HEIDENHAIN

Pilot

TNC 426B
TNC 430

NC Software
280 470 xx
280 471 xx

7/96

The Pilot

... is your concise programming guide for the HEIDENHAIN TNC 426 B 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
- 3D 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 CB, TNC 426 PB	280 470 xx
TNC 426 CF*, TNC 426 PF*	280 471 xx
TNC 430 CA, TNC 430 PA	280 470 xx
TNC 430 CE*, TNC 430 PE*	280 471 xx

*) Export version

Contents

Fundamentals	4
Contour Approach and Departure	13
Path Functions	18
FK Free Contour Programming	25
Subprograms and Program Section Repeats	33
Working with Cycles	36
Drilling Cycles	39
Pockets, Studs, and Slots	46
Point Patterns	55
SL Cycles	57
Multipass Milling	64
Coordinate Transformation Cycles	67
Special Cycles	73
Digitizing 3D Surfaces	75
Graphics and Status Displays	81
ISO Programming	84
Miscellaneous Functions M	88

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

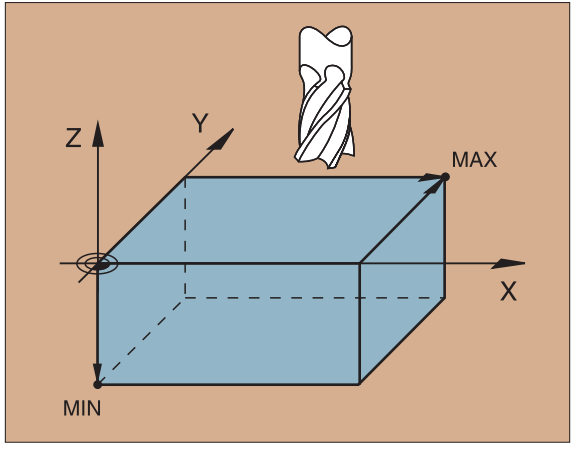
Files in the TNC	File type
Programs	
• in HEIDENHAIN format	.H
• in ISO format	.I
Tables for	
• Tools	.T
• Datums	.D
• Pallets	.P
• Points	.PNT
Texts as	
• ASCII files	.A

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
```



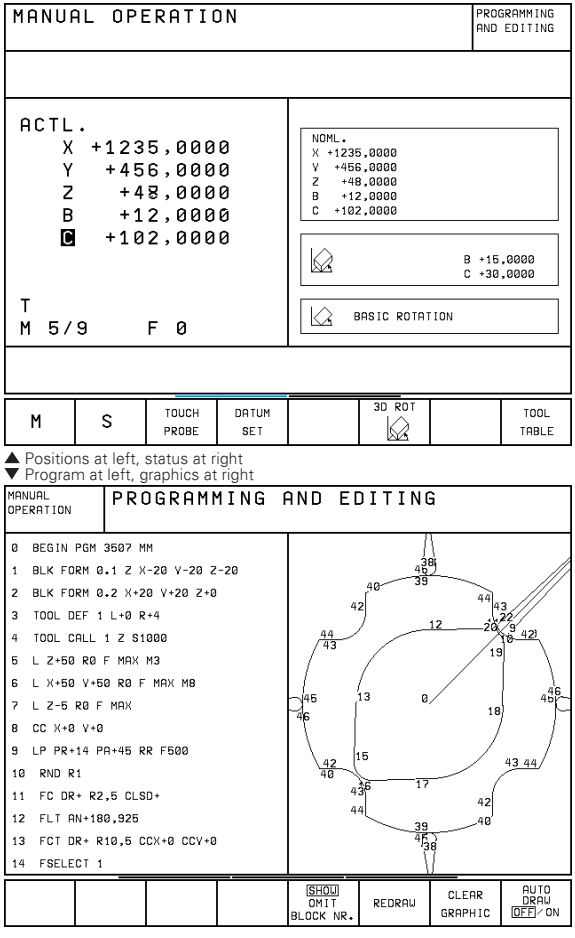
Choosing the Screen Layout

See “Introduction, the TNC 426 B, TNC 430”

▶ Show soft keys for setting the screen layout

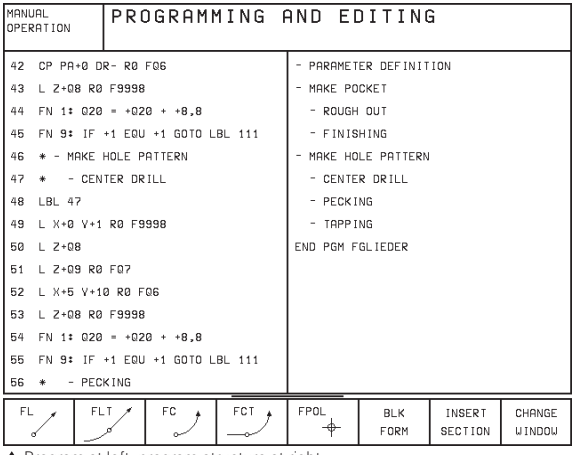
Mode of operation	Screen contents	
MANUAL OPERATION ELECTRONIC HANDWHEEL	Positions	POSITION
	Positions at left Status at right	POSITION + STATUS
POSITIONING WITH MANUAL DATA INPUT	Program	PGM
	Program at left Status at right	PGM + STATUS
PROGRAM RUN, FULL SEQUENCE	Program	PGM
PROGRAM RUN, SINGLE BLOCK TEST RUN	Program at left Program structure at right	PGM + SECTION
	Program at left Status at right	PGM + STATUS
	Program at left Graphics at right	PGM + GRAPHICS
	Graphics	GRAPHICS

Continued ▶



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

Mode of operation	Screen contents	
PROGRAMMING AND EDITING	Program	PGM
	Program at left Program structure at right	PGM + SECTION
	Program at left Programming graphics at right	PGM + GRAPHICS

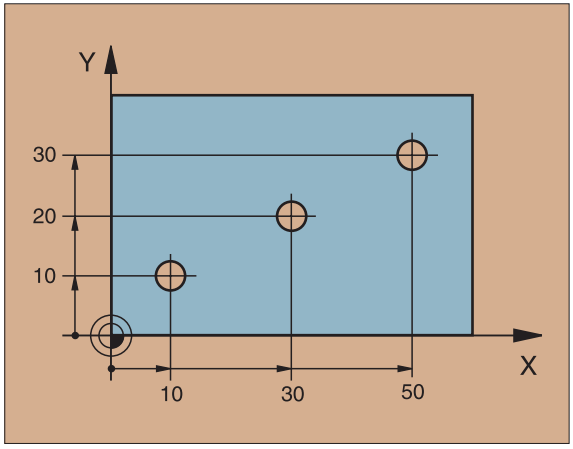


▲ Program at left, program structure at right

Absolute Cartesian Coordinates

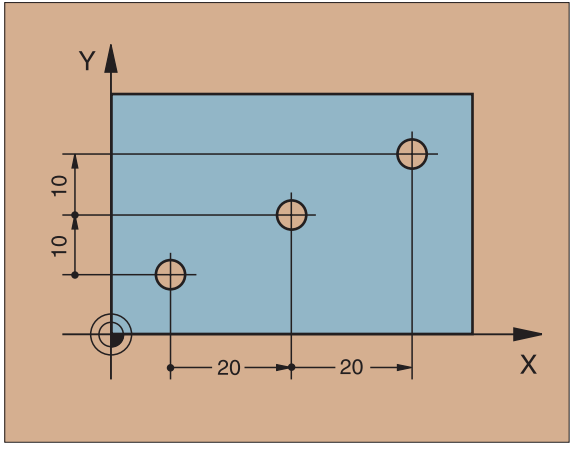
The dimensions are measured from the current datum.
The tool moves to the absolute coordinates.

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.



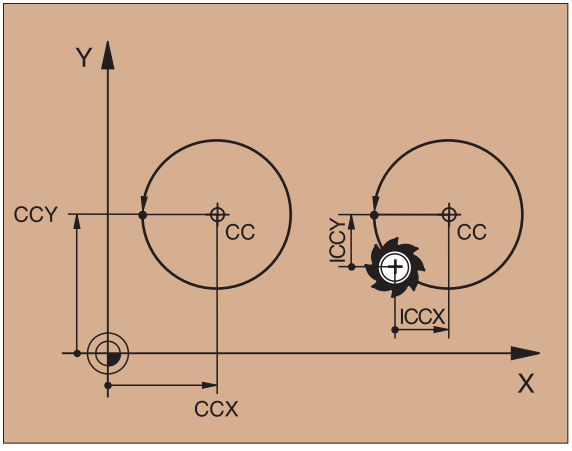
Circle Center and Pole: CC

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.

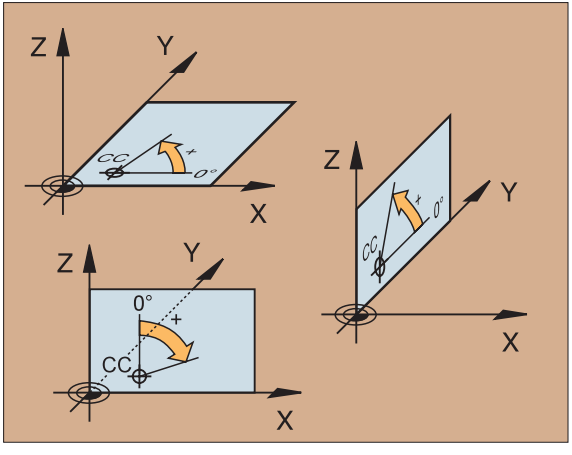
An incrementally defined circle center or pole is always measured from the last programmed position of the workpiece.



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

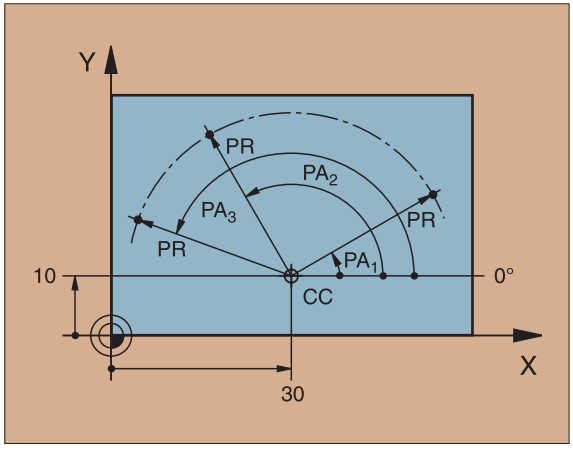
Tool data

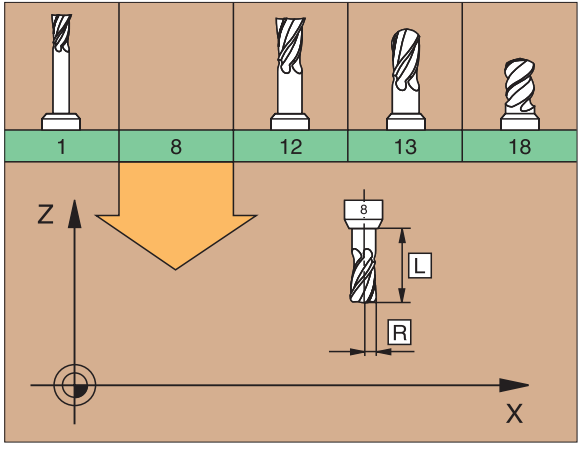
Every tool is designated by a tool number between 1 and 254 or, if you are using tool tables, by a tool name.

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 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

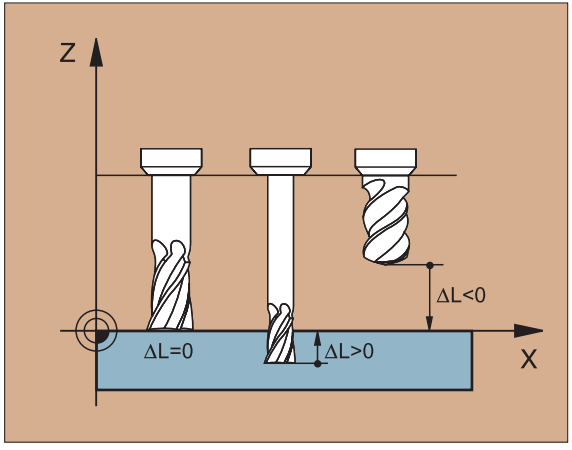
TOOL CALL

- TOOL NUMBER or name
- 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)

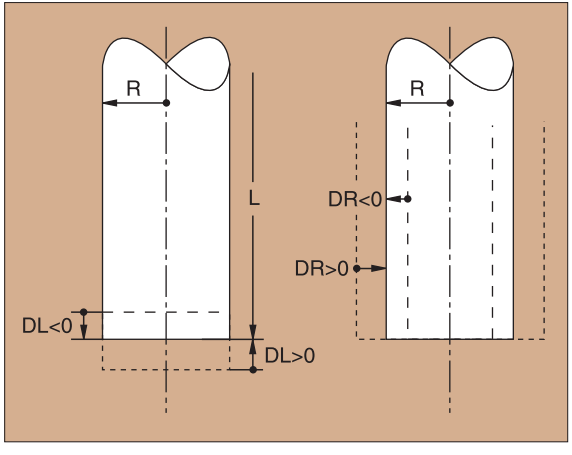
```
3 TOOL DEF 6 L+7.5 R+3
4 TOOL CALL 6 Z S2000 DL+1 DR+0.5
5 L Z+100 R0 FMAX
6 L X-10 Y-10 R0 FMAX M6
```

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!



▼ Oversizes on an end mill



Tool Compensation

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

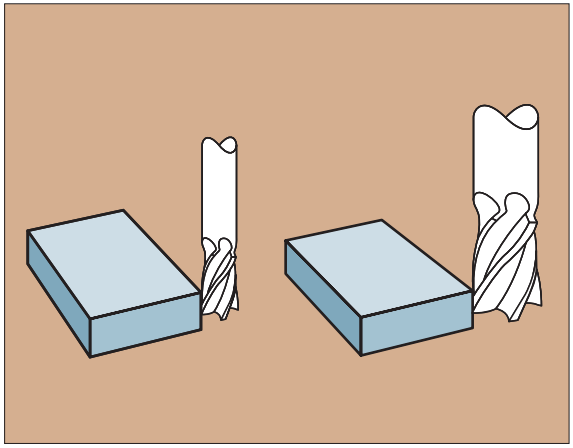
Length compensation
Beginning of effect:
▶ Tool movement in the spindle axis

End of effect:
▶ Tool exchange or tool with the length L=0

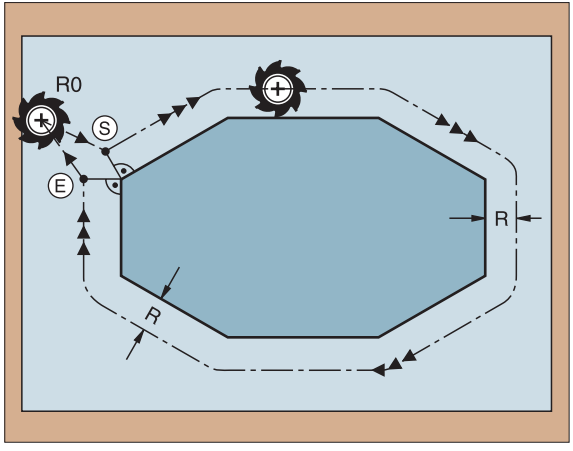
Radius compensation
Beginning of effect:
▶ Tool movement in the working plane with RR or RL

End of effect:
▶ Execution of a positioning block with R0

Working without radius compensation (e.g. drilling):
▶ Tool movement with R0



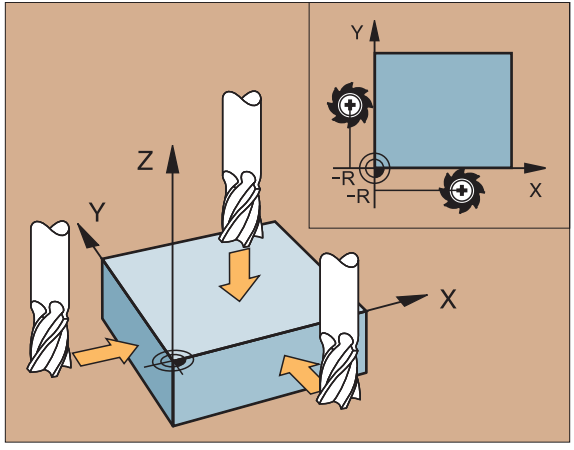
▼ S = Start; E = End



Datum Setting Without a 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.

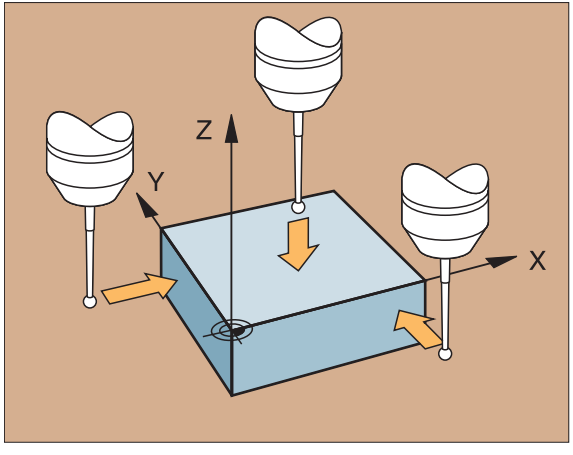
The following probe functions are provided by the MANUAL OPERATION and ELECTRONIC HANDWHEEL modes of operation:

Basic rotation

Datum setting in one axis

Datum setting at a corner

Datum setting at a circle center



Contour Approach and Departure

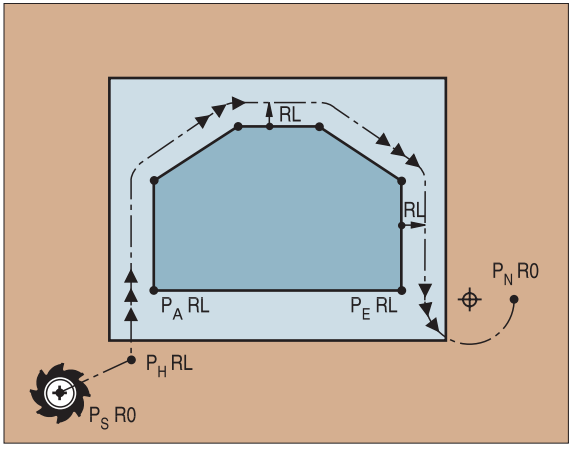
Starting point P_S
P_S lies outside of the contour and must be approached without radius compensation.

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

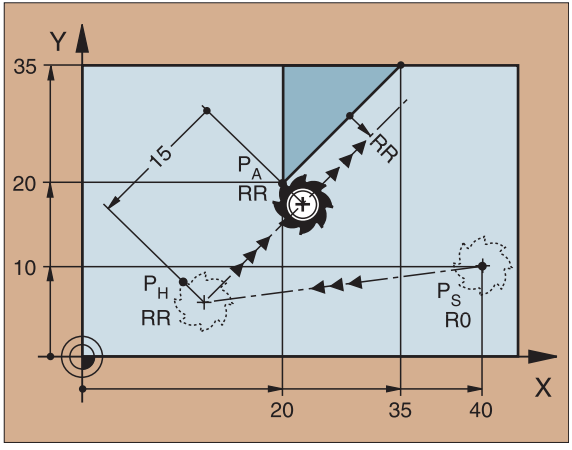
- Program a radius compensation in the APPR block!
- DEP blocks set the radius compensation to 0!

Approaching on a Straight Line with Tangential Connection

APPR LT

- 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
```

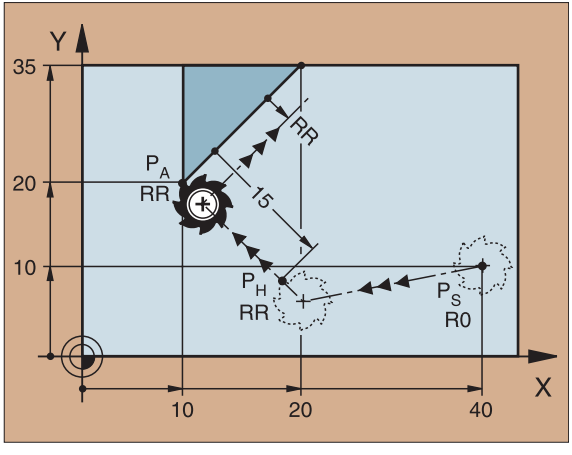


Approaching on a Straight Line Perpendicular to the First Contour Element

APPR LN

- 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

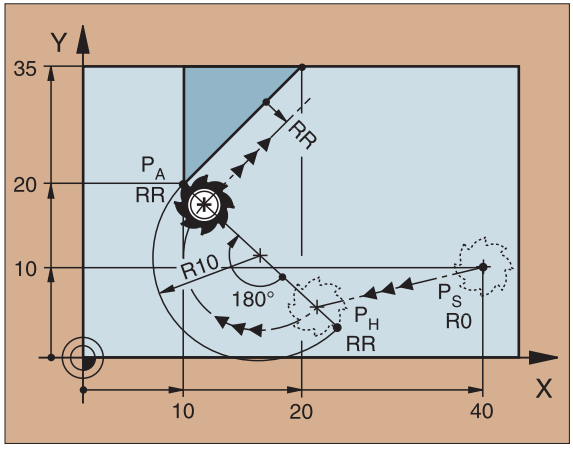
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LN X+10 Y+20 LEN 15 RR F100
9 L X+20 Y+35
```



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

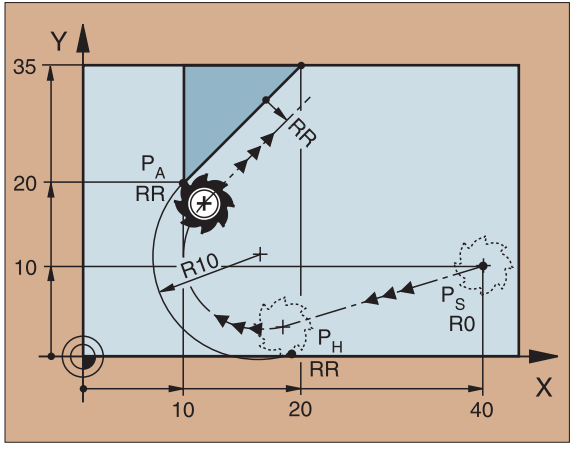
```
7 L X+40 Y+10 R0 FMAX M3
8 APPR CT X+10 Y+20 CCA 180 R10 RR F100
9 L X+20 Y+35
```



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
```

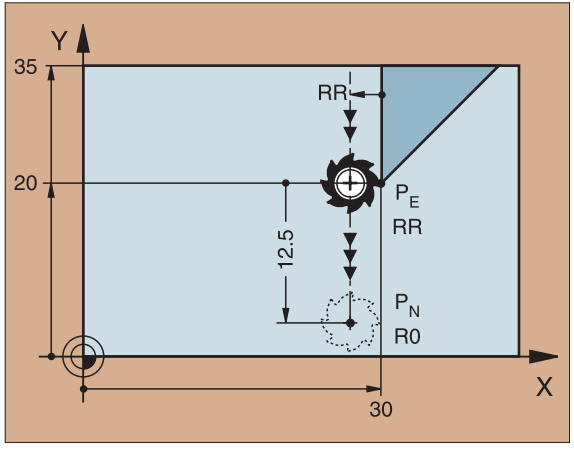


Departing Tangentially on a Straight Line

DEP LT

▶ 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
```

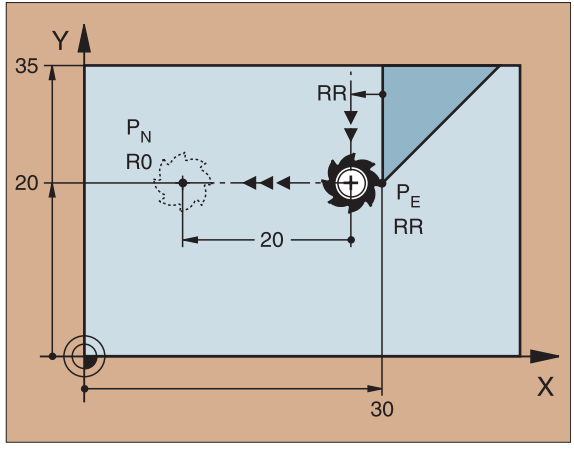


Departing on a Straight Line Perpendicular to the Last Contour Element

DEP LN

▶ 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
```

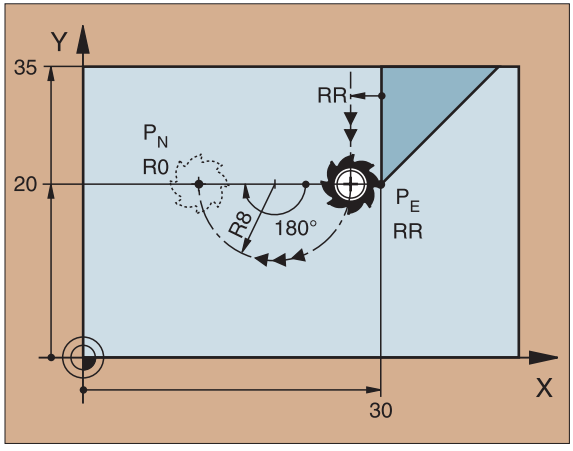


Departing Tangentially on an Arc

DEP CT

- 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
```

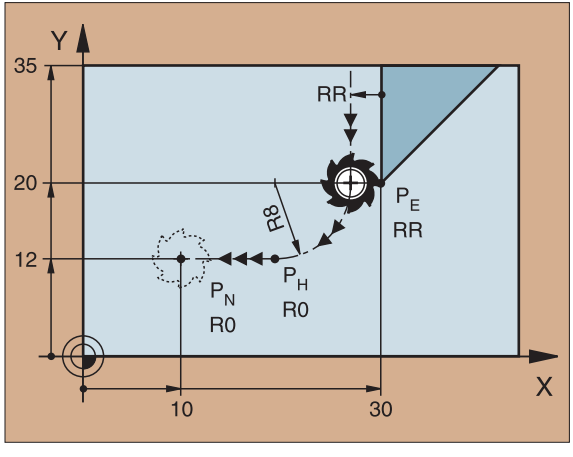


Departing on an Arc Tangentially Connecting the Contour and a Straight Line

DEP LCT

- 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!

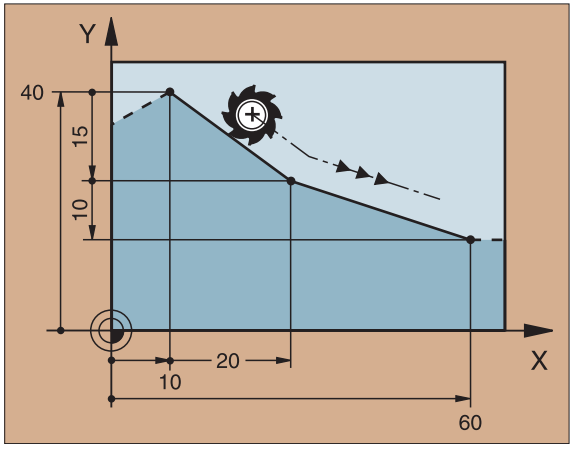
Path functions		
Straight line		Page 19
Chamfer between two straight lines		Page 20
Corner rounding		Page 20
Circle center or pole for polar coordinates		Page 21
Circular path around the circle center CC		Page 21
Circular path with known radius		Page 22
Circular path with tangential connection to previous contour		Page 23

Straight Line

- COORDINATES of the straight line end point
- TOOL RADIUS COMPENSATION RR/RL/R0
- FEED RATE F
- MISCELLANEOUS FUNCTION M

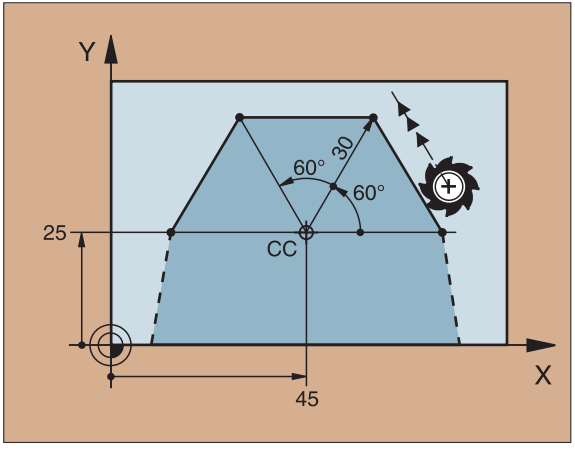
With Cartesian coordinates:

```
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
```



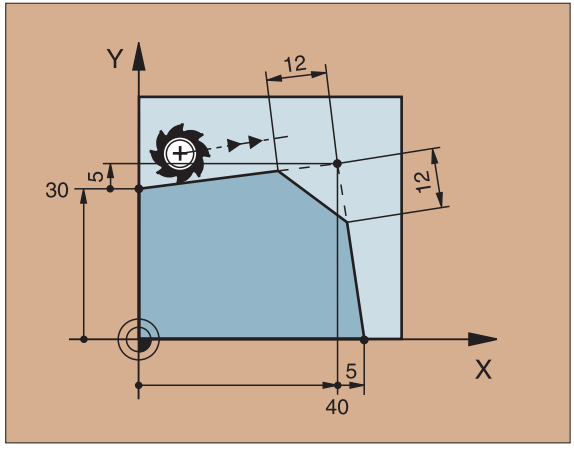
- You must first define the pole CC before you can program polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!

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!



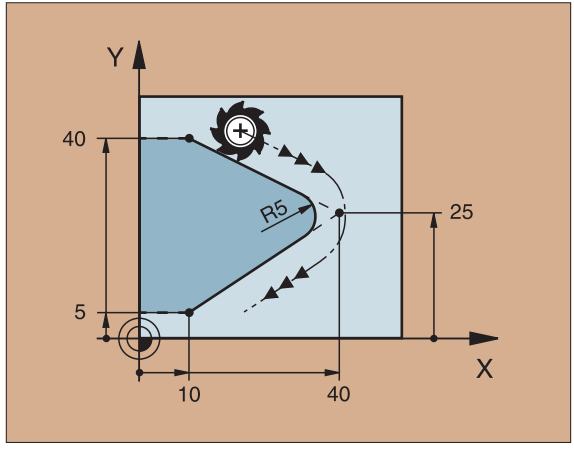
Corner Rounding

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

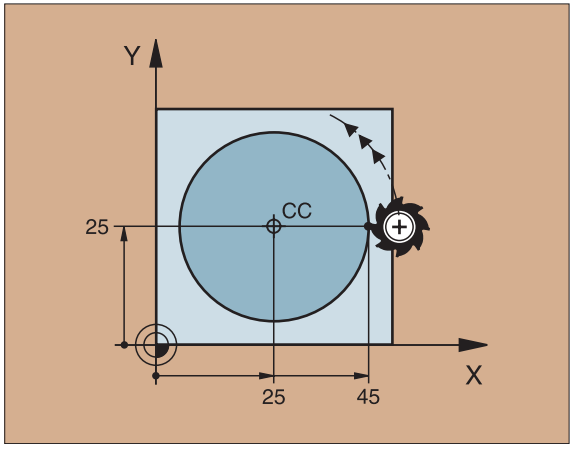
▶ COORDINATES of the circle center CC

▶ COORDINATES of the arc end point
▶ DIRECTION OF ROTATION DR

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

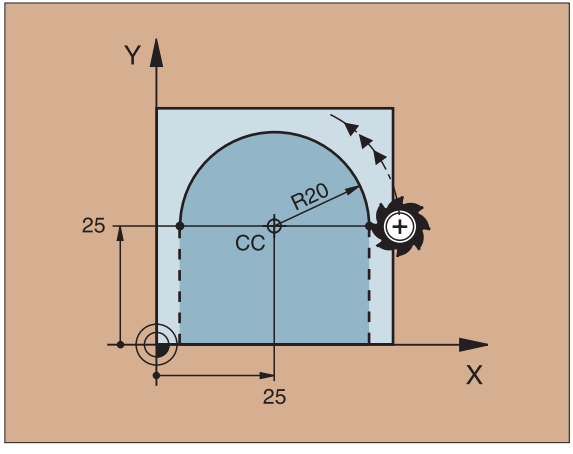
With Cartesian coordinates:

```
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+
```

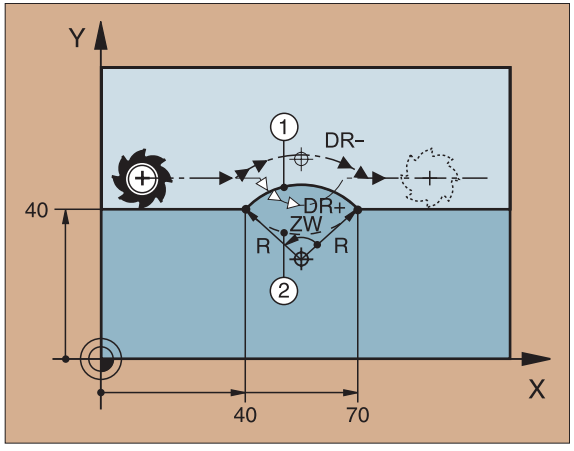


- Define the pole CC before programming polar coordinates!
- Program the pole CC only in Cartesian coordinates!
- The pole CC remains effective until you define a new one!
- The arc end point can be defined only with the polar coordinate angle (PA)!

Circular Path with Known Radius (CR)

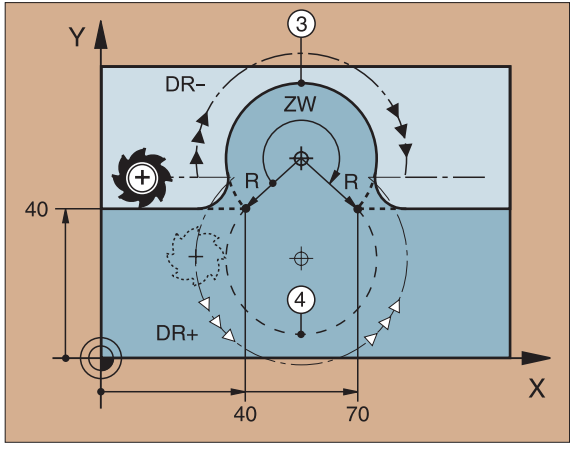
- COORDINATES of the arc end point
- RADIUS R
  If the central angle ZW > 180, R is negative.
  If the central angle ZW < 180, R is positive.
- DIRECTION OF ROTATION DR

`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



▲ Arcs 1 and 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 3 and 4

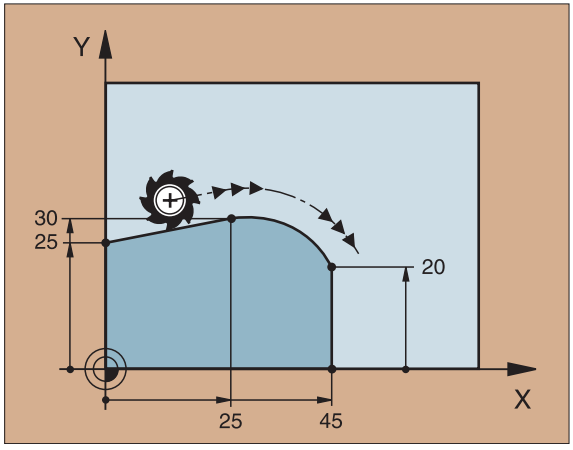
Circular Path CT with Tangential Connection

CT

- COORDINATES of the arc end point
- RADIUS COMPENSATION RR/RL/R0
- FEED RATE F
- MISCELLANEOUS FUNCTION M

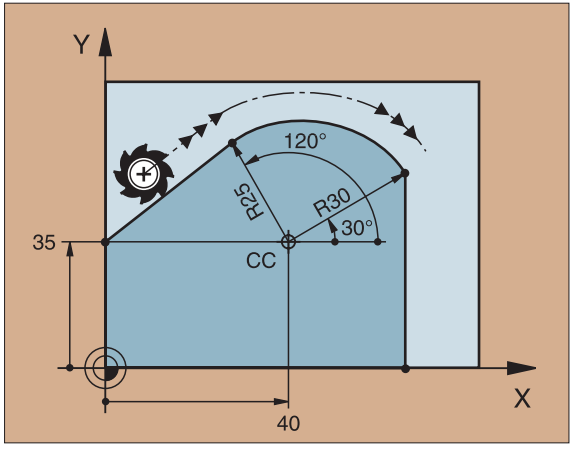
With Cartesian coordinates:

```
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)

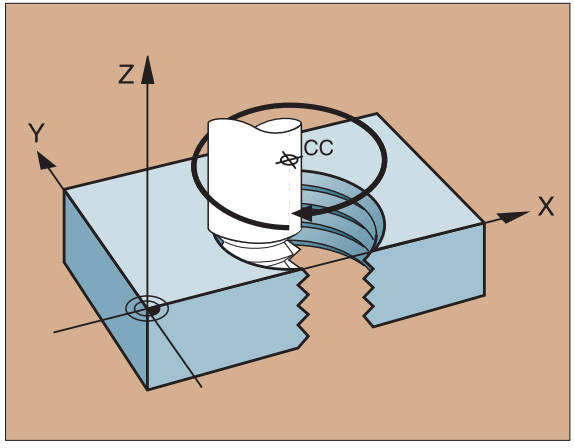
Path revolutions:	n	=	Thread revolutions + overrun at start and end of thread
Total height:	h	=	Pitch P x path revolutions n
Incr. coord. angle:	IPA	=	Path revolutions n x 360°
Start angle:	PA	=	Angle at start of thread + angle for overrun
Start coordinate:	Z	=	Pitch P x (thread revolutions + thread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

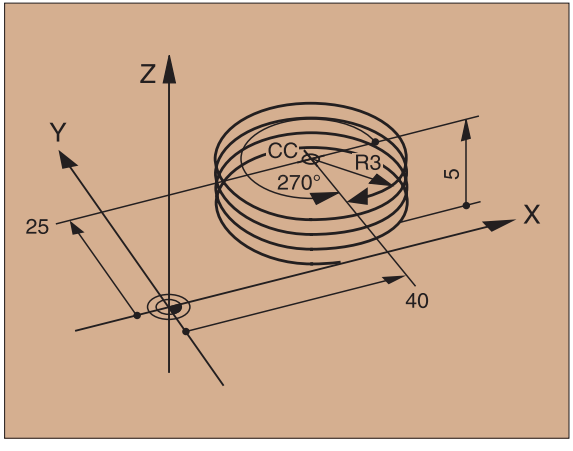
Shape of helix

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
```





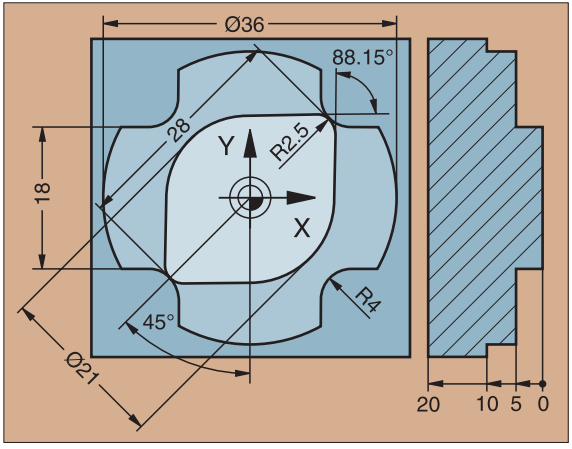
FK Free Contour Programming

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 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	To show the possible solutions
FSELECT	To enter the displayed solution in the part program
EDIT	To enter data for subsequent contour elements
START SINGLE	To graphically display the next programmed block

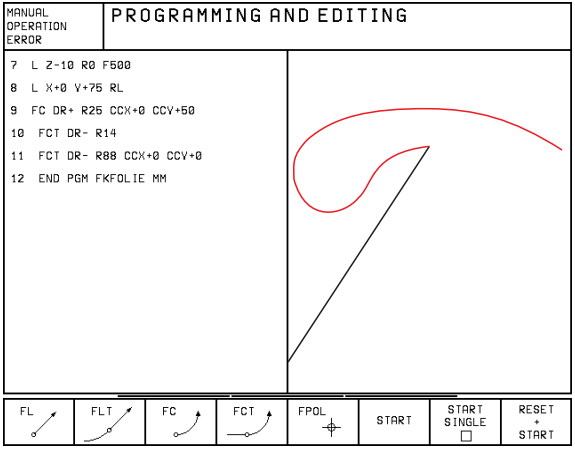
Standard colors of the interactive graphics

Fully defined contour element

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

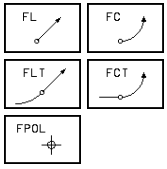
The element is one of an infinite number of solutions

Contour element from a subprogram



Initiating the FK Dialog

Straight Circular

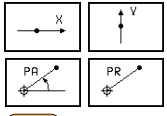


Contour element without tangential connection

Contour element with tangential connection

Pole for FK programming

End Point Coordinates X, Y or PA, PR

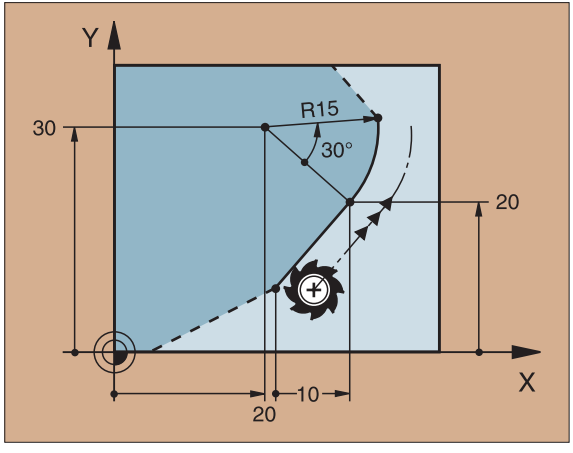


Cartesian coordinates X and Y

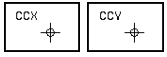
Polar coordinates referenced to FPOL

Incremental input

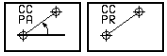
```
7 FPOL X+20 Y+30
8 FL IX+10 Y+20 RR F100
9 FCT PR+15 IPA+30 DR+ R15
```



Circle Center (CC) in an FC/FCT block



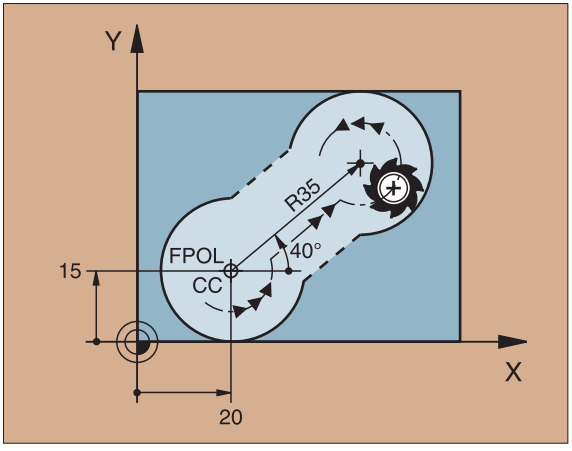
Cartesian coordinates of the circle center



Polar coordinates of the circle center referenced to FPOL

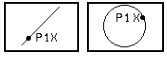
Incremental input

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



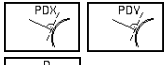
Auxiliary Points

... P1, P2, P3 on a contour



For straight lines: up to 2 auxiliary points
For circles: up to 3 auxiliary points

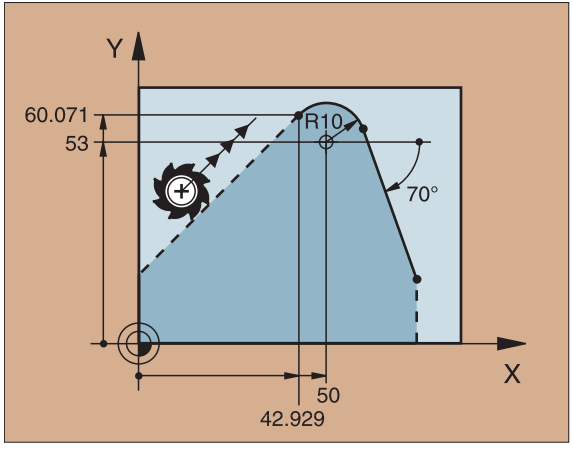
... next to a contour



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

Data on a circular path

Gradient angle of the entry tangent

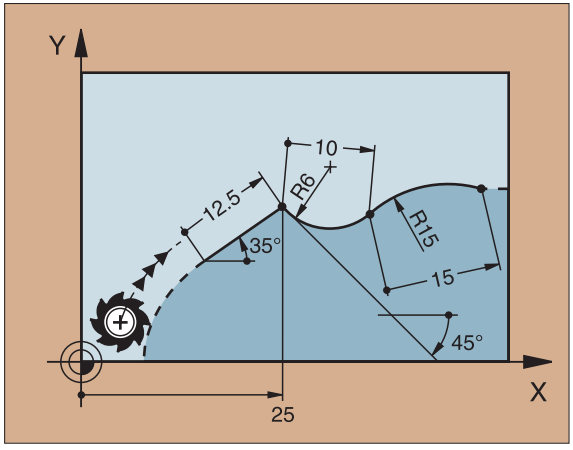
Length of an arc chord

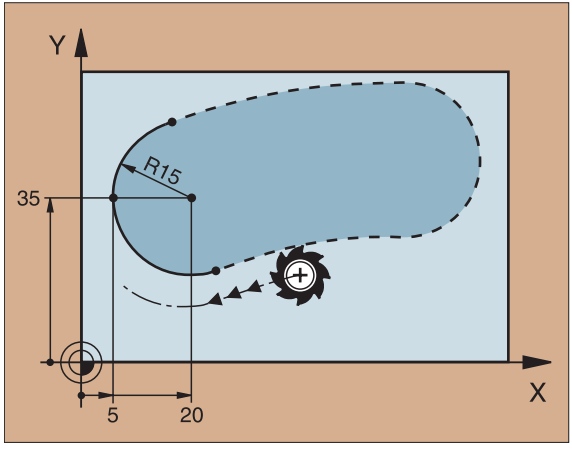
```
27 FLT X+25 LEN 12.5 AN+35 RL F200
28 FC DR+ R6 LEN 10 AN-45
29 FCT DR- R15 LEN 15
```

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-
```



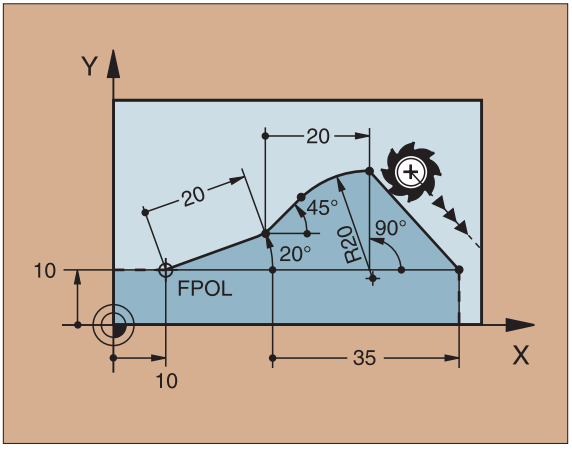


Values Relative to Block N: Entering Coordinates

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

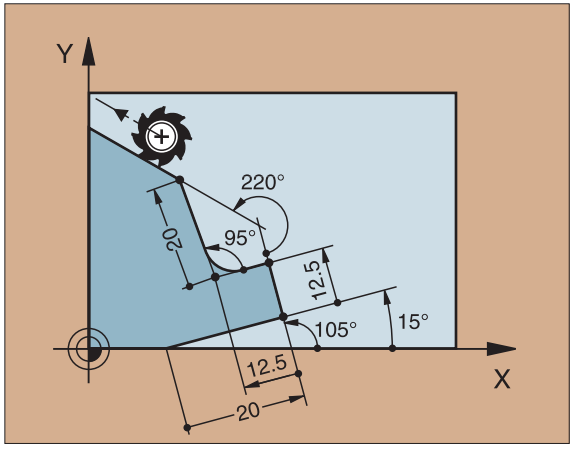
RAN N — Gradient angle

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
22 FL IAN+220 RAN 18
```



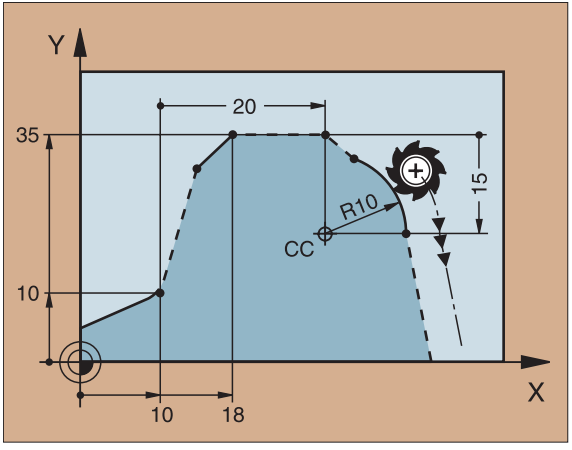
Values Relative to Block N: Circle Center CC

RCCX N RCCY N — Cartesian coordinates of a circle center relative to block N

RCCPR N RCCPA 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.

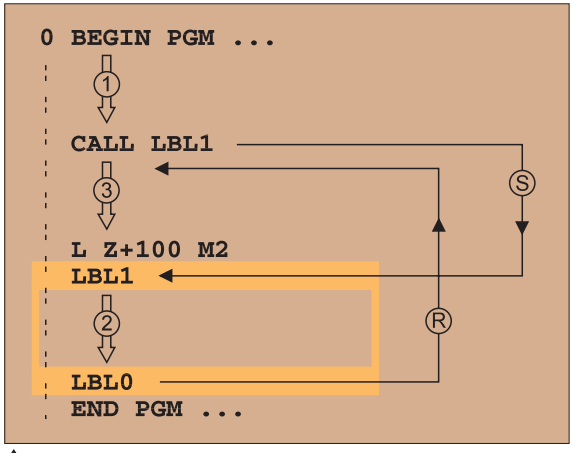
It's good practice to place subprograms after the main program end (M2).

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

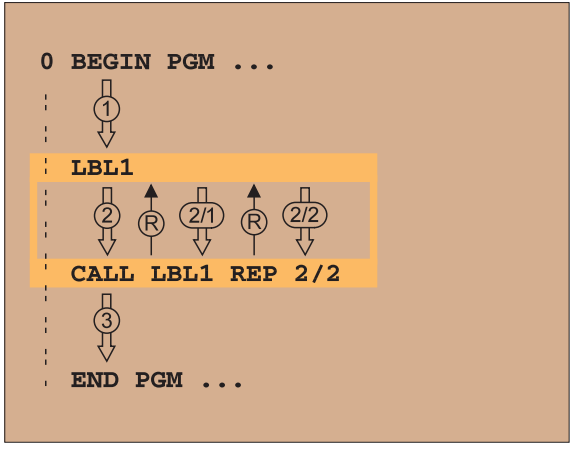
Working with Program Section Repeats

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!



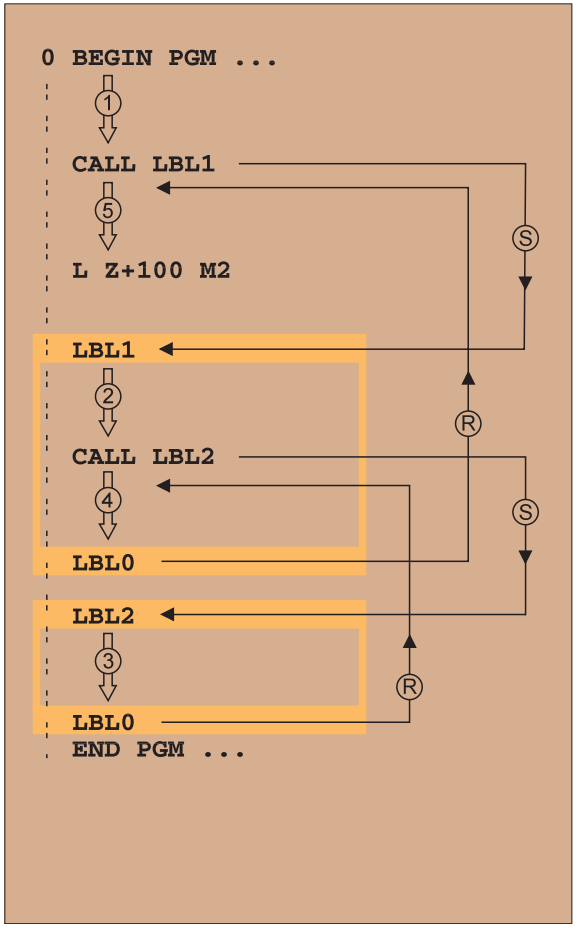
S = Jump; R = Return jump



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!

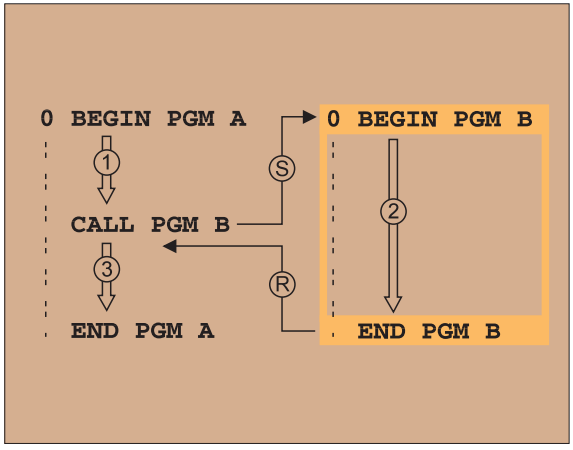


S = Jump; R = Return jump ▶

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

200 ▶ Select the cycle

Drilling Cycles		
1	PECKING	Page 39
200	DRILLING	Page 40
201	REAMING	Page 41
202	BORING	Page 42
203	UNIVERSAL DRILLING	Page 43
2	TAPPING	Page 44
17	RIGID TAPPING	Page 45
18	THREAD CUTTING	Page 45
Pockets, Studs, and Slots		
4	POCKET MILLING	Page 46
212	POCKET FINISHING	Page 47
213	STUD FINISHING	Page 48
5	CIRCULAR POCKET MILLING	Page 49
214	CIRCULAR POCKET FINISHING	Page 50
215	CIRCULAR STUD FINISHING	Page 51
3	SLOT MILLING	Page 52
210	SLOT WITH RECIP. PLUNGE	Page 53
211	CIRCULAR SLOT	Page 54
Point Patterns		
220	CIRCULAR PATTERN	Page 55
221	LINEAR PATTERN	Page 56
SL Cycles		
14	CONTOUR GEOMETRY	Page 58
20	CONTOUR DATA	Page 59
21	PILOT DRILLING	Page 60
22	ROUGH-OUT	Page 60
23	FLOOR FINISHING	Page 61
24	SIDE FINISHING	Page 61
25	CONTOUR TRAIN	Page 62
27	CYLINDER SURFACE	Page 63

Continued on next page ▶

Multipass Milling		
30	RUN DIGITIZED DATA	Page 64
230	MULTIPASS MILLING	Page 65
231	RULED SURFACE	Page 66
Cycles for Coordinate Transformations		
7	DATUM SHIFT	Page 67
8	MIRROR IMAGE	Page 68
10	ROTATION	Page 69
19	WORKING PLANE	Page 70
11	SCALING FACTOR	Page 71
26	AXIS-SPECIFIC SCALING	Page 72
Special Cycles		
9	DWELL TIME	Page 73
12	PGM CALL	Page 73
13	ORIENTED SPINDLE STOP	Page 74

Graphic Support During Cycle Programming

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

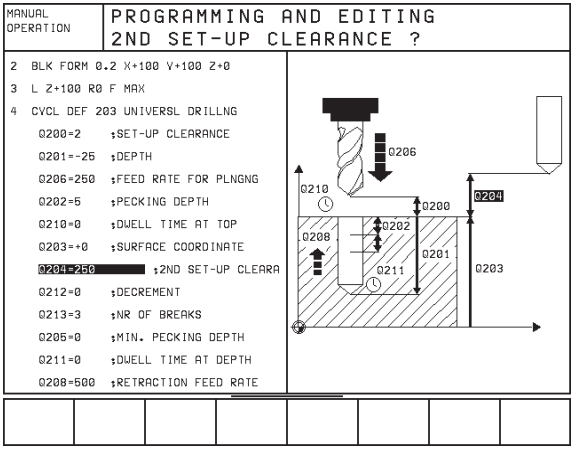
Calling a Cycle

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

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)

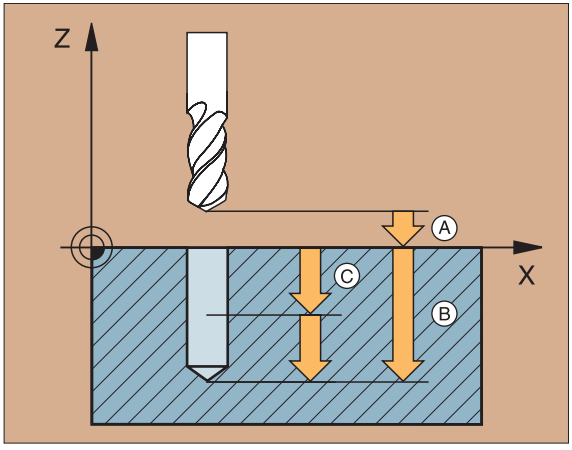


Drilling Cycles

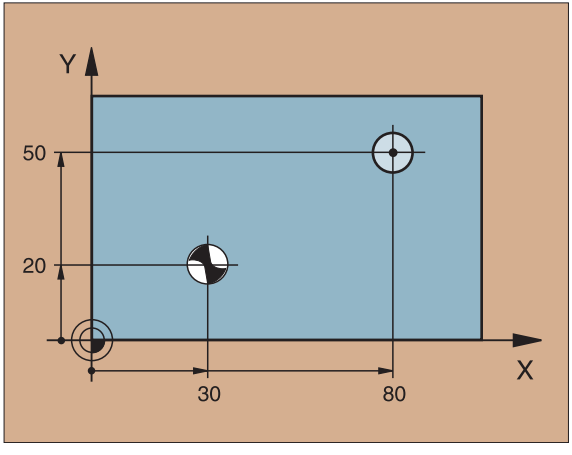
PECKING (1)

- 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.



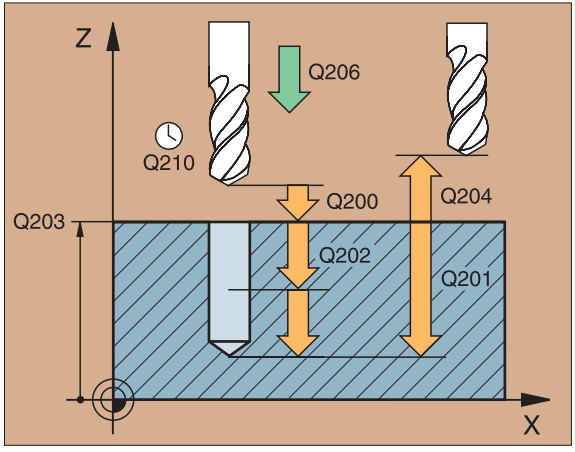

```
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP 2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG 7.5
10 CYCL DEF 1.4 DWELL 1
11 CYCL DEF 1.5 F80
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 L Z+2 FMAX M99
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```



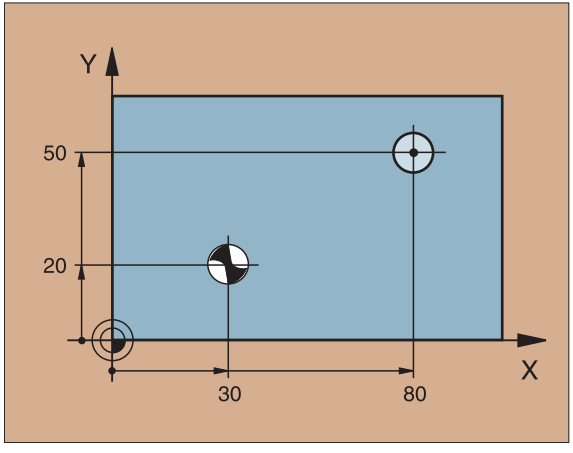
DRILLING (200)

- 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.



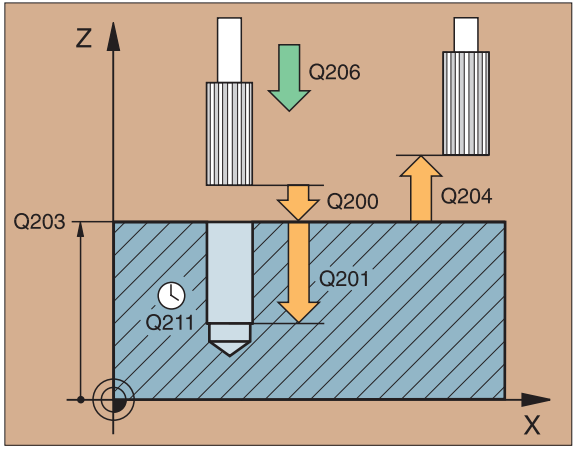

```
11 CYCL DEF 200 DRILLING
   Q200 = 2
   Q201 = -15
   Q206 = 250
   Q202 = 5
   Q210 = 0
   Q203 = +0
   Q204 = 100
12 L Z+100 R0 FMAX M6
13 L X+30 Y+20 FMAX M3
14 CYCL CALL
15 L X+80 Y+50 FMAX M99
16 L Z+100 FMAX M2
```



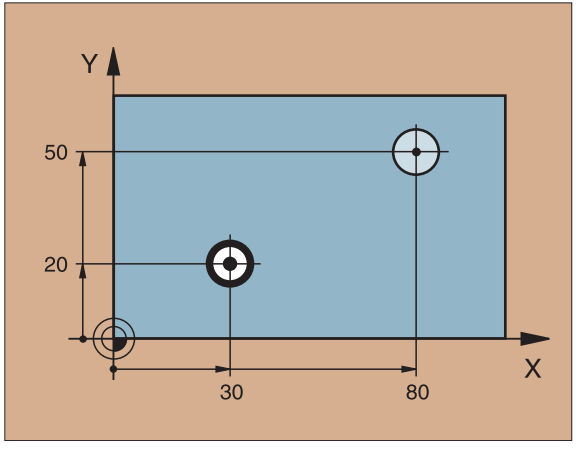
REAMING (201)

- 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
   Q201 = -15
   Q206 = 100
   Q211 = 0.5
   Q208 = 250
   Q203 = +0
   Q204 = 100
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
```



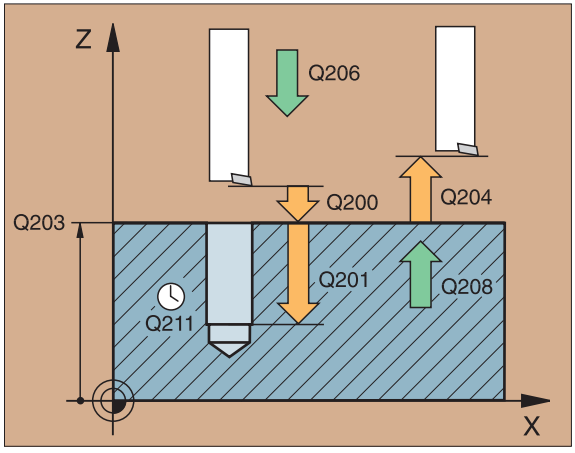
BORING (202)

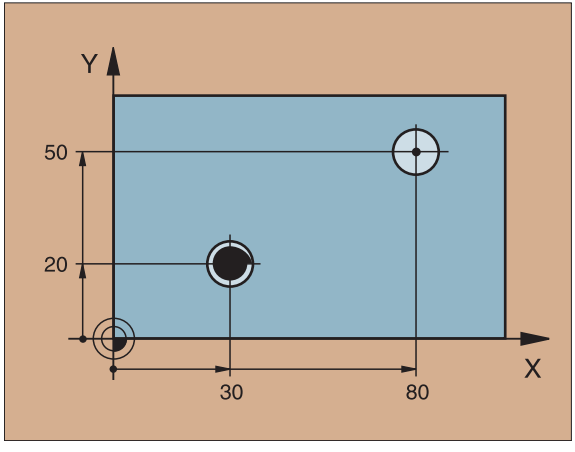
Danger of collision! Choose a disengaging direction that moves the tool away from the wall of the hole.

- 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 DIRECTN (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
   Q201 = -15
   Q206 = 100
   Q211 = 0.5
   Q208 = 250
   Q203 = +0
   Q204 = 100
   Q214 = 1
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
```

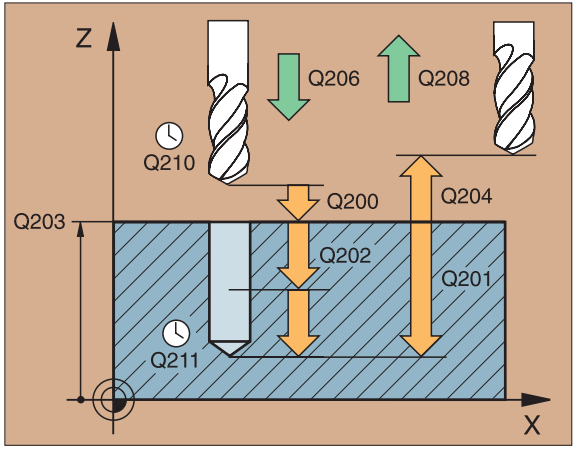




UNIVERSAL DRILLING (203)

- 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

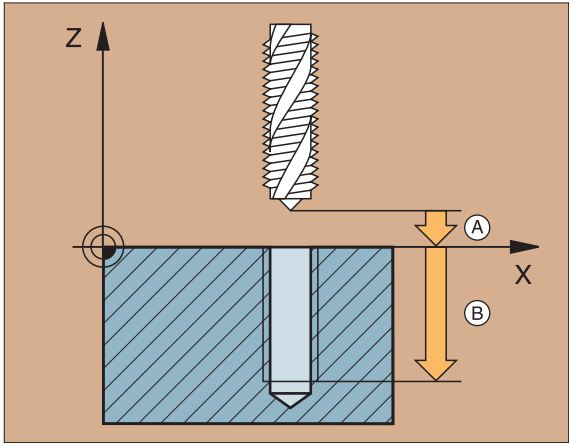
The TNC automatically pre-positions the tool in the tool axis. If the DEPTH is greater than or equal to the PECKING DEPTH, the tool drills to the DEPTH in one plunge.



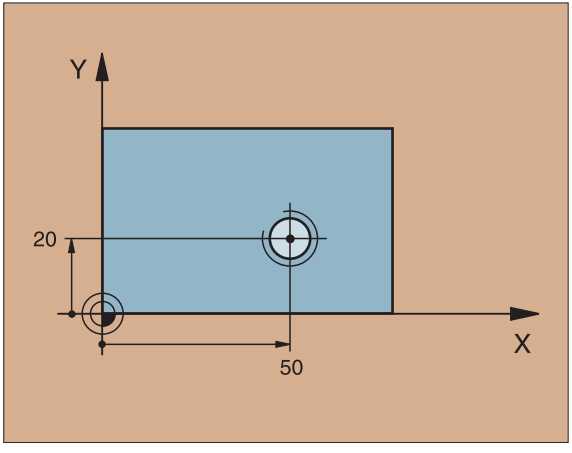
TAPPING with Floating Tap Holder (2)

- 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
```

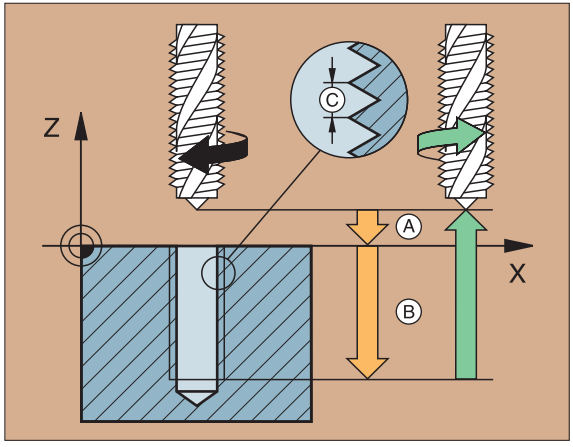


RIGID TAPPING (17)

- 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: –

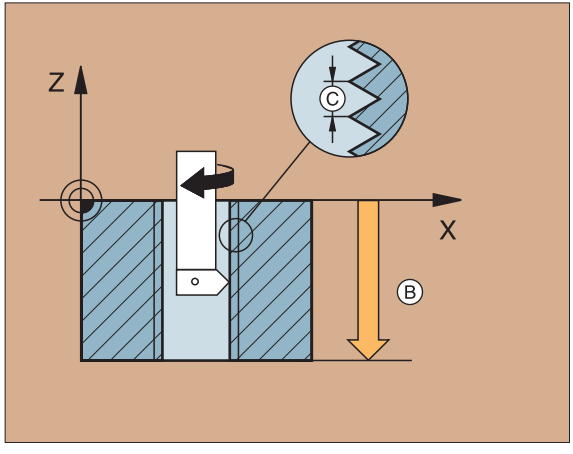


THREAD CUTTING (18)

- 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: –



Pockets, Studs, and Slots

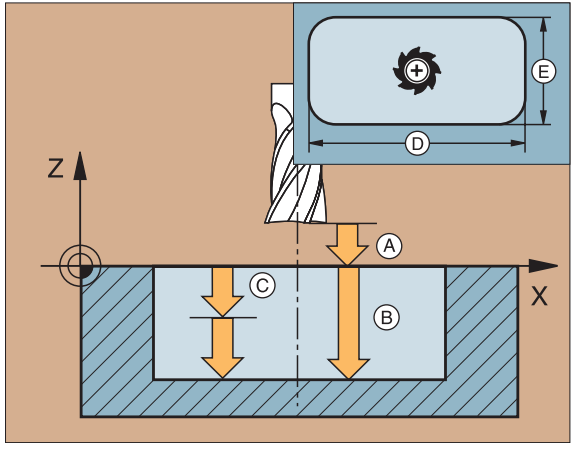
POCKET MILLING (4)

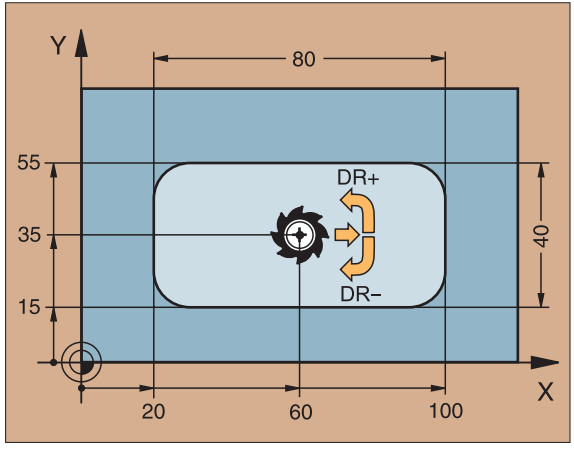
This cycle requires either a center-cut end mill (ISO 1641) or pilot drilling at the pocket center!

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
```

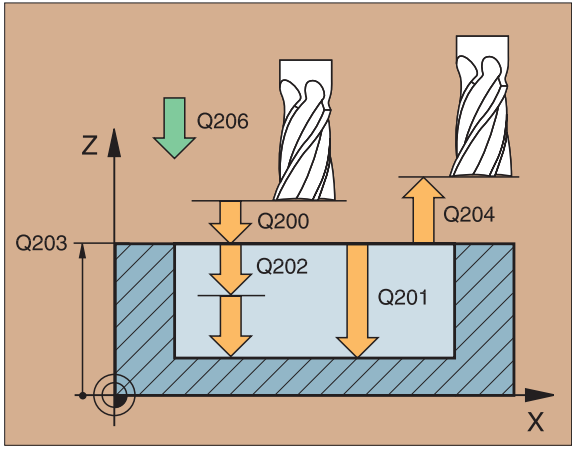


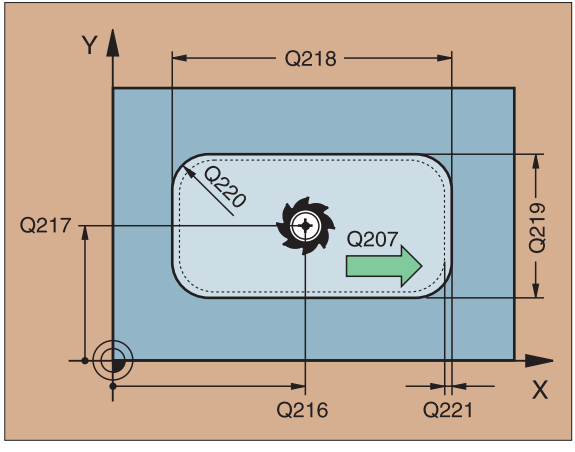


POCKET FINISHING (212)

- 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.

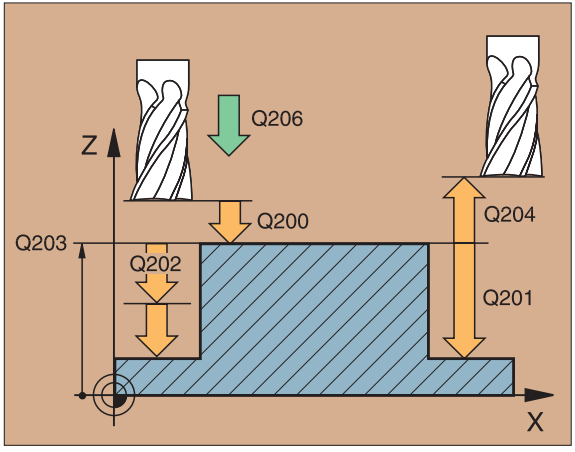


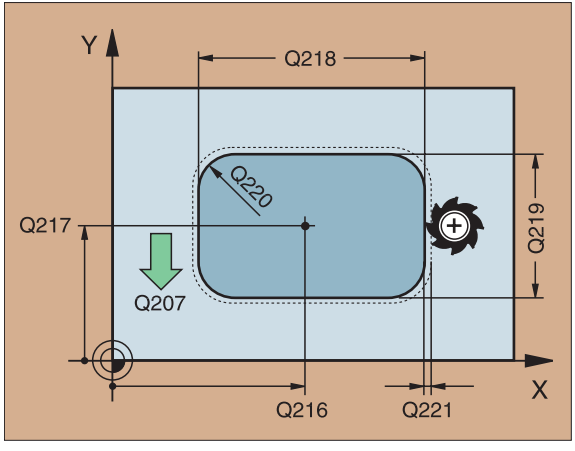


STUD FINISHING (213)

- 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
  - 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.



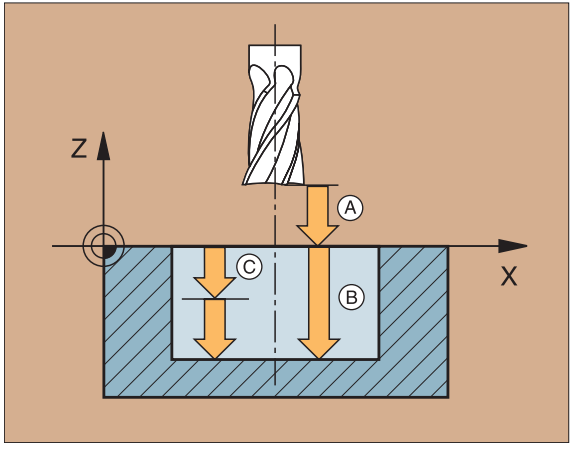


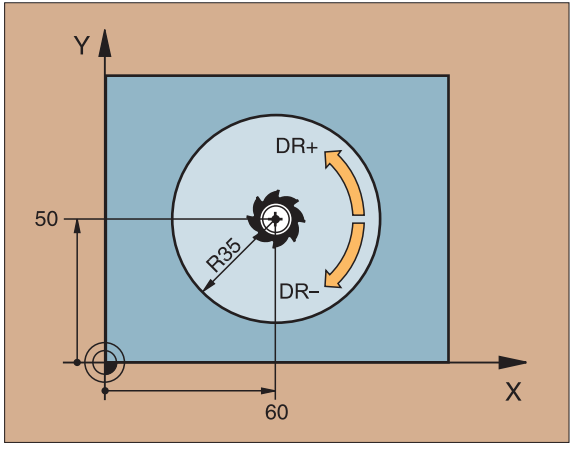
CIRCULAR POCKET MILLING (5)

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
```

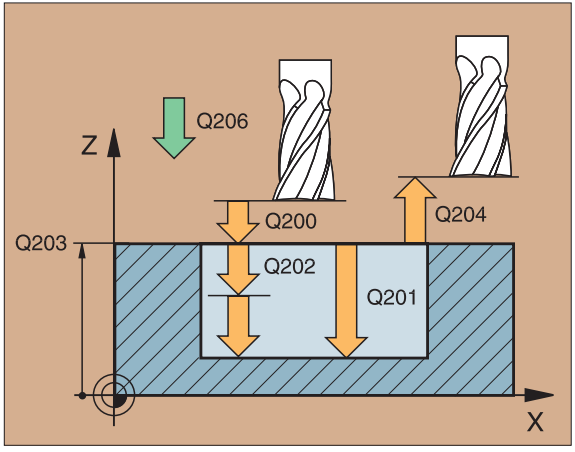


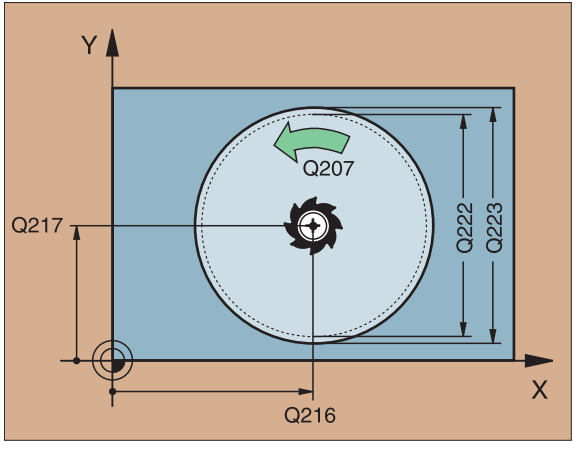


CIRCULAR POCKET FINISHING (214)

- 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.

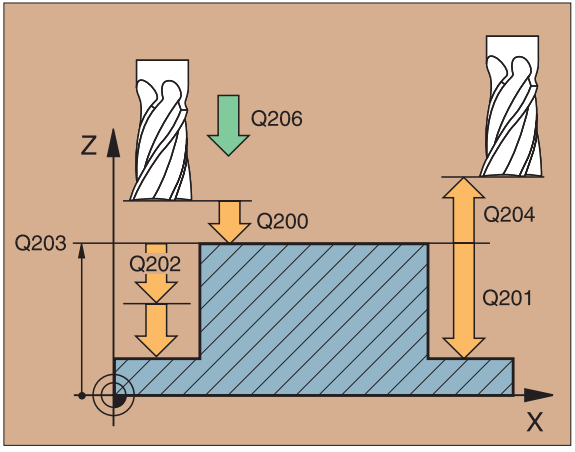


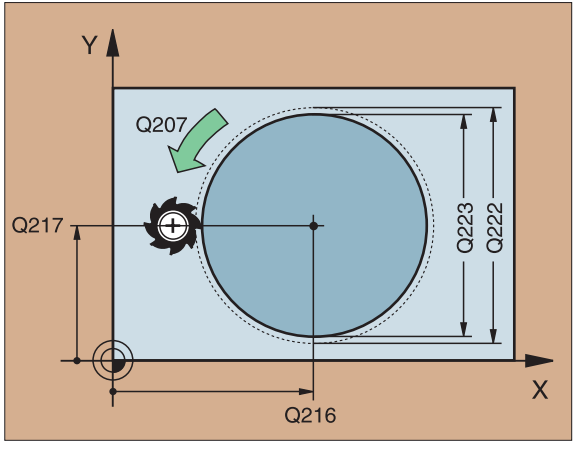


CIRCULAR STUD FINISHING (215)

- 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.

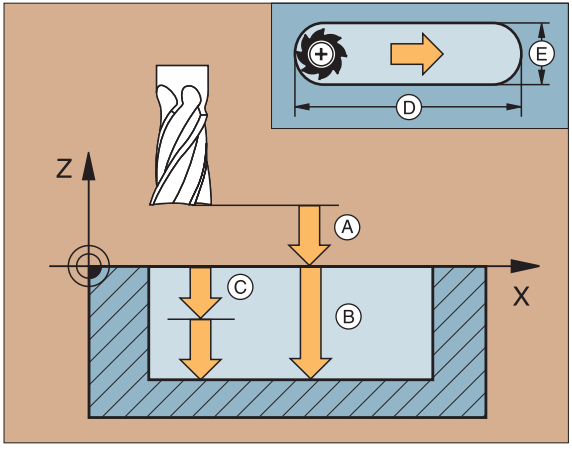


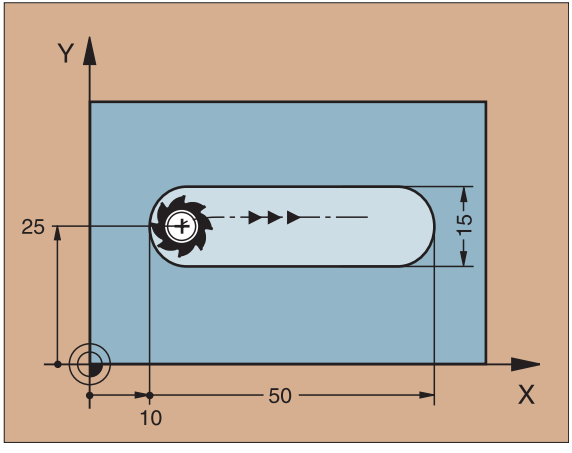


SLOT MILLING (3)

- 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)





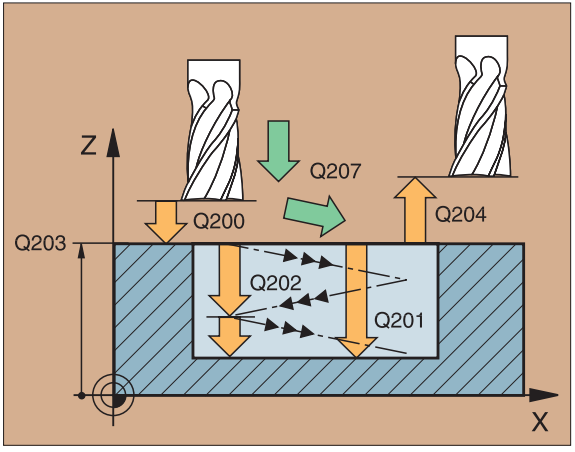

```
10 TOOL DEF 1 L+0 R+6
11 TOOL CALL 1 Z S1500
12 CYCL DEF 3.0 SLOT MILLING
13 CYCL DEF 3.1 SET UP 2
14 CYCL DEF 3.2 DEPTH -15
15 CYCL DEF 3.3 PECKG 5 F80
16 CYCL DEF 3.4 X+50
17 CYCL DEF 3.5 Y15
18 CYCL DEF 3.6 F120
19 L Z+100 R0 FMAX M6
20 L X+16 Y+25 R0 FMAX M3
21 L Z+2 M99
```

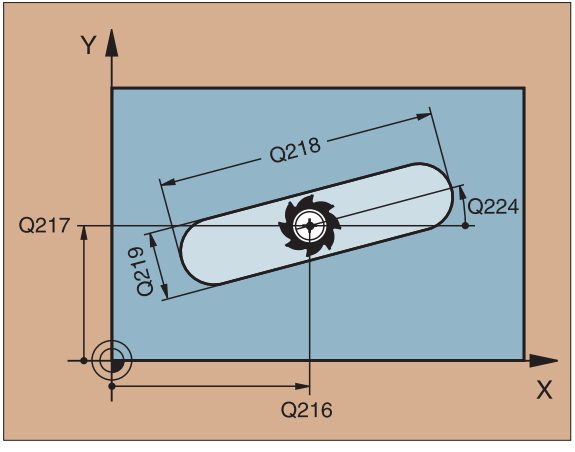
SLOT WITH RECIPROCATING PLUNGE-CUT (210)

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.



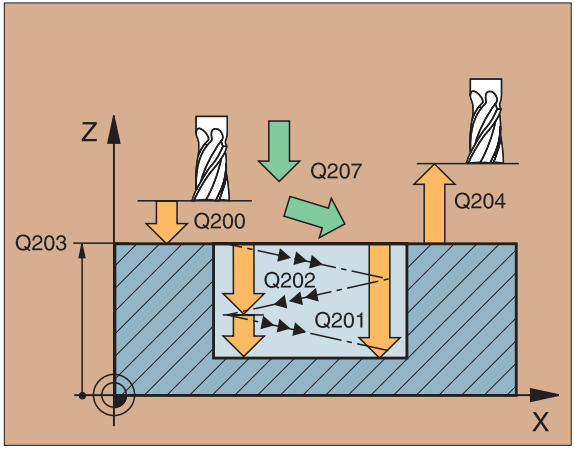


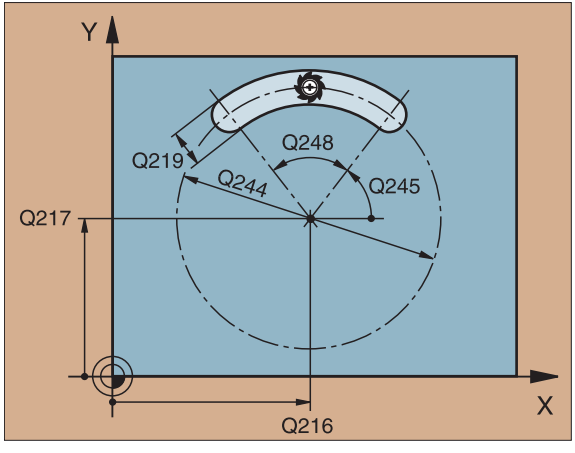
CIRCULAR SLOT with reciprocating plunge (211)

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.





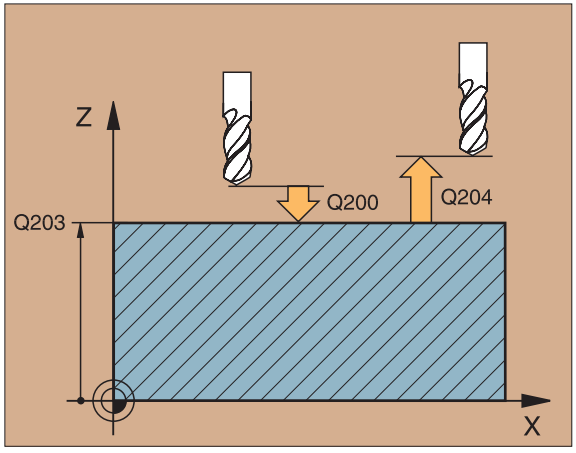
Point Patterns

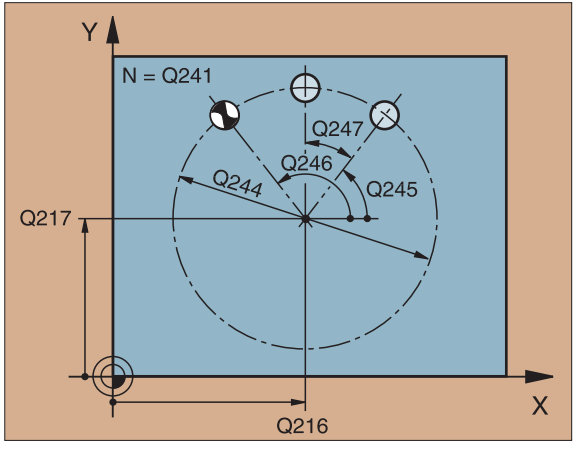
CIRCULAR PATTERN (220)

- 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 OF 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, 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.



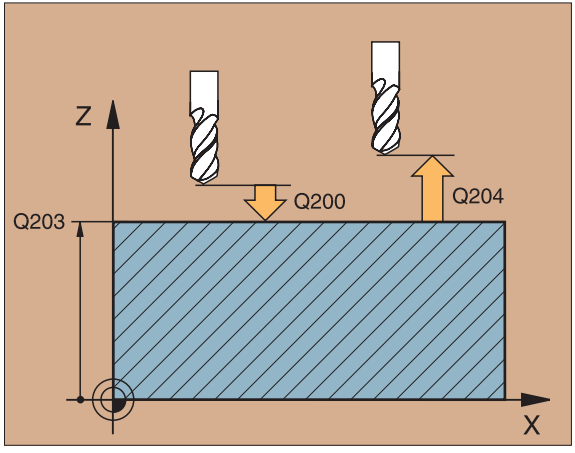


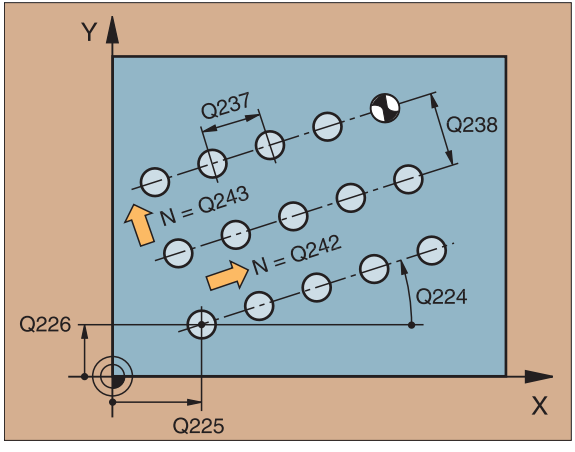
LINEAR PATTERN (221)

- CYCL DEF: Select Cycle 221 LINEAR PATTERN
  - 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

- 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, 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).

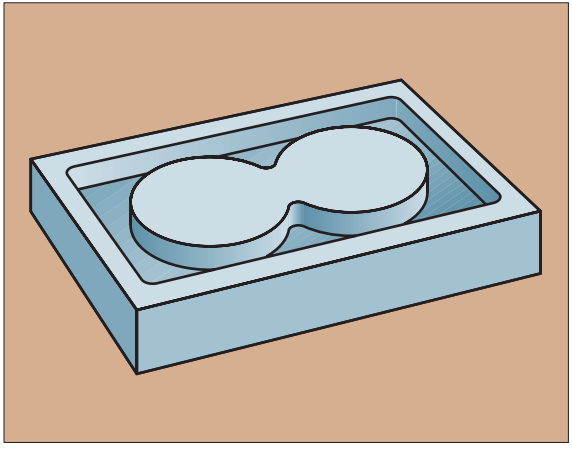
The subcontours are defined in subprograms.

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.

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!



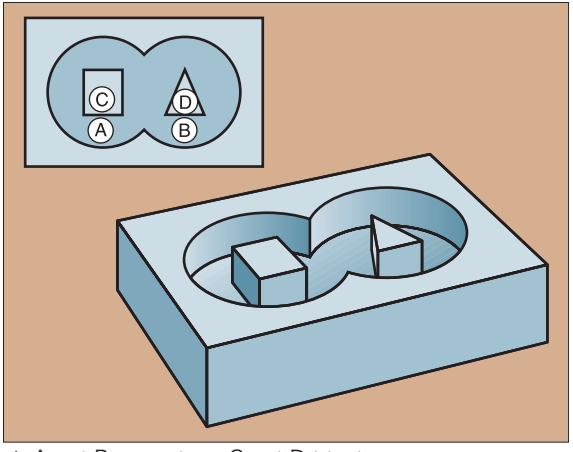
CONTOUR GEOMETRY (14)

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 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
```



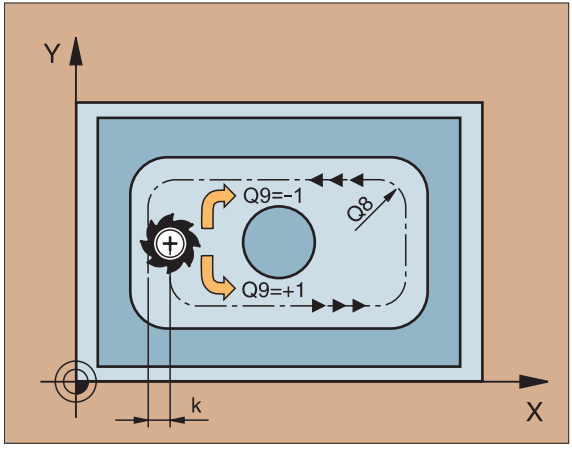
▲ A and B are pockets, C and D islands

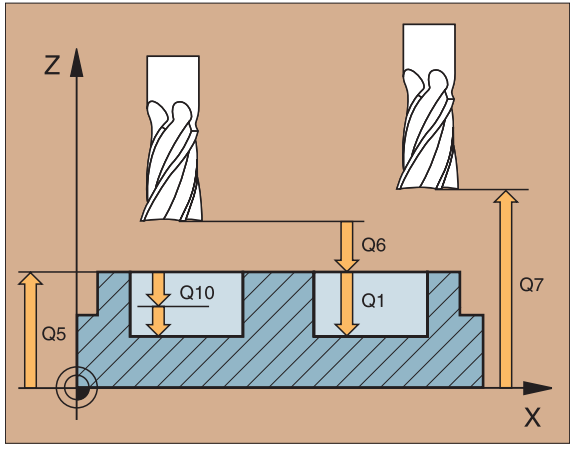
CONTOUR DATA (20)

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 x tool radius = 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
  - SETUP 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

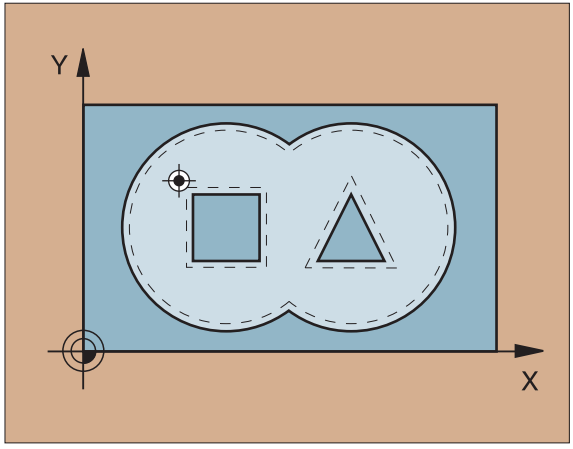
Cycle 20 CONTOUR DATA is effective immediately upon definition!





PILOT DRILLING (21)

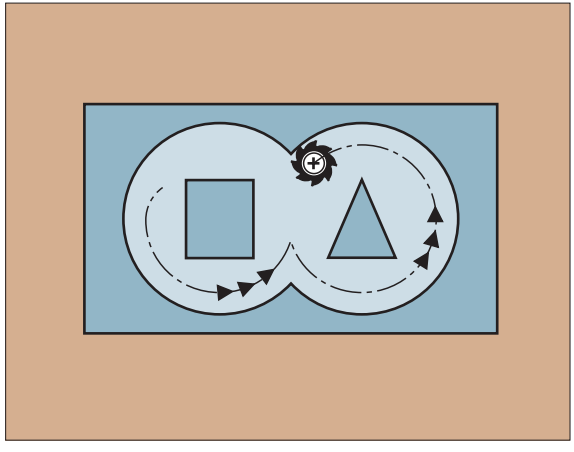
- CYCL DEF: Select Cycle 21 PILOT DRILLING
  - PECKING DEPTH Q10; incremental
  - FEED RATE FOR PECKING Q11
  - ROUGH MILL Q13: Number of the roughing tool



ROUGH-OUT (22)

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

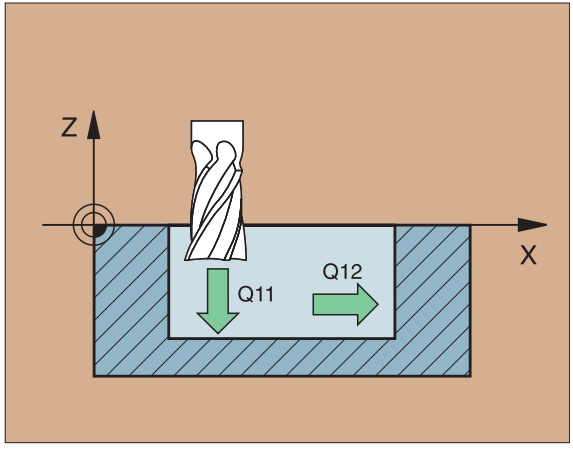
- CYCL DEF: Select Cycle 22 ROUGH-OUT
  - PECKING DEPTH Q10; incremental
  - FEED RATE FOR PECKING Q11
  - FEED RATE FOR MILLING Q12
  - COARSE ROUGHING TOOL NUMBER Q18
  - FEED RATE FOR RECIPROCATION Q19



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

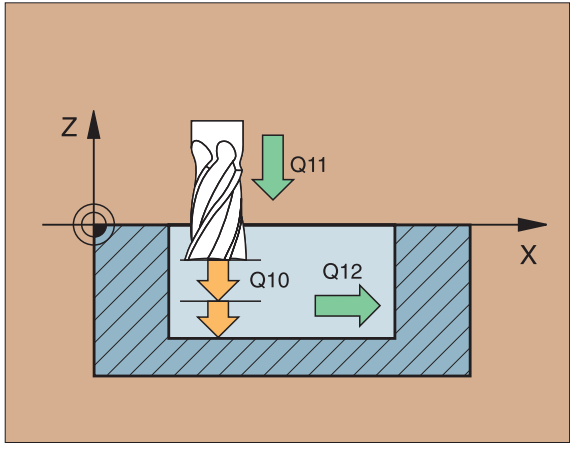


SIDE FINISHING (24)

Finishing the individual contour elements

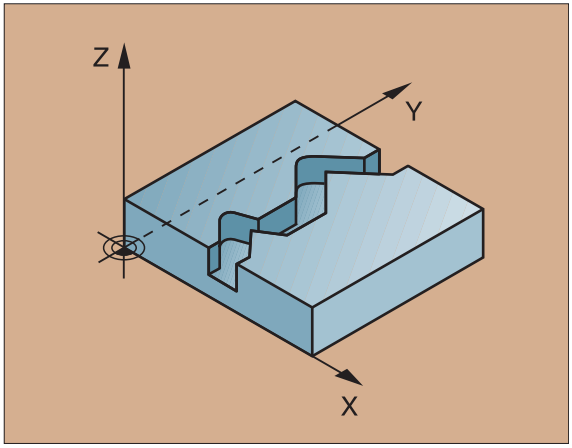
- CYCL DEF: Select Cycle 24 SIDE FINISHING
  - DIRECTION OF ROTATION? CLOCKWISE = –1 Q9:
    - Clockwise Q9 = –1
    - Counterclockwise Q9 = +1
  - PECKING DEPTH Q10; incremental
  - FEED RATE FOR PECKING Q11
  - FEED RATE FOR MILLING Q12
  - FINISHING ALLOWANCE FOR SIDE Q14: Allowance for finishing 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 subprogam.



- 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)

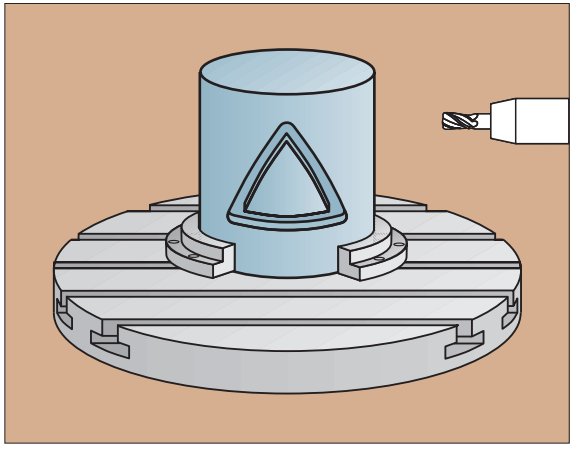
This cycle requires a center-cut end mill (ISO 1641)!

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.

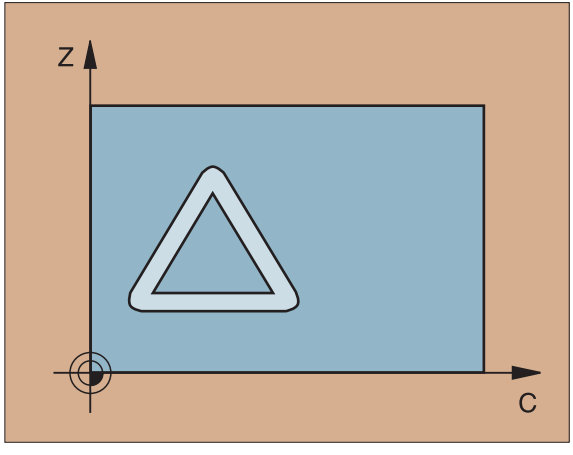
- Define a contour in a subprogram and list it in Cycle 14 CONTOUR GEOMETRY
- CYCL DEF: Select Cycle 27 CYLINDER SURFACE
  - MILLING DEPTH Q1
  - FINISHING ALLOWANCE FOR SIDE Q3: Enter the finishing allowance (Either Q3>0 or Q3<0)
  - SET-UP CLEARANCE ? Q6: Distance from the tool to the workpiece
  - PECKING DEPTH Q10
  - FEED RATE FOR PECKING 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



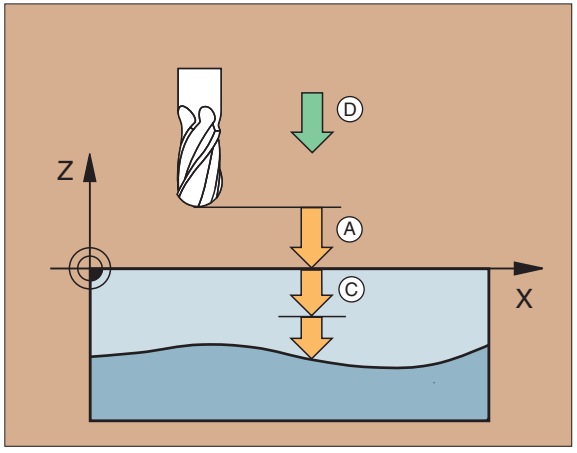
Multipass Milling

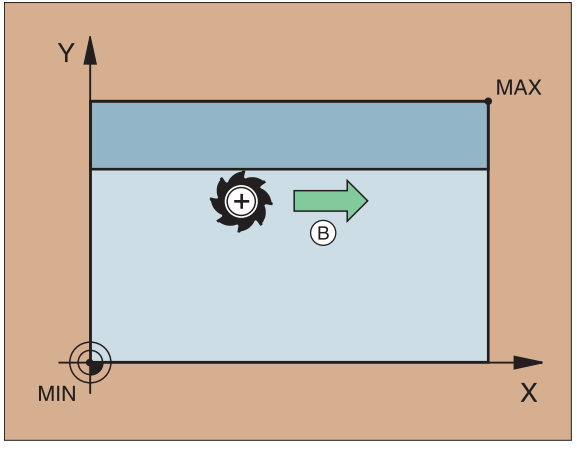
RUN DIGITIZED DATA (30)

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
```

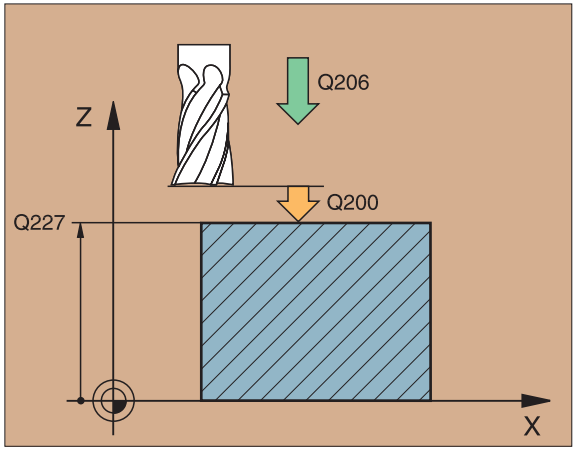


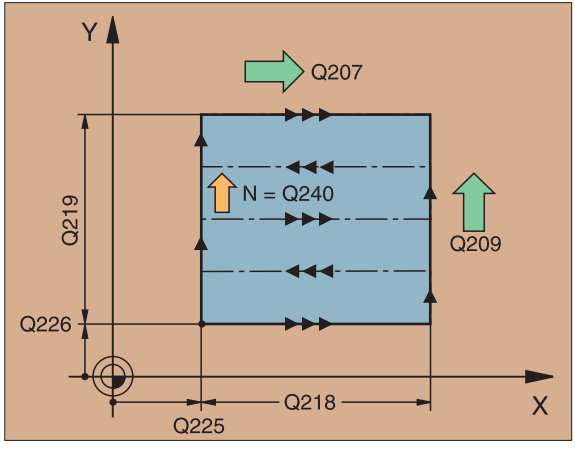


MULTIPASS MILLING (230)

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
  - STEPOVER FEED RATE: Q209
  - SET-UP CLEARANCE: Q200

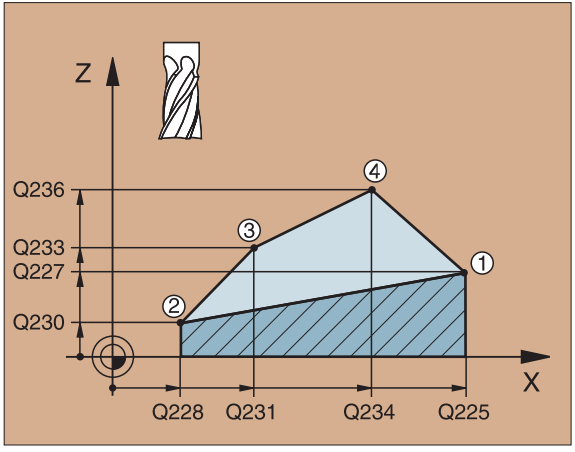


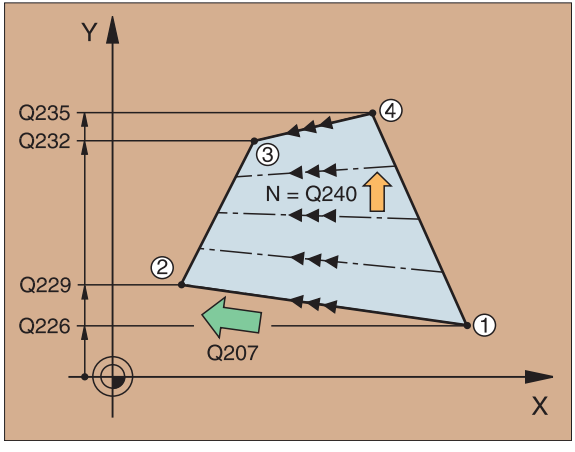


RULED SURFACE (231)

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



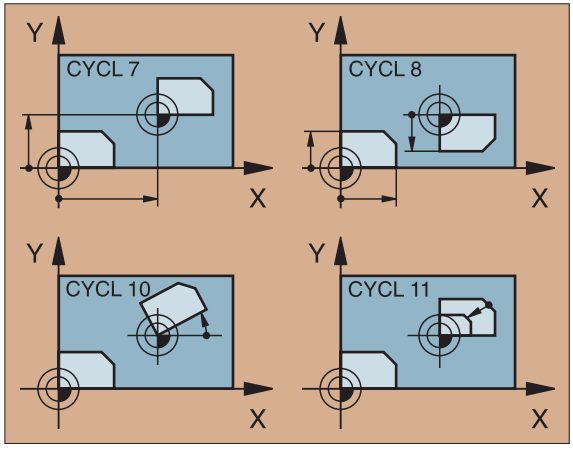


Cycles for Coordinate Transformation

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
• Tilted out of the plane	Cycle	19	WORKING PLANE
• 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.



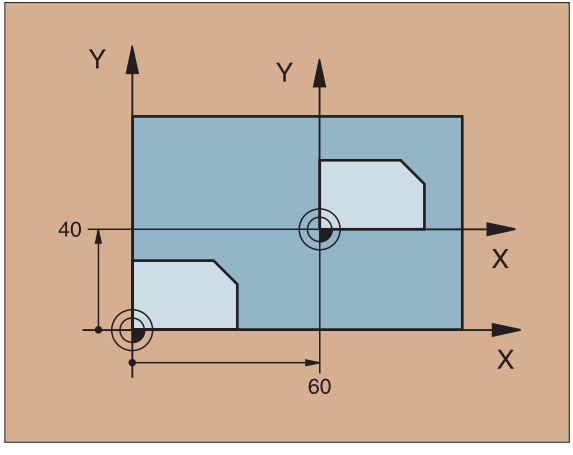
DATUM SHIFT (7)

▶ 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!



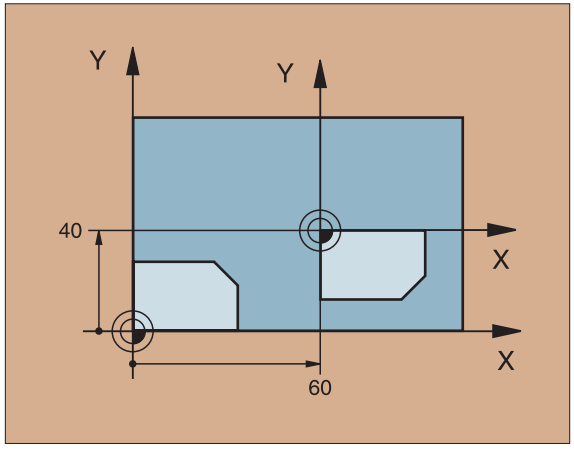
MIRROR IMAGE (8)

- 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)!



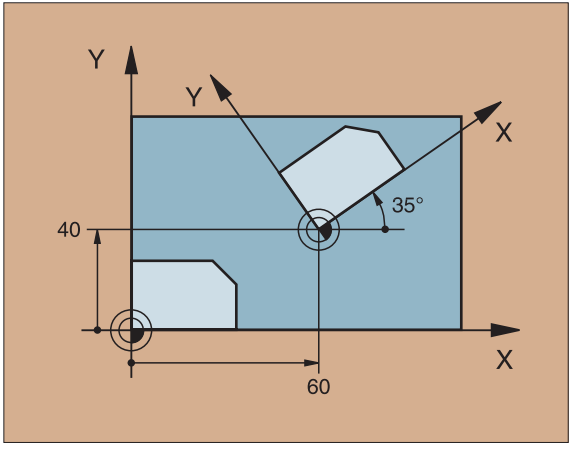
Rotation (10)

- 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
```



WORKING PLANE (19)

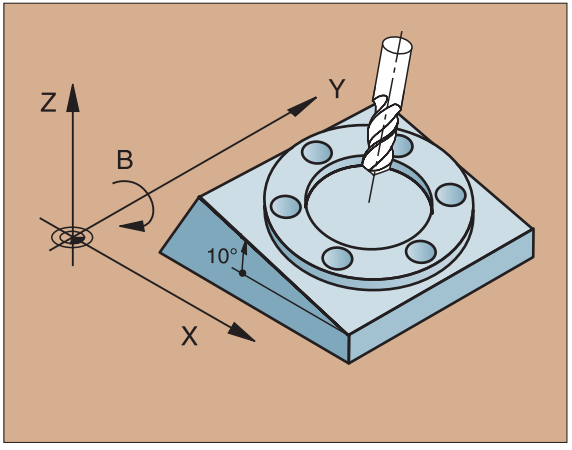
Cycle 19 WORKING PLANE supports machining operations with a swivel head and/or tilting table.

- Call the tool
- Retract the tool in the tool axis (to prevent collision)
- Position the rotary axis to the desired angle with an L-block
- CYCL DEF: Select Cycle 19 WORKING PLANE
  - Enter the ROTARY AXIS AND ANGLE
- 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
```



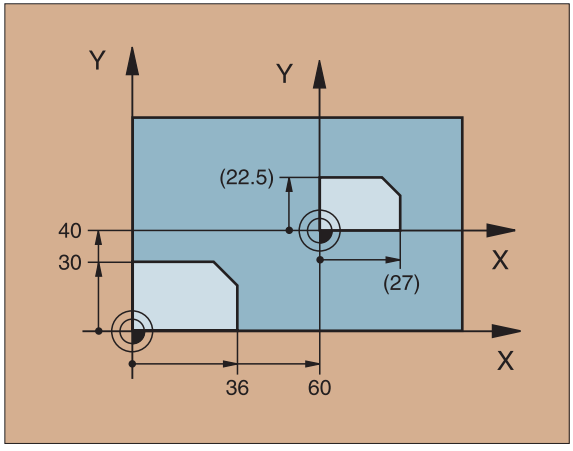
SCALING (11)

▶ 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)!



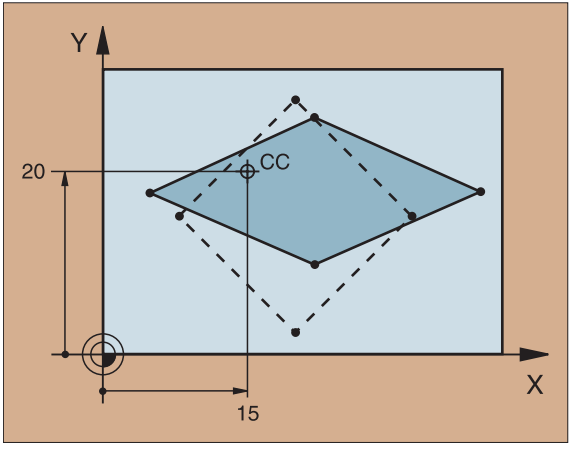
AXIS-SPECIFIC SCALING (26)

- 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
```

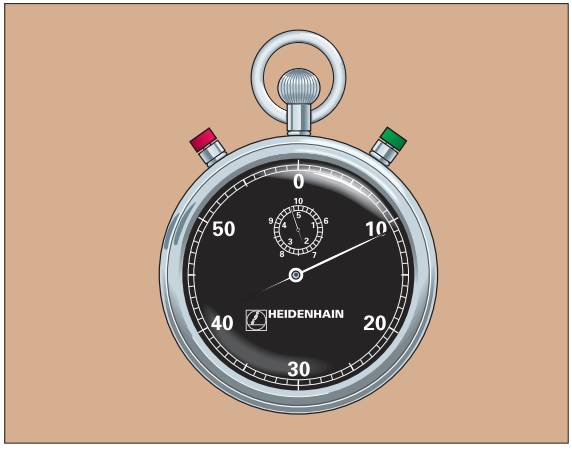


DWELL TIME (9)

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
```

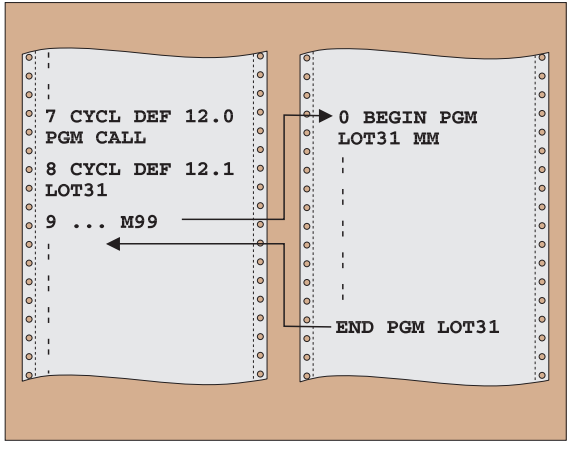


PGM CALL (12)

- 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
```

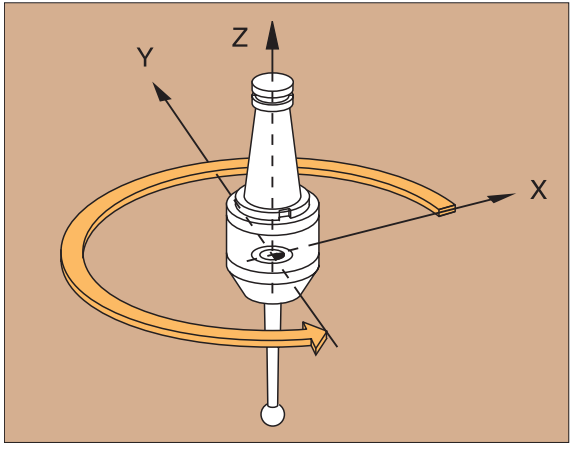


Spindle ORIENTATION

- 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
  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

TOUCH PROBE ▶ Call an overview of touch probe functions

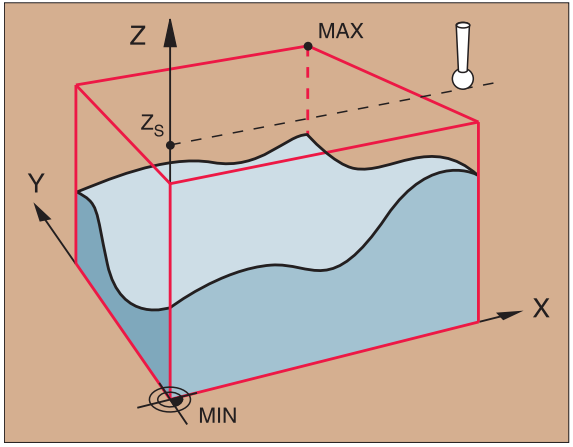
↑ ↓ ▶ Select with the arrow keys, or

GOTO □ 5 ▶ Press GOTO and enter the cycle number

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

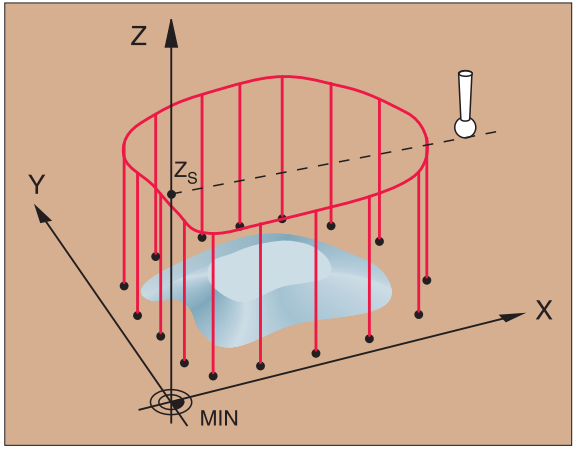
```
5 TCH PROBE 5.0 RANGE
6 TCH PROBE 5.1 PGM NAME: DIGI1
7 TCH PROBE 5.2 Z X+0 Y+0 Z+0
8 TCH PROBE 5.3 X+100 Y+100 Z+20
9 TCH PROBE 5.4 HEIGHT: +100
```



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
```



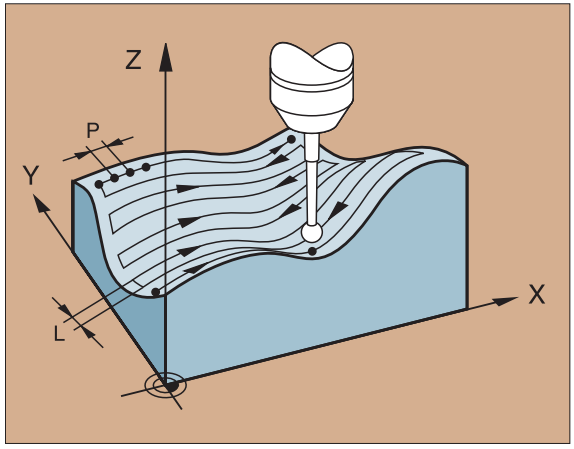
Digitizing Cycle MEANDER (16)

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
  - 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.
  - FEED RATE REDUCTION AT EDGES: Distance at which the TNC begins to reduce the scanning feed rate before steep edges

- 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!

```
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
```



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

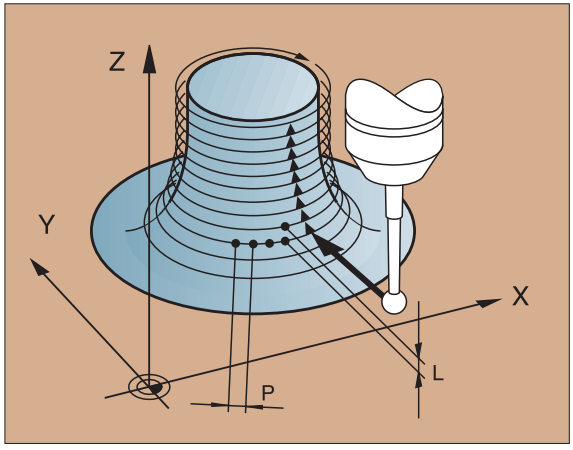
Digitizing Cycle CONTOUR LINES (17)

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
  - 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.
  - FEED RATE REDUCTION AT EDGES: Distance at which the TNC begins to reduce the scanning feed rate before steep edges

The LINE SPACING and MAX. PROBE POINT INTERVAL cannot exceed 20 mm!

```
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
```



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

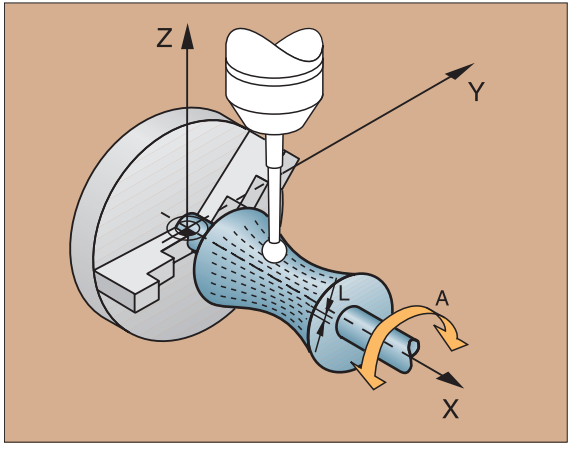
Digitizing Cycle LINE (18)

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
  - MIN. LINE SPACING: Minimum distance moved forward to start the next line at steep surfaces
  - LINE SPACING AND 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.
  - FEED RATE REDUCTION AT EDGES: Distance at which the TNC begins to reduce the scanning feed rate before steep edges

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:

AUTO DRAW OFF / ON

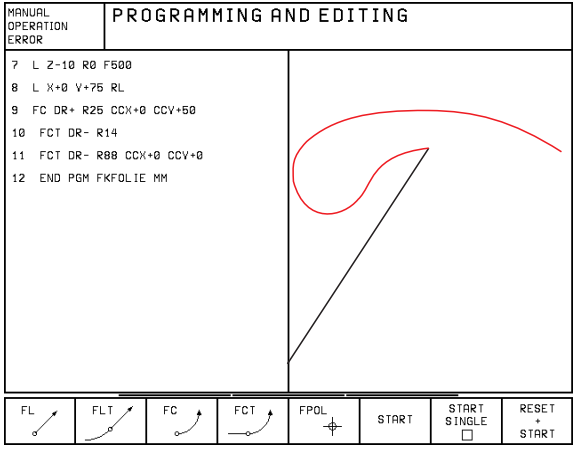
- Automatic graphic generation during programming

RESET + START

- Manually start graphic generation

START SINGLE

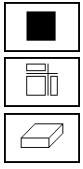
- Generate interactive graphics blockwise



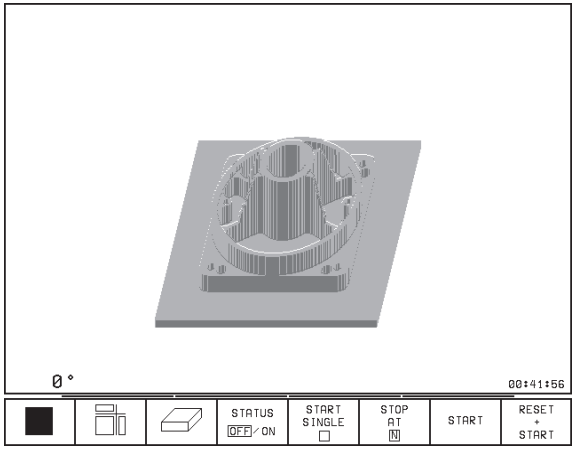
Test Graphics and Program Run Graphics

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

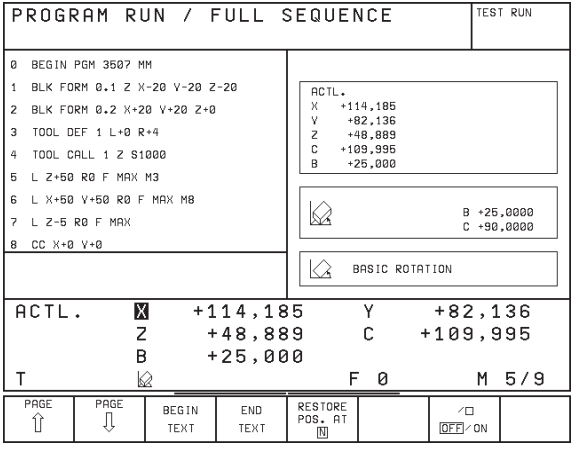
Select the PGM+STATUS or POSITION+STATUS screen layout!

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

- Tool position
- Feed rate
- Active M 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



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

Standard Fixed Cycles

G83	Pecking
G84	Tapping
G85	Rigid tapping
G86	Thread cutting
G74	Slot milling
G75	Rectangular pocket milling, clockwise machining direction
G76	Rectangular pocket milling, counterclockwise machining direction
G77	Circular pocket milling, clockwise machining direction
G78	Circular pocket milling, counterclockwise machining direction

SL Cycles, Group I

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

*) Effective blockwise

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

Coordinate Transformations

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
G80	Working plane

Special Cycles

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

*) 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)
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
D19	Transfer numerical values or Q parameters to the PLC
D68-D99	Call the user cycles

*) 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

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°

M95 Reserved

M96 Reserved

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

M105 Machine with second k_V factor

M106 Machine with first k_V factor

M107 See User's Manual

M108 Reset M107

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

M114 See User's Manual

M115 Reset M114

M116 Feed rate for rotary axes in mm/min

M118 Superimpose handwheel positioning during program run

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
M130*	Within the positioning block: points are referenced to the non-tilted coordinate system
M132	Jolt reduction during change of traverse direction
M200*	Miscellaneous function for laser cutting machines
⋮	
M204*	See User's Manual

*) Only with conversational programming

HEIDENHAIN

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

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

www.heidenhain.de

Ve 00
311 063-20 · SW12 · 6/97 · pdf · Subject to change without notice