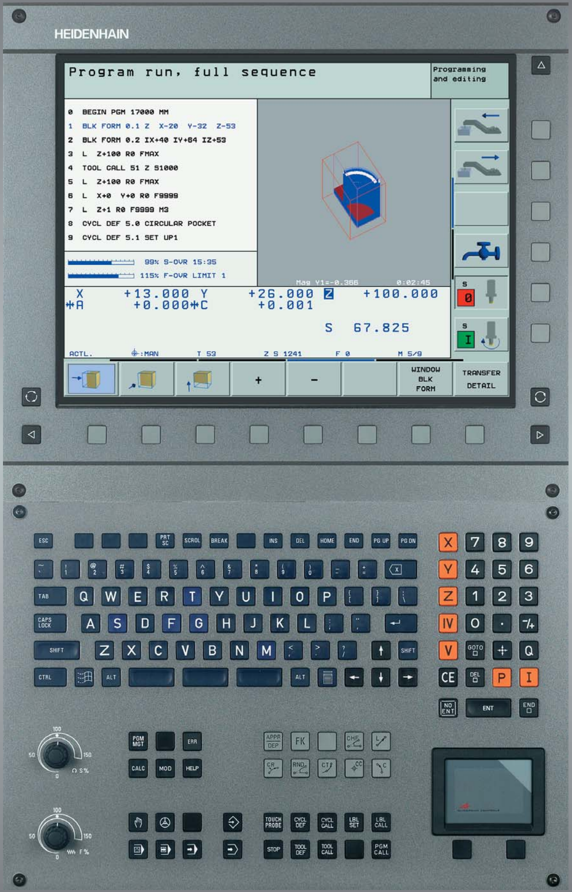


HEIDENHAIN

Pilot

iTNC 530

NC Software
340 422-xx
340 423-xx
340 480-xx
340 481-xx

English (en)
9/2004

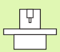
The Pilot

... is your concise programming guide for the HEIDENHAIN iTNC 530 contouring control. For more comprehensive information on programming and operating, refer to the TNC User’s Manual. There you will find complete information on:

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

Symbols in the Pilot

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

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

Control	NC software number
iTNC 530	340 422-xx
iTNC 530, export version	340 423-xx
iTNC 530 with Windows 2000	340 480-xx
iTNC 530 with Windows 2000, export version	340 481-xx
iTNC 530 programming station	374 150-xx

Contents

Fundamentals

Programs/Files

See "Programming, File Management."

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

PROG20	.H
File name	File type
Maximum Length	See table at right

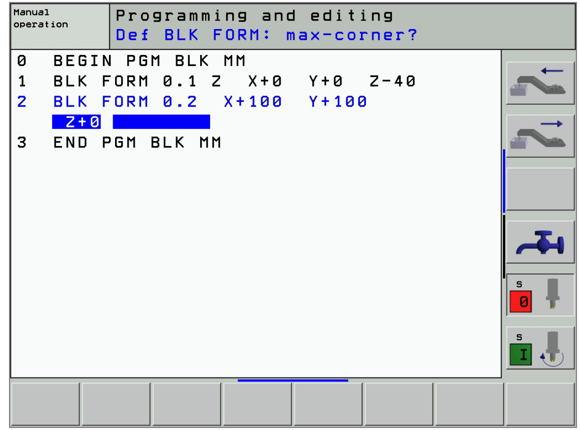
Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In ISO format	.I
Tables for	
Tools	.T
Tool changers	.TCH
Pallets	.P
Datums	.D
Points	.PNT
Presets (reference points)	.PR
Cutting data	.CDT
Cutting materials, workpiece materials	.TAB
Texts as	
ASCII files	.A

Creating a New Part Program

PGM MGT

- Select the directory in which the program is stored.
- Enter the new program name and confirm your entry with the ENT key.
- To select the unit of measure, press the MM or INCH soft key. The TNC switches the screen layout and initiates the dialog for defining the **BLK FORM** (workpiece blank).
- Enter the spindle axis.
- Enter in sequence the X, Y and Z coordinates of the MIN point.
- Enter in sequence the X, Y and Z coordinates of the MAX point.

```
1 BLK FORM 0.1 Z X+0 Y+0 Z-50
2 BLK FORM 0.2 X+100 Y+100 Z+0
```

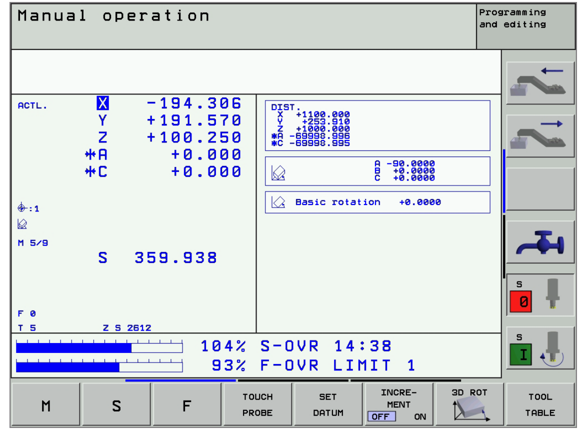


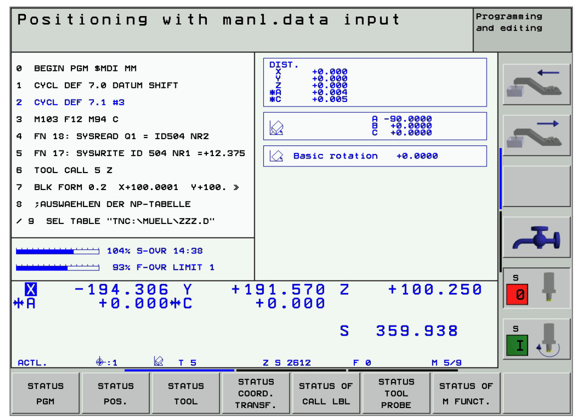
Choosing the Screen Layout

See "Introduction, the iTNC 530."

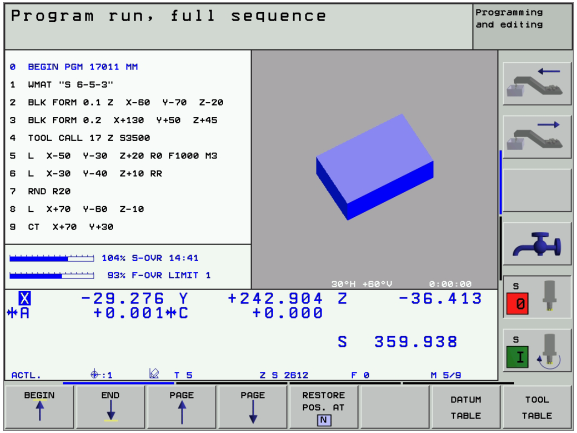
▶ Show soft keys for setting the screen layout.

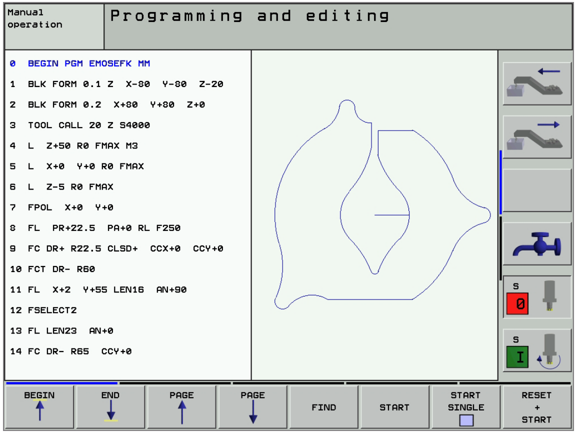
Operating mode	Screen contents	
Manual Operation / Electronic Handwheel	Positions	POSITION
	Positions at left, status at right	POSITION + STATUS
Positioning with Manual Data Input (MDI)	Program	PGM
	Positions at left, status at right	POSITION + STATUS





Operating mode	Screen contents	
Program run, full sequence Program run, single block Test run	Program	PGM
	Program at left, program structure at right	PGM + SECTS
	Program at left, status at right	POSITION + STATUS
	Program at left, graphics at right	PGM + GRAPHICS
	Graphics	GRAPHICS
Programming and editing	Program	PGM
	Program at left, program structure at right	PGM + SECTS
	Program at left, programming graphics at right	PGM + GRAPHICS



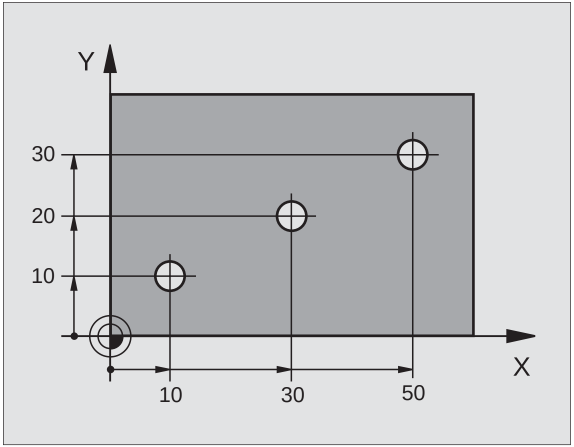


Absolute Cartesian Coordinates

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

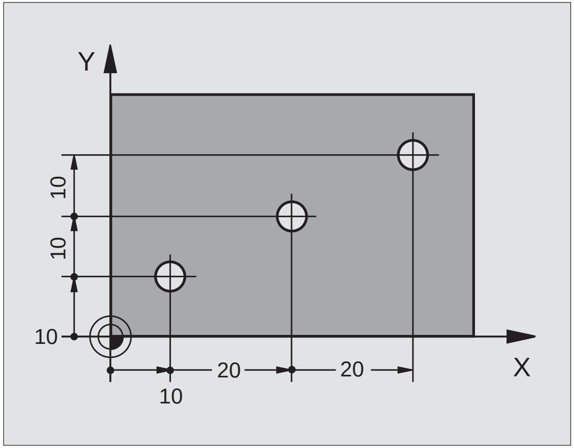
Programmable NC axes in an NC block

Straight movement	5 axes
Circular movement	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 absolute coordinates.



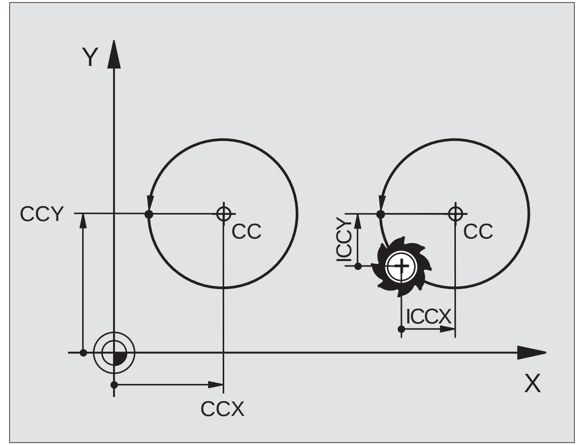
Circle Center and Pole: CC

The circle center **CC** must be entered to program circular tool movements with the path function **C** (see page 26). **CC** is also needed to define the pole for polar coordinates.

CC is entered in Cartesian coordinates.

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

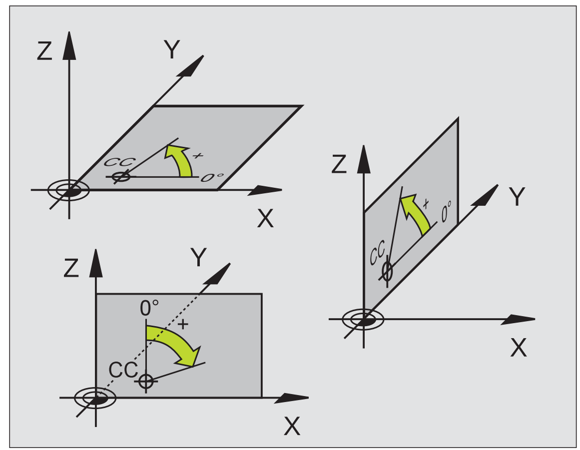
An incrementally defined circle center or pole **CC** is always measured from the last programmed position of the tool.



Angle Reference Axis

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

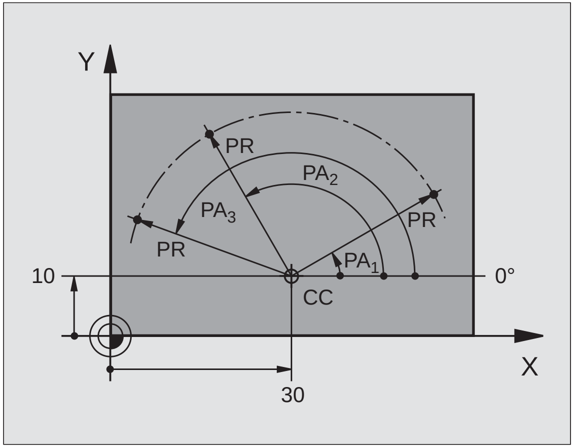
Working plane	Ref. axis and 0° direction
X/Y	+X
Y/Z	+Y
Z/X	+Z



Polar Coordinates

Dimensional data in polar coordinates is entered relative to the pole **CC.** A position in the working plane is defined by

- Polar coordinate radius **PR** = Distance of the position to the pole **CC**
- 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.

P

▶ Press the P key.
▶ Answer the dialog prompts.

Defining Tools

Tool data

Each tool is identified by a tool number between 0 and 254. If you are working with tool tables, you can use higher numbers and you can also enter a tool name for each tool.

Entering tool data

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

- in a tool table (centrally, Program TOOL.T)

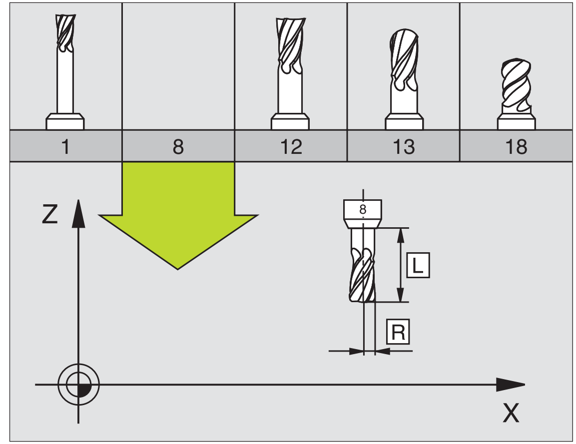
or

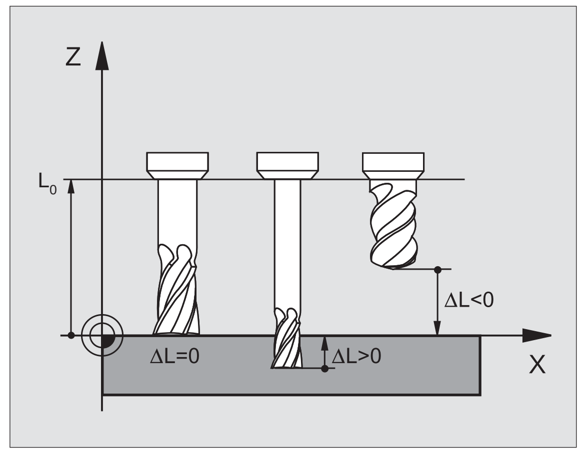
- within the part program in **TOOL DEF** blocks (locally)

TOOL DEF

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

- ▶ Program the tool length as the length difference L0 to the zero tool:
  - L>L0: The tool is longer than the zero tool
  - L<L0: The tool is shorter than the zero tool
- ▶ With a tool presetter you can measure the actual tool length, then program that length.

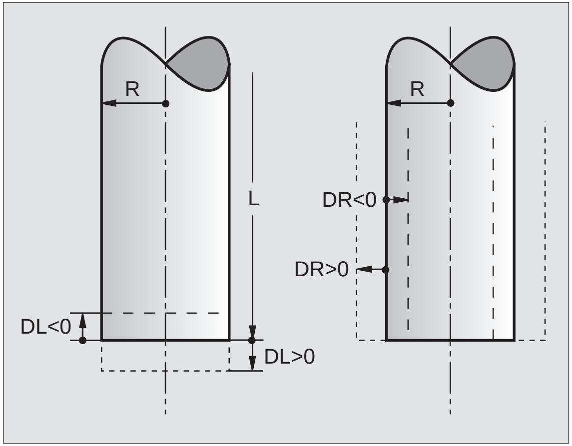




Calling tool data

TOOL CALL

- **`Tool number`** or name
- **`Working spindle axis X/Y/Z:`** Tool axis
- **`Spindle speed S`**
- **`Feed rate F`**
- **`Tool length oversize DL`** (e.g. to compensate wear)
- **`Tool radius oversize DR`** (e.g. to compensate wear)
- **`Tool radius oversize DR2`** (e.g. to compensate wear)




```
3 TOOL DEF 6 L+7.5 R+3
4 TOOL CALL 6 Z S2000 F650 DL+1 DR+0.5 DR2+0.1
5 L Z+100 R0 FMAX
6 L X-10 Y-10 RO FMAX M6
```

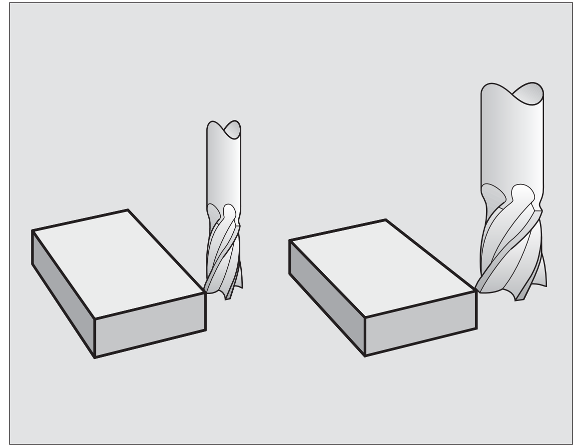
Tool change

- 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!

Fundamentals

Tool Compensations

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



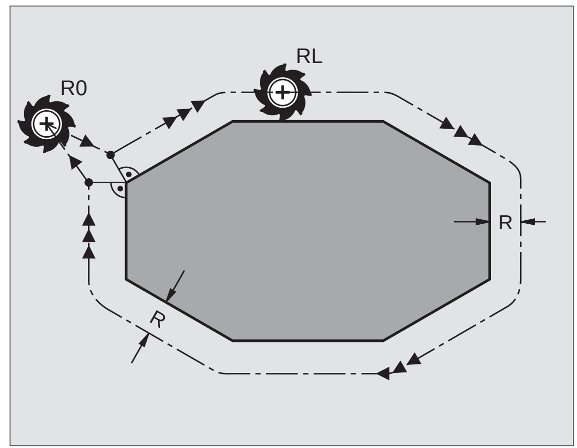
Length compensation

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

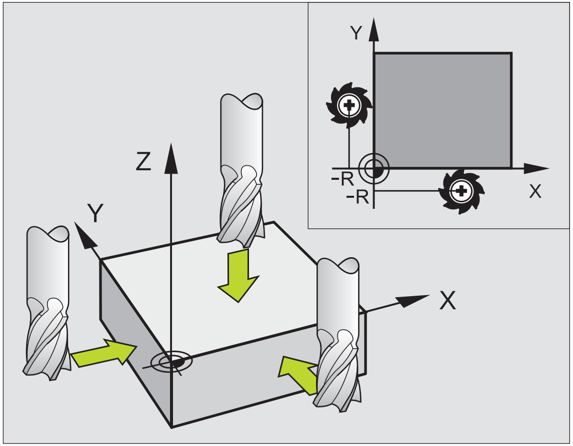
Working **without radius compensation** (e.g. drilling):

- Execution of a positioning block with R0

Datum Setting without a 3-D Touch Probe

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

- Insert the zero tool with known radius into the spindle.
- Select the Manual Operation or Electronic Handwheel mode of operation.
- Touch the reference surface in the tool axis with the tool and enter its length.
- Touch the reference surface in the working plane with the tool and enter the position of the tool center.

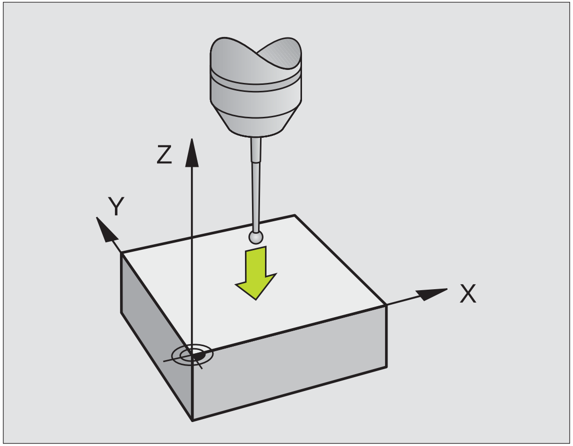


Setup and Measurement with 3-D Touch Probes

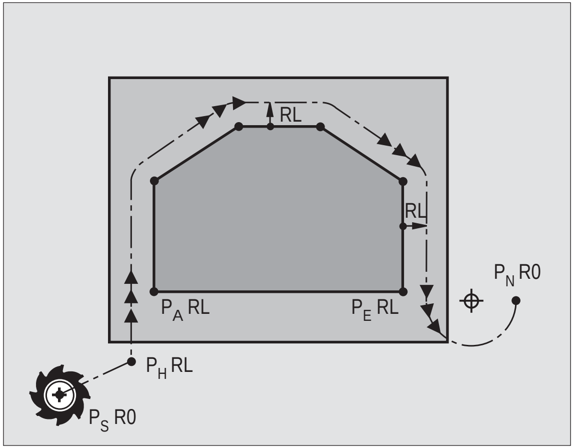
A HEIDENHAIN 3-D touch probe enables you to setup the machine very quickly, simply and precisely.

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

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



Contour Approach and Departure



Starting point P_S

P_S lies outside the contour and must be approached without radius compensation (R0).

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 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.

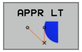
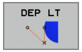
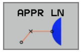
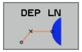
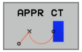
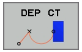
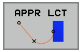
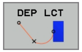
End point P_N

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

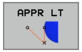
APPR DEP

▶ Press the soft key with the desired path function:

		Straight line with tangential connection
		Straight line perpendicular to a contour point
		Circular arc with tangential connection
		Straight line segment tangentially connected to the contour through an arc

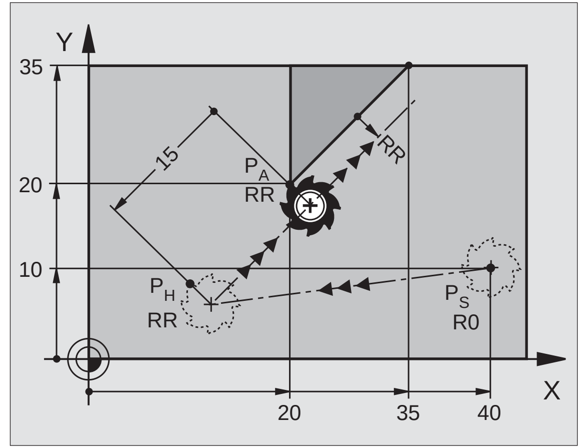
- Program a radius compensation in the **APPR** block.
- **DEP** blocks set the radius compensation to **R0!**

Approaching on a straight line with tangential connection: APPR LT

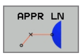


- Coordinates of the first contour point P_A
- LEN: Distance from the auxiliary point P_H to the first contour point P_A
- Radius compensation RR/RL

```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100
9 L Y+35 Y+35
10 L ...
```

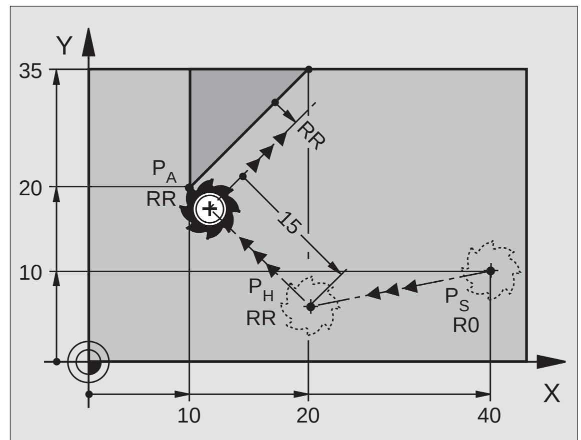


Approaching on a straight line perpendicular to the first contour point: APPR LN

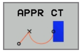


- Coordinates of the first contour point P_A
- LEN: Distance from the auxiliary point P_H to the first contour point P_A
- Radius compensation RR/RL

```
7 L X+40 Y+10 R0 FMAX M3
8 APPR LN X+10 Y+20 Z-10 LEN15 RR F100
9 L X+20 Y+35
10 L ...
```

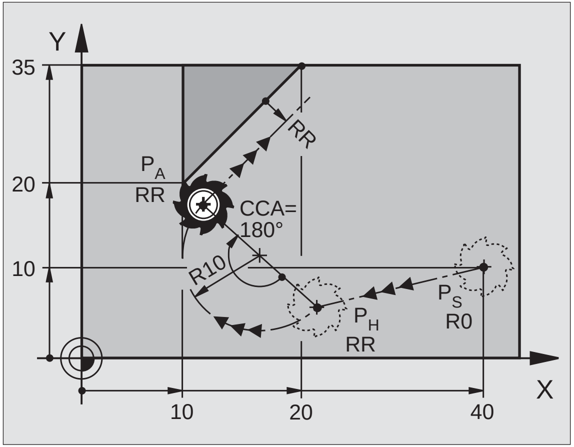


Approaching on a circular path with tangential connection: APPR CT

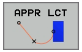


- Coordinates of the first contour point P_A
- Radius R
  Enter R > 0
- Circle center angle (CCA)
  Enter CCA > 0
- Radius compensation RR/RL

```
7 L X+40 Y+10 RO FMAX M3
8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100
9 L X+20 Y+35
10 L ...
```

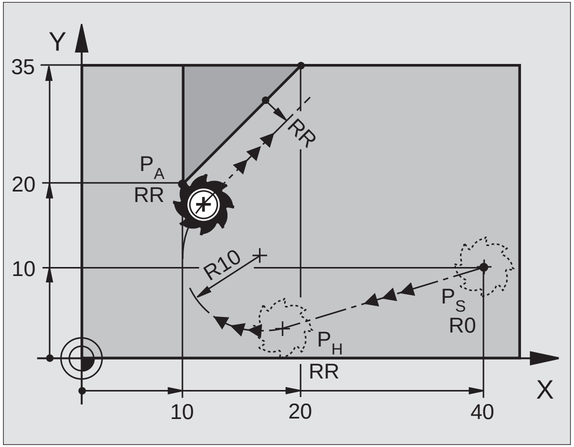


Approaching on a circular arc tangentially connecting the contour and a straight line: APPR LCT

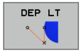


- Coordinates of the first contour point P_A
- Radius R
  Enter R > 0
- Radius compensation RR/RL

```
7 L X+40 Y+10 RO FMAX M3
8 APPR LCT X+10 Y+20 Z-10 R10 RR F100
9 L X+20 Y+35
10 L ...
```

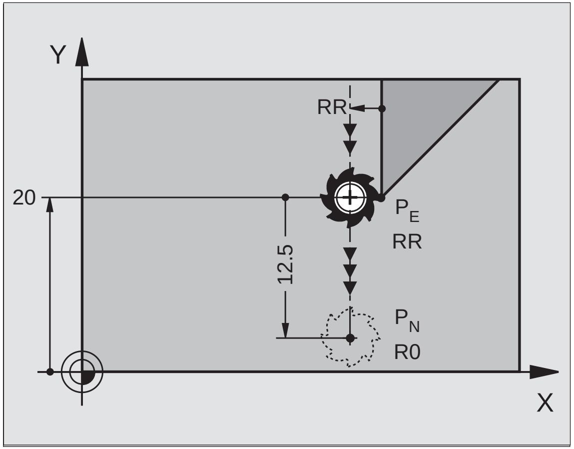


Departing tangentially on a straight line: DEP LT

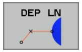


- Enter the distance between P_E and P_N as Enter LEN > 0

```
23 L Y+20 RR F100
24 DEP LT LEN12.5 F100
25 L Z+100 FMAX M2
```

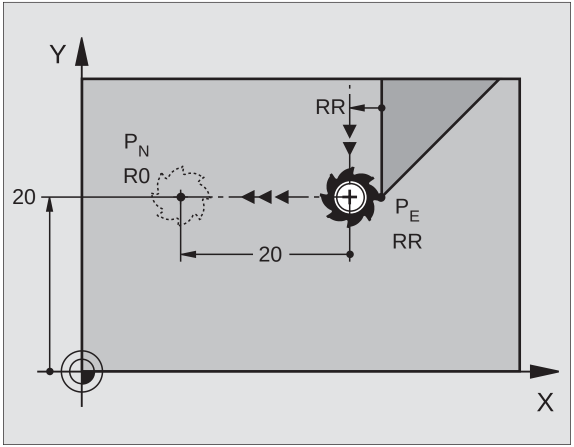


Departing on a straight line perpendicular to the last contour point: DEP LN

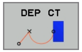


- Enter the distance between P_E and P_N as Enter LEN > 0

```
23 L Y+20 RR F100
24 DEP LN LEN+20 F100
25 L Z+100 FMAX M2
```

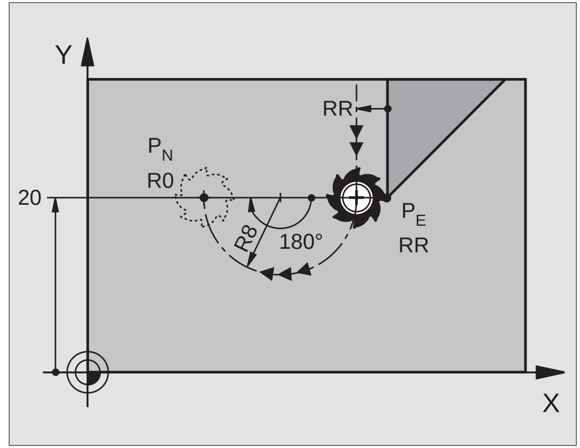


Departing tangentially on a circular arc: DEP CT

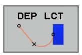


- Radius R
  Enter R > 0
- Circle center angle (CCA)

```
23 L Y+20 RR F100
24 DEP CT CCA 180 R+8 F100
25 L Z+100 FMAX M2
```

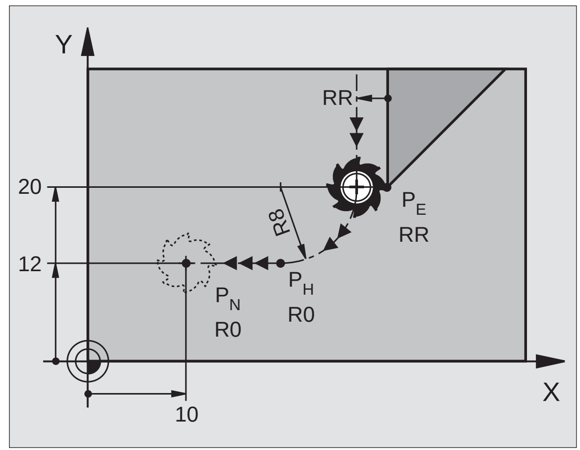


Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT



- Coordinates of the end point P_N
- Radius R
  Enter R > 0

```
23 L Y+20 RR F100
24 DEP LCT X+10 Y+12 R+8 F100
25 L Z+100 FMAX M2
```



Path Functions

Path Functions for Positioning Blocks

See "Programming, Programming Contours."

Assumption

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!

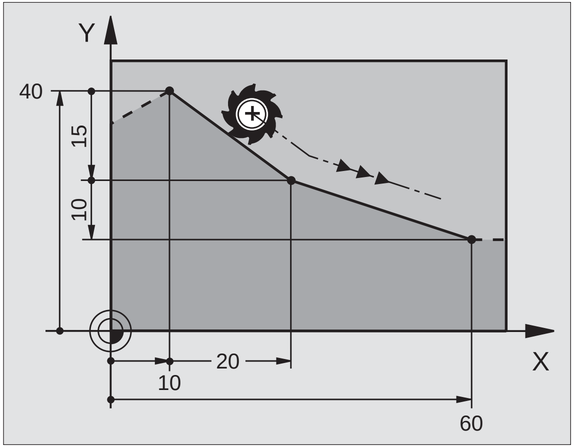
Path Functions		
Straight line	L	page 23
Chamfer between two straight lines	CHF	page 24
Corner rounding	RND	page 25
Circle center or **pole for polar coordinates**	CC	page 26
Circular path around circle center CC	C	page 26
Circular arc with radius	CR	page 27
Circular arc with tangential connection to the preceding contour element	CT	page 28
FK free contour programming	FK	page 31

Straight Line L

- Coordinates of the end points of the straight line
- Radius compensation **RR/RL/R0**
- Feed rate **F**
- Miscellaneous function **M**

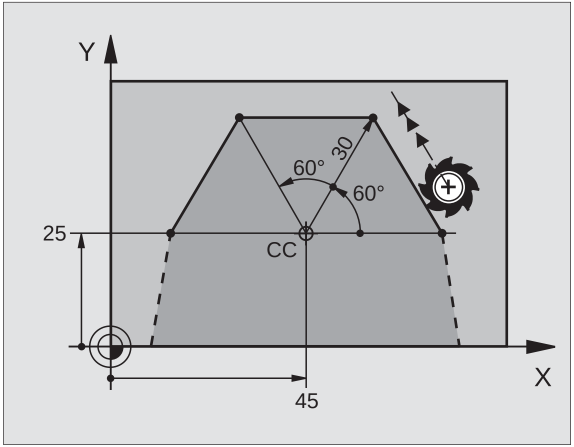
With Cartesian coordinates

```
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
```

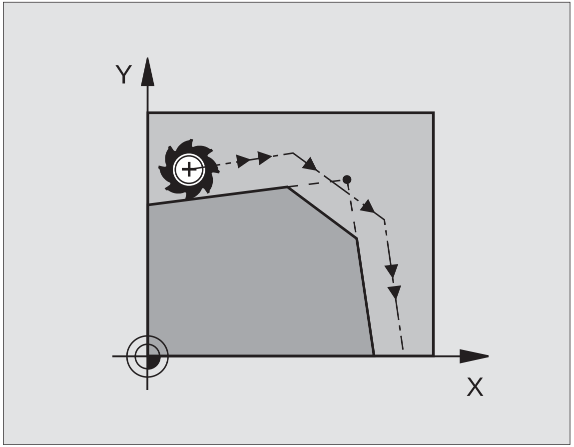


- Define the pole **CC** before programming polar coordinates.
- You can define the pole **CC** only in Cartesian coordinates.
- The pole **CC** remains in effect until you define a new pole **CC**.

Inserting a Chamfer CHF between Two Straight Lines

- Chamfer side length
- Feed rate F

```
7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12 F250
10 L IX+5 Y+0
```



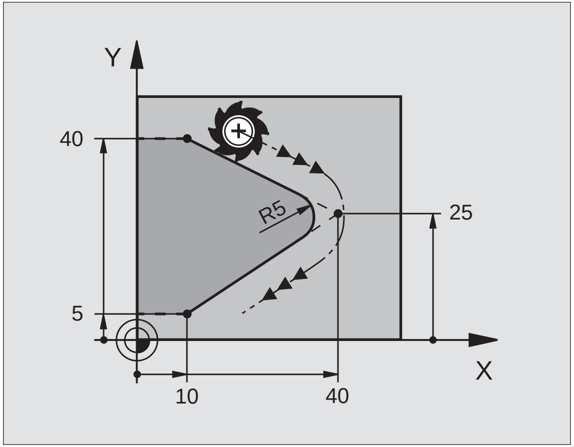
- You cannot start a contour with a **CHF** block.
- The radius compensation before and after the **CHAMFER** block must be the same.
- An inside chamfer must be large enough to accommodate the called tool.

Corner Rounding RND

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

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

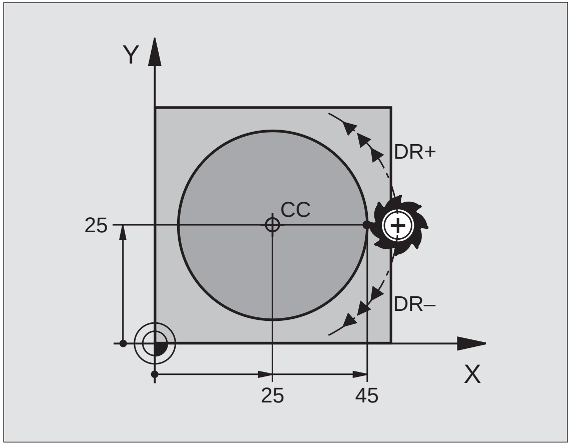
```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
```



Circular Path around 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.

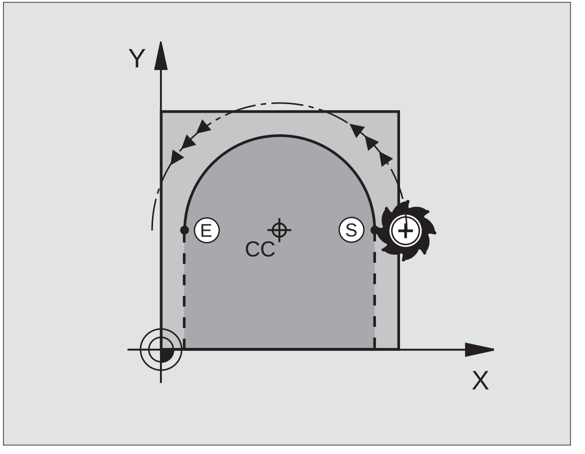


With Cartesian coordinates

```
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.
- You can define the pole **CC** only in Cartesian coordinates.
- The pole **CC** remains in effect until you define a new pole **CC.**
- The arc end point can be defined only with the polar coordinate angle (**PA**)!

Circular Arc CR with Radius

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

or

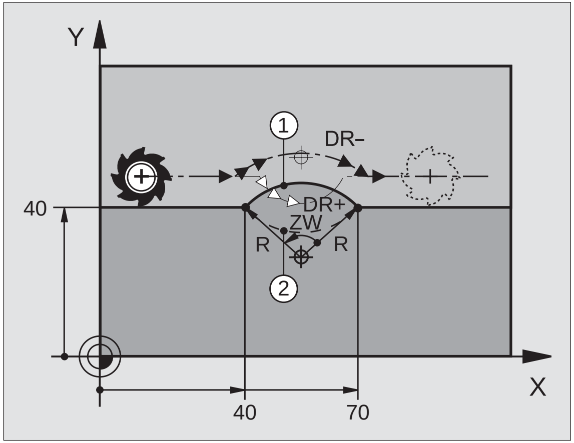
```
11 CR X+70 Y+40 R+20 DR+ (ARC 2)
```

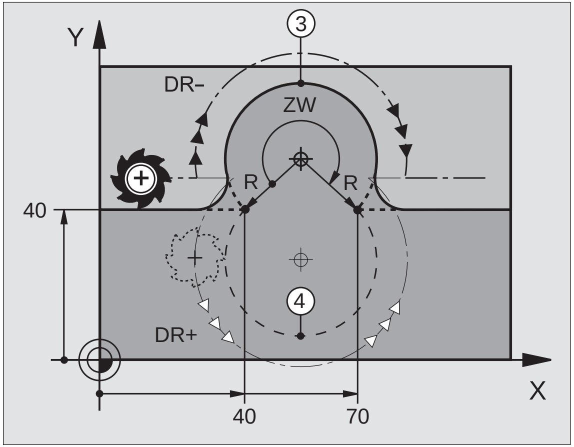
or

```
10 L X+40 Y+40 RL F200 M3
11 CR X+70 Y+40 R-20 DR- (ARC 3)
```

or

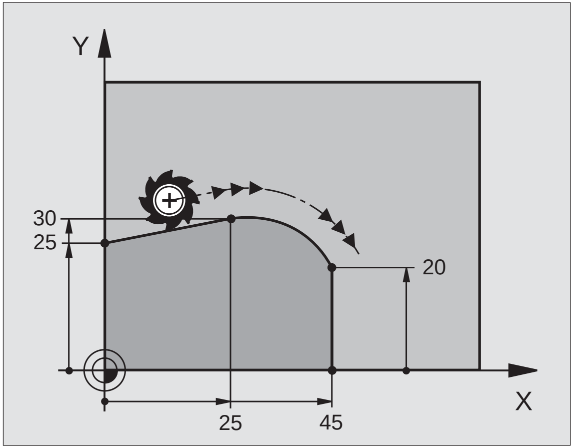
```
11 CR X+70 Y+40 R-20 DR+ (ARC 4)
```





Circular Path CT with Tangential Connection

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



With Cartesian coordinates

```
7 L X+0 Y+25 RL F300 M3
8 L X+25 Y+30
9 CT X+45 Y+20
10 L Y+0
```

With polar coordinates

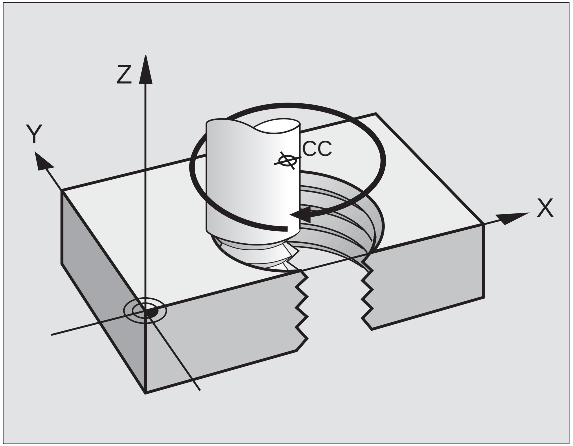
```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```

- Define the pole **CC** before programming polar coordinates.
- You can define the pole **CC** only in Cartesian coordinates.
- The pole **CC** remains in effect until you define a new pole **CC**.

Helix (Only in Polar Coordinates)

Calculations (upward milling direction)

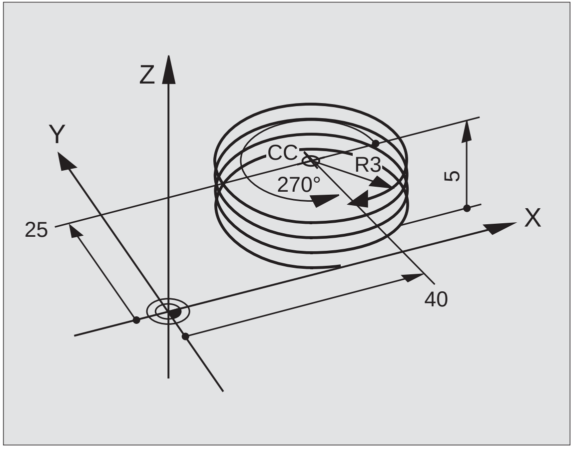
Path revolutions:	**n**	Thread revolutions + overrun at start and end of thread
Total height:	**h**	Thread pitch P x path revolutions *n*
Incr. coord. angle:	**IPA**	Path revolutions *n* x 360°
Start angle:	**PA**	Angle for start of thread + angle for thread overrun
Start coordinate:	**Z**	Pitch P x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



Shape of the helix

Internal thread	Work direction	Direction	Radius compens.
Right-hand Lefthand	Z+ Z+	DR+ DR-	RL RR
Right-hand Lefthand	Z- Z-	DR- DR+	RR RL

External thread	Work direction	Direction	Radius compens.
Right-hand Lefthand	Z+ Z+	DR+ DR-	RR RL
Right-hand Lefthand	Z- Z-	DR- DR+	RL RR



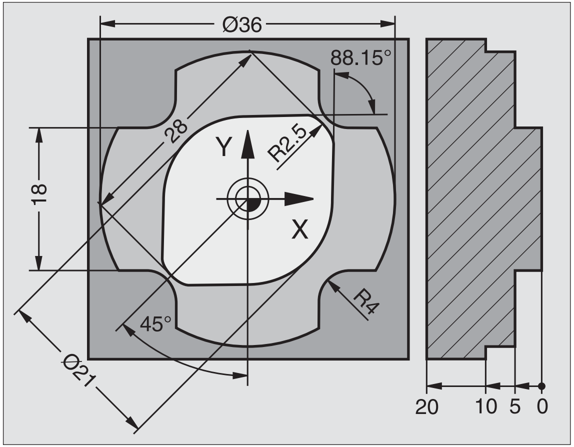
Thread M6 x 1 mm with 5 revolutions:

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

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. Only then will the TNC allow you to enter conventional path functions.

Working with the Interactive Graphics

Select the PROGRAM+GRAPHICS screen layout.

SHOW SOLUTION
- Show the possible solutions.

SELECT SOLUTION
- Enter the displayed solution in the part program.

END SELECT
- Enter data for subsequent contour elements.

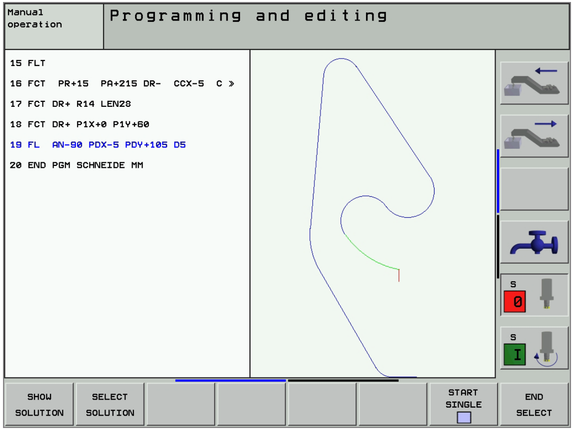
START SINGLE
- Graphically display the next programmed block.

Standard colors of the interactive graphics

White The contour element is fully defined.

Green The entered data describe a limited number of possible solutions: select the correct one.

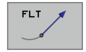
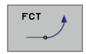
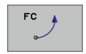
Red The entered data are not sufficient to determine the contour element: enter further data.



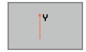
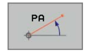
Initiating the FK dialog

FK

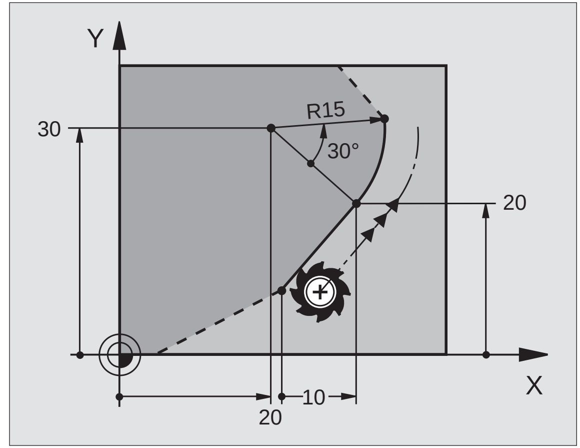
▶ Initiate the FK dialog. The following functions are available:

Contour element	Soft keys
Straight line with tangential connection	
Straight line without tangential connection	
Circular arc with tangential connection	
Circular arc without tangential connection	
Pole for FK programming	

End point coordinates X, Y or PA, PR

Known data	Soft keys	
Cartesian coordinates **X** and **Y**		
Polar coordinates referenced to **FPOL**		
Incremental input	**I**	

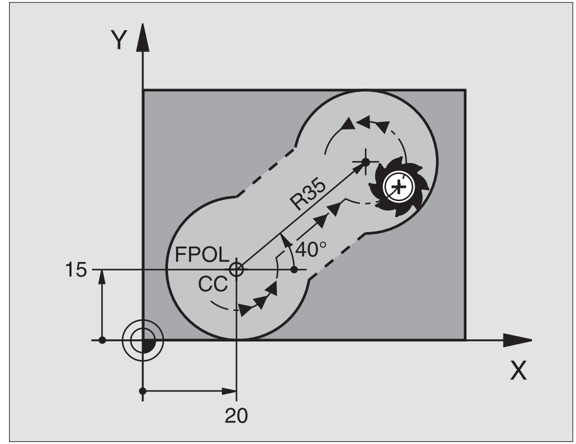
```
7 FPOL X+20 Y+30
8 FL IX+10 Y-20 RR F100
9 FCT PR+15 IPA+30 DR+ R15
```



Circle center (CC) in an FC/FCT block

Known data	Soft keys	
Circle center in Cartesian coordinates	CCX	CCY
Circle center in polar coordinates	CC PR	CC PA
Incremental input	I	

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

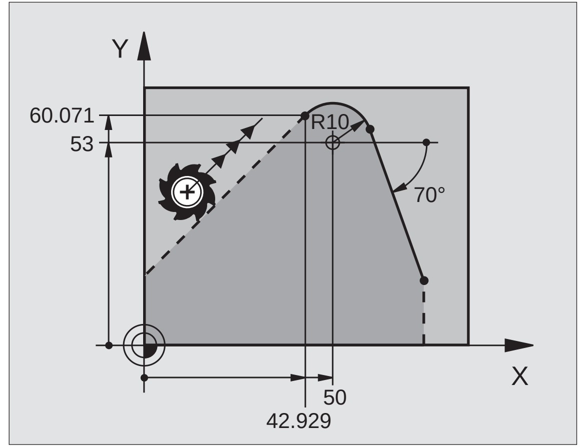


Auxiliary points on or next to a contour

Known data	Soft keys		
X coordinate of an auxiliary point P1 or P2 of a straight line	P1X	P2X	
Y coordinate of an auxiliary point P1 or P2 of a straight line	P1Y	P2Y	
X coordinate of an auxiliary point P1, P2 or P3 of a circular path	P1X	P2X	P3X
Y coordinate of an auxiliary point P1, P2 or P3 of a circular path	P1Y	P2Y	P3Y

Known data	Soft keys	
X and Y coordinates of the auxiliary point near a straight line	PDX	PDY
Distance auxiliary point/straight line	D	
X and Y coordinates of the auxiliary point near a circular arc	PDX	PDY
Distance auxiliary point/circular arc	D	

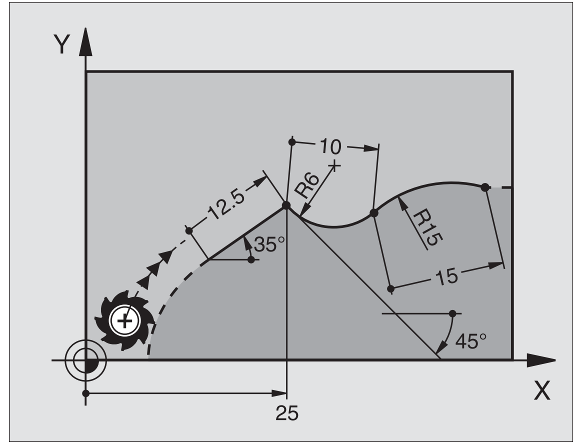
```
13 FC DR- R10 P1X+42.929 P1Y+60.071
14 FLT AH-70 PDX+50 PDY+53 D10
```



Direction and length of the contour element

Known data	Soft keys
Length of a straight line	LEN
Gradient angle of a straight line	AN
Chord length **LEN** of the arc	LEN
Gradient angle **AN** of the entry tangent	AN

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

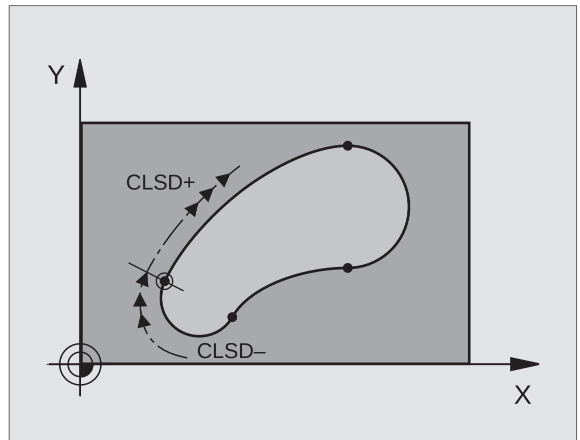


Identifying a closed contour

CLSD

Beginning of contour: **CLSD+**
End of contour: **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-
```

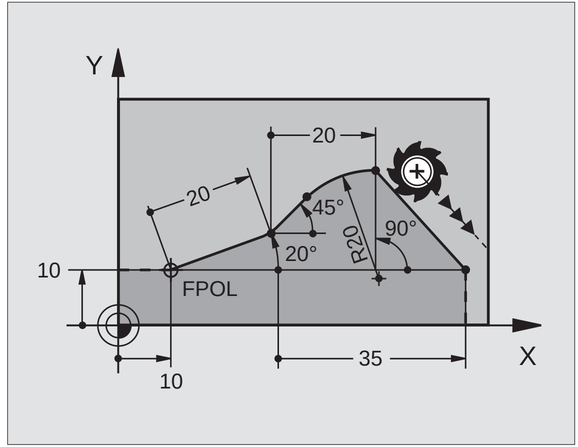


Data relative to block N: End point coordinates

The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the block number of the contour element on which the data are based.

Known data	Soft keys	
Cartesian coordinates relative to block N	RX N	RY N
Polar coordinates relative to block N	RPR N	RPA N

```
12 FPOL X+10 Y+10
13 FL PR+20 PA+20
14 FL AH+45
15 FCT IX+20 DR- R20 CCA+90 RX 13
16 FL IPR+35 FA+0 RPR 13
```

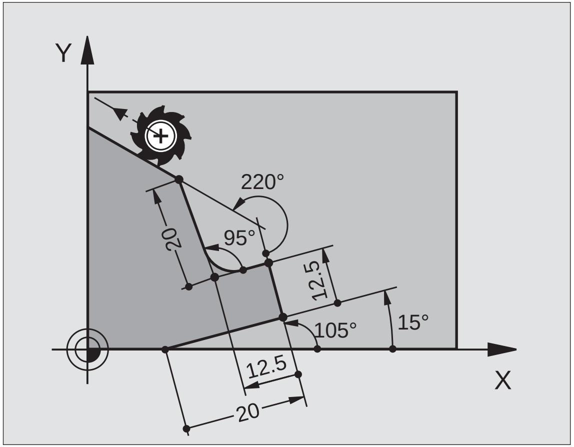


Data relative to block N: Direction and distance of the contour element

The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the block number of the contour element on which the data are based.

Known data	Soft keys
Angle between a straight line and another element or between the entry tangent of the arc and another element	RAN N
Straight line parallel to another contour element	PAR N
Distance from a straight line to a parallel contour element	DP

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

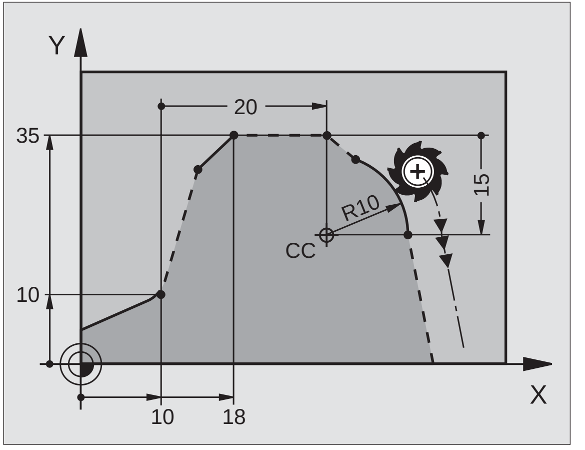


Data relative to block N: Circle center CC

> The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the block number of the contour element on which the data are based.

Known data	Soft keys	
Cartesian coordinates of the circle center relative to block N	RCCX N	RCCY N
Polar coordinates of the circle center relative to block N	RCCPR N	RCCPA N

```
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 desired.

Working with Subprograms

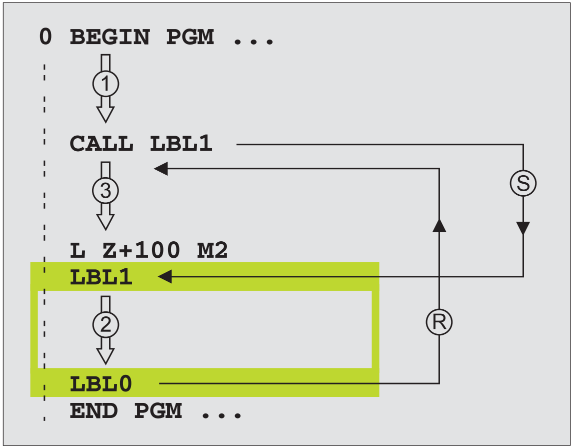
1 The main program runs up to the subprogram call **CALL LBL 1.**

2 The subprogram—labeled with **LBL 1**—runs through to its end at **LBL 0.**

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 NO ENT key.
- You cannot call **CALL LBL0!**



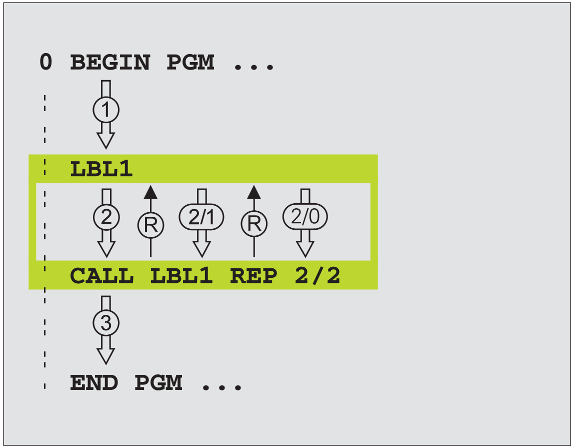
Working with Program Section Repeats

1 The main program runs up to the call for a section repeat **CALL LBL 1 REP2.**

2 The program section between **LBL 1** and **CALL LBL 1 REP2** 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.



Subprogram Nesting

Subprogram within a subprogram

1. The main program runs up to the first subprogram call **CALL LBL 1.**
2. Subprogram 1 runs up to the second subprogram call **CALL LBL 2.**
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.

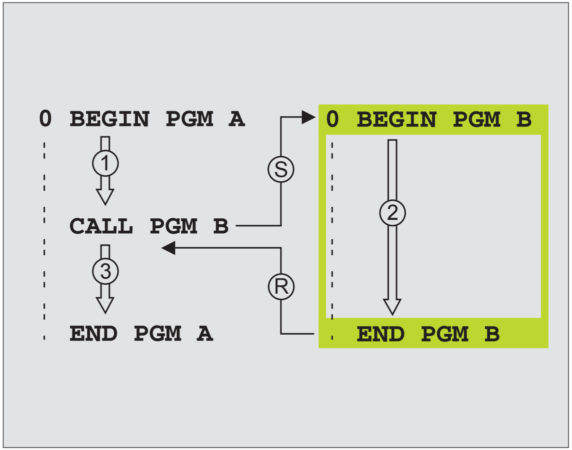
Program as subprogram

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.**



Working with Cycles

Certain frequently needed machining sequences are stored in the TNC as cycles. Coordinate transformations and other special cycles are also provided as standard cycles.

- In order to avoid erroneous entries during cycle definition, you should run a graphical program test before machining.
- The algebraic sign for the cycle parameter DEPTH determines the machining direction.
- For all cycles with numbers above 200 the TNC automatically pre-positions the tool in the tool axis.

Group of cycles	
Cycles for pecking, reaming, boring, counterboring, tapping and thread milling	DRILLING/ THREAD
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS
Cycles for producing point patterns, such as circular or linear hole patterns	PATTERN
SL (Subcontour List) cycles which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation	SL II
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Special cycles such as dwell time, program call, oriented spindle stop and tolerance	SPECIAL CYCLES

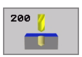
Cycle definition

CYCL DEF

- Select the Cycle Overview:

DRILLING/ THREAD

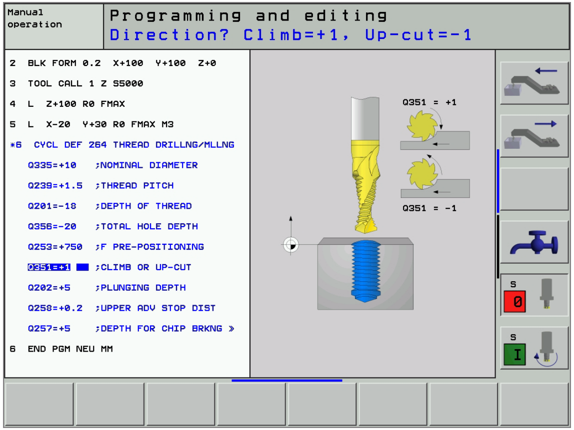
- Select the cycle group.



- Select the cycle.

Graphic support for programming cycles

The TNC supports you during cycle definition with graphic representations of the input parameters.



Calling cycles

The following cycles become effective automatically as soon as they are defined in the machining program:

- Coordinate Transformation Cycles
- DWELL TIME cycle
- The SL cycles CONTOUR and CONTOUR DATA
- Point Patterns
- Cycle TOLERANCE

All other cycles take effect after they are called with

- **CYCL CALL**: effective blockwise
- **CYCL CALL PAT**: effective blockwise in combination with point tables
- **CYCL CALL POS**: effective blockwise after the position defined in the **CYCL CALL POS** block was approached
- **M99**: effective blockwise
- **M89**: effective modally (depends on machine parameters)

Cycles for Drilling, Tapping and Thread Milling

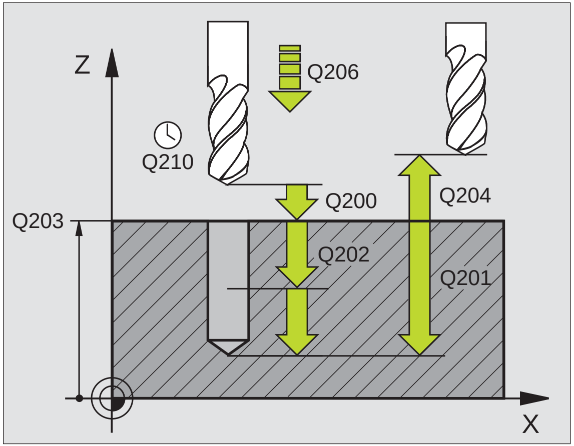
Overview

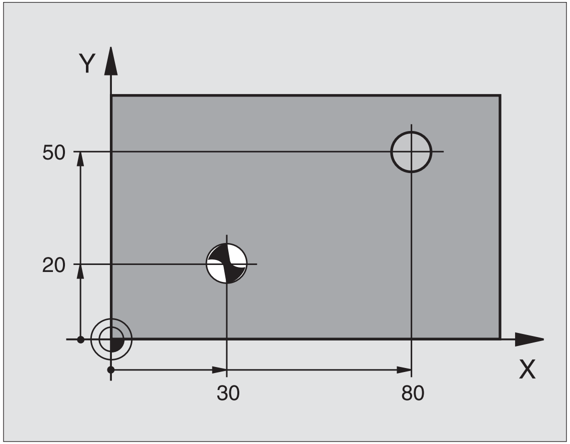
Available cycles		
200	DRILLING	page 47
201	REAMING	page 48
202	BORING	page 49
203	UNIVERSAL DRILLING	page 50
204	BACK BORING	page 51
205	UNIVERSAL PECKING	page 52
208	BORE MILLING	page 53
206	TAPPING NEW	page 54
207	RIGID TAPPING NEW	page 55
209	TAPPING W/ CHIP BRKG	page 56
262	THREAD MILLING	page 57
263	THREAD MILLING/COUNTERSINKING	page 58
264	THREAD DRILLING/MILLING	page 59
265	HELICAL THREAD DRLLNG/MLLNG	page 60
267	OUTSIDE THREAD MILLING	page 61

DRILLING (Cycle 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**
  - Plunging depth: **Q202**
  - Dwell time at top: **Q210**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Dwell time at depth: **Q211**

```
11 CYCL DEF 200 DRILLING
    Q200=2          ;SET-UP CLEARANCE
    Q201=-15        ;DEPTH
    Q206=250        ;FEED RATE FOR PLUNGING
    Q202=5          ;PLUNGING DEPTH
    Q210=0          ;DWELL TIME AT TOP
    Q203=+20        ;SURFACE COORDINATE
    Q204=100        ;2ND SET-UP CLEARANCE
    Q211=0.1        ;DWELL TIME AT DEPTH
12 CYCL CALL POS X+30 Y+20 M3
13 CYCL CALL POS X+80 Y+50
```

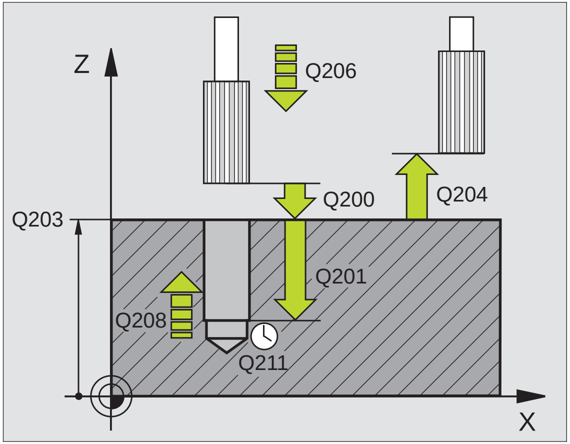


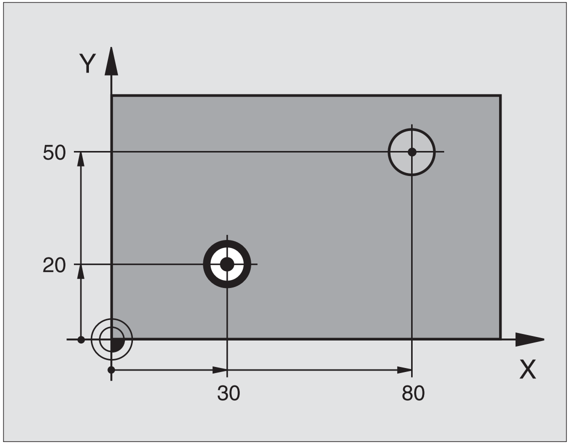


REAMING (Cycle 201)

- 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**
  - Feed rate for retraction: **Q208**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**

```
10 L Z+100 R0 FMAX
11 CYCL DEF 201 REAMING
     Q200=2        ;SET-UP CLEARANCE
     Q201=-15      ;DEPTH
     Q206=100      ;FEED RATE FOR PLUNGING
     Q211=0.5      ;DWELL TIME AT DEPTH
     Q208=250      ;RETRACTION FEED RATE
     Q203=+20      ;SURFACE COORDINATE
     Q204=100      ;2ND SET-UP CLEARANCE
12 CYCL CALL POS X+30 Y+20 M3
13 CYCL CALL POS X+80 Y+50
```

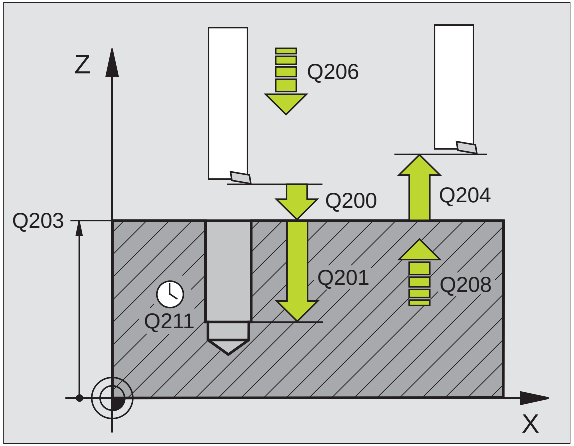




BORING (Cycle 202)

- The TNC and the machine tool must be specially prepared by the machine tool builder for the use of the BORING Cycle.
- This cycle requires a position-controlled spindle.

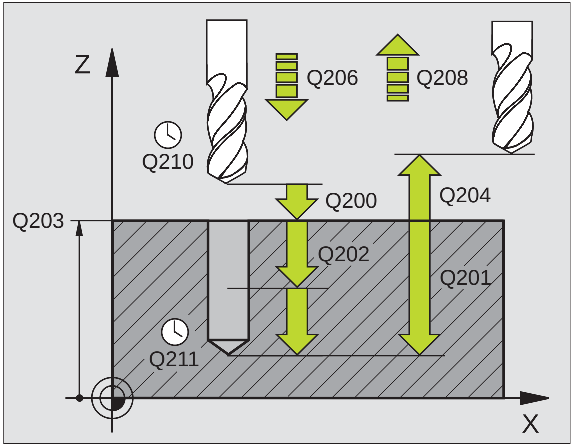
Danger of collision! Choose a disengaging direction that moves the tool away from the wall of the hole.



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

UNIVERSAL DRILLING (Cycle 203)

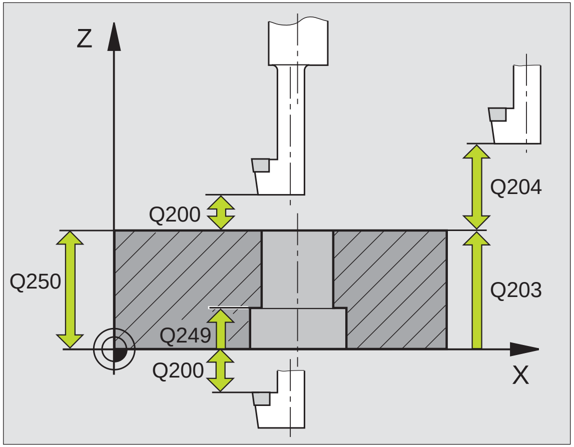
- CYCL DEF: Select Cycle **203 UNIVERSAL DRILLING**
  - set-up clearance: **Q200**
  - Depth: Distance between workpiece surface and bottom of hole: **Q201**
  - Feed rate for plunging: **Q206**
  - Plunging depth: **Q202**
  - Dwell time at top: **Q210**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Decrement after each pecking depth: **Q212**
  - No. of chip breaks before retraction: **Q213**
  - Min. pecking depth if a decrement has been entered: **Q205**
  - Dwell time at depth: **Q211**
  - Feed rate for retraction: **Q208**
  - Retraction rate for chip breaking: **Q256**

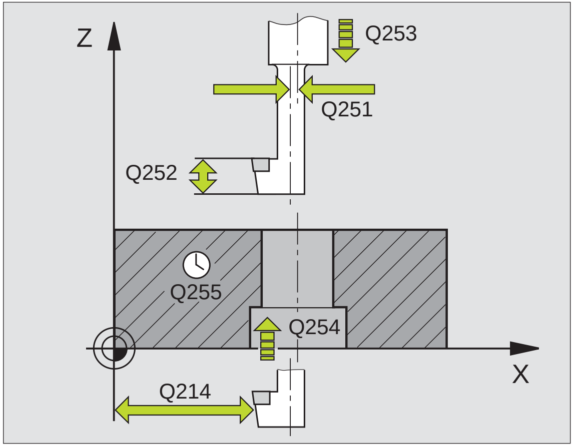


BACK BORING (Cycle 204)

- The TNC and the machine tool must be specially prepared by the machine tool builder for the use of the COUNTERBORE BACK Cycle.
- This cycle requires a position-controlled spindle.

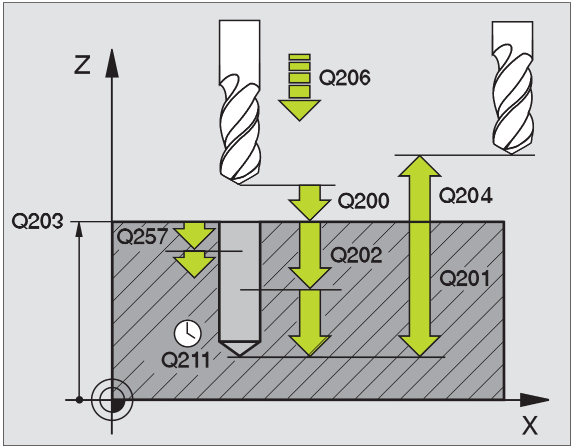
- Danger of collision! Choose a disengaging direction that moves the tool away from the counterbore floor.
- Use this cycle only with a reverse boring bar.





- CYCL DEF: Select Cycle **204 COUNTERBORE BACK**
  - set-up clearance: **Q200**
  - Depth of counterbore: **Q249**
  - Material thickness: **Q250**
  - Tool edge off-center distance: **Q251**
  - Tool edge height: **Q252**
  - Feed rate for pre-positioning: **Q253**
  - Feed rate for counterboring: **Q254**
  - Dwell time at counterbore floor: **Q255**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Disengaging direction (0/1/2/3/4): **Q214**
  - Angle for oriented spindle stop: **Q336**

UNIVERSAL PECKING (Cycle 205)

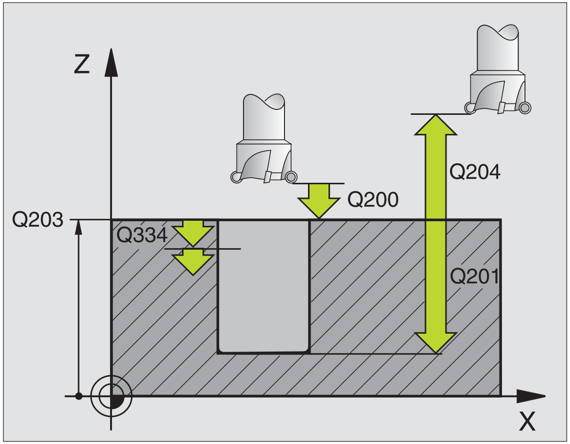


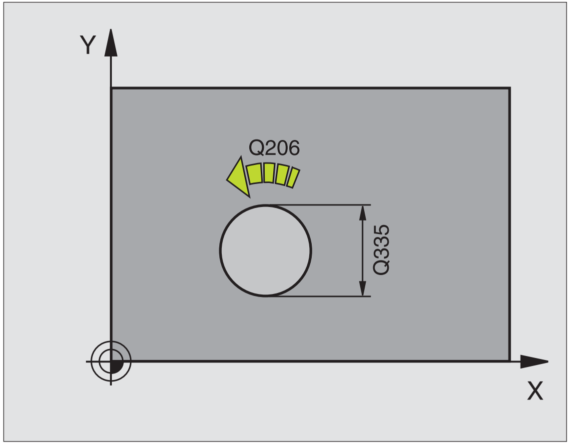
- CYCL DEF: Select Cycle **205 UNIVERSAL PECKING**
  - set-up clearance: **Q200**
  - Depth: Distance between workpiece surface and bottom of hole: **Q201**
  - Feed rate for plunging: **Q206**
  - Plunging depth: **Q202**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Decrement after each pecking depth: **Q212**
  - Min. pecking depth if a decrement has been entered: **Q205**
  - Upper advanced stop distance: **Q258**
  - Lower advanced stop distance: **Q259**
  - Infeed depth for chip breaking: **Q257**
  - Retraction rate for chip breaking: **Q256**
  - Dwell time at depth: **Q211**
  - Deepened starting point: **Q379**
  - Feed rate for pre-positioning: **Q253**

BORE MILLING (Cycle 208)

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

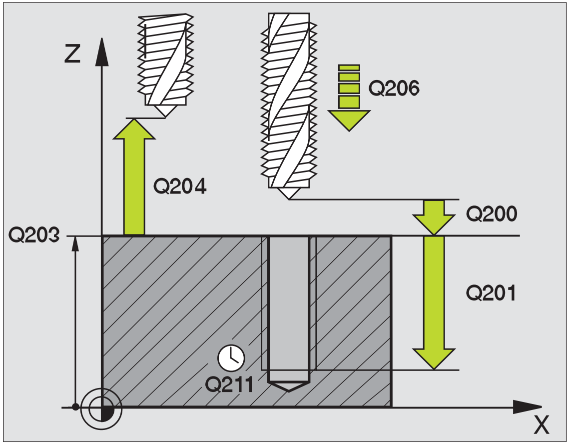
```
12 CYCL DEF 208 BORE MILLING
     Q200=2          ;SET-UP CLEARANCE
     Q201=-80        ;DEPTH
     Q206=150        ;FEED RATE FOR PLUNGING
     Q334=1.5        ;PLUNGING DEPTH
     Q203=+100       ;SURFACE COORDINATE
     Q204=50         ;2ND SET-UP CLEARANCE
     Q335=25         ;NOMINAL DIAMETER
     Q342=0          ;ROUGHING DIAMETER
```





TAPPING NEW with floating tap holder (Cycle 206)

For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

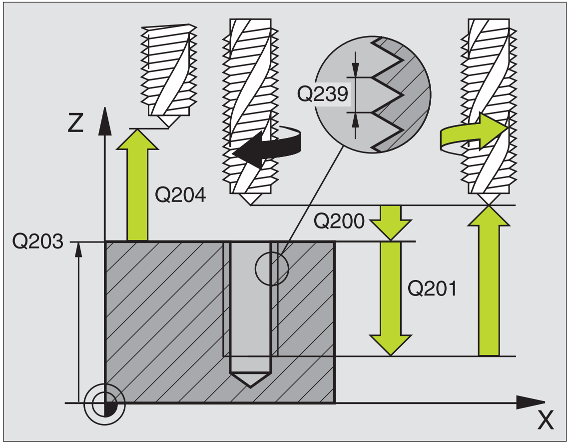


- Insert the floating tap holder
- CYCL DEF: Select Cycle **206 TAPPING NEW**
  - set-up clearance: **Q200**
  - Total hole depth: thread length = distance between the workpiece surface and the end of the thread: **Q201**
  - Feed rate F = Spindle speed S x thread pitch P: **Q206**
  - Enter the dwell time (a value between 0 and 0.5 seconds): **Q211**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**

```
25 CYCL DEF 206 TAPPING NEW
    Q200=2          ;SET-UP CLEARANCE
    Q201=-20        ;DEPTH
    Q206=150        ;FEED RATE FOR PLUNGING
    Q211=0.25       ;DWELL TIME AT DEPTH
    Q203=+25        ;SURFACE COORDINATE
    Q204=50         ;2ND SET-UP CLEARANCE
```

RIGID TAPPING without a floating tap holder NEW (Cycle 207)

- Machine and control must be specially prepared by the machine tool builder to enable rigid tapping.
- This cycle requires a position-controlled spindle.

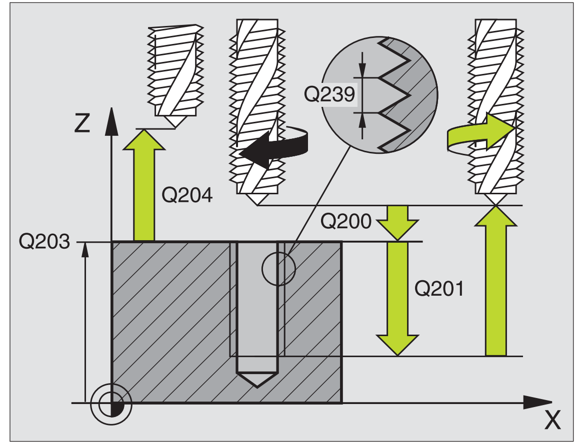


- CYCL DEF: Select Cycle **207 TAPPING NEW**
  - set-up clearance: **Q200**
  - Total hole depth: thread length = distance between the workpiece surface and the end of the thread: **Q201**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**

```
26 CYCL DEF 207 RIGID TAPPING NEW
    Q200=2         ;SET-UP CLEARANCE
    Q201=-20       ;DEPTH
    Q239=+1        ;PITCH
    Q203=+25       ;SURFACE COORDINATE
    Q204=50        ;2ND SET-UP CLEARANCE
```

TAPPING WITH CHIP BREAKING (Cycle 209)

- Machine and control must be specially prepared by the machine tool builder to enable tapping.
- This cycle requires a position-controlled spindle.

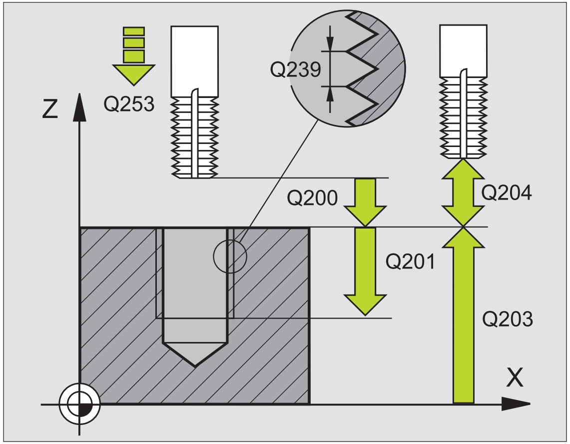


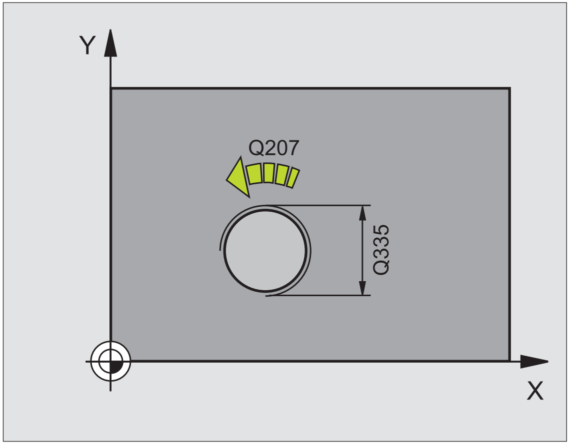
- CYCL DEF: Select Cycle **209 TAPPING WITH CHIP BREAKING**
  - set-up clearance: **Q200**
  - Total hole depth: thread length = distance between the workpiece surface and the end of the thread: **Q201**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Infeed depth for chip breaking: **Q257**
  - Retraction rate for chip breaking: **Q256**
  - Angle for oriented spindle stop: **Q336**

THREAD MILLING (Cycle 262)

- Pre-position to the center of the hole with **R0**
- CYCL DEF: Select Cycle **262 THREAD MILLING**
  - Nominal diameter of the thread: **Q335**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Thread depth: distance between the workpiece surface and the end of the thread: **Q201**
  - Number of threads per step: **Q355**
  - Feed rate for pre-positioning: **Q253**
  - Type of milling: **Q351**
    Climb: +1
    Up-cut: –1
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Feed rate for milling: **Q207**

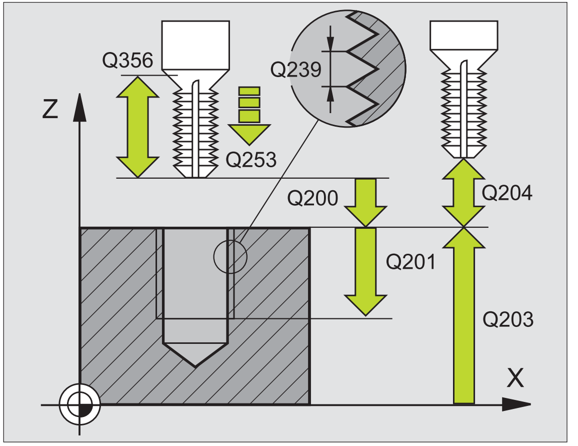
Note that the TNC makes a compensating movement in the tool axis before the approach movement. The length of the compensating movement depends on the thread pitch. Ensure sufficient space in the hole!

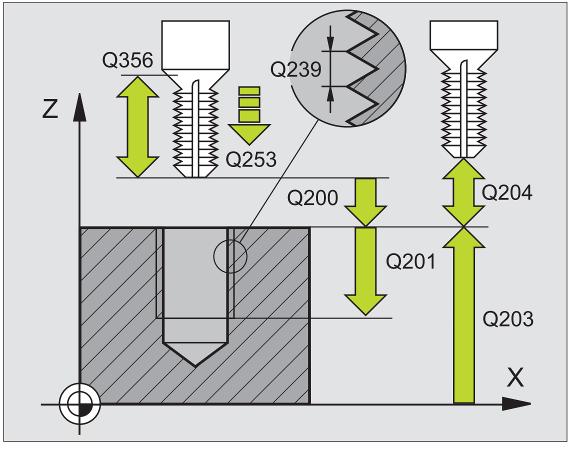




THREAD MILLING/COUNTERSINKING (Cycle 263)

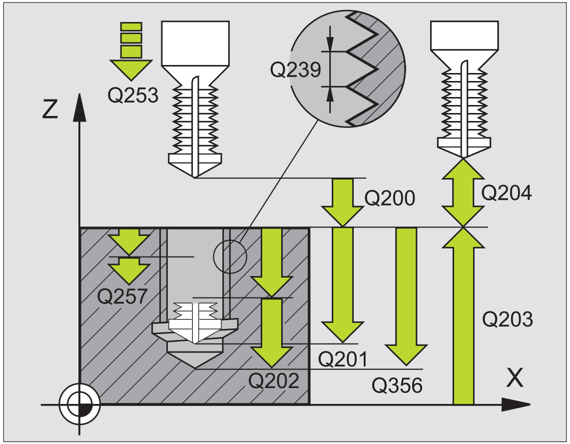
- Pre-position to the center of the hole with **R0**
- CYCL DEF: Select Cycle **263 THREAD MILLING AND COUNTERSINKING**
  - Nominal diameter of the thread: **Q335**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Thread depth: distance between the workpiece surface and the end of the thread: **Q201**
  - Countersinking depth: Distance between workpiece surface and bottom of hole: **Q356**
  - Feed rate for pre-positioning: **Q253**
  - Type of milling: **Q351**
    Climb: +1
    Up-cut: –1
  - set-up clearance: **Q200**
  - Lateral set-up clearance: **Q357**
  - Sinking depth at front: **Q358**
  - Countersinking offset at front: **Q359**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Feed rate for counterboring: **Q254**
  - Feed rate for milling: **Q207**

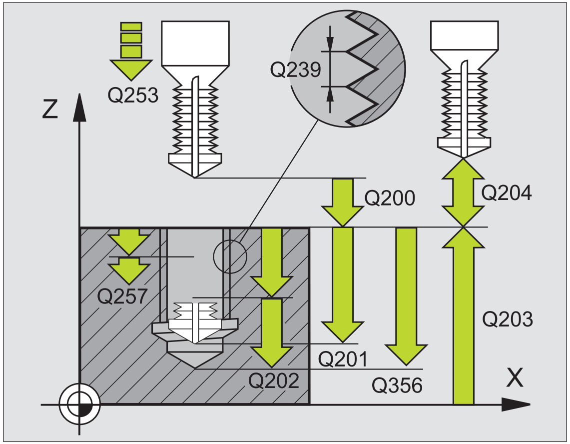




THREAD DRILLING/MILLING (Cycle 264)

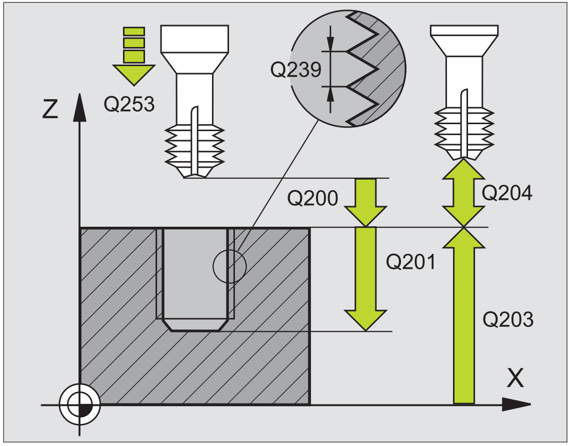
- Pre-position to the center of the hole with **R0**
- CYCL DEF: Select Cycle **264 THREAD DRILLING AND MILLING**
  - Nominal diameter of the thread: **Q335**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Thread depth: distance between the workpiece surface and the end of the thread: **Q201**
  - Total hole depth: Distance between workpiece surface and bottom of hole: **Q356**
  - Feed rate for pre-positioning: **Q253**
  - Type of milling: **Q351**
    Climb: +1
    Up-cut: –1
  - Plunging depth: **Q202**
  - Upper advanced stop distance: **Q258**
  - Infeed depth for chip breaking: **Q257**
  - Retraction rate for chip breaking: **Q256**
  - Dwell time at depth: **Q211**
  - Sinking depth at front: **Q358**
  - Countersinking offset at front: **Q359**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Feed rate for plunging: **Q206**
  - Feed rate for milling: **Q207**

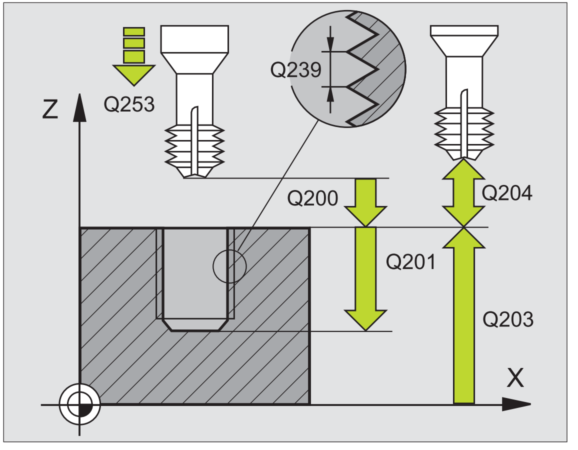




HELICAL THREAD DRILLING/MILLING (Cycle 265)

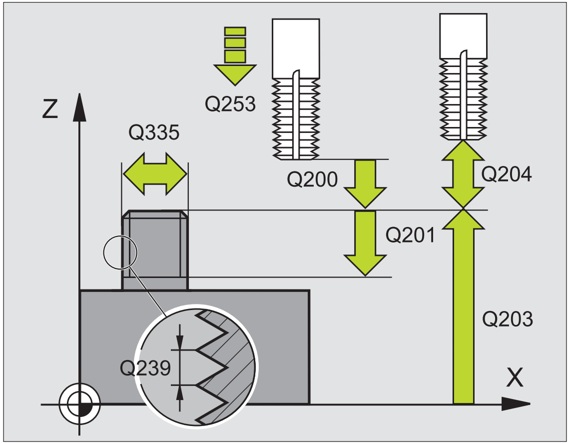
- Pre-position to the center of the hole with **R0**
- CYCL DEF: Select Cycle **265 HELICAL THREAD DRILLING AND MILLING**
  - Nominal diameter of the thread: **Q335**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Thread depth: distance between the workpiece surface and the end of the thread: **Q201**
  - Feed rate for pre-positioning: **Q253**
  - Sinking depth at front: **Q358**
  - Countersinking offset at front: **Q359**
  - Countersink: **Q360**
  - Plunging depth: **Q202**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Feed rate for counterboring: **Q254**
  - Feed rate for milling: **Q207**

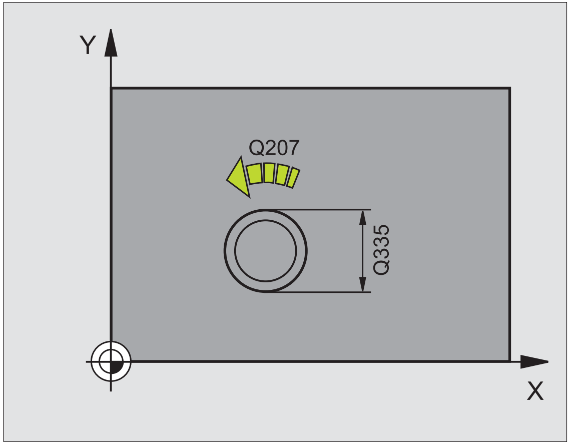




OUTSIDE THREAD MILLING (Cycle 267)

- Pre-position to the center of the hole with **R0**
- CYCL DEF: Select Cycle **267 OUTSIDE THREAD MILLING**
  - Nominal diameter of the thread: **Q335**
  - Pitch: **Q239**
    The algebraic sign differentiates between right-hand and left-hand threads:
    Right-hand thread: +
    Left-hand thread: –
  - Thread depth: distance between the workpiece surface and the end of the thread: **Q201**
  - Number of threads per step: **Q355**
  - Feed rate for pre-positioning: **Q253**
  - Type of milling: **Q351**
    Climb: +1
    Up-cut: –1
  - set-up clearance: **Q200**
  - Sinking depth at front: **Q358**
  - Countersinking offset at front: **Q359**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Feed rate for counterboring: **Q254**
  - Feed rate for milling: **Q207**





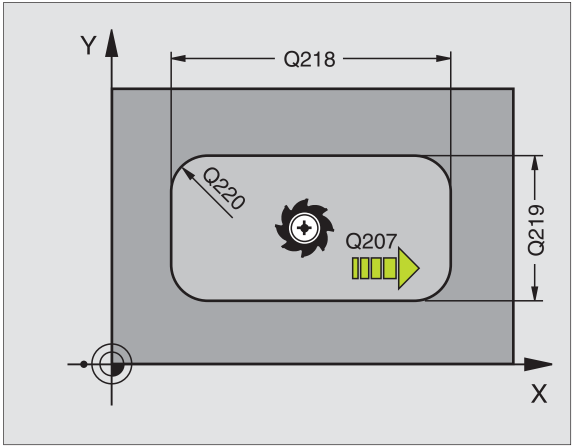
Pockets, Studs and Slots

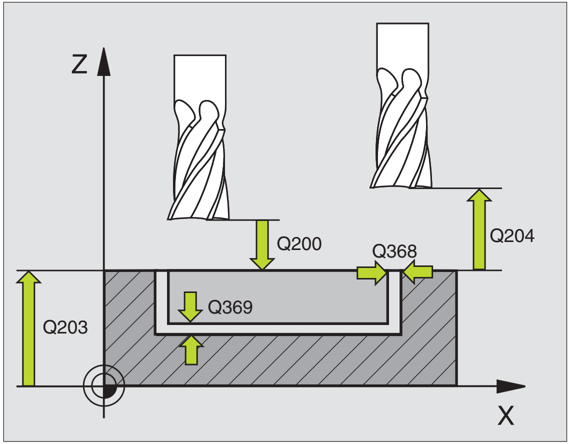
Overview

Available cycles		
251	RECTANGULAR POCKET complete	page 63
252	CIRCULAR POCKET complete	page 64
253	SLOT complete	page 65
254	ROUNDED SLOT complete	page 66
212	POCKET FINISHING	page 67
213	STUD FINISHING	page 68
214	CIRCULAR POCKET FINISHING	page 69
215	CIRCULAR STUD FINISHING	page 70

RECTANGULAR POCKET (Cycle 251)

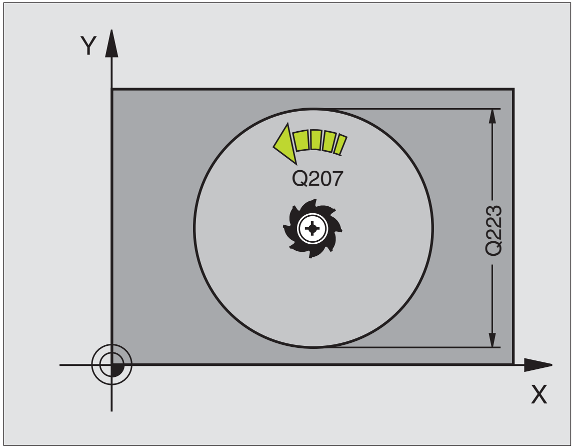
- CYCL DEF: Select Cycle **251 RECTANGULAR POCKET**
  - Machining operation (0/1/2): **Q215**
  - 1. side length: **Q218**
  - 2. side length: **Q219**
  - Corner radius: **Q220**
  - Finishing allowance for side: **Q368**
  - Angle of rotation: **Q224**
  - Pocket position: **Q367**
  - Feed rate for milling: **Q207**
  - Type of milling: **Q351.** Climb: +1; Up-cut: –1
  - Depth: Distance between workpiece surface and bottom of pocket: **Q201**
  - Plunging depth: **Q202**
  - Finishing allowance for floor: **Q369**
  - Feed rate for plunging: **Q206**
  - Infeed for finishing: **Q338**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Path overlap factor: **Q370**
  - Plunging strategy: **Q366**. 0 = vertical plunging; 1 = helical plunging; 2 = reciprocating plunging
  - Feed rate for finishing: **Q385**

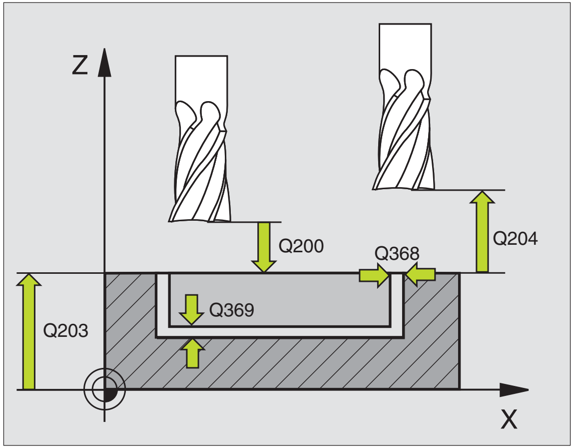




CIRCULAR POCKET (Cycle 252)

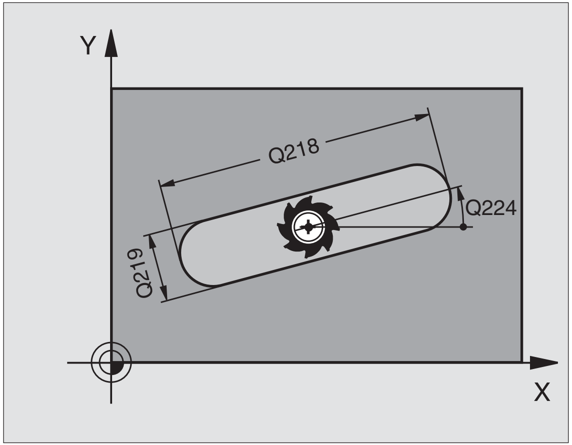
- CYCL DEF: Select Cycle **252 CIRCULAR POCKET**
  - Machining operation (0/1/2): **Q215**
  - Finished part diameter: **Q223**
  - Finishing allowance for side: **Q368**
  - Feed rate for milling: **Q207**
  - Type of milling: **Q351.** Climb: +1; Up-cut: –1
  - Depth: Distance between workpiece surface and bottom of pocket: **Q201**
  - Plunging depth: **Q202**
  - Finishing allowance for floor: **Q369**
  - Feed rate for plunging: **Q206**
  - Infeed for finishing: **Q338**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Path overlap factor: **Q370**
  - Plunging strategy: **Q366**. 0 = vertical plunging; 1 = helical plunging
  - Feed rate for finishing: **Q385**

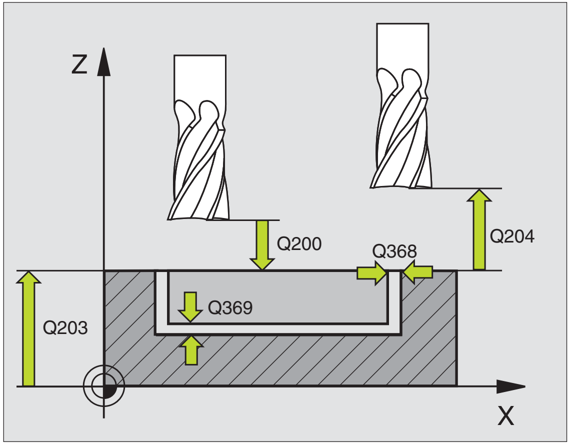




SLOT MILLING (Cycle 253)

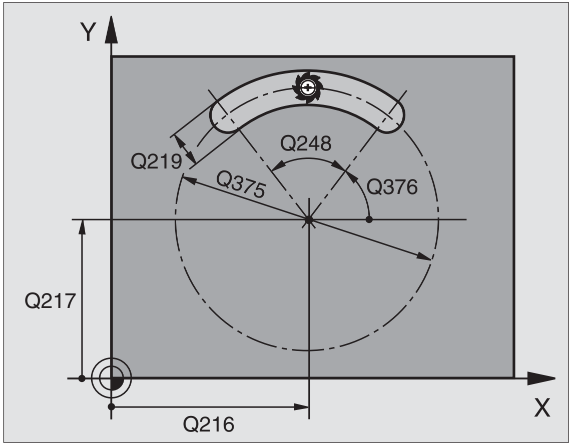
- CYCL DEF: Select Cycle **253 SLOT MILLING**
  - Machining operation (0/1/2): **Q215**
  - 1. side length: **Q218**
  - 2. side length: **Q219**
  - Finishing allowance for side: **Q368**
  - Angle by which the entire slot is rotated: **Q374**
  - Slot position (0/1/2/3/4): **Q367**
  - Feed rate for milling: **Q207**
  - Type of milling: **Q351.** Climb: +1; Up-cut: –1
  - Depth: Distance between workpiece surface and bottom of slot: **Q201**
  - Plunging depth: **Q202**
  - Finishing allowance for floor: **Q369**
  - Feed rate for plunging: **Q206**
  - Infeed for finishing: **Q338**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Plunging strategy: **Q366**. 0 = vertical plunging; 1 = reciprocating plunging
  - Feed rate for finishing: **Q385**

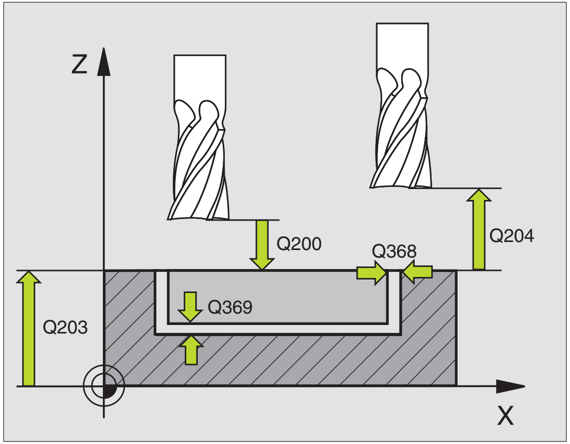




CIRCULAR SLOT (Cycle 254)

- CYCL DEF: Select Cycle **254 CIRCULAR SLOT**
  - Machining operation (0/1/2): **Q215**
  - 2. side length: **Q219**
  - Finishing allowance for side: **Q368**
  - Pitch circle diameter: **Q375**
  - Slot position (0/1/2/3): **Q367**
  - Center in 1st axis: **Q216**
  - Center in 2nd axis: **Q217**
  - Starting angle: **Q376**
  - Angular length: **Q248**
  - Angle increment: **Q378**
  - Number of repetitions: **Q377**
  - Feed rate for milling: **Q207**
  - Type of milling: **Q351.** Climb: +1; Up-cut: –1
  - Depth: Distance between workpiece surface and bottom of slot: **Q201**
  - Plunging depth: **Q202**
  - Finishing allowance for floor: **Q369**
  - Feed rate for plunging: **Q206**
  - Infeed for finishing: **Q338**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Plunging strategy: **Q366**. 0 = vertical plunging; 1 = helical plunging
  - Feed rate for finishing: **Q385**

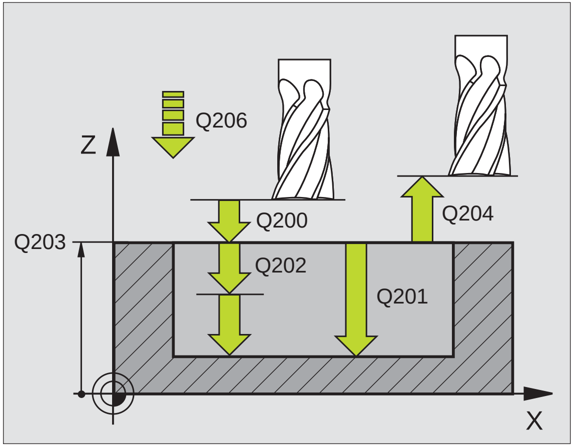


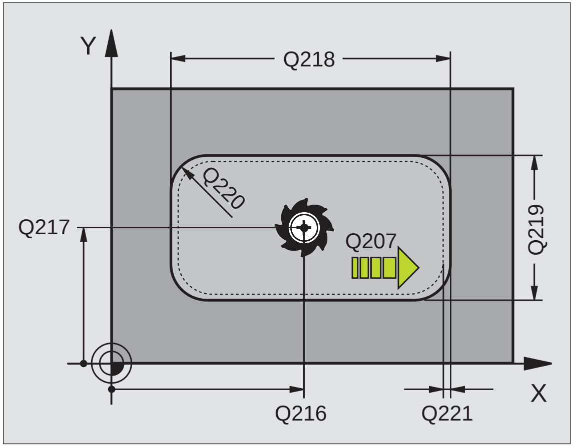


POCKET FINISHING (Cycle 212)

- CYCL DEF: Select Cycle **212 POCKET FINISHING**
  - set-up clearance: **Q200**
  - Depth: Distance between workpiece surface and bottom of pocket: **Q201**
  - Feed rate for plunging: **Q206**
  - Plunging depth: **Q202**
  - Feed rate for milling: **Q207**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Center in 1st axis: **Q216**
  - Center in 2nd axis: **Q217**
  - 1. side length: **Q218**
  - 2. side length: **Q219**
  - Corner radius: **Q220**
  - Oversize in 1st axis: **Q221**

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

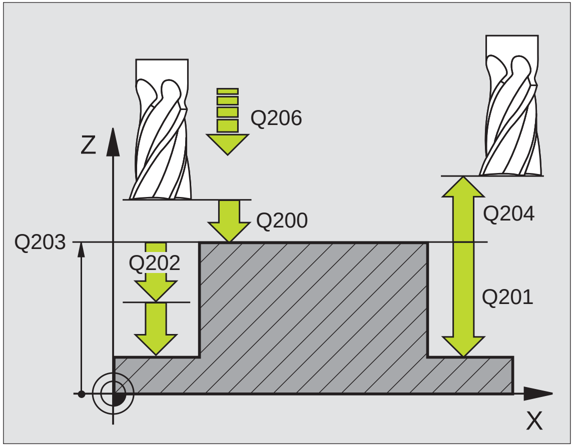


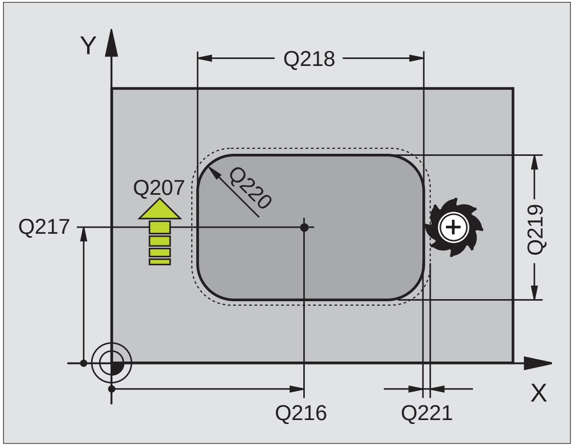


STUD FINISHING (Cycle 213)

- CYCL DEF: Select Cycle **213 STUD FINISHING**
  - set-up clearance: **Q200**
  - Depth: Distance between workpiece surface and bottom of stud: **Q201**
  - Feed rate for plunging: **Q206**
  - Plunging depth: **Q202**
  - Feed rate for milling: **Q207**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Center in 1st axis: **Q216**
  - Center in 2nd axis: **Q217**
  - 1. side length: **Q218**
  - 2. side length: **Q219**
  - Corner radius: **Q220**
  - Oversize in 1st axis: **Q221**

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

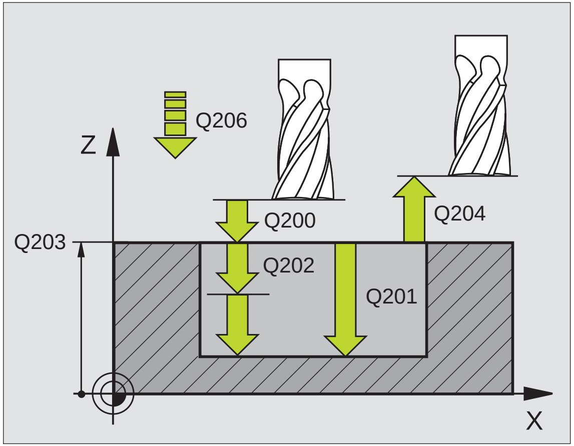


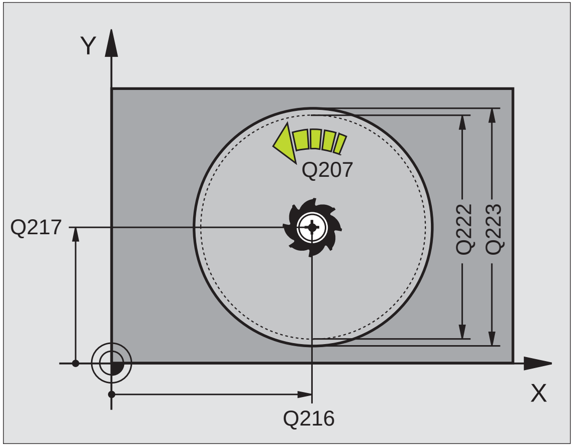


CIRCULAR POCKET FINISHING (Cycle 214)

- CYCL DEF: Select Cycle **214 C. POCKET FINISHING**
  - set-up clearance: **Q200**
  - Depth: Distance between workpiece surface and bottom of pocket: **Q201**
  - Feed rate for plunging: **Q206**
  - Plunging depth: **Q202**
  - Feed rate for milling: **Q207**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Center in 1st axis: **Q216**
  - Center in 2nd axis: **Q217**
  - Workpiece blank diameter: **Q222**
  - Finished part diameter: **Q223**

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

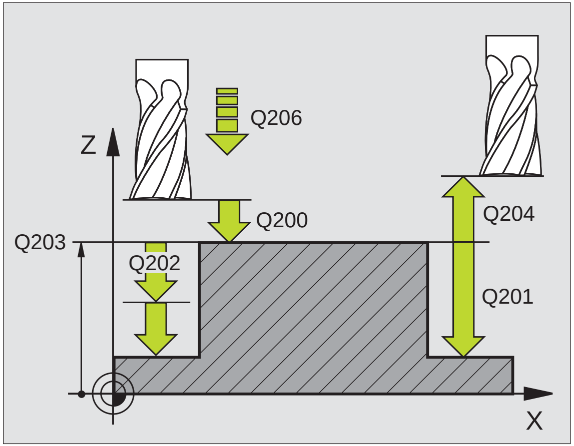


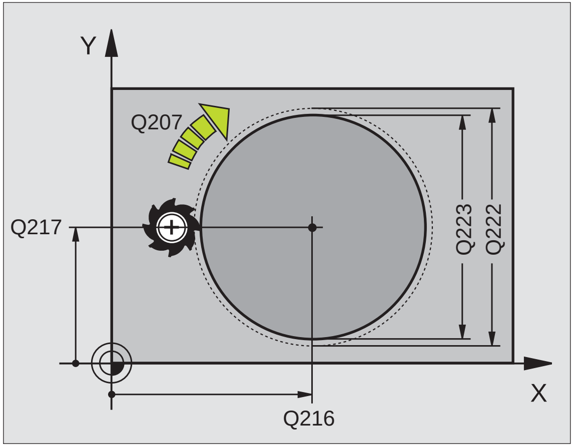


CIRCULAR STUD FINISHING (Cycle 215)

- CYCL DEF: Select Cycle **215 C. STUD FINISHING**
  - set-up clearance: **Q200**
  - Depth: Distance between workpiece surface and bottom of stud: **Q201**
  - Feed rate for plunging: **Q206**
  - Plunging depth: **Q202**
  - Feed rate for milling: **Q207**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Center in 1st axis: **Q216**
  - Center in 2nd axis: **Q217**
  - Workpiece blank diameter: **Q222**
  - Finished part diameter: **Q223**

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





Point Patterns

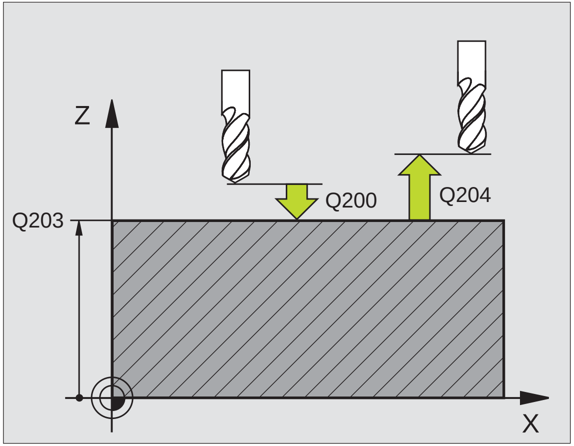
Overview

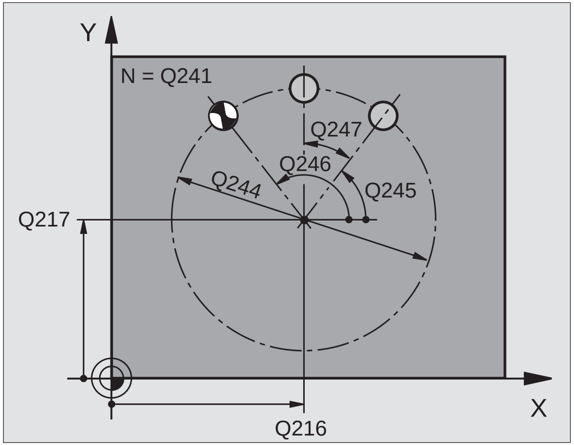
Available cycles		
220	POLAR PATTERN	page 71
221	LINEAR PATTERN	page 72

CIRCULAR PATTERN (Cycle 220)

- CYCL DEF: Select Cycle **220 CIRCULAR PATTERN**
  - Center in 1st axis: **Q216**
  - Center in 2nd axis: **Q217**
  - Pitch circle diameter: **Q244**
  - Starting angle: **Q245**
  - Stopping angle: **Q246**
  - Angle increment: **Q247**
  - Number of repetitions: **Q241**
  - set-up clearance: **Q200**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Move to clearance height: **Q301**
  - Type of traverse: **Q365**

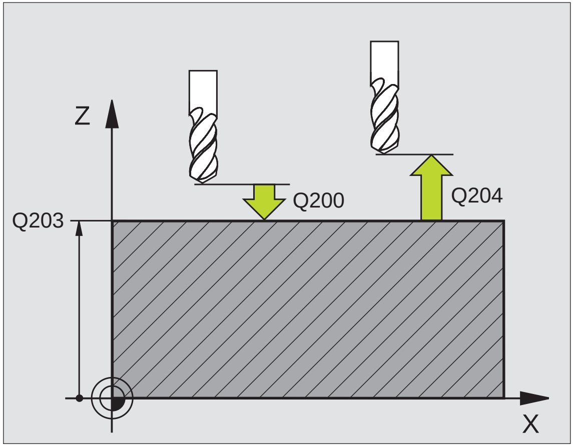
You can combine the following cycles with Cycle 200: 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 212, 213, 214, 215, 251, 252, 253, 254, 262, 263, 264, 265, 267.



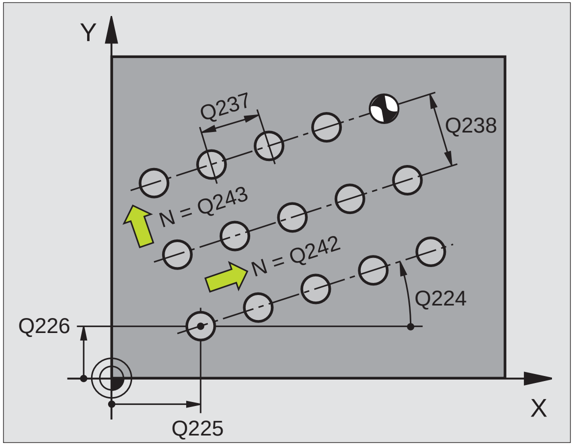


LINEAR PATTERN (Cycle 221)

- CYCL DEF: Select Cycle **221 LINEAR PATTERN**
  - Starting point in 1st axis: **Q225**
  - Starting point in 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**
  - Workpiece surface coordinate: **Q203**
  - 2. set-up clearance: **Q204**
  - Move to clearance height: **Q301**



- Cycle **221 LINEAR PATTERN** is effective immediately upon definition.
- Cycle 221 automatically calls the last defined fixed cycle.
- You can combine the following cycles with Cycle 221: 1, 2, 3, 4, 5, 17, 200, 201, 202, 203, 204, 205, 206, 207, 208, 209, 212, 213, 214, 215, 251, 252, 253, 262, 263, 264, 265, 267.
- In combined cycles, the set-up clearance, surface coordinate and 2nd set-up-clearance are always taken from Cycle 221.



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

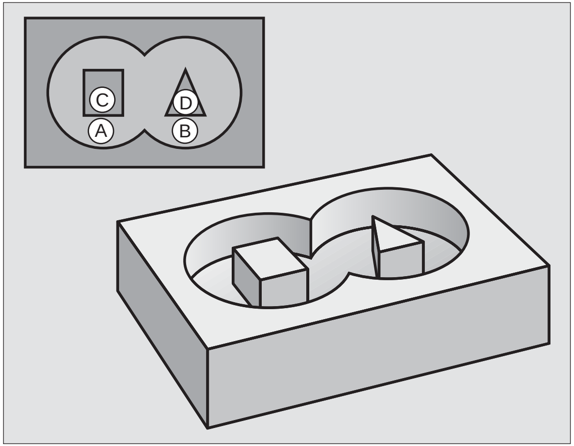
SL Cycles

Overview

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 **infeeds** in the **tool axis cannot** be programmed in SL cycles.
- Each contour listed in Cycle 14 contour Geometry must be a closed contour.
- The memory capacity for programming an SL cycle is limited. For example, you can program approximately 2048 straight-line blocks in one 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.

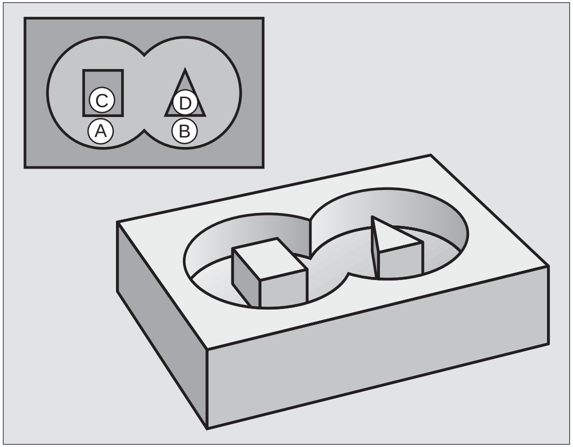
CONTOUR GEOMETRY (Cycle 14)

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 GEOMETRY
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
...
```

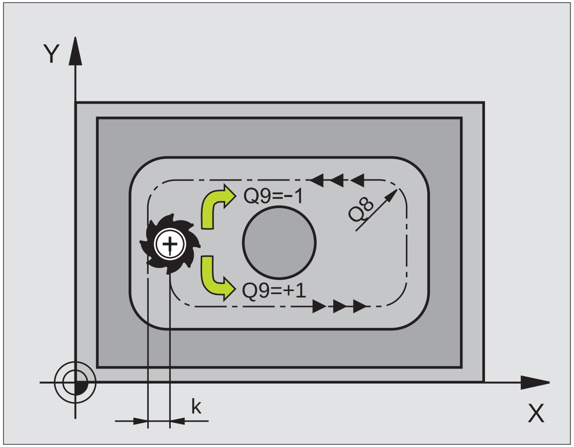


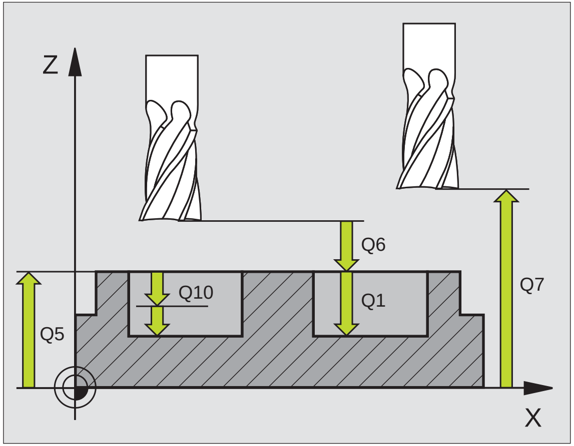
CONTOUR DATA (Cycle 20)

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

- CYCL DEF: Select Cycle **20 CONTOUR DATA**
  - Milling depth: Distance between workpiece surface and bottom of pocket: **Q1**
  - Path overlap factor: **Q2**
  - Finishing allowance for side: **Q3**
  - Finishing allowance for floor: **Q4**
  - Workpiece surface coordinate: Coordinate of the workpiece surface referenced to the current datum: Q5 **Q5**
  - Set-up clearance: Distance from the tool to the workpiece surface: **Q6**
  - Clearance height: Height at which collision with the workpiece is impossible: **Q7**
  - Inside corner radius: Rounding radius at inside corners referenced to the tool midpoint path: **Q8**
  - Direction of rotation: **Q9**: Clockwise Q9 = –1; counterclockwise Q9 = +1

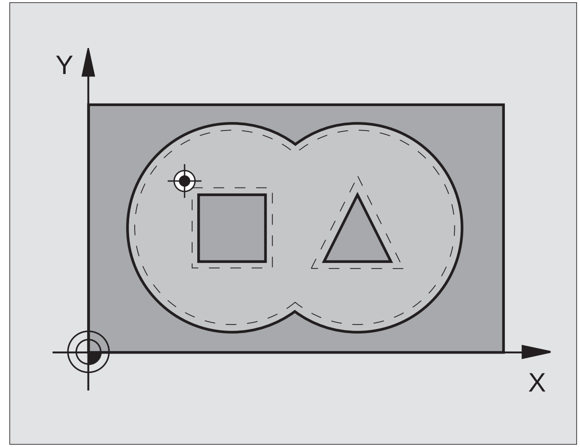
Cycle **20 CONTOUR DATA** is effective immediately upon definition.





PILOT DRILLING (Cycle 21)

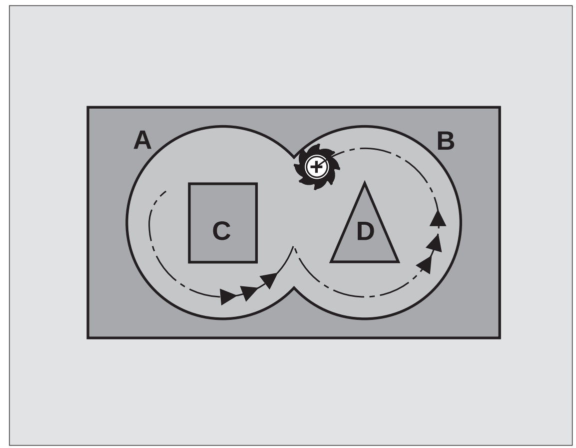
- CYCL DEF: Select Cycle **21 PILOT DRILLING**
  - Plunging depth: **Q10** incremental
  - Feed rate for plunging: **Q11**
  - Roughing tool number: **Q13**



ROUGH-OUT (Cycle 22)

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

- CYCL DEF: Select Cycle **22 ROUGH-OUT**
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Course roughing tool number: **Q18**
  - Reciprocation feed rate: **Q19**
  - Feed rate for retraction: **Q208**

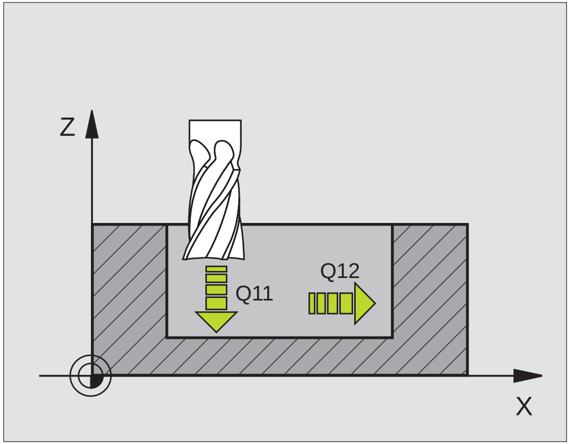


FLOOR FINISHING (Cycle 23)

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 plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Feed rate for retraction: **Q208**

Call Cycle **22 ROUGH-OUT** before calling Cycle 23.

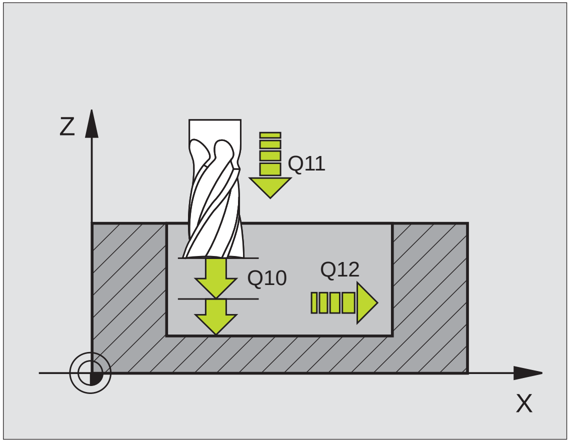


SIDE FINISHING (Cycle 24)

Finishing the individual contour elements

- CYCL DEF: Select Cycle **24 SIDE FINISHING**
  - Direction of rotation: **Q9**. Clockwise Q9 = –1; counterclockwise Q9 = +1
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Finishing allowance for side: **Q14;** Allowance for finishing in several passes

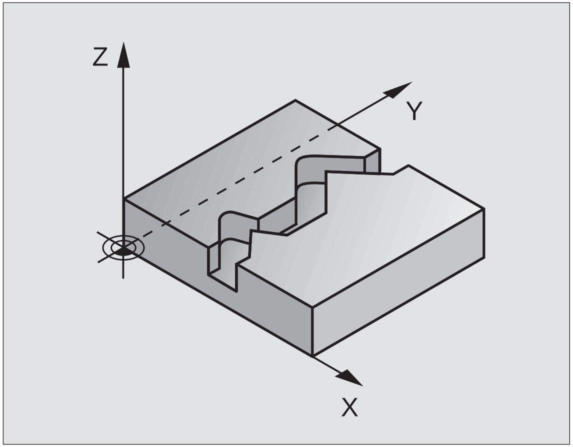
Call Cycle **22 ROUGH-OUT** before calling Cycle 24.



CONTOUR TRAIN (Cycle 25)

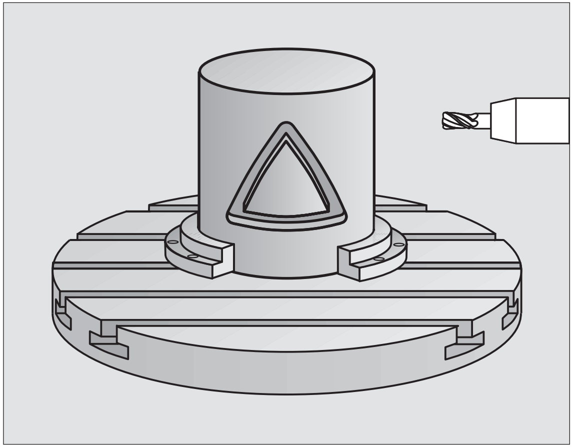
This cycle is for entering data for machining an open contour that has been defined in a contour subprogram.

- CYCL DEF: Select Cycle **25 CONTOUR TRAIN**
  - Milling depth: **Q1**
  - Finishing allowance for side: **Q3**. Finishing allowance in the working plane
  - Workpiece surface coordinate: **Q5**) Coordinate of the workpiece surface
  - Clearance height: **Q7:** Height at which the tool cannot collide with the workpiece
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Type of milling: **Q15.** Climb: Q15 = +1; Up-cut: Q15 = –1; reciprocatingly, in several infeeds: Q15 = 0



- Cycle **14 CONTOUR GEOMETRY** can have only one label number.
- A subprogram can hold approx. 2048 line segments.
- Do not program incremental dimensions after calling the cycle: danger of collision.
- After calling the cycle, move to a defined absolute position.

CYLINDER SURFACE (Cycle 27, Software Option 1)

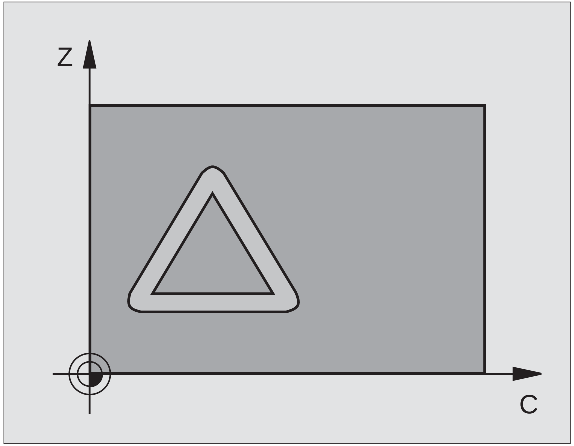


The TNC and the machine tool must be specially prepared by the machine tool builder for the use of the **27 CYLINDER SURFACE** Cycle.

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**
  - Set-up clearance: **Q6**. Distance between tool and workpiece surface
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Cylinder radius: **Q16**. Radius of the cylinder
  - Dimension type: **Q17**. Degrees = 0; mm/inch = 1

- 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 approx. 1024 line segments.



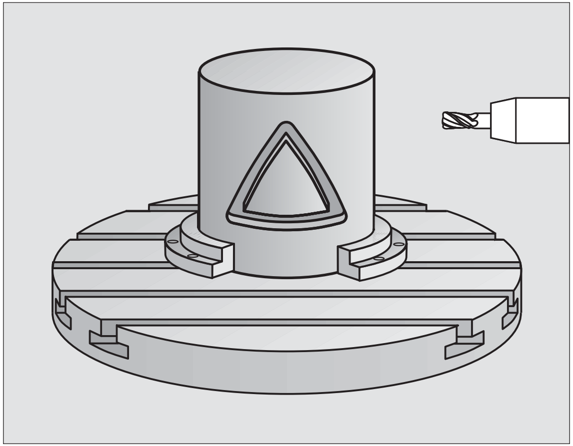
CYLINDER SURFACE (Cycle 28, Software Option 1)

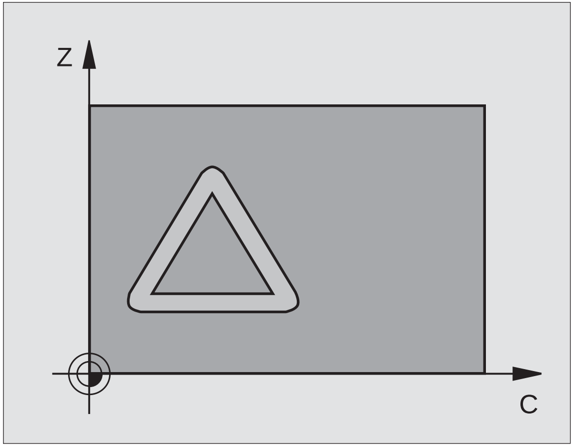
The TNC and the machine tool must be specially prepared by the machine tool builder for the use of the **28 CYLINDER SURFACE** Cycle.

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

- Define a contour in a subprogram and list it in Cycle **14 CONTOUR GEOMETRY**
- CYCL DEF: Select Cycle **28 CYLINDER SURFACE**
  - Milling depth: **Q1**
  - Finishing allowance for side: **Q3**
  - Set-up clearance: **Q6**. Distance between tool and workpiece surface
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Cylinder radius: **Q16**. Radius of the cylinder
  - Dimension type: **Q17**. Degrees = 0; mm/inch = 1
  - Slot width: **Q20**
  - Tolerance: **Q21**

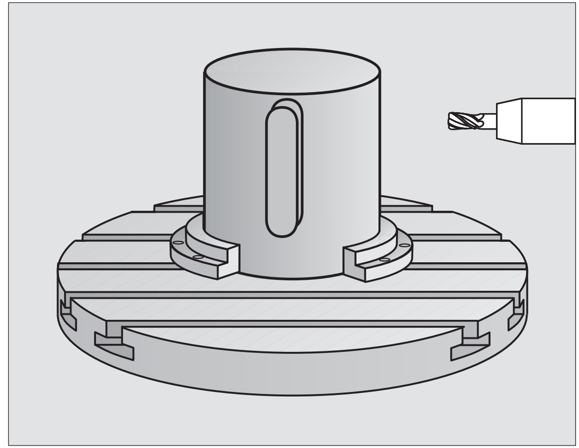
- The workpiece must be set up concentrically on the rotary table.
- The tool axis must be perpendicular to the axis of the rotary table.
- Cycle **14 CONTOUR GEOMETRY** can have only one label number.
- A subprogram can hold approx. 2048 line segments.

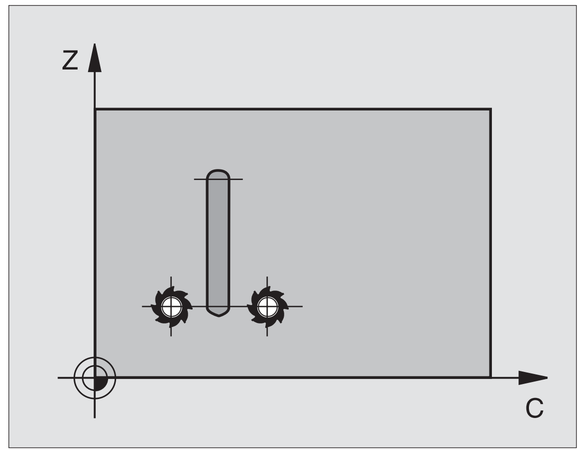




CYLINDER SURFACE (Cycle 29, Software Option 1)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of the **29 CYLINDER SURFACE** Cycle.





Cycle **29 CYLINDER SURFACE** enables you to program a ridge in only two axes, and then machine it on a cylindrical surface without distorting the angle of the slot walls.

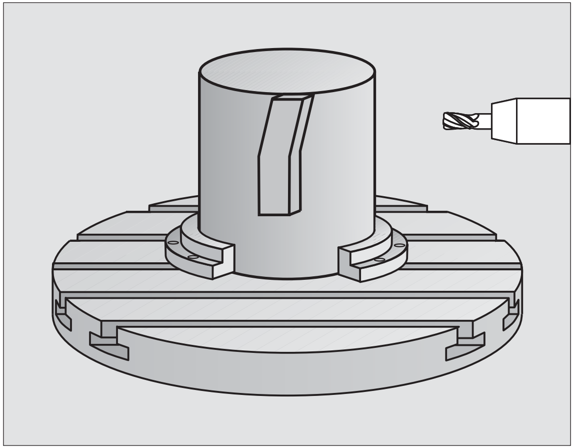
- Define a contour in a subprogram and list it in Cycle **14 CONTOUR GEOMETRY**
- CYCL DEF: Select Cycle **29 CYL SURFACE RIDGE**
  - Milling depth: **Q1**
  - Finishing allowance for side: **Q3**
  - Set-up clearance: **Q6**. Distance between tool and workpiece surface
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for roughing: **Q12**
  - Cylinder radius: **Q16**. Radius of the cylinder
  - Dimension type: **Q17**. Degrees = 0; mm/inch = 1
  - Ridge width: **Q20**

- 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 approx. 2048 line segments.

CYLINDER SURFACE (Cycle 39, Software Option 1)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of the **39 CYL SURFACE CONTOUR** Cycle.

Cycle **39 CYL SURFACE CONTOUR** enables you to program an open contour in only two axes, as if in a plane. The TNC then converts it for a cylindrical surface.



- Define a contour in a subprogram and list it in Cycle **14 CONTOUR GEOMETRY**
- CYCL DEF: Select Cycle **39 CYL SURFACE CONTOUR**
  - Milling depth: **Q1**
  - Finishing allowance for side: **Q3**
  - Set-up clearance: **Q6**. Distance between tool and workpiece surface
  - Plunging depth: **Q10**
  - Feed rate for plunging: **Q11**
  - Feed rate for milling: **Q12**
  - Cylinder radius: **Q16**. Radius of the cylinder
  - Dimension type: **Q17**. Degrees = 0; mm/inch = 1

- 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 approx. 2048 line segments.

Cycles for Multipass Milling

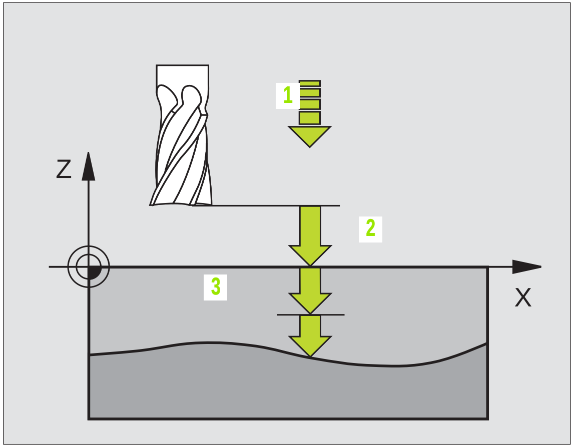
Overview

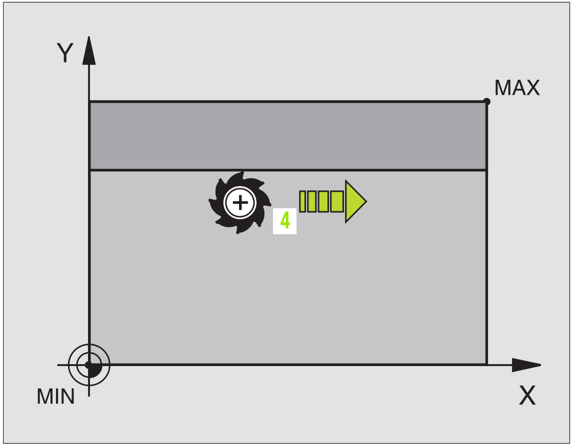
Available cycles		
30	3-D DATA	page 84
230	MULTIPASS MILLING	page 85
231	RULED SURFACE	page 86
232	FACE MILLING	page 87

3-D DATA (Cycle 14)

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

- CYCL DEF: Select Cycle **30 3-D DATA**
  - Program name for digitized data
  - Minimum point of range
  - Maximum point of range
  - Set-up clearance: **1**
  - Plunging depth: **2**
  - Feed rate for plunging: **3**
  - Feed rate: **4**
  - Miscellaneous function M.

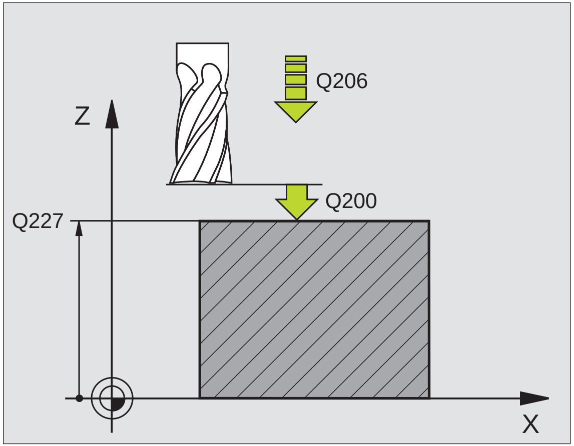


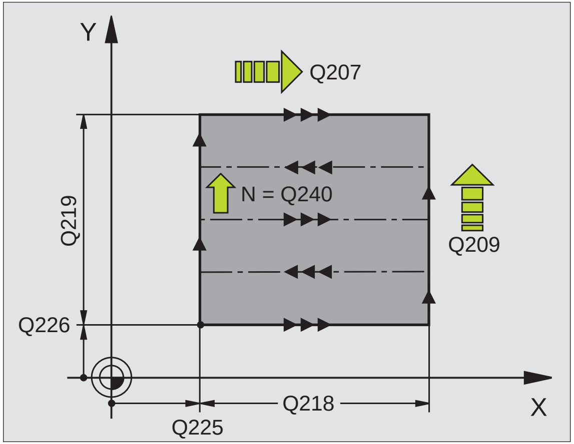


MULTIPASS MILLING (Cycle 230)

> ⚠ From the current position, the TNC positions the tool at the starting point, first in the working plane and then in the tool axis. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

- 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**
  - 1. side length: **Q218**
  - 2. 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**

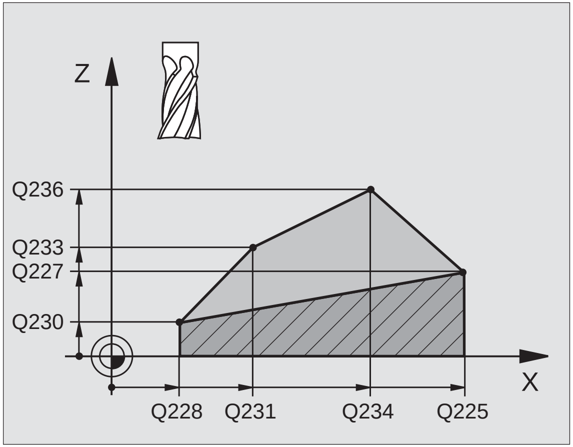


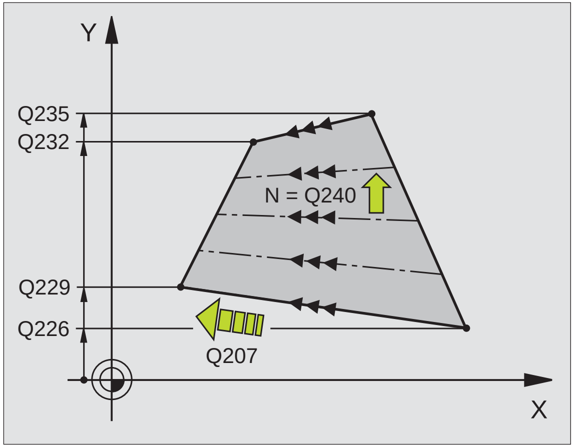


RULED SURFACE (Cycle 231)

From the current position, the TNC positions the tool at the starting point (point 1), first in the working plane and then in the tool axis. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

- 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**
  - 2. point in 1st axis: **Q228**
  - 2. point in 2nd axis: **Q229**
  - 2. point in 3rd axis: **Q230**
  - 3. point in 1st axis: **Q232**
  - 3. point in 2nd axis: **Q232**
  - 3. point in 3rd axis: **Q233**
  - 4. point in 1st axis: **Q234**
  - 4. point in 2nd axis: **Q235**
  - 4. point in 3rd axis: **Q236**
  - Number of cuts: **Q240**
  - Feed rate for milling: **Q207**

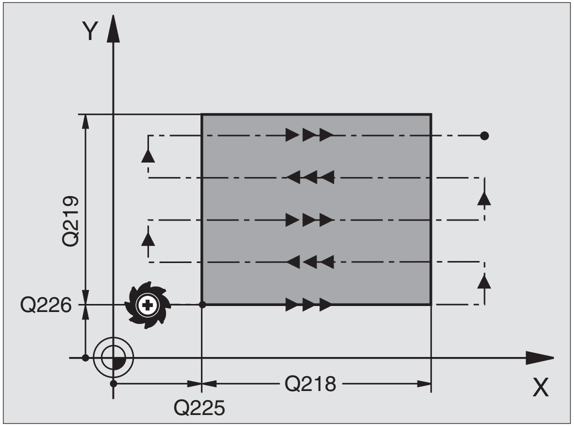


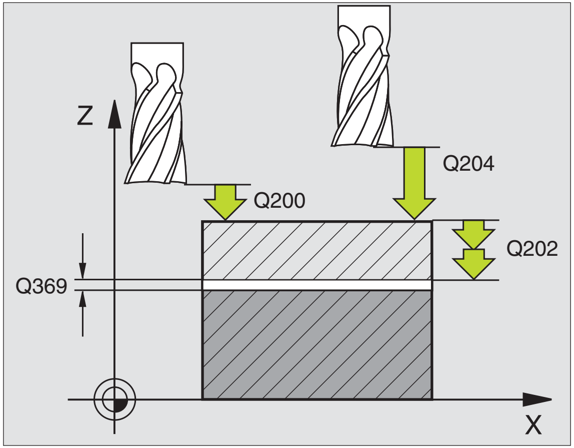


FACE MILLING (Cycle 232)

> ⚠ 2. Enter the 2nd set-up clearance in Q204 so that no collision between tool and clamping devices can occur.

- CYCL DEF: Select Cycle **232 FACE MILLING**
  - Machining strategy: **Q389**
  - Starting point in 1st axis: **Q225**
  - Starting point in 2nd axis: **Q226**
  - Starting point in 3rd axis: **Q227**
  - End point in 3rd axis: **Q386**
  - 1. side length: **Q218**
  - 2. side length: **Q219**
  - Maximum plunging depth: **Q202**
  - Finishing allowance for floor: **Q369**
  - Max. path overlap factor: **Q370**
  - Feed rate for milling: **Q207**
  - Feed rate for finishing: **Q385**
  - Feed rate for pre-positioning: **Q253**
  - set-up clearance: **Q200**
  - Lateral set-up clearance: **Q357**
  - 2. set-up clearance: **Q204**





Coordinate Transformation Cycles

Overview

Cycles for coordinate transformation are used to shift, mirror, rotate (in the plane), tilt (out of the plane), reduce and enlarge contours.

Available cycles		
7	DATUM SHIFT	page 89
247	DATUM SETTING	page 90
8	MIRROR IMAGE	page 91
10	ROTATION	page 92
11	SCALING FACTOR	page 93
26	AXIS-SPECIFIC SCALING	page 94
19	WORKING PLANE (software option)	page 95

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.

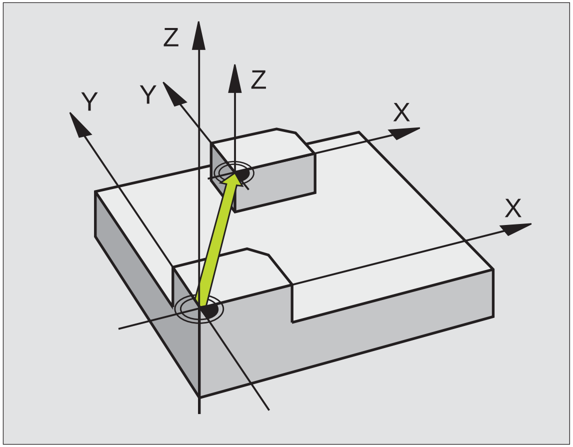
DATUM SHIFT (Cycle 7)

- 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.

```
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
16 CYCL DEF 7.3 Z-5
15 CYCL DEF 7.2 Y+40
```

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



DATUM SETTING (Cycle 247)

- CYCL DEF: Select Cycle **247 DATUM SETTING**
  - Number for datum: **Q339**. Enter the number of the new datum from the preset table.

```
13 CYCL DEF 247 DATUM SETTING
    Q339=4        ;DATUM NUMBER
```

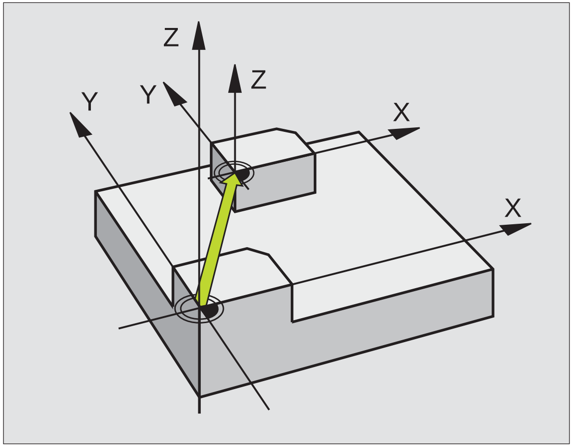
When activating a datum from the preset table, the TNC resets all coordinate transformations that were activated with the following cycles:

- Cycle 7, Datum Shift
- Cycle 8, Mirroring
- Cycle 10, Rotation
- Cycle 11, Scaling
- Cycle 26, Axis-Specific Scaling

However, the coordinate transformation from Cycle 19, Tilted Working Plane, remains active.

If you activate preset number 0 (line 0), then you activate the datum that you last set by hand in a manual operating mode.

Cycle 247 is not functional in Test Run mode.



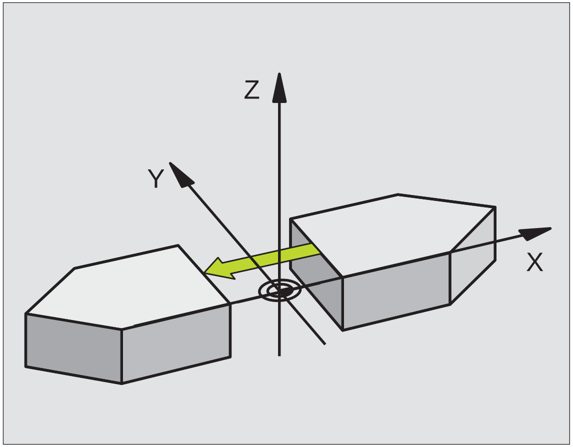
MIRROR IMAGE (Cycle 8)

- CYCL DEF: Select Cycle **8 MIRROR IMAGE**
  - Enter the mirrored axis: **X** or **Y** or **X** and **Y**

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

```
15 CALL LBL1
16 CYCL DEF 7.0 NULLPUNKT
17 CYCL DEF 7.1 X+60
18 CYCL DEF 7.2 Y+40
19 CYCL DEF 8.0 SPIEGELN
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).



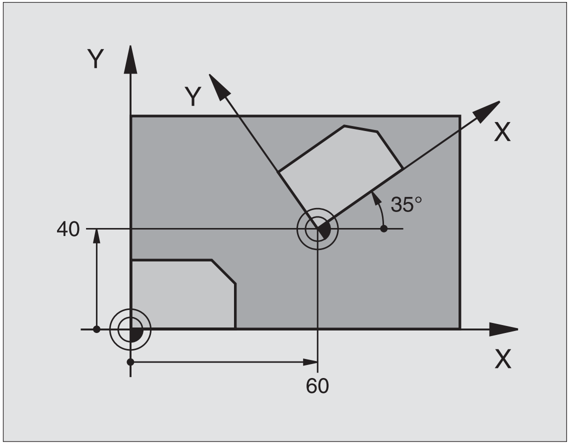
ROTATION (Cycle 10)

- CYCL DEF: Zyklus **10 DREHUNG** wählen
  - Enter the rotation angle:
    Input range: –360° to +360°
    Reference axis for the rotation angle

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

To cancel a rotation: Re-enter the cycle definition with the rotation angle 0.

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



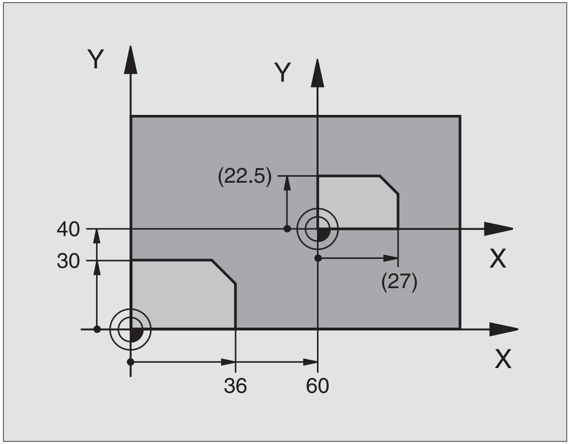
SCALING FACTOR (Cycle 11)

- CYCL DEF: Zyklus **11 MASSFAKTOR** wählen
  - Enter the scaling factor (SCL):
    Input range: 0.000 001 to 99.999 999
    Reduction... SCL<1
    Enlargement... SCL>1

To cancel the scaling: Re-enter the cycle definition with **SCL1.**

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

SCALING FACTOR can be effective in the working plane only or in all three main axes (depending on MP7410).



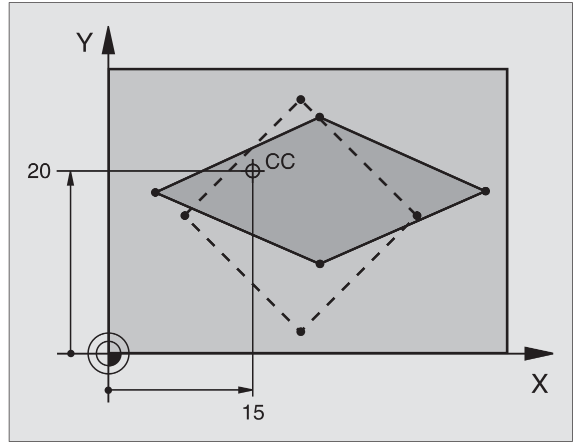
AXIS-SPECIFIC SCALING FACTOR (Cycle 26)

- CYCL DEF: Select Cycle **26 AXIS-SPECIFIC SCALING**
  - Axis and scaling factor: Enter the coordinate axes as well as the factors involved in enlarging or reducing.
  - Center coordinates: Enter the center of the enlargement or reduction.

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

Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

```
25 CALL LBL1
26 CYCL DEF 26.0 MASSFAKTOR ACHSSP.
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL1
```



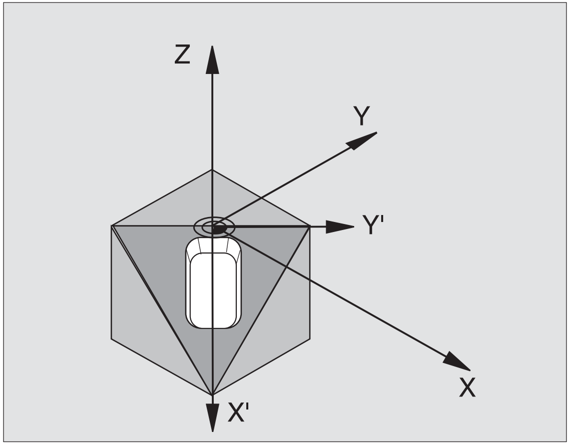
WORKING PLANE (Cycle 19, Software Option 1)

The TNC and the machine tool must be specially prepared by the machine tool builder in order to tilt the WORKING PLANE.

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

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

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




```
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 BEARBEITUNGSEBENE
8 CYCL DEF 19.1 B+10 C+90 F1000 ABST 50
```

Special Cycles

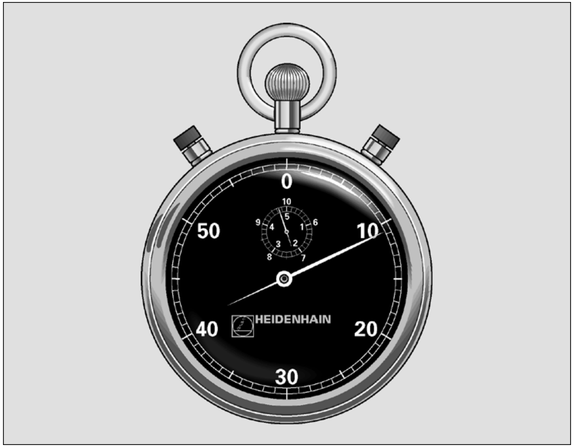
Overview

DWELL TIME (Cycle 9)

This causes the execution of the next block within a running program to be delayed by the programmed 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
```

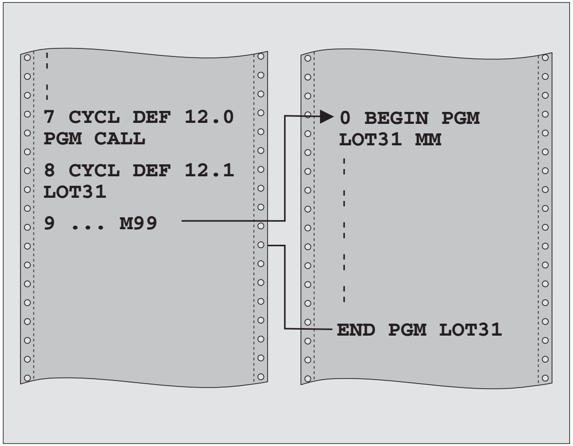


PGM CALL (Cycle 12)

- CYCL DEF: Select Cycle **12 PGM CALL**
  - Enter the name of the program to be called

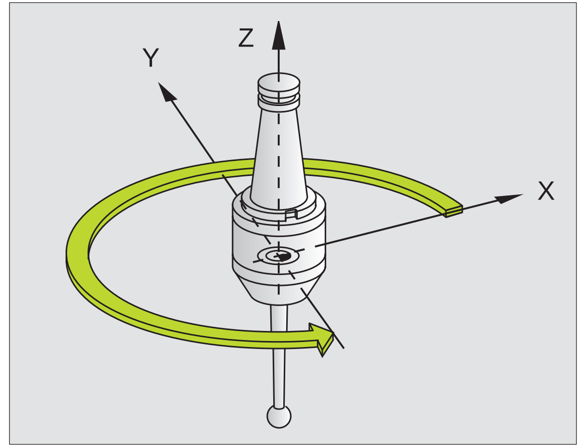
Cycle **12 PGM CALL** must be called to become active.

```
7 CYCL DEF 12.0 PGM CALL
8 CYCL DEF 12.1 LOT31
9 L X+37.5 Y-12 R0 FMAX M99
```



ORIENTED SPINDLE STOP (Cycle 13)

The TNC and the machine tool must be specially prepared by the machine tool builder for the ORIENTED SPINDLE STOP.



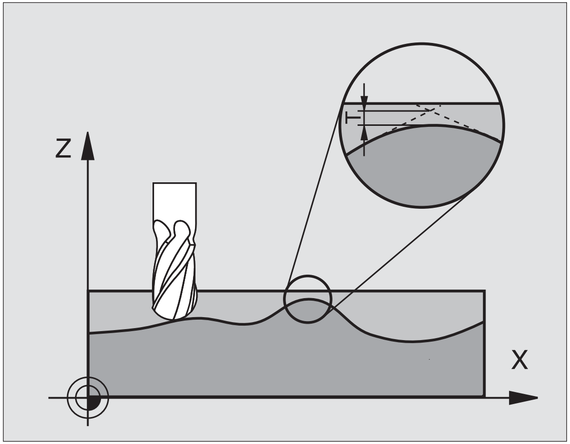
- CYCL DEF: Select Cycle **13 ORIENTED SPINDLE STOP**
  - Enter the orientation angle referenced to the angle reference axis of the working plane:
    Input range: 0° to 360°
    Input resolution: 0.1°
- Call the cycle with M19 or M20.

```
12 CYCL DEF 13.0 ORIENTATION
13 CYCL DEF 13.1 ANGLE 90
```

TOLERANCE (Cycle 32)

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

Cycle 32 TOLERANCE is effective immediately upon definition.



The TNC automatically smoothes the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the **fastest possible** speed without short pauses for computing time.

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

- CYCL DEF: Select Cycle **32 TOLERANCE**
  - Tolerance T: Permissible contour deviation in mm
  - Finishing/Roughing: (software option)
    Select the filter setting
    0: Milling with increased contour accuracy
    1: Milling at increased feed rate
  - Tolerance for rotary axes: (software option)
    Permissible position error of rotary axes in degrees with active M128.

The PLANE Function (Software Option 1)

Overview

The TNC and the machine tool must be specially prepared by the machine tool builder for tilting with the **PLANE** function.

The **PLANE** function is a powerful function for defining tilted working planes in various manners.

All **PLANE** functions available on the TNC describe the desired working plane independently of the rotary axes actually present on your machine. The following possibilities are available:

Available plane definitions	
Space-angle definition	page 101
Projection angle definition	page 102
Euler angle definition	page 103
Vector definition	page 104
Points definition	page 105
Incremental space angle	page 106
Reset the plane definition	page 107

Space Angle Definition (PLANE SPATIAL)

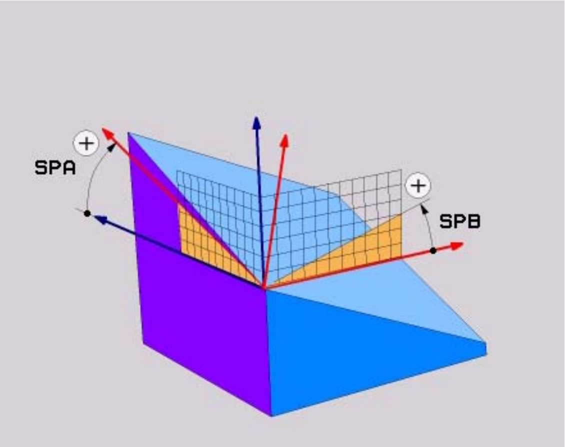
- Press SPECIAL TNC FUNCTIONS.
- Press TILT MACHINING PLANE, and then **PLANE SPATIAL**
  - **Spatial angle A?:** Rotational angle **SPA** around the fixed machine axis X (see figure at top right).
  - **Space angle B?:** Rotational angle **SPB** around the fixed machine axis Y (see figure at top right).
  - **Space angle C?:** Rotational angle **SPC** around the fixed machine axis Z (see figure at lower right).
  - Continue with the positioning properties (see "Automatic Positioning (MOVE/STAY/TURN)" on page 108).

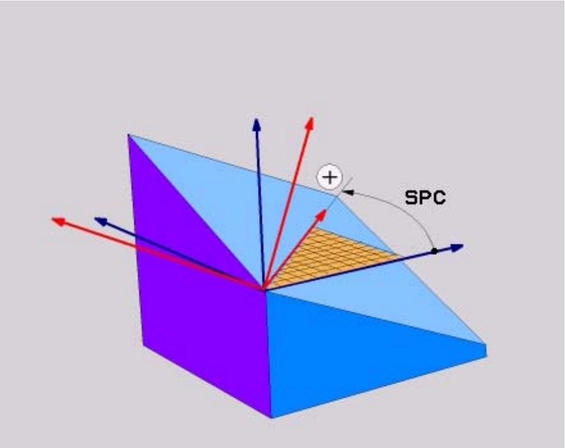
```
5 PLANE SPATIAL SPA+27 SPB+0 SPC+45 MOVE SETUP10 F500 SEQ-
```

Before programming, note the following:

You must always define the three space angles **SPA, SPB,** and **SPC,** even if one of them = 0.

The sequence of the rotations described above is independent of the active tool axis.

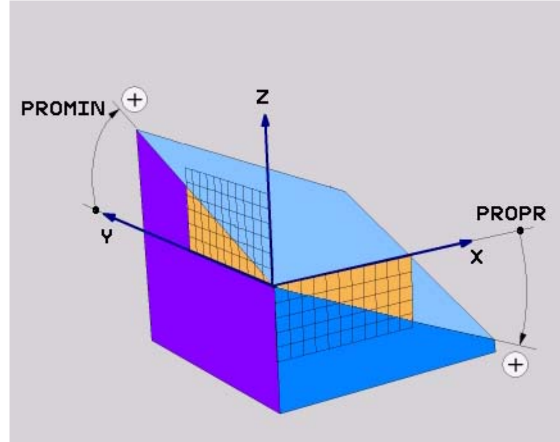




Projection Angle Definition (PLANE PROJECTED)

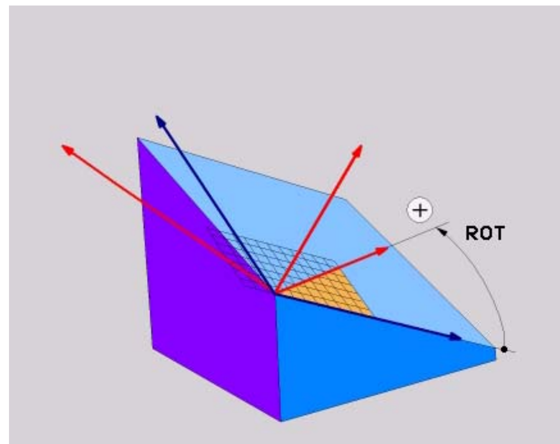
- Press SPECIAL TNC FUNCTIONS.
- Press TILT MACHINING PLANE, and then **PLANE PROJECTED**
  - **Proj. angle 1st coordinate plane?:** Projected angle of the tilted machining plane in the 1st coordinate plane of the fixed machine coordinate system (see figure at top right)
  - **Proj. angle 2nd coordinate plane?:** Projected angle in the 2nd coordinate plane of the fixed machine coordinate system (see figure at top right)
  - **ROT angle of the tilted plane?:** Rotation of the tilted coordinate system around the tilted tool axis (corresponds to a rotation with Cycle 10 ROTATION, see figure at lower right)
  - Continue with the positioning properties (see “Automatic Positioning (MOVE/STAY/TURN)” on page 108).

```
5 PLANE PROJECTED PROPR+24 PROMIN+24 PROROT+30 MOVE SETUP10
 F500
```



Before programming, note the following:

You can only use projection angles if a rectangular cuboid is to be machined. Otherwise distortions could occur on the workpiece.



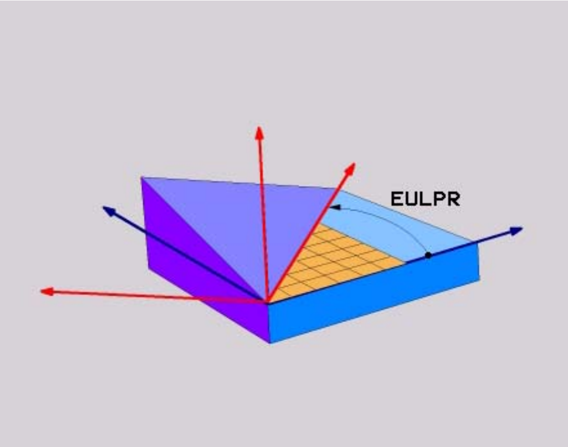
Euler Angles Definition (PLANE EULER)

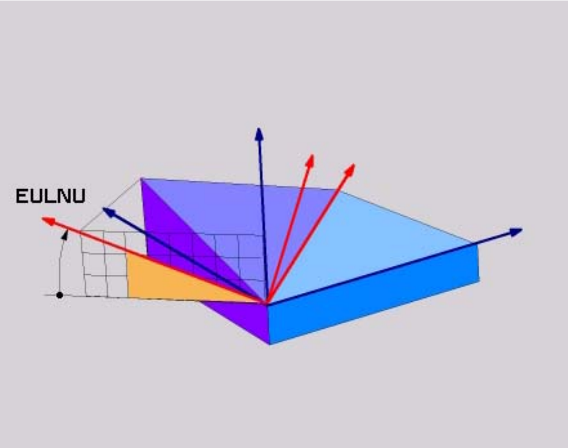
- Press SPECIAL TNC FUNCTIONS.
- Press TILT MACHINING PLANE, and then **PLANE EULER**
  - **Rot. angle main coordinate plane?:** Rotary angle **EULPR** around the Z axis (see figure at top right)
  - **Tilting angle tool axis?:** Tilting angle **EULNUT** of the coordinate system around the X axis shifted by the precession angle (see figure at lower right)
  - **ROT angle of the tilted plane?**: Rotation **EULROT** of the tilted coordinate system around the tilted Z axis (corresponds to a rotation with Cycle 10 ROTATION). Use the rotation angle to simply define the direction of the X axis in the tilted machining plane
  - Continue with the positioning properties (see “Automatic Positioning (MOVE/STAY/TURN)” on page 108).

```
5 PLANE EULER EULPR45 EULNU20 EULROT22 MOVE SETUP10 F500
```

Before programming, note the following:

The sequence of the rotations is independent of the active tool axis.





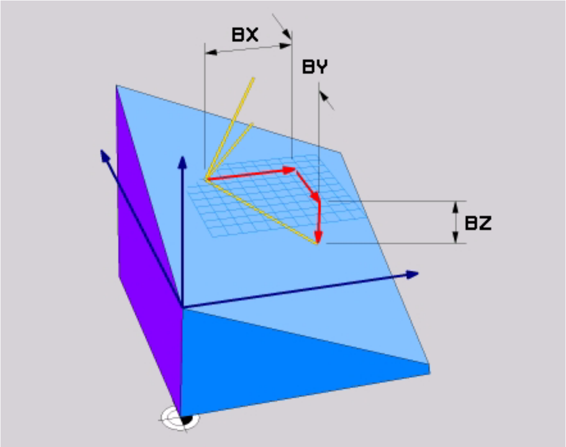
Vector Definition (PLANE VECTOR)

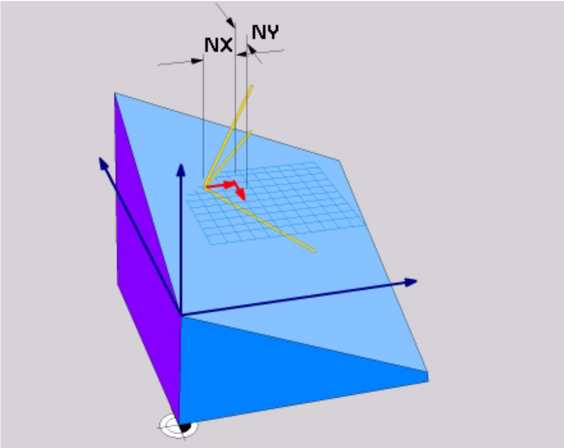
- Press SPECIAL TNC FUNCTIONS.
- Press TILT MACHINING PLANE, and then **PLANE VECTOR**
  - **X component of base vector?:** X component **BX** of the base vector B (see figure at top right)
  - **Y component of base vector?:** Y component **BY** of the base vector B (see figure at top right)
  - **Z component of base vector?:** Z component **BZ** of the base vector B (see figure at top right)
  - **X component of normal vector?:** X component **NX** of the normal vector N (see figure at lower right)
  - **Y component of normal vector?:** Y component **NY** of the normal vector N (see figure at lower right)
  - **Z component of normal vector?:** Z component **NZ** of the normal vector N
  - Continue with the positioning properties (see "Automatic Positioning (MOVE/STAY/TURN)" on page 108).

```
5 PLANE VECTOR BX0.8 BY-0.4 BZ-
0.4472 NX0.2 NY0.2 NZ0.9592 MOVE SETUP10 F500
```

Before programming, note the following:

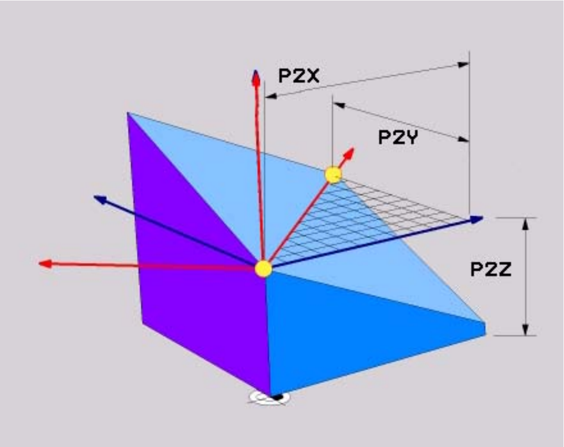
The TNC calculates standardized vectors from the values you enter.



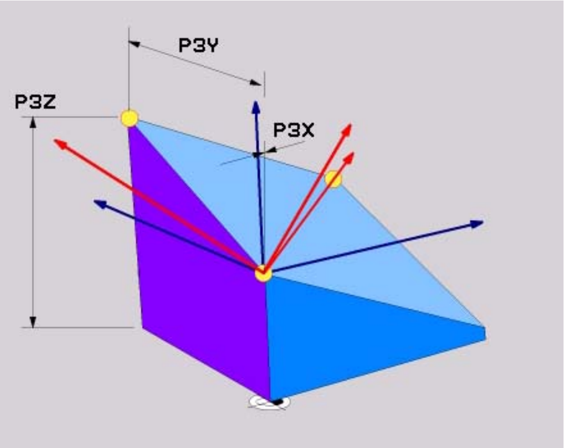


Points Definition (PLANE POINTS)

- Press SPECIAL TNC FUNCTIONS.
- Press TILT WORKING PLANE, and then **PLANE POINTS**
  - **X coordinate of 1st plane point?:** X coordinate **P1X**
  - **Y coordinate of 1st plane point?:** Y coordinate **P1Y**
  - **Z coordinate of 1st plane point?:** Z coordinate **P1Z**
  - **X coordinate of 2nd plane point?:** X coordinate **P2X**
  - **Y coordinate of 2nd plane point?:** Y coordinate **P2Y**
  - **Z coordinate of 2nd plane point?:** Z coordinate **P2Z**
  - **X coordinate of 3rd plane point?:** X coordinate **P3X**
  - **Y coordinate of 3rd plane point?:** Y coordinate **P3Y**
  - **Z coordinate of 3rd plane point?:** Z coordinate **P3Z**
  - Continue with the positioning properties (see "Automatic Positioning (MOVE/STAY/TURN)" on page 108).




```
5 POINTS P1X+0 P1Y+0 P1Z+20 P2X+30 P2Y+31 P2Z+20
P3X+0 P3Y+41 P3Z+32.5 MOVE SETUP10 F500
```



Before programming, note the following:

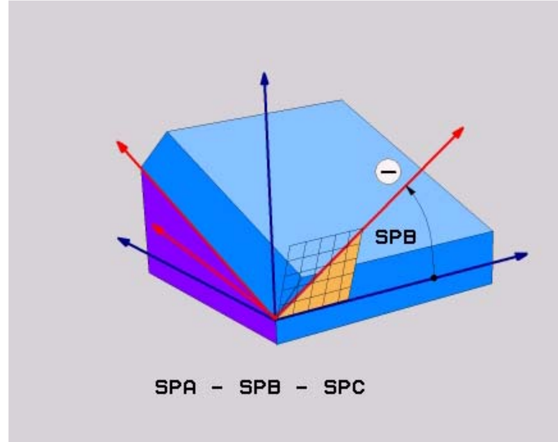
The connection from Point 1 to Point 2 determines the direction of the tilted principal axis (X for tool axis Z).

The three points define the slope of the plane. The position of the active datum is not changed by the TNC.

Incremental Space Angle (PLANE RELATIVE)

- Press SPECIAL TNC FUNCTIONS.
- Press TILT MACHINING PLANE, and then **PLANE RELATIVE**
  - **Incremental angle?:** Spatial angle about which the active machining plane is to be rotated additionally (see figure at right). Use a soft key to select the axis to be rotated about.
  - Continue with the positioning properties (see "Automatic Positioning (MOVE/STAY/TURN)" on page 108).

```
5 PLANE RELATIVE SPB-45 MOVE SETUP10 F500 SEQ-
```



Before programming, note the following:

The defined angle always applies to the active machining plane, no matter which function you used to activate it.

You can program any number of **PLANE RELATIVE** functions in a row.

If you want to return to the machining plane that was active before the **PLANE RELATIVE** function, define the **PLANE RELATIVE** function again with the same angle but with the opposite algebraic sign.

If you use the **PLANE RELATIVE** function on an untilted machining plane, then you simply rotate the untilted plane about the space angle defined in the **PLANE** function.

Reset Plane Definition (PLANE RESET)

- Press SPECIAL TNC FUNCTIONS.
- Press TILT MACHINING PLANE, and then **PLANE RESET.**
  - Continue with the positioning properties (see "Automatic Positioning (MOVE/STAY/TURN)" on page 108).

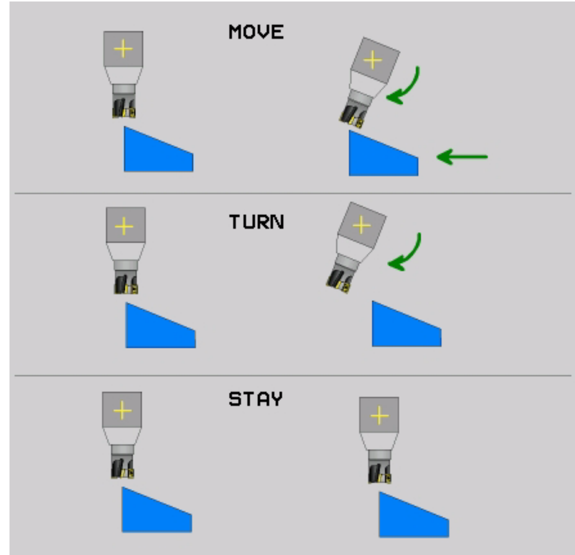
```
5 PLANE RESET MOVE SETUP10 F500 SEQ-
```

Before programming, note the following:

The **PLANE RESET** function resets the current **PLANE** function—or an active Cycle 19—completely (angles = 0 and function is inactive). It does not need to be defined more than once.

Automatic Positioning (MOVE/STAY/TURN)

After you have entered all parameters for the plane definition, you must specify how the rotary axes will be positioned to the calculated axis values:



MOVE

- The PLANE function is to automatically position the rotary axes to the calculated position values. The position of the tool relative to the workpiece is to remain the same. The TNC carries out a compensating motion in the linear axes.

TURN

- The PLANE function is to automatically position the rotary axes to the calculated position values, but only the rotary axes are positioned. The TNC does **not** carry out a compensating motion in the linear axes.

STAY

- You will position the rotary axes later in a separate positioning block.

If you select either the **MOVE** or the **TURN** option (**PLANE** function is to position the axes automatically), then the following two parameters must still be defined:

- **Dist. tool tip – center of rot.** (incremental): The TNC tilts the tool (or table) relative to the tool tip. The **SETUP** parameter shifts the center of rotation of the positioning movement relative to the current position of the tool tip.
- **Feed rate ? F=**: Contour speed at which the tool should be positioned.

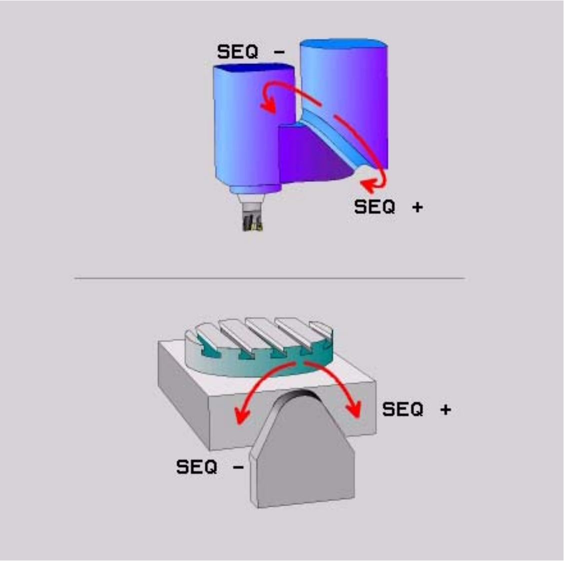
Select a Possible Solution (SEQ +/–)

The position you define for the machining plane is used by the TNC to calculate the appropriate positioning of the rotary axes present on the machine. In general there are always two solution possibilities.

Use the **SEQ** switch to specify which possibility the TNC should use:

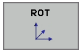
- **SEQ+** positions the master axis so that it assumes a positive angle. The master axis is the 2nd rotary axis from the table, or the 1st axis from the tool (depending on the machine configuration (see figure at top right)).
- **SEQ–** positions the master axis so that it assumes a negative angle.

If the solution you chose with **SEQ** is not within the machine's range of traverse, the TNC displays the **Entered angle not permitted** error message.

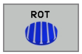


Selection of the type of transformation

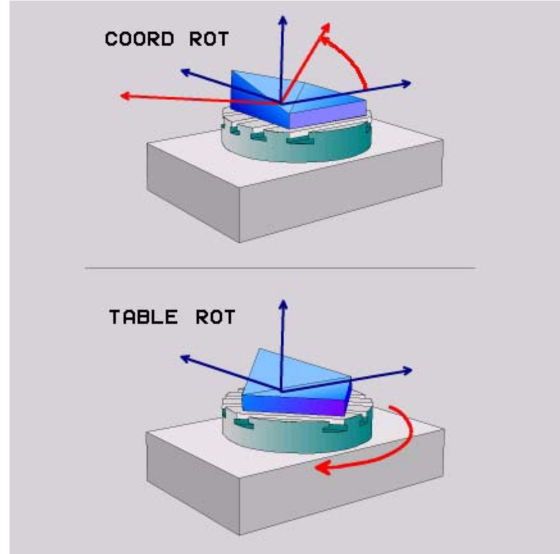
On machines with C-rotary tables, a function is available for specifying the type of transformation:



- **COORD ROT** specifies that the PLANE function should only rotate the coordinate system to the defined tilting angle. The rotary table is not moved; the compensation is purely mathematical.



- **TABLE ROT** specifies that the PLANE function should position the rotary table to the defined tilting angle. Compensation results from rotating the workpiece.



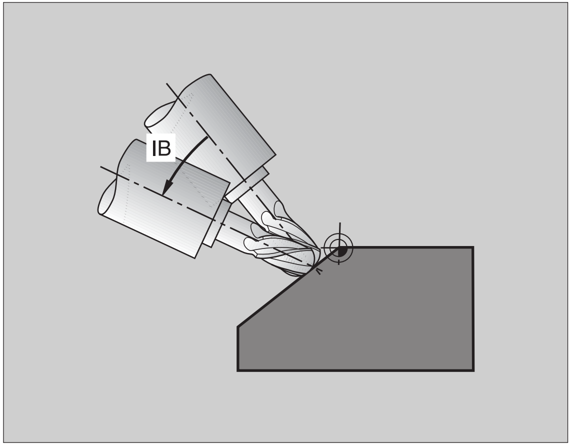
Inclined-Tool Machining in the Tilted Plane

In combination with M128 and the new **PLANE** functions, **inclined-tool machining** in a tilted machining plane is now possible. Two possibilities are available for definition:

- Inclined-tool machining via incremental traverse of a rotary axis
- Inclined-tool machining via normal vectors

Inclined-tool machining in a tilted machining plane only functions with spherical cutters.

With 45° swivel heads and tilting tables you can also define the incline angle as a space angle. Use the **FUNCTION TCPM** for this.



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

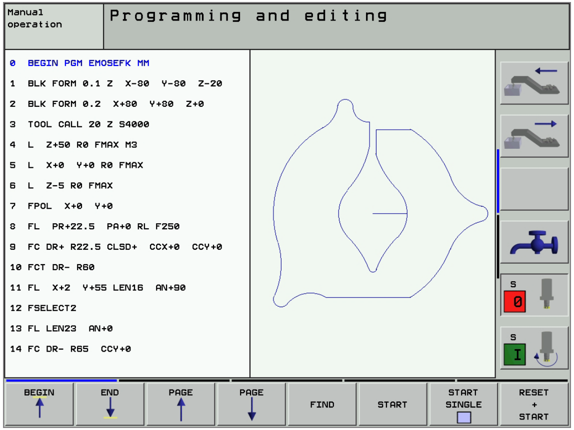
Select the PROGRAM+GRAPHICS layout.

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

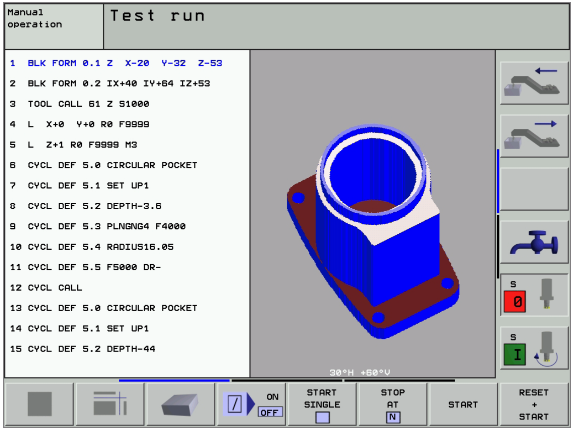
RESET + START ▶ Manually start graphic generation

START SINGLE ▶ Generate interactive graphics blockwise



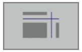
Test Graphics and Program Run Graphics

Select the GRAPHICS or PROGRAM+GRAPHICS layout.

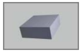


In the test run and program run modes the TNC can graphically simulate the machining process. The following display types are available via soft key:

- Plan view



- Projection in 3 planes



- 3-D view

Status displays

Select the PROGRAM+STATUS or POSITION+STATUS layout.

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

- Tool position
- Feed rate
- Active miscellaneous functions

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

STATUS PGM
- Program information

STATUS POS.
- Tool positions

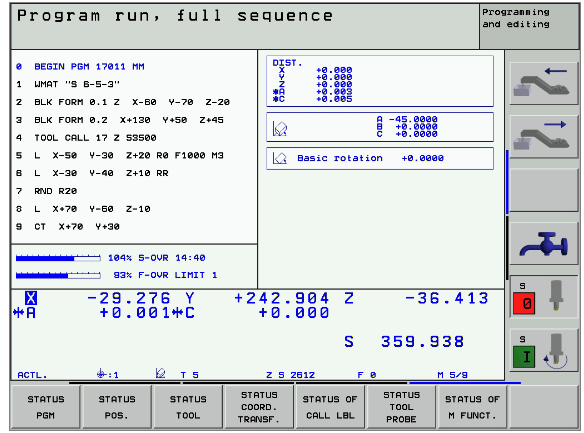
STATUS TOOL
- Tool data

STATUS COORD. TRANSF.
- Coordinate transformations

STATUS OF CALL LBL
- Subprograms, program section repeats

STATUS TOOL PROBE
- Tool measurement

STATUS OF M FUNCT.
- Active miscellaneous functions M



ISO Programming

Programming tool movements with Cartesian coordinates	
G00	Linear motion at rapid traverse
G01	Straight movement
G02	Circular motion, clockwise
G03	Circular motion, counterclockwise
G05	Circular motion without directional data
G06	Circular motion with tangential contour connection
G07*	Paraxial positioning block

Programming tool movements with Polar coordinates	
G10	Linear motion at rapid traverse
G11	Straight movement
G12	Circular motion, clockwise
G13	Circular motion, counterclockwise
G15	Circular motion without directional data
G16	Circular motion with tangential contour connection

*) Non-modal function

Drilling cycles	
G200	Drilling
G201	Reaming
G202	Boring
G203	Universal drilling
G204	Back boring
G205	Universal pecking
G208	Bore milling
G206	Tapping NEW
G207	Rigid tapping (controlled spindle) NEW
G209	Tapping with chip breaking
G262	Thread milling
G263	Thread milling/countersinking
G264	Thread drilling/milling
G265	Helical thread drilling/milling
G267	Outside thread milling

Pockets, Studs and Slots	
G251	Rectangular pocket, complete
G252	Circular pocket, complete
G253	Slot, complete
G254	Circular slot, complete
G212	Pocket finishing
G213	Stud finishing
G214	Circular pocket finishing
G215	Circular stud finishing
G210	Slot with reciprocating plunge
G211	Circular slot

Point Patterns	
G220	Circular point pattern
G221	Linear point pattern

*) Non-modal function

SL Cycles Group II	
G37	Define contour subprogram
G120	Contour data
G121	Pilot drilling
G122	Rough-out
G123	Floor finishing
G124	Side finishing
G125	Contour train
G127	Cylinder surface (software option)
G128	Cylinder surface slot milling (software option)
G129	Cylinder surface ridge milling (software option)
G139	Cylinder surface contour milling (software option)

Multipass milling	
G60	3-D data
G230	Multipass milling
G231	Ruled surface
G232	Face milling

Coordinate Transformation Cycles

G53	Datum shift in datum table
G54	Enter datum shift directly
G247	Datum setting
G28	Mirror image of contours
G73	Rotate coordinate system
G72	Scaling factor: reduce or enlarge contours
G80	Working plane (software option)

Special Cycles

G04*	Dwell time
G36	Oriented spindle stop
G39	Designating a program as a cycle
G79*	Call the cycle
G62	Tolerance (software option)

Touch Probe Cycles

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

Touch Probe Cycles

G410*	Datum at center of rectangular pocket
G411*	Datum at center of rectangular stud
G412*	Datum at center of hole
G413*	Datum at center of circular stud
G414*	Datum at outside corner
G415*	Datum at inside corner
G416*	Datum at center of bolt hole circle
G417*	Datum in touch probe axis
G418*	Datum at center of 4 holes
G419*	Datum in single axis
G420*	Measure angle
G421*	Measure hole
G422*	Measure cylindrical stud
G423*	Measure rectangular pocket
G424*	Measure rectangular stud
G425*	Measure slot width
G426*	Measure ridge width
G427*	Measure any coordinate
G430*	Measure bolt hole circle
G431*	Measure plane
G440*	Thermal compensation
G480*	Calibrating the TT
G481*	Measure tool length
G482*	Measure tool radius
G483*	Measure tool length and radius

Define machining plane	
G17	Working plane: X/Y; tool axis: Z
G18	Working plane: Z/X; tool axis: Y
G19	Working plane: Y/Z; tool axis: X
G20	Fourth axis is tool axis

Chamfer, Rounding, Approach /Depart Contour	
G24*	Chamfer with length R
G25*	Corner rounding with radius R
G26*	Tangential contour approach on arc with radius R
G27*	Tangential contour departure on arc with radius R

Define the tool	
G99*	Tool definition in the program with length L and radius R

Tool radius compensation	
G40	No radius compensation
G41	Tool radius compensation, left of the contour
G42	Tool radius compensation, right of the contour
G43	Paraxial radius compensation: the path is lengthened
G44	Paraxial radius compensation: the path is shortened

*) Non-modal function

Dimensions	
G90	Absolute dimensions
G91	Incremental (chain) dimensions

Unit of Measure (at Beginning of Program)	
G70	Unit of measure: **Inch**
G71	Unit of measure: **mm**

Blank Form Definition for Graphics	
G30	Set the working plane, MIN point coordinates
G31	Dimensional data (with G90, G91), coordinates of the MAX point

Other G functions	
G29	Assume the last position as the pole
G38	Stop program run
G51*	Call next tool number (only with central tool file)
G98*	Set marker (label number)

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

Addresses	
%	Start of program
A	Swiveling axis around X
B	Swiveling 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 for positioning blocks
F	Dwell time in seconds with G04
F	Scaling factor with G72
G	G function (see list of G functions)
H	Polar coordinate angle
H	Rotation angle with G73
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Set marker (label number) with G98
L	Jump to a marker (label number)
L	Tool length with G99
M	Miscellaneous function
N	Block number
P	Cycle parameter with machining cycles
P	Value or Q parameter for Q-parameter definitions
Q	Variable Q parameter
R	Polar coordinate radius with G10/G11/G12/G13/ G15/G16
R	Circular radius with G02/G03/G05
R	Rounding 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/Spindle stop/Coolant off
M01	Optional program-run interruption
M02	Stop program run/Stop spindle/Coolant off/Jump back to block1/Clear status display
M03	Spindle ON clockwise
M04	Spindle ON counterclockwise
M05	Spindle stop
M06	Tool change/Stop program run (depending on MPs)/Spindle stop
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 MPs)
M90	Constant contouring speed at corners (effective only in lag mode)
M91	Within the positioning block: Coordinates are referenced to machine datum
M92	Within the positioning block: Coordinates are referenced to position defined by the machine manufacturer
M93	Reserved
M94	Reduce display of rotary axis to value under 360°
M95	Reserved
M96	Reserved
M97	Machine small contour steps
M98	Suspend tool path compensation
M99	Cycle call, non-modal
M101	Automatic tool change after tool lifetime expires
M102	Reset M101
M103	Reduce plunging feed rate to factor F
M104	Reactivate the datum as last defined
M105	Machining with second k_V factor
M106	Machining with first k_V factor
M107	See User's Manual
M108	Reset M107
M109	Constant contouring speed of tool cutting edge on arcs (increase and decrease feed rate)

M110	Constant contouring speed of tool cutting edge on arcs (only feed-rate decrease)
M111	Reset M109/M110
M114	Automatic compensation of machine geometry when working with tilted axes (software option)
M115	Reset M114
M116	Feed rate for rotary axes in mm/min (software option)
M117	Reset M116
M118	Superimpose handwheel positioning during program run
M120	Pre-calculate radius-compensated position (LOOK AHEAD)
M124	Do not include points when executing non-compensated line blocks
M126	Shortest-path traverse of rotary axes
M127	Reset M126
M128	Maintain the position of the tool tip when positioning with tilted axes (TCPM)[1] (software option)
M129	Reset M126

[1] TCPM: Tool Center Point Management

M130	Moving to position in an untilted coordinate system with a tilted working plane
M134	Exact stop for positioning with rotary axes
M135	Reset M134
M136	Feed rate F in millimeters per spindle revolution
M137	Feed rate F in millimeters per minute
M138	Selection of tilted axes for M114, M128 and the Tilt Working Plane cycle
M140	Retraction from the contour in the tool-axis direction
M141	Suppress touch probe monitoring
M142	Delete modal program information
M143	Delete basic rotation
M144	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block (software option)
M145	Reset M144
M148	Automatically retract tool from the contour at an NC stop
M149	Reset M148
M200	Miscellaneous functions for laser cutting machines
.	
.	
.	
M204	See User's Manual

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 02
387 952-22 · SW10 · 9/2004 · pdf · Subject to change without notice