Q255 = 0.5    ;DWELL TIME
   Q203 = +0     ;SURFACE COORDINATE
   Q204 = 50     ;2ND SET-UP CLEARANCE
   Q214 = 1      ;DISENGAGING DIRECTN
```

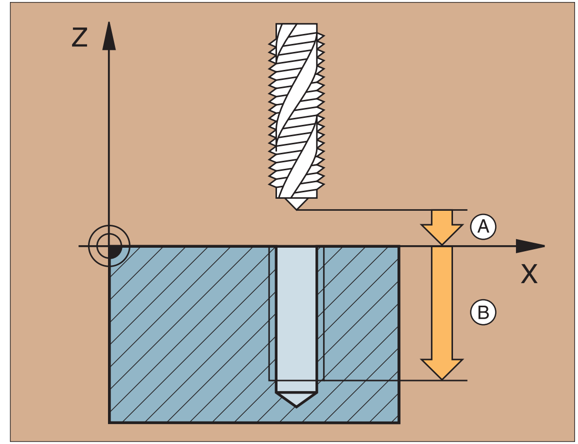




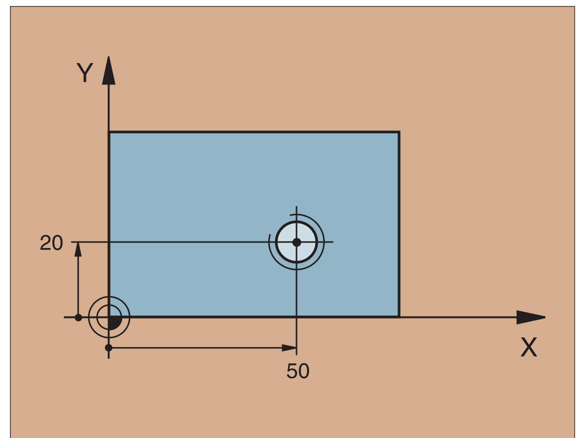
TAPPING with Floating Tap Holder (2)

- 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!

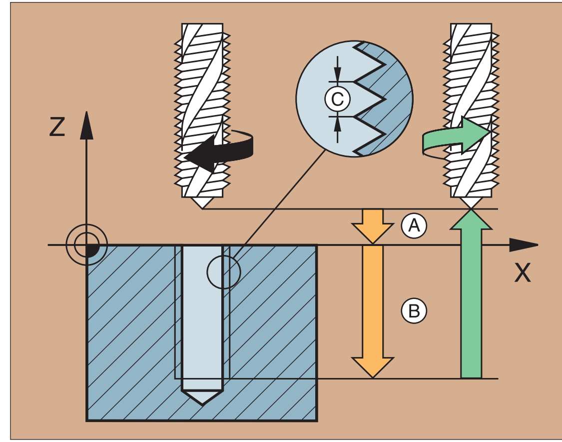



```
25 CYCL DEF 2.0 TAPPING
26 CYCL DEF 2.1 SET UP 3
27 CYCL DEF 2.2 DEPTH -20
28 CYCL DEF 2.3 DWELL 0.4
29 CYCL DEF 2.4 F100
30 L Z+100 R0 FMAX M6
31 L X+50 Y+20 FMAX M3
32 L Z+3 FMAX M99
```



RIGID TAPPING (17)

- 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: –

Pockets, Studs, and Slots

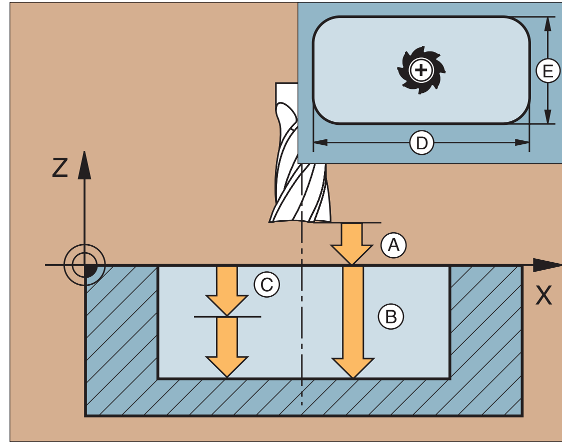
POCKET MILLING (4)

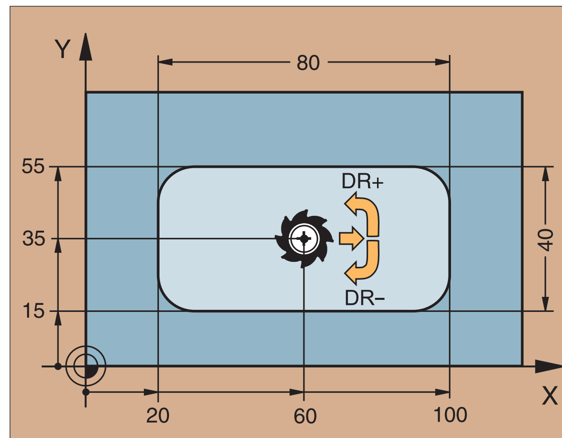
> ⚠ This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at the pocket center!

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–

```
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+
19 L Z+100 R0 FMAX M6
20 L X+60 Y+35 FMAX M3
21 L Z+2 FMAX M99
```

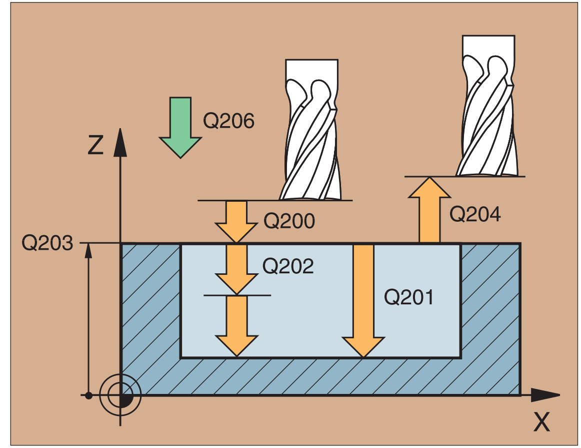


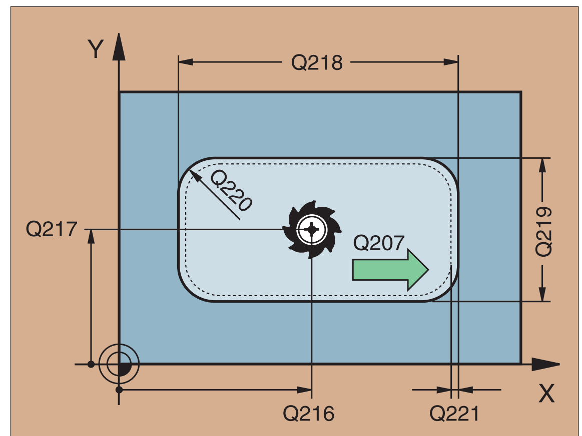


POCKET FINISHING (212)

- 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.

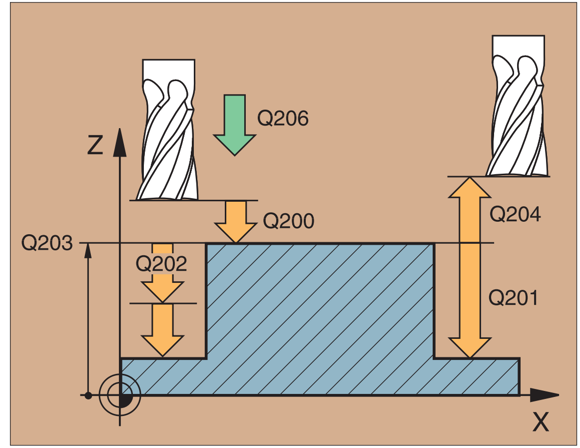


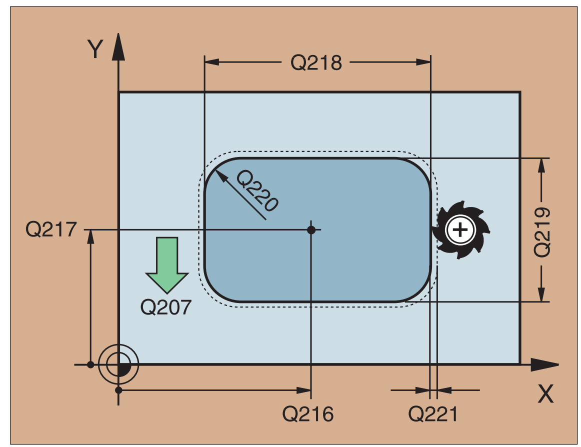


STUD FINISHING (213)

- 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
  - 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.

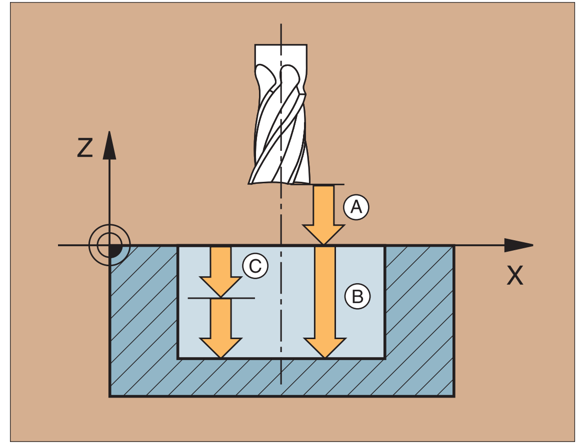




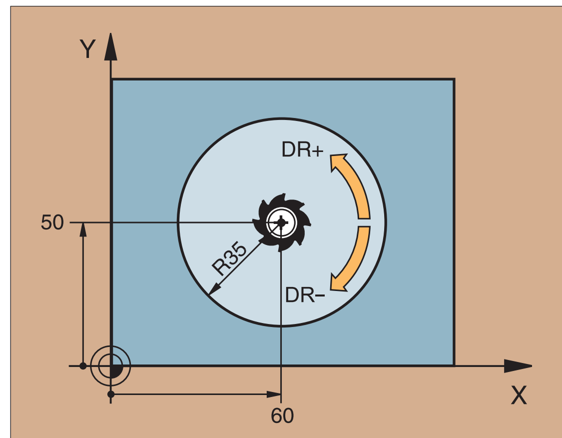
CIRCULAR POCKET MILLING (5)

> ⚠ 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–



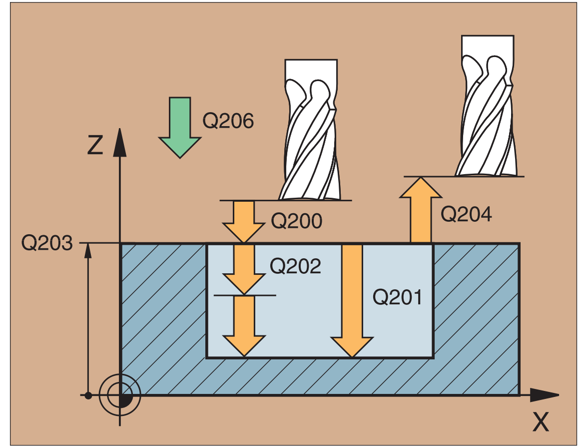

```
17 CYCL DEF 5.0 CIRCULAR POCKET
18 CYCL DEF 5.1 SET UP 2
19 CYCL DEF 5.2 Depth -12
20 CYCL DEF 5.3 PECKG 6 F80
21 CYCL DEF 5.4 RADIUS 35
22 CYCL DEF 5.5 F100 DR+
23 L Z+100 R0 FMAX M6
24 L X+60 Y+50 FMAX M3
25 L Z+2 FMAX M99
```

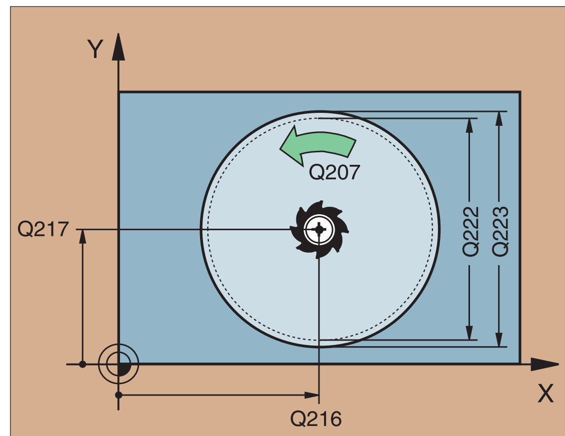


CIRCULAR POCKET FINISHING (214)

- 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.

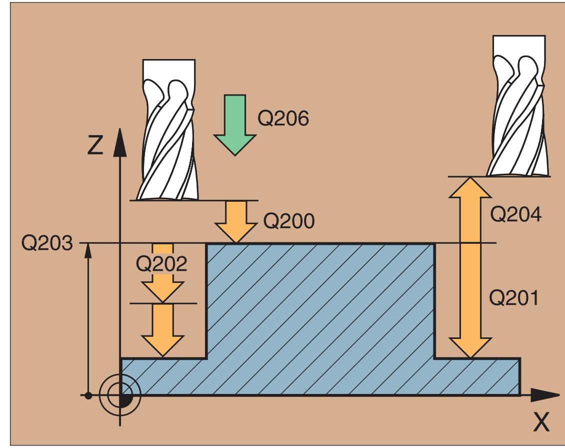


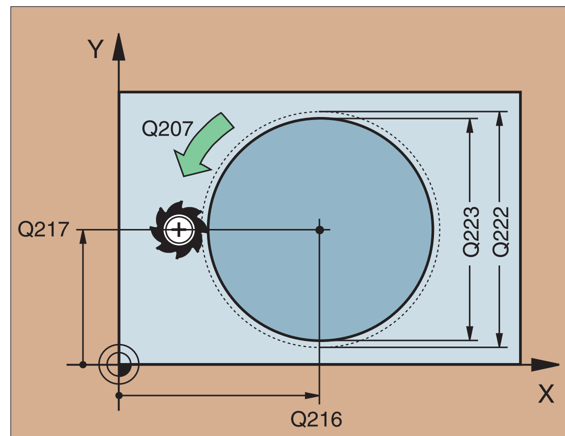


CIRCULAR STUD FINISHING (215)

- 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.

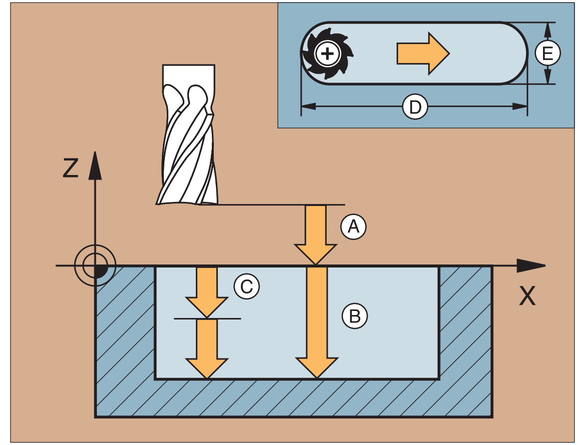




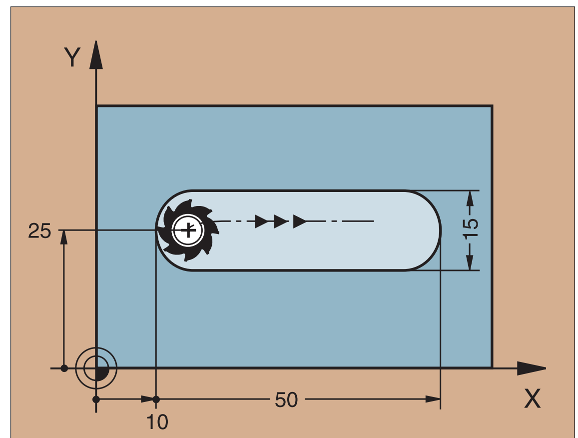
SLOT MILLING (3)

> ⚠
> - 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
  - Safety 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)

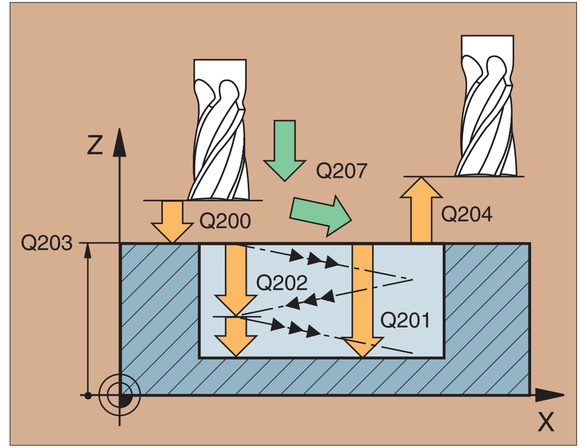



```
10 TOOL DEF 1 L+0 R+6
11 TOOL CALL 1 Z S1500
12 CYCL DEF 3.0 SLOT MILLING
13 CYCL DEF 3.1 SET UP 2
14 CYCL DEF 3.2 Depth -15
15 CYCL DEF 3.3 PECKG 5 F80
16 CYCL DEF 3.4 X50
17 CYCL DEF 3.5 Y15
18 CYCL DEF 3.6 F120
19 L Z+100 R0 FMAX M6
20 L X+16 Y+25 R0 FMAX M3
21 L Z+2 M99
```



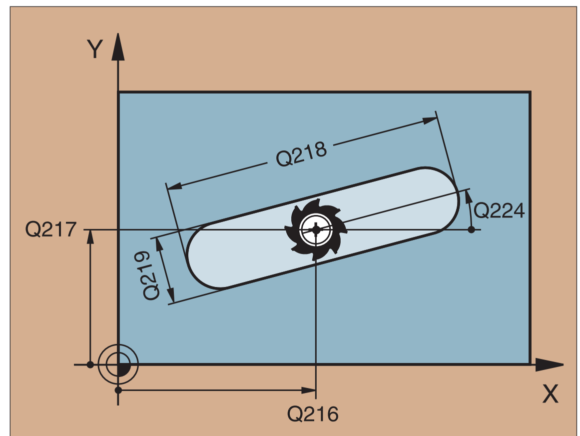
SLOT WITH RECIPROCATING PLUNGE-CUT (210)

> ⚠ 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

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.

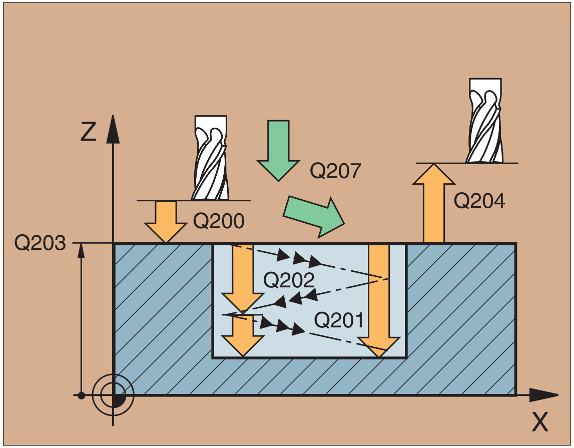


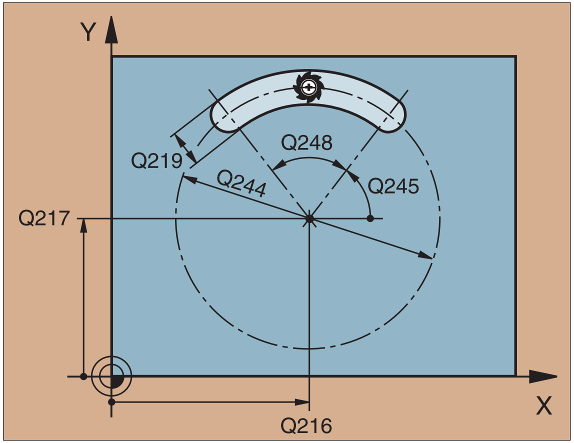
CIRCULAR SLOT with reciprocating plunge (211)

> ⚠ 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 circle dia.: Q244
  - Second side length: Q219
  - Starting angle of the slot: Q245
  - Angular length of the slot: Q248

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.

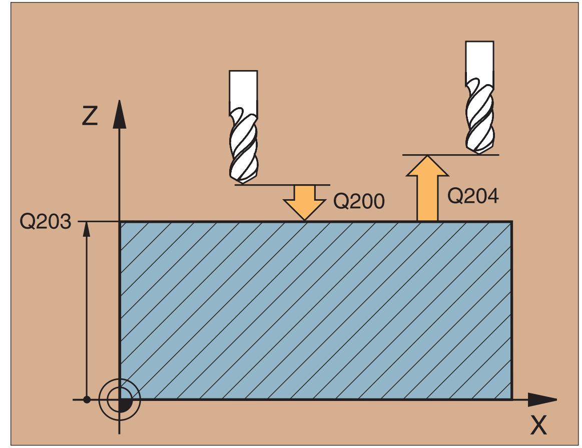




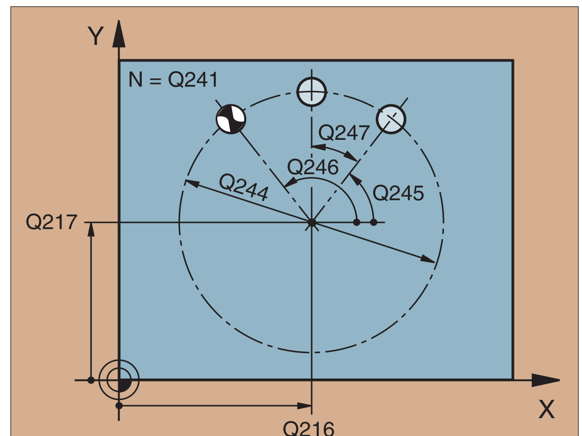
Point Patterns

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 or repetitions: Q241
  - ▶ Set-up clearance: Q200
  - ▶ Surface coordinate: Q203
  - ▶ 2nd set-up clearance: Q204



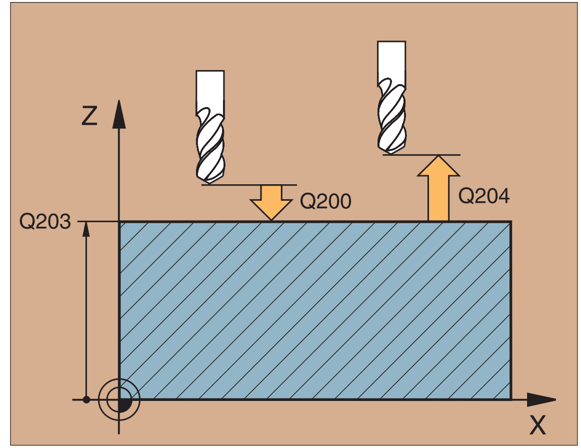
- Cycle 220 POLAR PATTERN is effective immediately upon definition!
- Cycle 220 automatically calls the last defined fixed cycle!
- Cycle 220 can be combined with Cycles 1, 2, 3, 4, 5, 17, 200, 201, 202, 203, 204, 212, 213, 214, 215
- In combined cycles, the SET-UP CLEARANCE, SURFACE COORDINATE and 2ND 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.

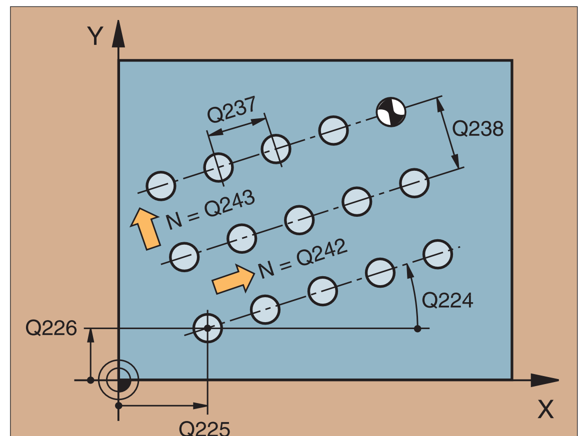
LINEAR PATTERN (221)

- CYCL DEF: Select Cycle 221 LINEAR PATTERN
  - Starting pnt 1st axis: Q225
  - Starting 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



- 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, 212, 213, 214, 215
- 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

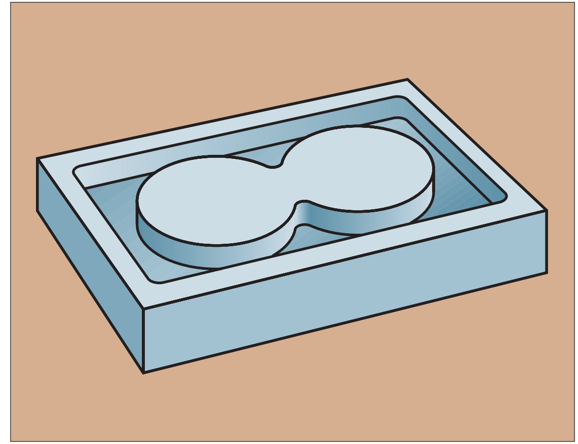
SL cycles are useful when you wish to machine a contour consisting of several subcontours (up to 12 islands or pockets).

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.

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



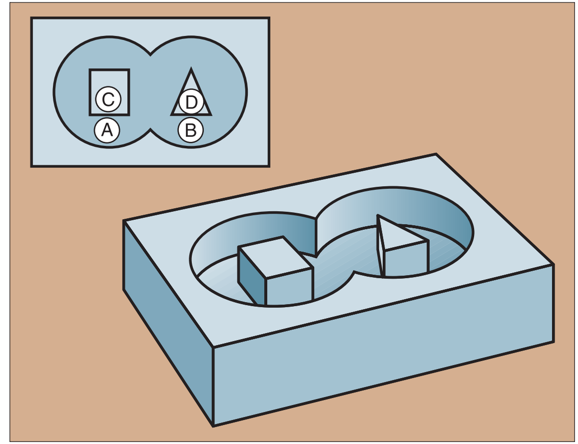
CONTOUR GEOMETRY (14)

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 numbers for contour: List the LABEL numbers of the subprograms that you wish to superimpose to make a complete closed contour.

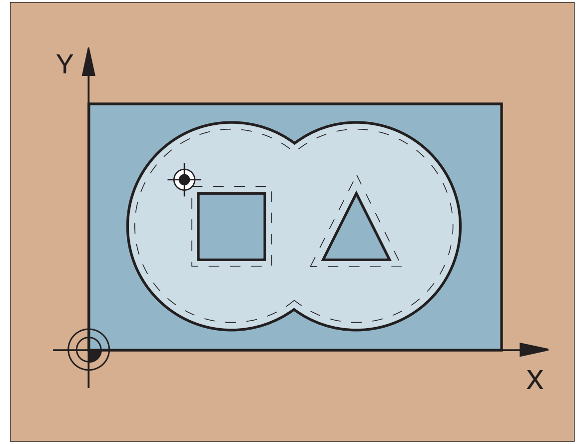
Cycle 14 CONTOUR GEOMETRY is effective immediately upon definition!

```
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
```



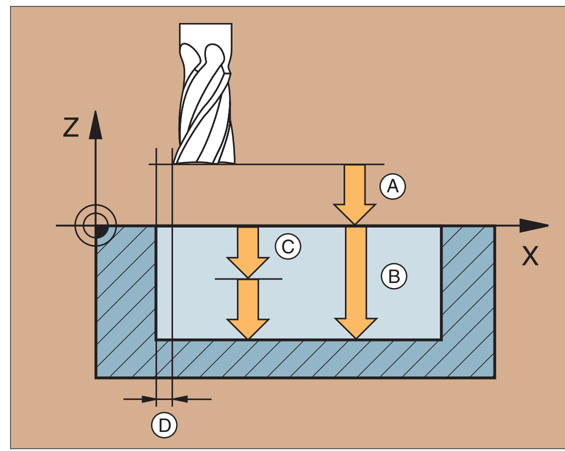
▲ A and B are pockets, C and D islands

PILOT DRILLING (15)



- CYCL DEF: Select cycle 15 PILOT DRILLING
  - Set-up clearance
  - Total hole depth Distance from the top surface of the workpiece to the hole bottom
  - Pecking depth
  - Finishing allowance D
  - Feed rate F

ROUGH-OUT (6)



There are two steps in the rough-out cycle:

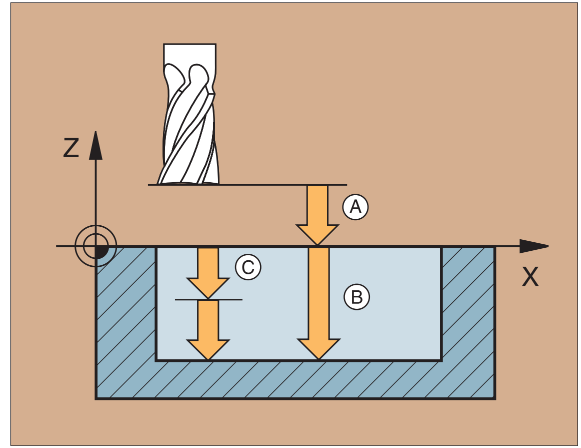
1. Milling a channel around subcontours
2. Area clearance

- CYCL DEF: Select Cycle 6 ROUGH-OUT
  - Set-up clearance: A
  - Milling depth: B
  - Pecking depth: C
  - Feed rate for pecking
  - Finishing allowance: D
  - Rough-out angle
  - Feed rate F

CONTOUR MILLING (16)

Finishing the individual subcontours.

- CYCL DEF: Select Cycle 16 CONTOUR MILLING
  - Set-up clearance: A
  - Milling depth: B
  - Pecking depth: C
  - Feed rate for pecking
  - Rotation clockwise: DR–
    - Climb milling for pocket and island: –
    - Up-cut milling for pocket and island: +
  - Feed rate F

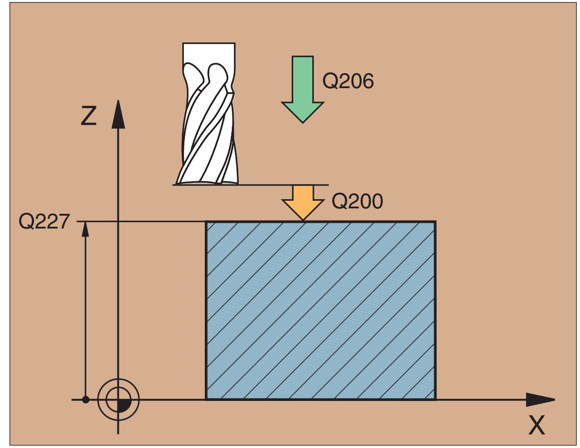


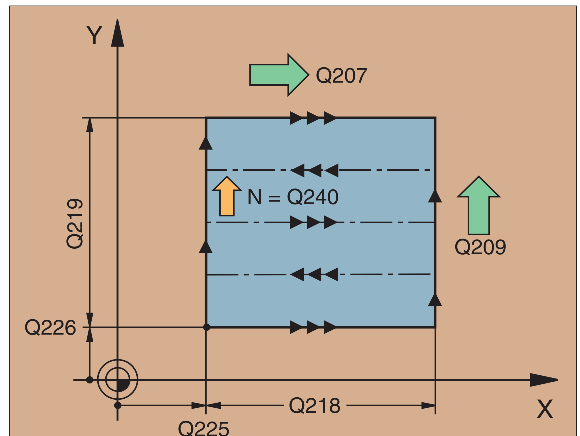
Multipass Milling

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 lengthIRST: Q218
  - Second side length: Q219
  - Number of cuts: Q240
  - Feed rate for plunging: Q206
  - Feed rate for milling: Q207
  - Stepover feed rate: Q209
  - Set-up clearance: Q200

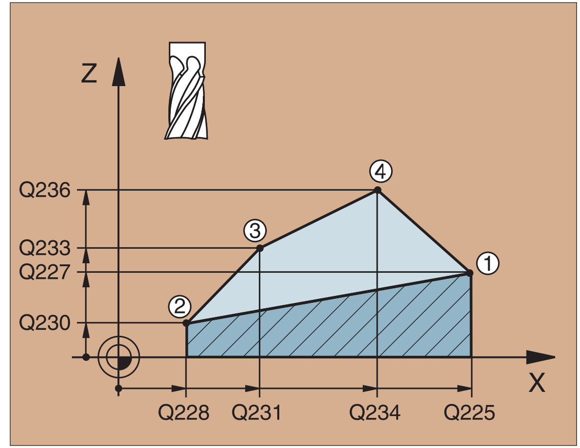


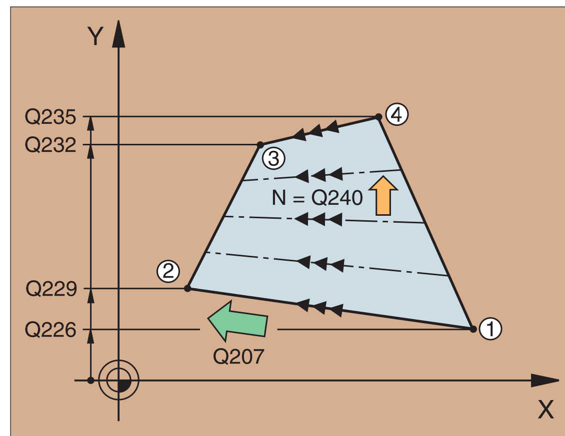


RULED SURFACE (231)

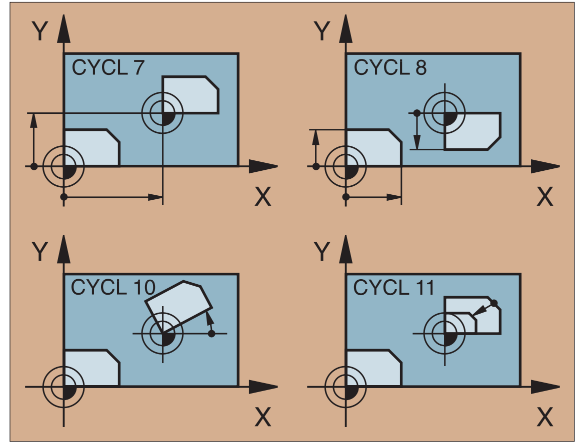
> ⚠ Starting from the initial position, the TNC positions the tool at the starting point (point 1), first in the working plane and then in the tool axis.

- 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

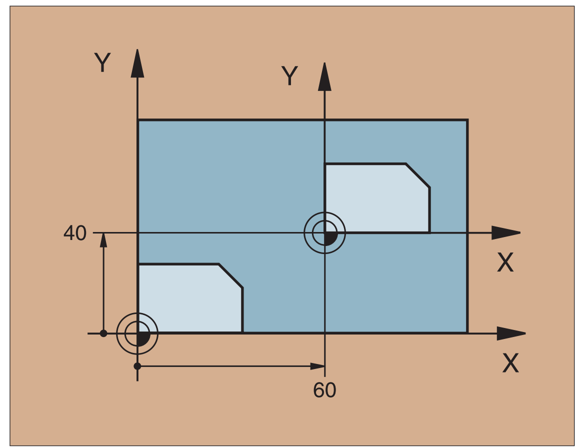


Cycles for coordinate transformation permit contours to be

• Shifted	Cycle 7	DATUM SHIFT
• Mirrored	Cycle 8	MIRROR IMAGE
• Rotated (in the plane)	Cycle 10	ROTATION
• Enlarged or reduced	Cycle 11	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.

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

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

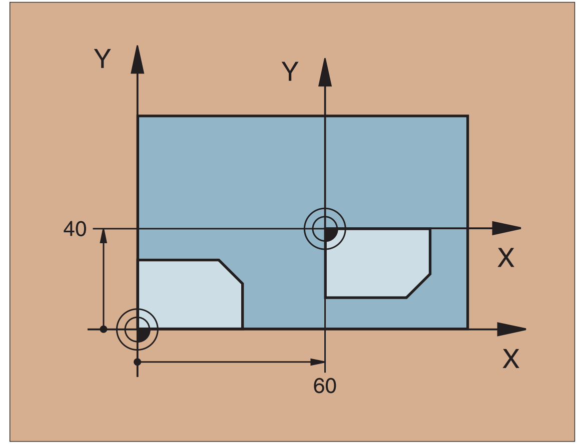
MIRROR IMAGE (8)

▶ 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)!



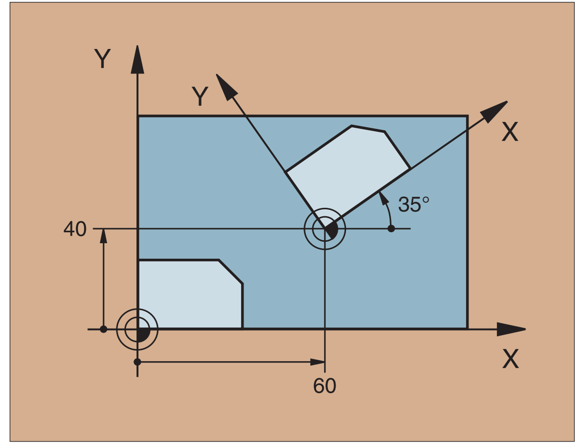
Rotation (10)

- ▶ CYCL DEF: Select Cycle 10 ROTATION
  - ▶ Enter the rotation angle:
    - Input range –360° to +360°
    - 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
```



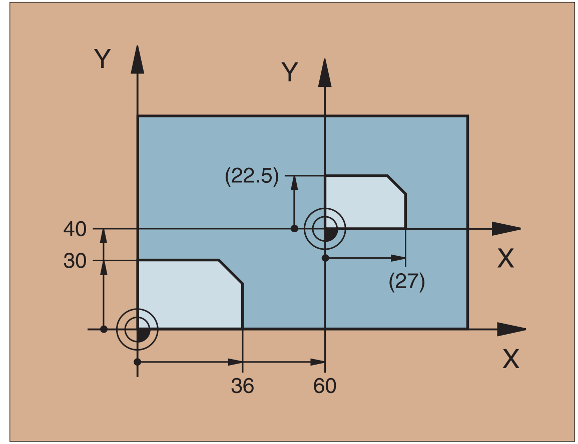
SCALING (11)

▶ 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

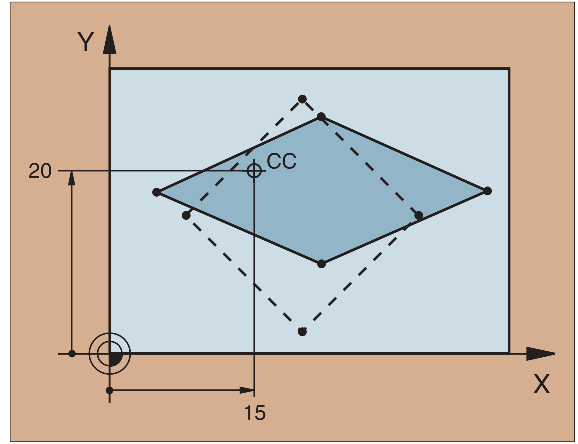
To cancel the SCALING, re-enter the cycle definition with SCL1.

```
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
```

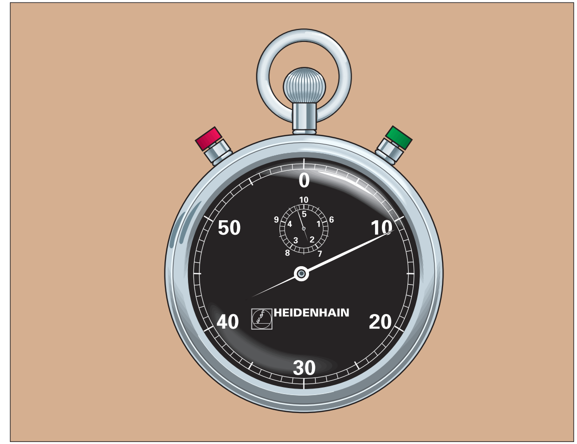
Special Cycles

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
```

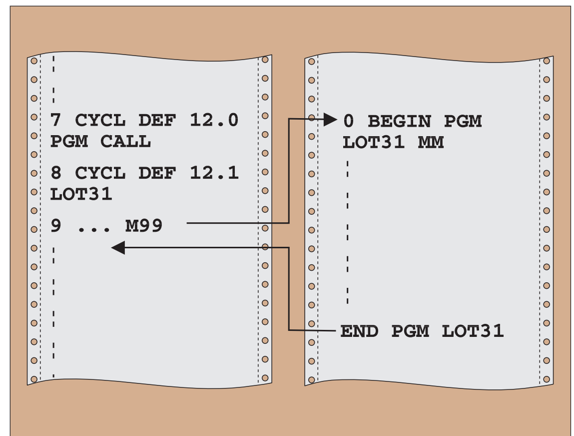


PGM CALL (12)

- 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

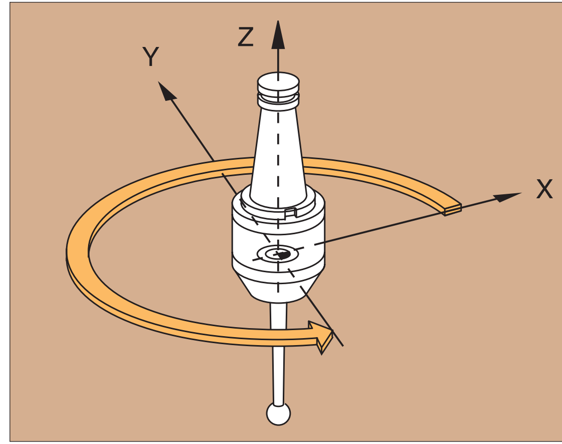
  ▶ Enter the orientation angle referenced to the angle reference axis of the working plane:

  - Input range 0 to 360°
  - Input resolution 0.1°

▶ Call the cycle with M19

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
```



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
- Digitize in reciprocating lines: TCH PROBE 6 MEANDER
- Digitize level by level: TCH PROBE 7 CONTOUR LINES

The digitizing cycles can be programmed only in plain language dialog. They can be programmed for the main axes X, Y and Z.

- 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!

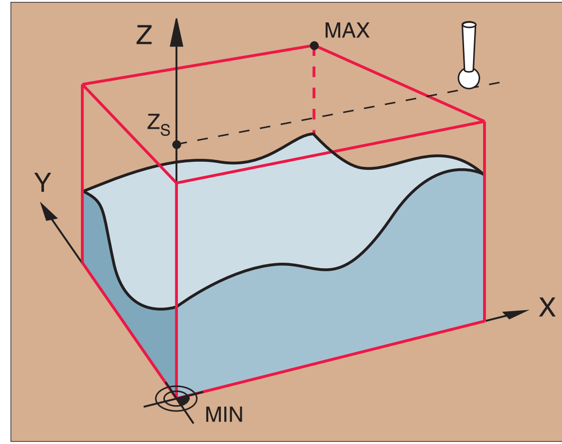
Selecting digitizing cycles

TOUCH PROBE

▶ Call an overview of touch probe functions
▶ Select a digitizing cycle via soft key

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
```

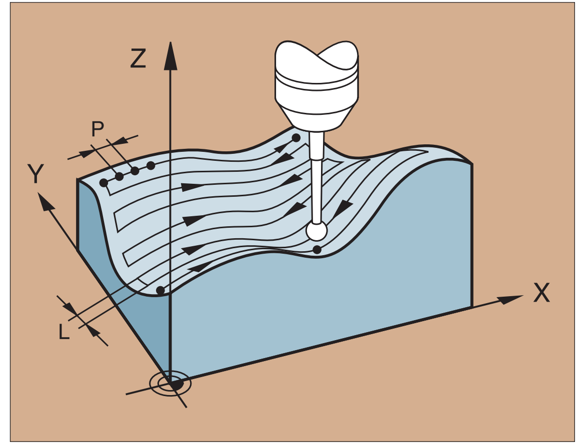
Digitizing Cycle 6: MEANDER

A 3D surface can be scanned in a reciprocating line-by-line process in Cycle 6 MEANDER.

▶ Define the RANGE with Cycle 5
▶ TOUCH PROBE: Select Cycle 6 MEANDER
  ▶ Line direction: Coordinate axis in whose positive direction the probe moves after touching the first contour point
  ▶ Limit in normal lines direction (travel): Distance by which the probe lifts off from the model surface after each deflection
  ▶ Line spacing: Distance moved forward to start the next line
  ▶ MAX. probe point interval

- The line spacing and MAX. probe point interval cannot exceed 5 mm.
- Set a line direction that is as perpendicular as possible to surface inclinations.

```
7 TCH PROBE 6.0 MEANDER
8 TCH PROBE 6.1 DIRECTN X
9 TCH PROBE 6.2 TRAVEL: 0.5 L.SPAC: 0.2 PP.INT:0.8
```



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

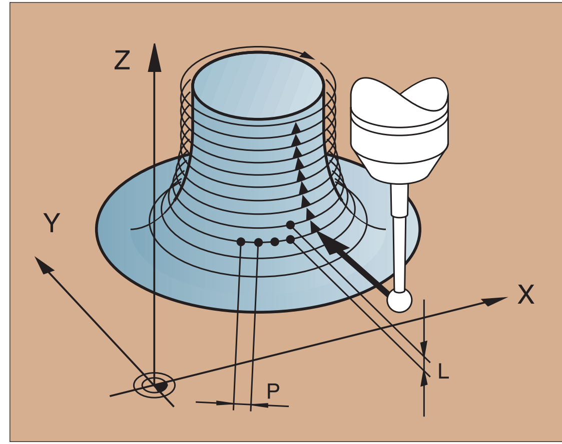
Digitizing Cycle 7: CONTOUR LINES

Cycle 7 CONTOUR LINES enables you to digitize a 3D surface level by level.

- Define Cycle 5 RANGE
- TOUCH PROBE: Select Cycle 7 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
  - Limit in normal lines direction (travel): Distance by which the probe lifts off from the model surface after each deflection
  - Line spacing and direction: Distance moved upward to start the next contour line
  - MAX. probe point interval

> The line spacing and MAX. probe point interval cannot exceed 5 mm.

```
10 TCH PROBE 7.0 CONTOUR LINES
11 TCH PROBE 7.1 TIME:200 X+50 Y+0
12 TCH PROBE 7.2 ORDER Y+/X+
13 TCH PROBE 7.3 TRAVEL 0.5 L.SPAC+1 PP.INT 0.2
```



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

Graphics and Status Displays

See "Test run and program run, graphics"

Defining the Workpiece in the Graphic Window

- In the open program, press the BLK FORM soft key

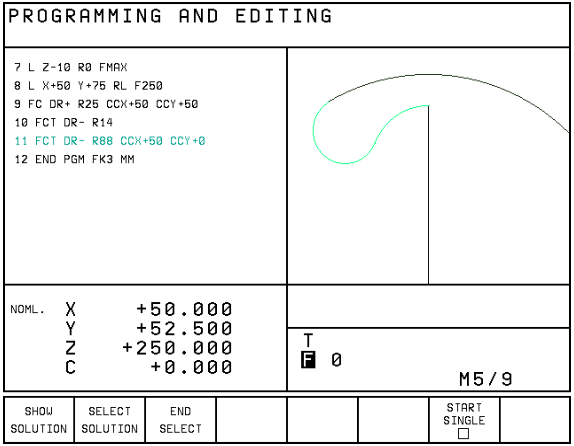
BLK FORM
- Spindle axis
- MIN and MAX POINT

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:



AUTO DRAW OFF / ON
- Automatic graphic generation during programming

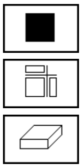
RESET + START
- Manually start graphic generation

START SINGLE
- Generate interactive graphics blockwise

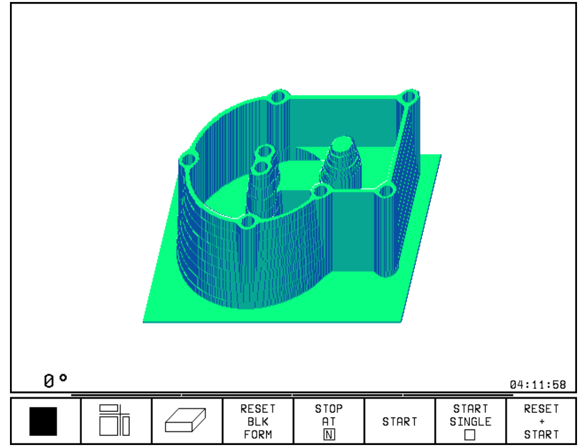
Test Graphics

Select the GRAPHICS or PGM+GRAPHICS screen layout!

In the test run mode 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 a screen layout showing the status information that you need.

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

- Tool position
- Feed rate
- Active M functions

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

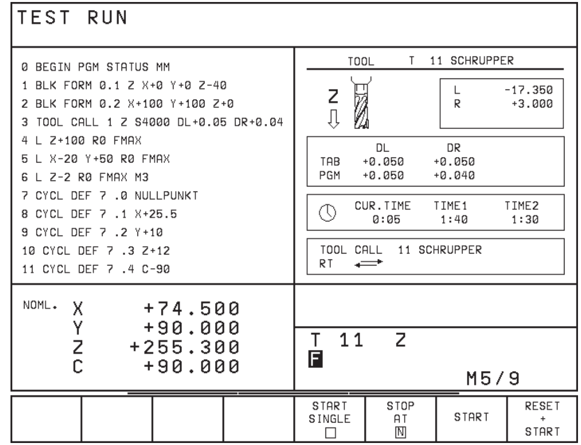
PGM + PGM STATUS ▶ Program information

PGM + POS. STATUS ▶ Tool positions

PGM + TOOL STATUS ▶ Tool data

PGM + C.TRANS. STATUS ▶ Coordinate transformations

PGM + T.PROBE STATUS ▶ Tool measurement



ISO-Programming

Programming Tool Movements with Cartesian Coordinates

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

Programming Tool Movements with Polar Coordinates

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

*) Effective blockwise

Drilling Cycles

G83	Pecking
G200	Drilling
G201	Reaming
G202	Boring
G203	Universal boring
G204	Counterbore back
G84	Tapping
G85	Rigid tapping (controlled spindle)

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

Multipass milling	
G230	Multipass milling
G231	Ruled surface

Cycles for Coordinate Transformation	
G53	Datum shift from datum tables
G54	Entering datum shift directly
G28	Mirror image
G73	Rotating the coordinate system
G72	Scaling factor: enlarging/reducing contours

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

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

*) Effective blockwise

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

*) Effective blockwise

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)
G55*	Automatic measurement with the 3D touch probe
G98*	Setting a label number

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)
D13	Find and assign an angle from the arc tangent of two sides or from the sine and cosine of an angle
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
D14	Output text to screen
D15	Output text or parameter contents through the data interface
D18	Read system data
D19	Transfer numerical values or Q parameters to the PLC

*) Effective blockwise

Addresses

%	Program beginning
A	Swivelling axis around X
B	Swivelling axis around Y
C	Rotary axis around Z
D	Define Q-parameter functions
E	Tolerance for rounding arc with M112
F	Feed rate in mm/min in positioning blocks
F	Dwell time in seconds with G04
F	Scaling factor with G72
G	G functions (see list of G functions)
H	Polar coordinate angle
H	Angle of rotation with G73
I	X coordinate of the circle center or pole
J	Y coordinate of the circle center or pole
K	Z coordinate of the circle center or pole
L	Label number with G98
L	Jump to a label number
L	Tool length with G99
M	Miscellaneous function
N	Block number
P	Cycle parameter for fixed cycles
P	Value or Q parameter with Q parameter definitions
Q	Variable Q parameter
R	Polar coordinate radius with G10/G11/G12/ G13/G15/G16/
R	Circle radius with G02/G03/G05
R	Corner radius with G25/G26/G27
R	Chamfer length with G24
R	Tool radius with G99
S	Spindle speed in rpm
S	Angle for spindle orientation with G36
T	Tool number with G99
T	Tool call
T	Call next tool with G51
U	Parallel axis to X
V	Parallel axis to Y
W	Parallel axis to Z
X	X axis
Y	Y axis
Z	Z axis
*	Character for end of block

Miscellaneous Functions M

M00	Stop program run/Stop spindle/Coolant off
M01	Optional program stop
M02	Stop program run/Stop spindle/Coolant off Jump back to block 1/Clear status display (depending on machine parameters)
M03	Spindle on clockwise
M04	Spindle on counterclockwise
M05	Stop spindle
M06	Tool change/Stop program run (depending on machine parameters) Stop spindle
M08	Coolant on
M09	Coolant off
M13	Spindle on clockwise/Coolant on
M14	Spindle on counterclockwise/Coolant on
M30	Same function as M02
M89	Vacant miscellaneous function or Cycle call, modally effective (depending on machine parameters)
M90	Constant contour speed at corners (effective only in lag mode)
M91	Within the positioning block: Coordinates are referenced to the machine datum
M92	Within the positioning block: The coordinates are referenced to a position defined by the machine tool builder
M93	Reserved
M94	Reduce rotary axis display to a value below 360°
M97	Machine small contour steps
M98	Suspend tool path compensation

M99	Cycle call, effective blockwise
M101	Automatic tool change after tool lifetime expires
M102	Reset M101
M103	Reduce the feed rate during plunging to factor F
M109	Constant contouring speed of tool cutting edge on arcs (increasing and decreasing the feed rate)
M110	Constant contouring speed of tool cutting edge on arcs (only decreasing the feed rate)
M111	Reset M109/M110
M112	Insert a rounding arc between two lines, with tolerance and limit angle
M113	Reset M112
M120	LOOK AHEAD: Calculate the radius-compensated tool path ahead of time
M124	Ignore points when calculating the rounding arc with M112
M126	Permit zero crossover on 360° rotary axes
M127	Cancel M126

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49/86 69/31-0
FAX +49/86 69/50 61
e-mail: info@heidenhain.de
www.heidenhain.de

HEIDENHAIN (G.B.) Limited
200 London Road, Burgess Hill
West Sussex RH15 9RD, Great Britain
☏ (0 14 44) 24 77 11
FAX (0 14 44) 87 00 24

309 742-23 · SW19 · 5 · 10/2002 · Bi · Printed in Germany · Subject to change without notice