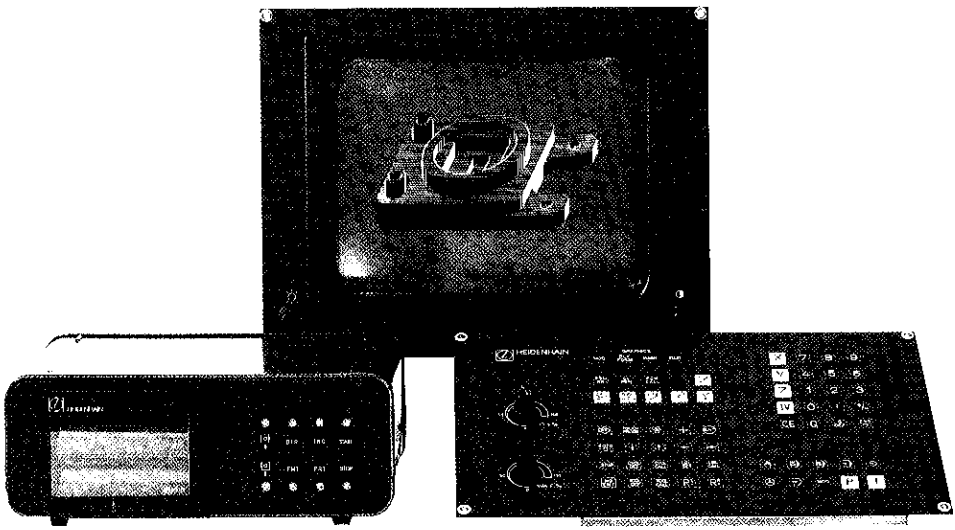
PILOT

TNC 355



August '89

HEIDENHAIN

Contents

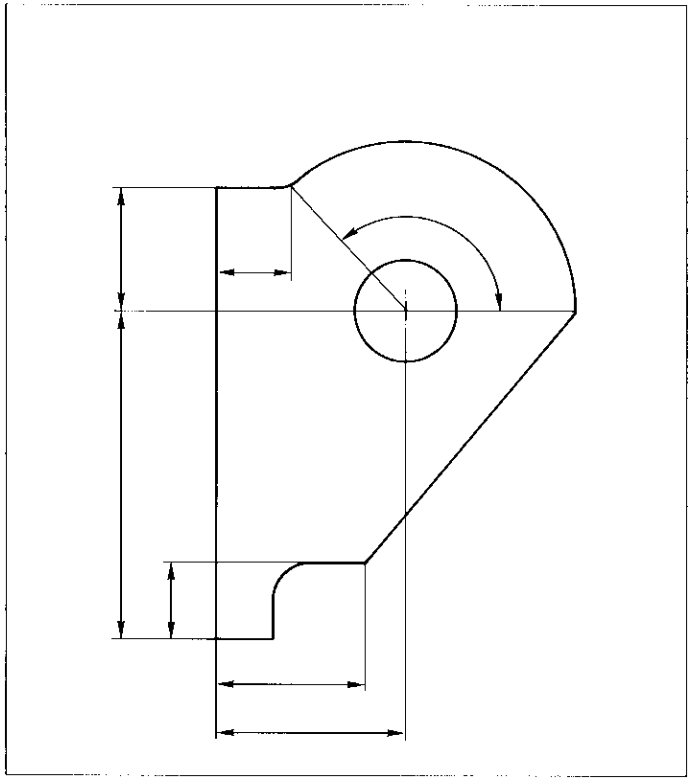
The PILOT is valid for all TNC 355 versions. Comprehensive information on programming possibilities can be found in the operator's manual.

HEIDENHAIN is constantly working on further developments of its TNC controls. It is therefore possible that details of certain control versions may deviate from the versions explained in this programming PILOT.

Introduction

The PILOT will help you with programming and trouble shooting based on the workpiece drawing.

Workpiece drawings are color-coded for clarity reasons. For explanations of symbols and colors please refer to the next page.



Key to symbols

Dialog initiation key

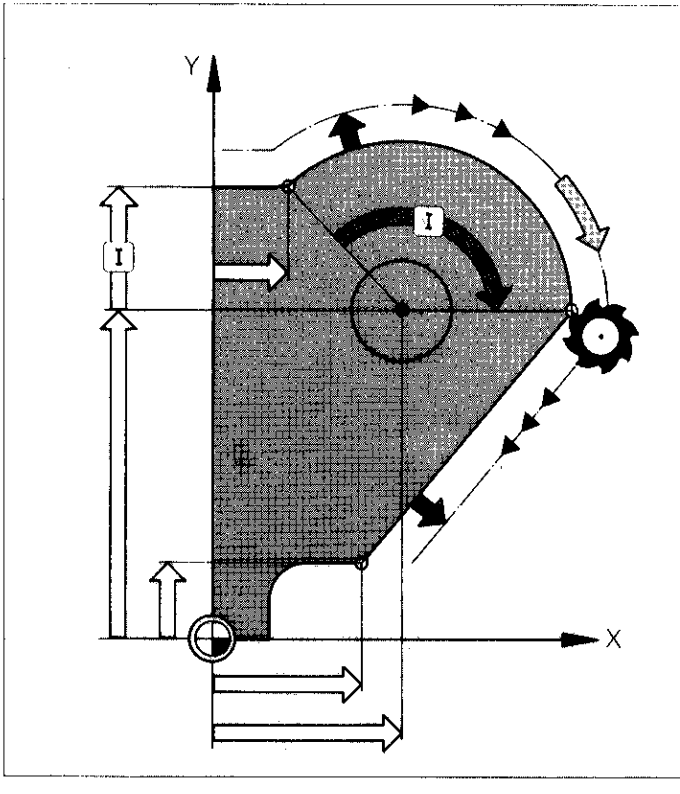
Program block on screen

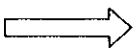
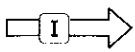
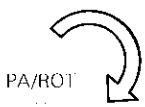
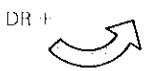
```
11  L  X+20,000      Y+10,000
                     RL  F180   M13
```

Notes

Tool position in the illustrations: Prior to program block execution, the tool is at the start position.

The color designation has the following meaning unless otherwise stated in the specific diagrams (e.g. explanation of cycles).

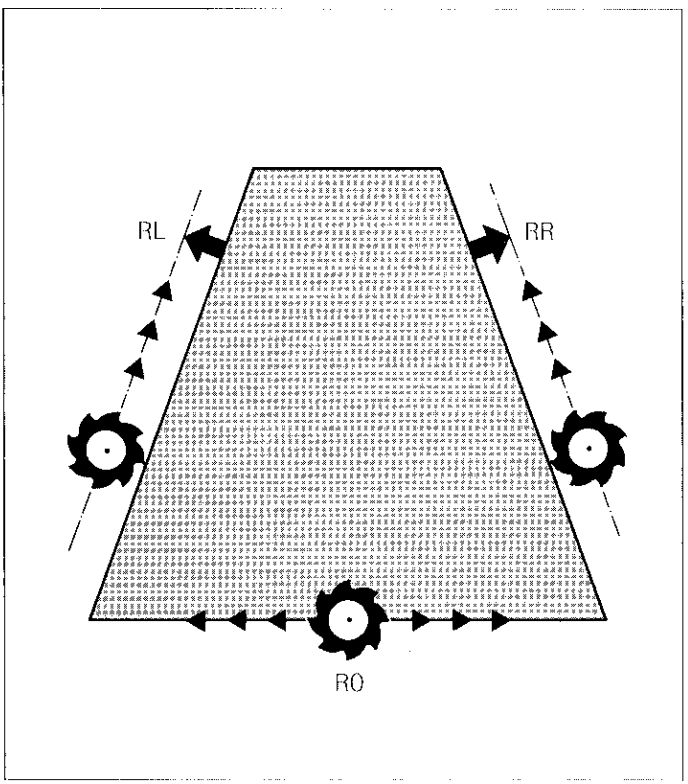


Dimensions	Absolute	
	Incremental	
	Positive angle (counter-clockwise)	
	Negative angle (clockwise)	
	Positive rotation	
	Negative rotation	
Tool path	Straight	
	Circular	
Radius compensation		

Path compensation

Radius compensation

Tool in milling direction		
to the **left** of contour:	RL	
to the **right** of contour:	RR	
on the contour:	R0	

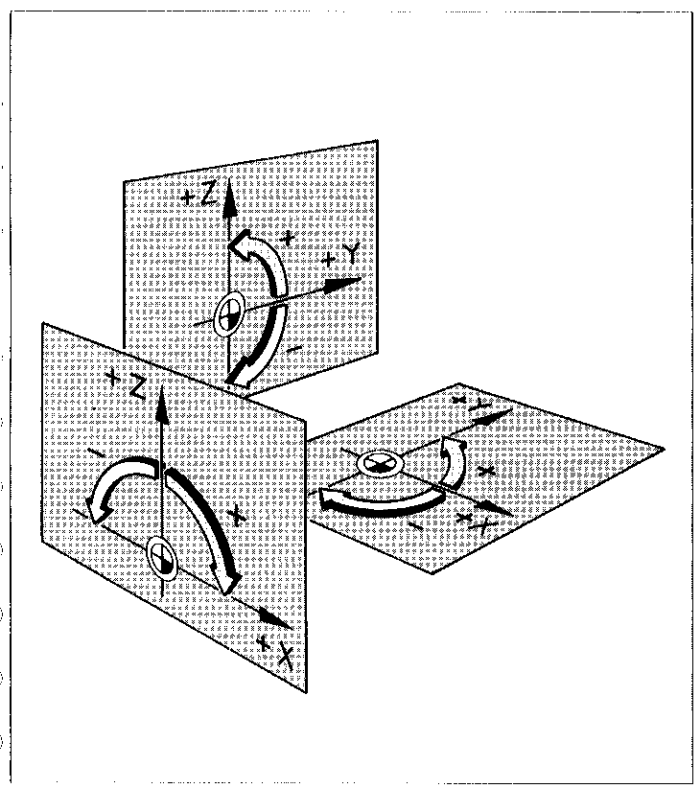


Reference axes

Working planes

The polar coordinates angle PA and rotation angle ROT are referenced to the reference axis:

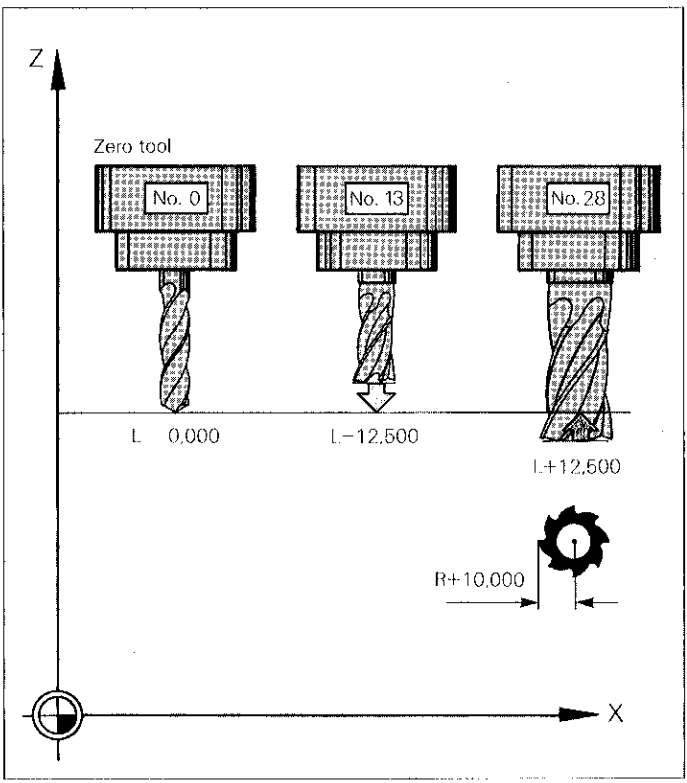
Working plane	Reference axis	+ 90° axis
X, Y	+ X	+ Y
Y, Z	+ Y	+ Z
Z, X	+ Z	+ X



TOOL DEF

Tool definition

```
15 TOOL DEF 28        L+12,500
                      R+10,000
```



Tool length compensation:

Compensation values are referenced to the "zero tool".

Tools longer than the zero tool:
+ sign

Tools shorter than the zero tool:
– sign

Tool call

TOOL CALL

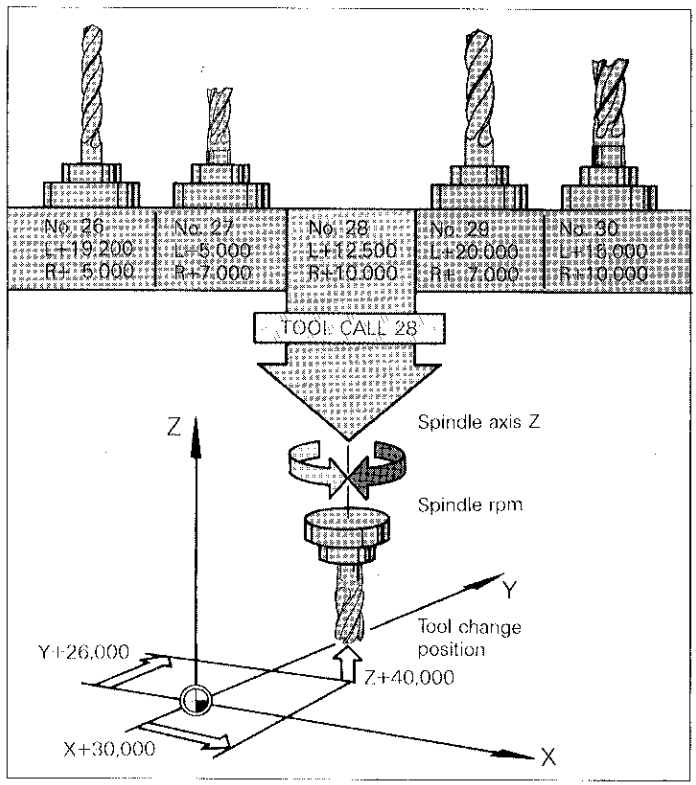
```
16 TOOL CALL 0          Z
             S
17 L  X+30,000      Y+26,000
      Z+40,000      R0 F       M
18 STOP
                                M05
19 TOOL CALL 28         Z
             S 125,000
```

Cancellation of current tool compensation values

Tool change position.

Program run STOP (for manual tool change)

Call-up of next tool with appropriate compensation values.



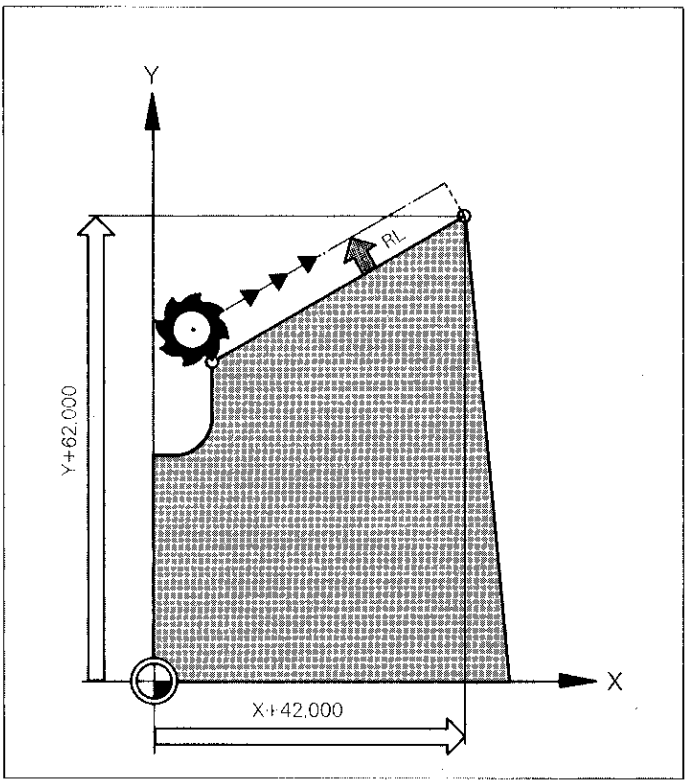
When traversing to the tool change position, take care that the tool does not collide with the workpiece!

The rotating direction of the spindle is determined by an auxiliary function: M03/M04 or M13/M14 (right/left)

Linear milling

Cartesian (right-angled) coordinates

```
49 L  X+42,000      Y+62,000
                    RL F180   M
```

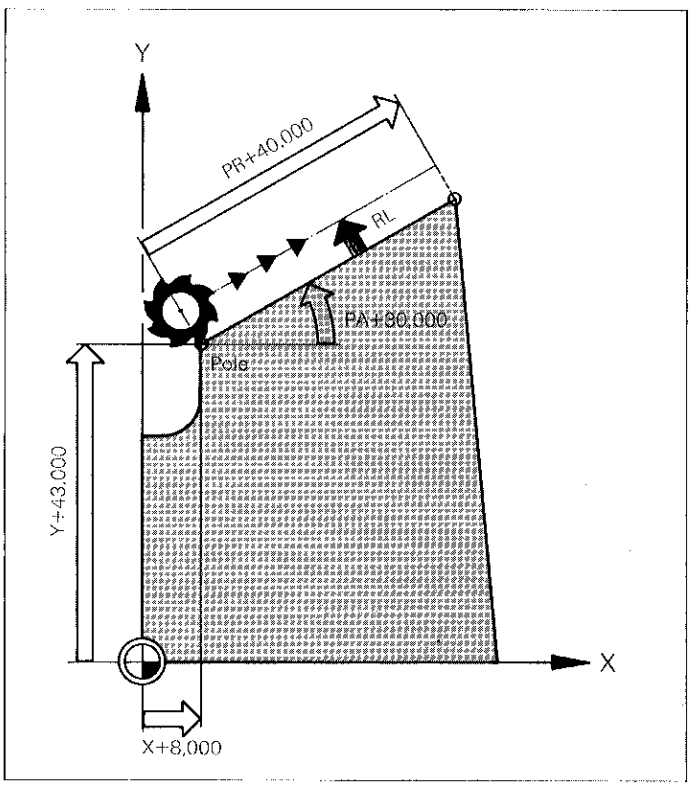


Linear milling

P

Polar coordinates

```
57 CC X+8,000      Y+43,000
58 LP  PR+40,000   PA+30,000
                   RL F     M
```



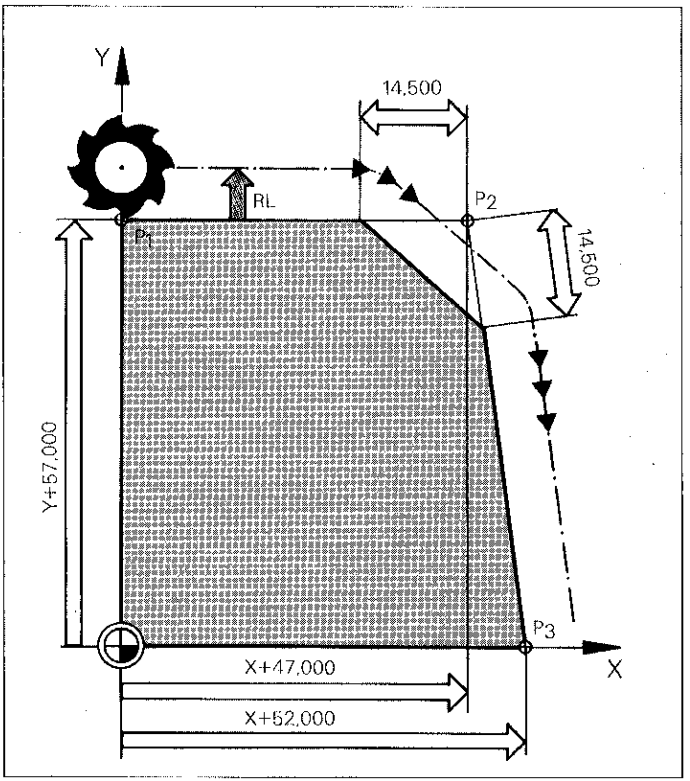
Before entry of polar coordinates, define pole via the CC-key.

The pole must be programmed in Cartesian coordinates only!

If the pole remains the same for several blocks, a new entry is not required.

Chamfer

Auxiliary position P_2	95	L	X+47,000	Y+57,000
				RL F180 M
Chamfer side length L	96	L	14,500	
Contour position P_3	97	L	X+52,000	Y+0,000
				R F M



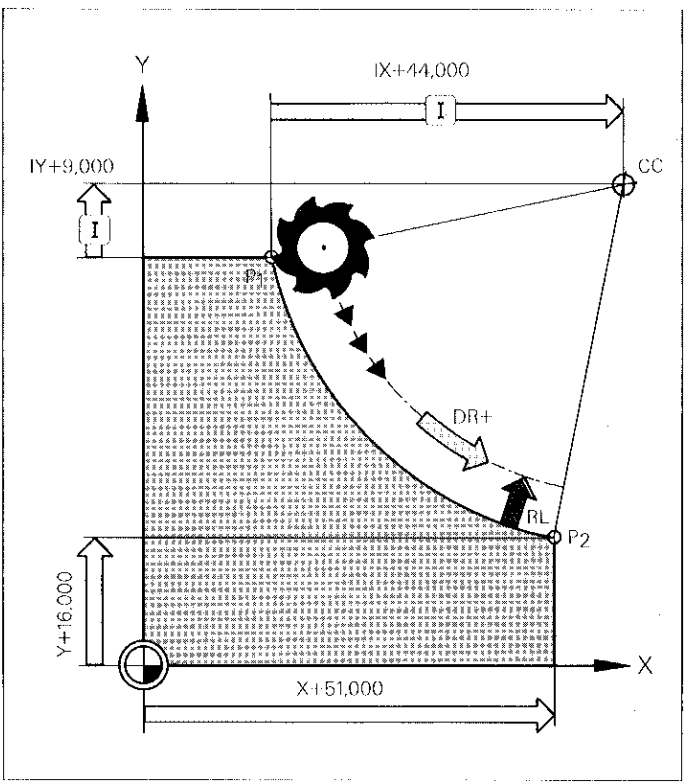
Notes

Circular interpolation

Specification via center and end position
End position in Cartesian (right-angled) coordinates

Circle center	5	CC	IX+44,000	IY+9,000	
Contour position P_2	6	C	X+51,000	Y+16,000	
			DR+	RL	F120 M

Contour position P_1 (X16/Y51) has been previously approached.



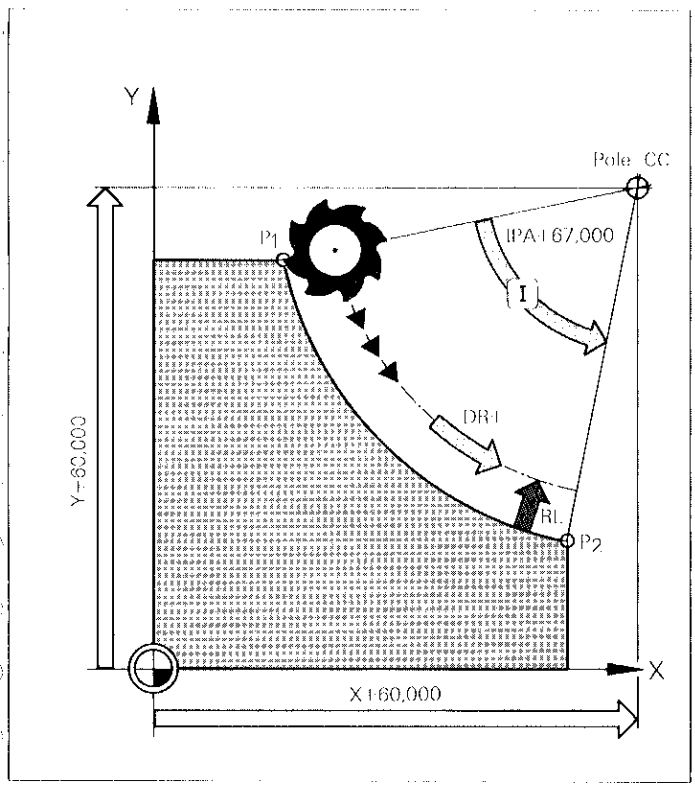
- CC in absolute dimension is referenced to the program zero point.
- CC in incremental dimension is referenced to the previous contour position.

Circular interpolation

Specification via center and end position
End position in polar coordinates

```
5  CC X+60,000      Y+60,000
6  CP IPA+67,000
                DR+ RL F120    M
```

Circle center/pole
Contour position P_2



Before entry of polar coordinates, define pole via the CC-key.

The pole must be programmed in Cartesian coordinates only!

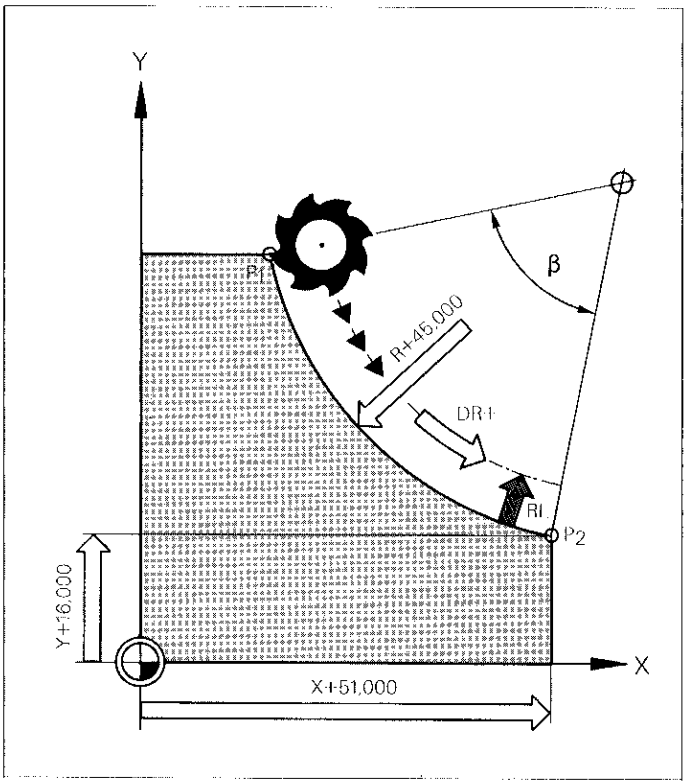
Determination of the arc end position only with polar coordinate angle PA.

Circular milling

Specification via radius and end position

Contour position P_2

```
6 CR X+51,000 Y+16,000
  R+45,000 DR+ RL F120 M
```

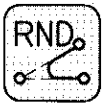


Circle radius Sign R:

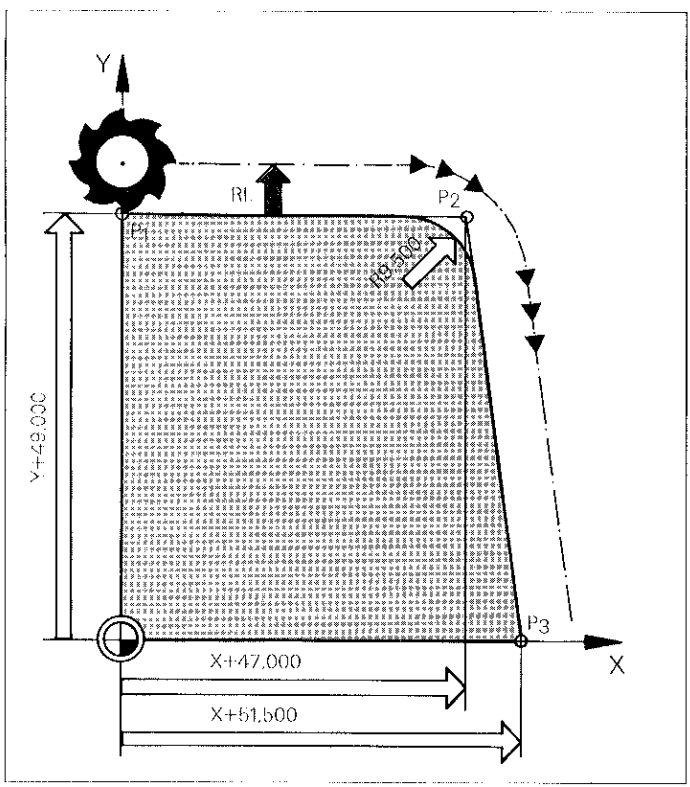
Center angle β less than 180°: R+

Center angle β greater than 180°: R–

Rounding of corners




```
39 L   X+47,000      Y+49,000                  Auxiliary position P2
                     RL F120    M
40 RND R9,500                                  Rounding off radius R
                       F80
41 L   X+51,500      Y+0,000                   Contour position P3
                     R  F120    M
```



Before and after an RND block a positioning block containing both coordinates of the machining plane must be programmed.

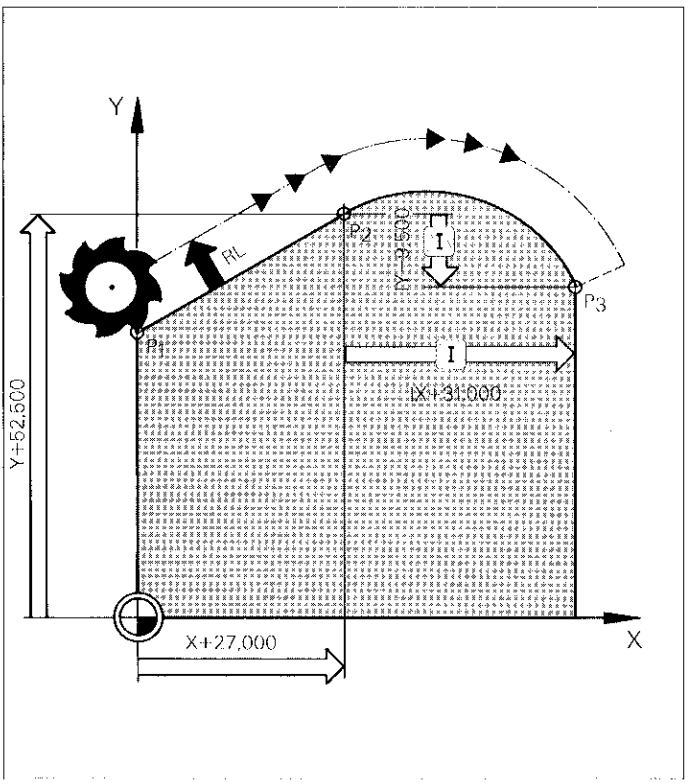
Circular milling

Tangential contour connection
Cartesian coordinates

End position P_2 of straight

End position P_3 of connecting arc

```
21  L   X+27,000     Y+52,500
                     RL F100    M
22  CT  IX+31,000    IY-9,500
                     RL F       M
```



A contour element (straight or arc) having coordinates within the same working plane must already be programmed before the connecting arc.

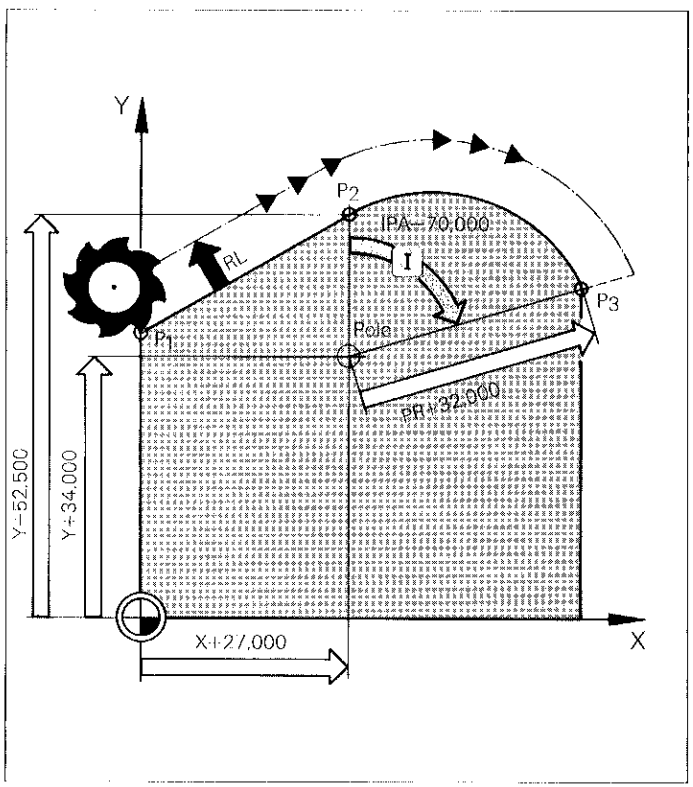
Circular milling

Tangential contour connection
Polar coordinates

```
45  L   X+27,000      Y+52,500
                      RL  F100    M
46  CC  X+27,000      Y+34,000
47  CIP PR+32,000     IPA-70,000
                      R   F       M
```

End position P_2 of straight

End position P_3 of connecting arc



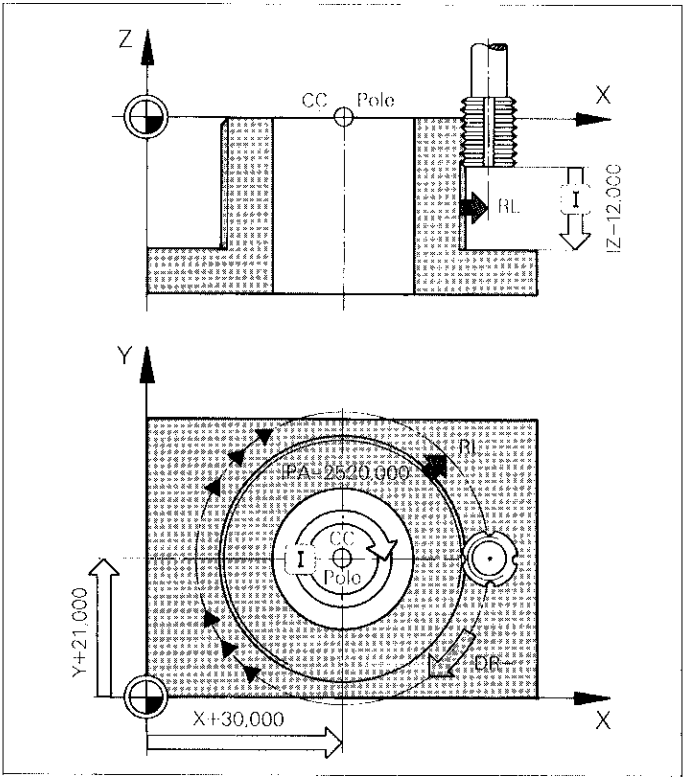
Before entry of polar coordinates, define pole via the CC-key.

The pole must be defined in Cartesian coordinates only.

A contour element (straight or arc) having coordinates within the same working plane must already be programmed before the connecting arc.

Helical interpolation

```
64 CC X+30,000      Y+21,000
65 CP IPA-2520,000 IZ-12,000
                DR- RL F80     M
```



Define pole CC beforehand and approach to starting position.

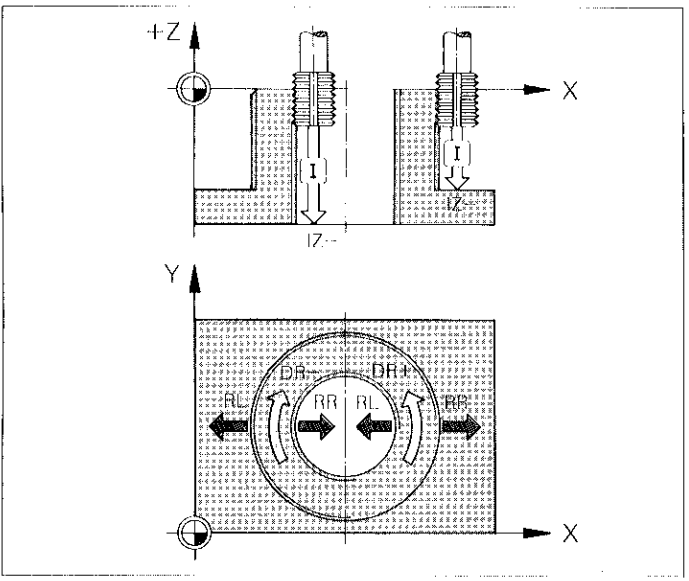
Specify angle PA in incremental only!

Polar co-ordinates angle PA is calculated as follows:

PA = 360 x IZ/P

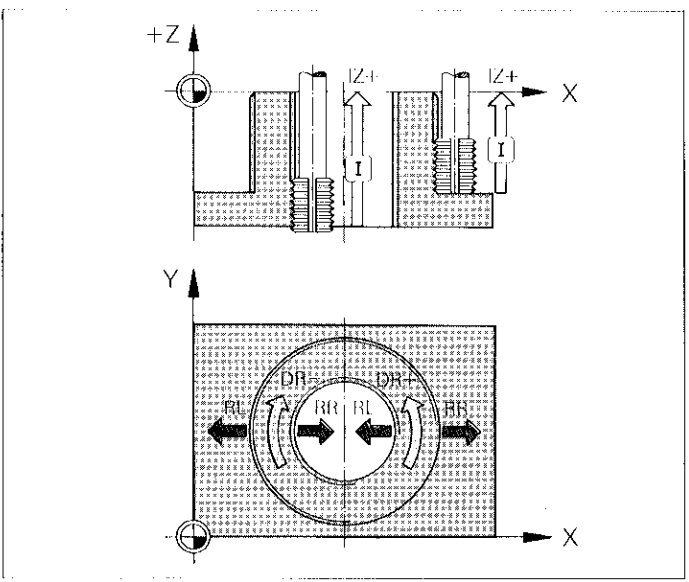
IZ: Downfeed distance
P: Pitch of thread

Helical interpolation



Right-hand thread
External: down-cut milling
Internal: up-cut milling

Left-hand thread
External: up-cut milling
Internal: down cut milling

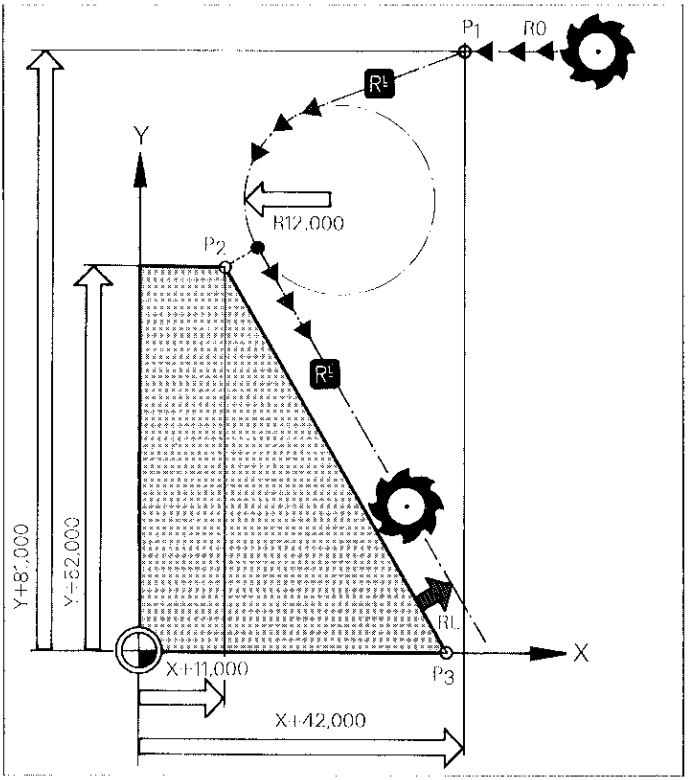


Right-hand thread
External: up-cut milling
Internal: down-cut milling

Left-hand thread
External: down-cut milling
Internal: up-cut milling

Contour approach on an arc

Starting position P_1 with R0	79	L	X+42,000	Y+81,000 R0 F15999	M
Contour position P_2 with RL (RR)	80	L	X+11,000	Y+52,000 RL F80	M
Rounding off radius	81	RND	R12,000	F	



Before and after an RND block a positioning block containing both coordinates of the machining plane must be programmed.

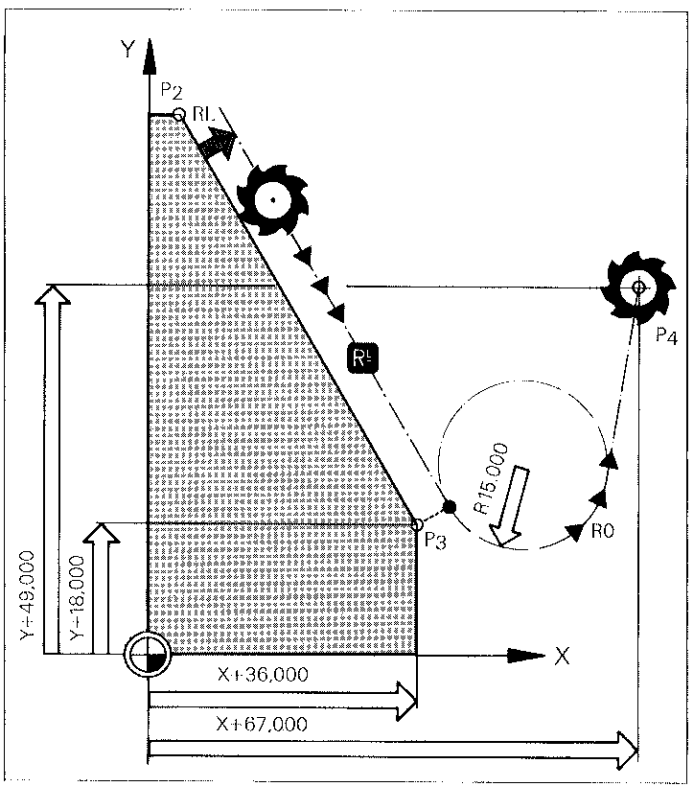
Contour departure on an arc

```
89 L   X+36,000    Y+18,000
                   RL F80    M
90 RND R15,000
                   F
91 L   X+67,000    Y+49,000
                   R0 F15999 M
```

Contour position P_3 with RL (RR)

Rounding off radius

End position P_4 with R0



Before and after an RND block a positioning block containing both coordinates of the machining plane must be programmed.

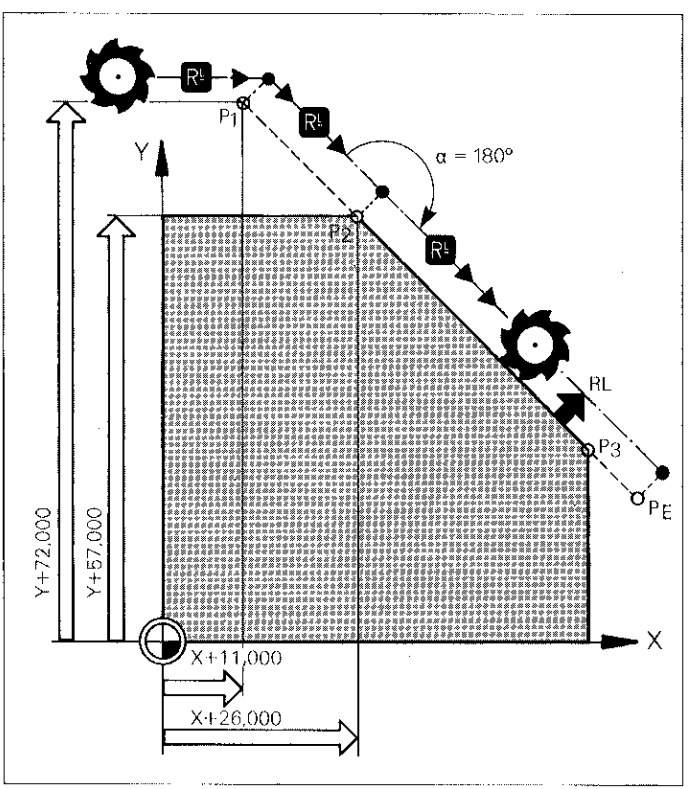
Contour approach and departure on a straight path

Path angle α = 180°

Starting position P_1 with RL (RR)

Contour position P_2 with RL (RR)

```
28 L  X+11,000  Y+72,000
                RL F15999 M
29 L  X+26,000  Y+57,000
                RL F120   M
```



Program structure for contour departure:

Last contour position with RL (RR),

Finishing position P_E with RL (RR).

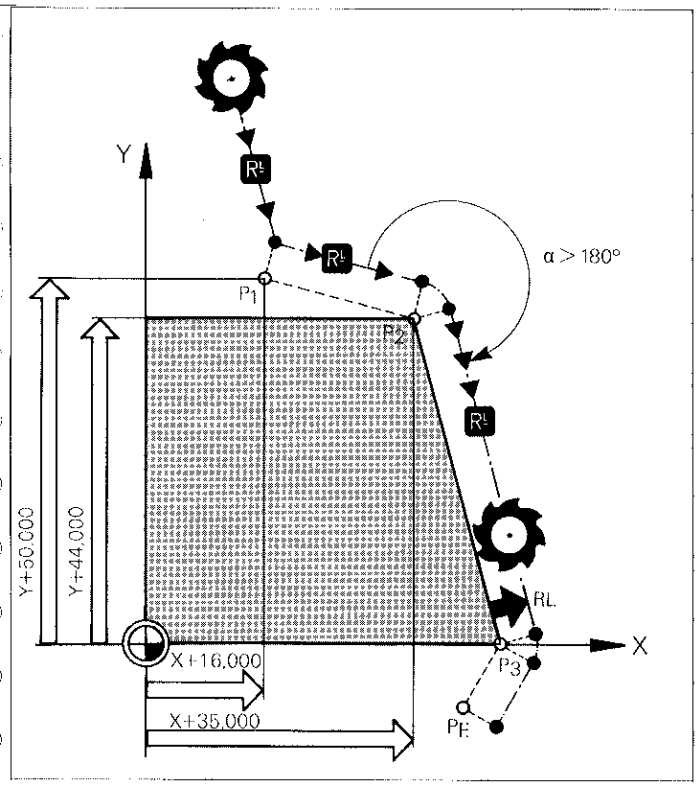
Contour approach and departure on a straight path

Path angle α greater than 180°

```
12 L  X+16,000    Y+50,000
                  RL F15999 M
13 L  X+35,000    Y+44,000
                  RL F80    M
```

Starting position P_1 with RL (RR)

Contour position P_2 with RL (RR)



Program structure for contour departure:

Last contour position with RL (RR),

Finishing position P_E with RL (RR).

Contour approach and departure on a straight path

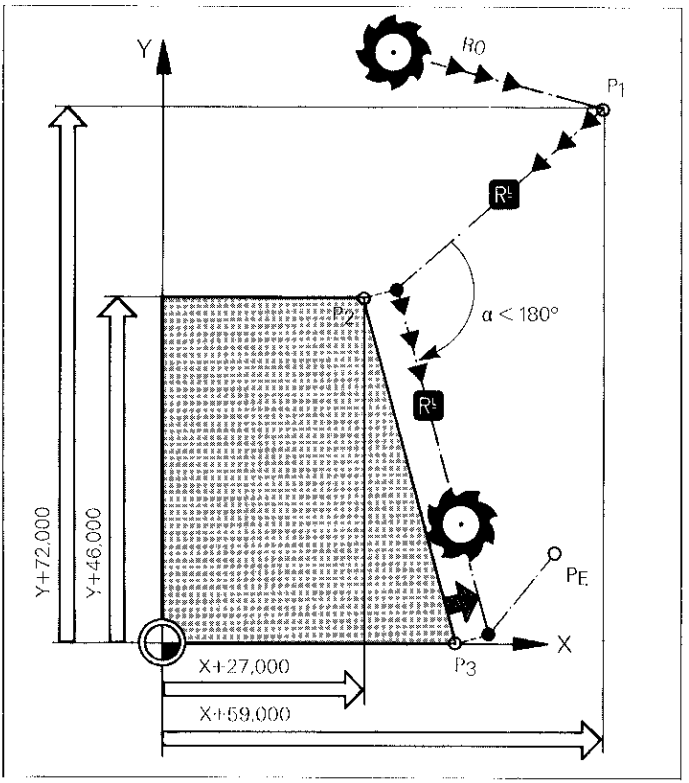
Path angle α less than 180°

Starting position P_1 with RL (RR)

```
75 L  X+59,000  Y+72,000
                R0 F15999 M
```

Contour position P_2 with RL (RR)

```
76 L  X+27,000  Y+46,000
                RL F120   M
```



Program structure for contour departure:

Last contour position with RL (RR),

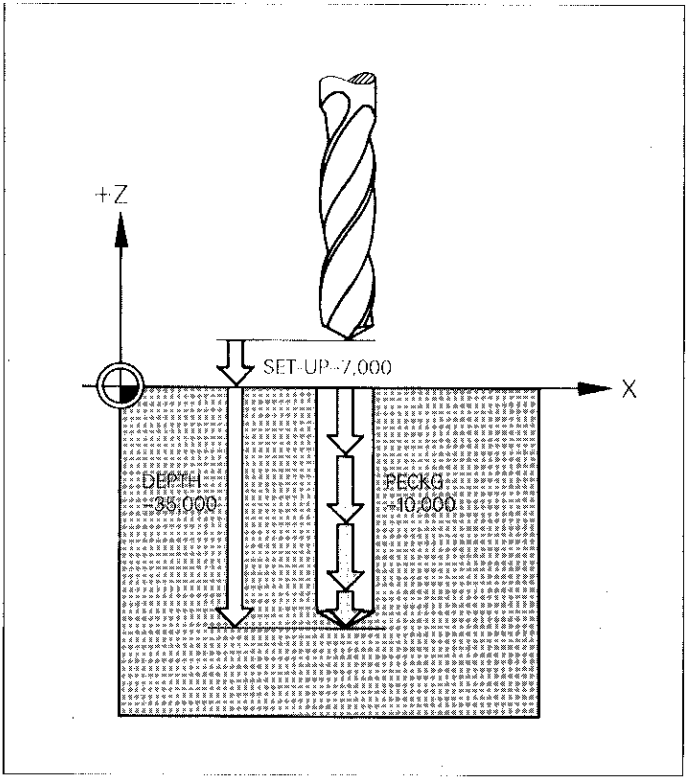
Finishing position P_E with R0.

Notes

CYCL DEF

Peck-drilling

```
12  CYCL DEF 1.0  PECKING
13  CYCL DEF 1.1  SET-UP -7,000
14  CYCL DEF 1.2  DEPTH -35,000
15  CYCL DEF 1.3  PECKG -10,000
16  CYCL DEF 1.4  DWELL 1,000
17  CYCL DEF 1.5  F180
```

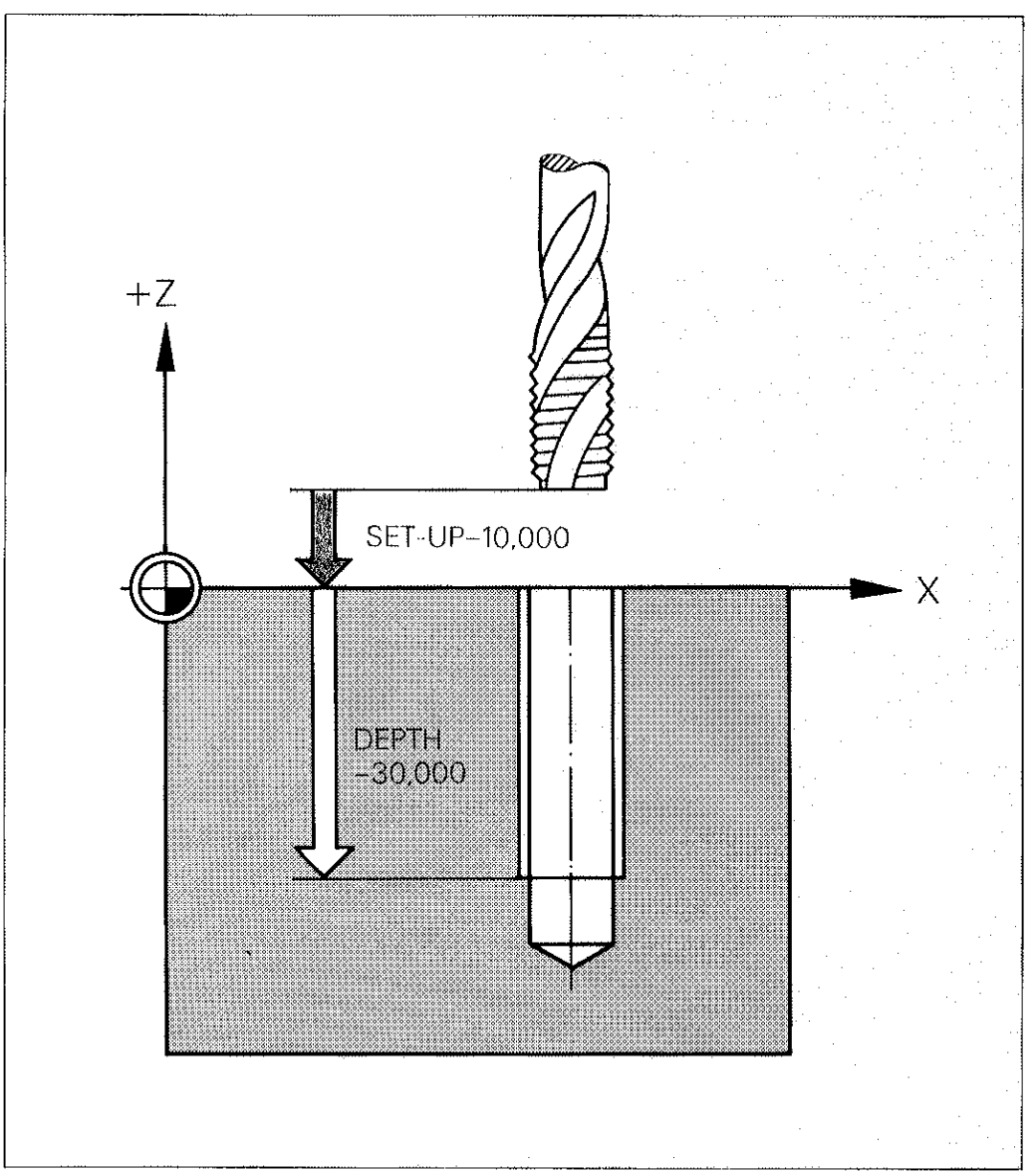


Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

Tapping

CYCL DEF

```
55  CYCL DEF 2.0  TAPPING
56  CYCL DEF 2.1  SET-UP -10,000
57  CYCL DEF 2.2  DEPTH -30,000
58  CYCL DEF 2.3  DWELL 1,000
59  CYCL DEF 2.4  F100
```



A chuck with length compensation is recommended for the tapping cycle.

Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

Feed rate calculation F:
F = S x P

S: Spindle rpm
P: Thread pitch

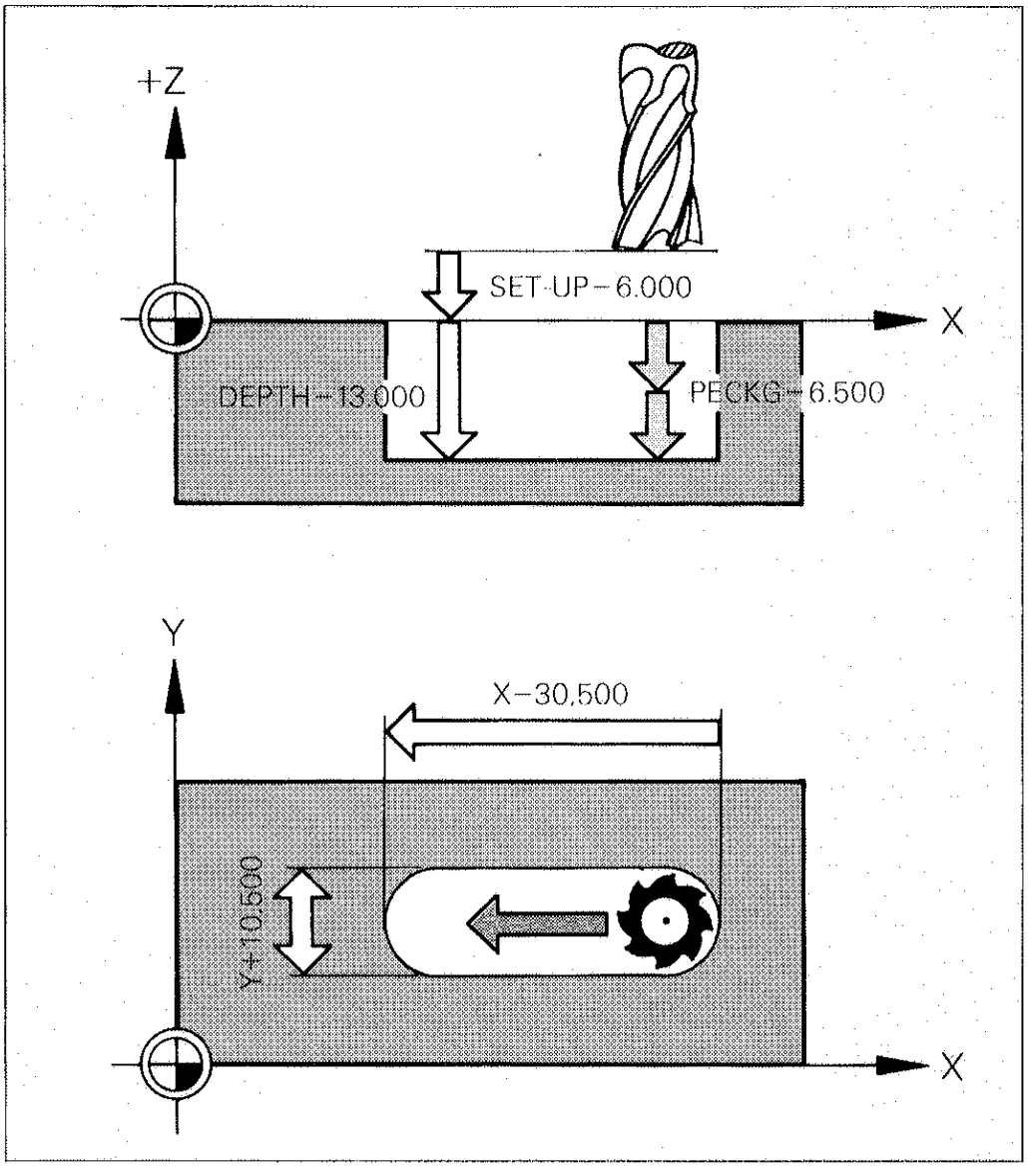
CYCL DEF

Slot milling

```
14  CYCL DEF 3.0  SLOT MILLING
15  CYCL DEF 3.1  SET-UP -6,000
16  CYCL DEF 3.2  DEPTH -13,000
17  CYCL DEF 3.3  PECKG -6,500
        F60
18  CYCL DEF 3.4  X-30,500
19  CYCL DEF 3.5  Y+10,500
20  CYCL DEF 3.6  F180
```

Slot length

Slot width



Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

The sign for the first side length depends on the direction of tool movement:
Positive axis direction +
Negative axis direction –

The sign for the second side length is always positive.

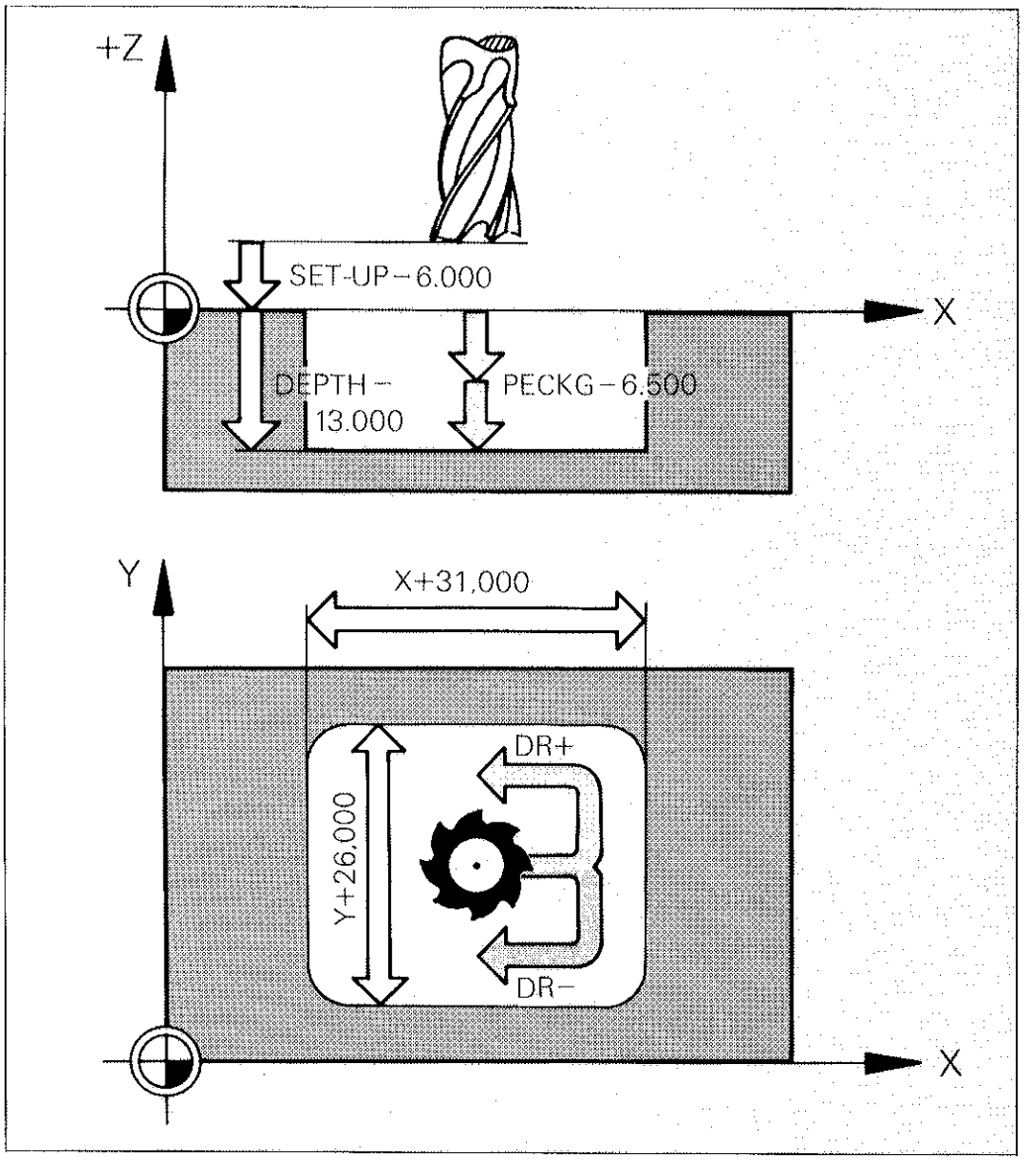
Rectangular pocket milling

CYCL DEF

```
45  CYCL  DEF  4.0  POCKET MILLING
46  CYCL  DEF  4.1  SET-UP -6,000
47  CYCL  DEF  4.2  DEPTH -13,000
48  CYCL  DEF  4.3  PECKG -6,500
    F40
49  CYCL  DEF  4.4  X+31,000
50  CYCL  DEF  4.5  Y+26,000
51  CYCL  DEF  4.6  F140   DR+
```

First side length

Second side length



Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

Signs for both side lengths are always positive.

DR+: down-cut milling

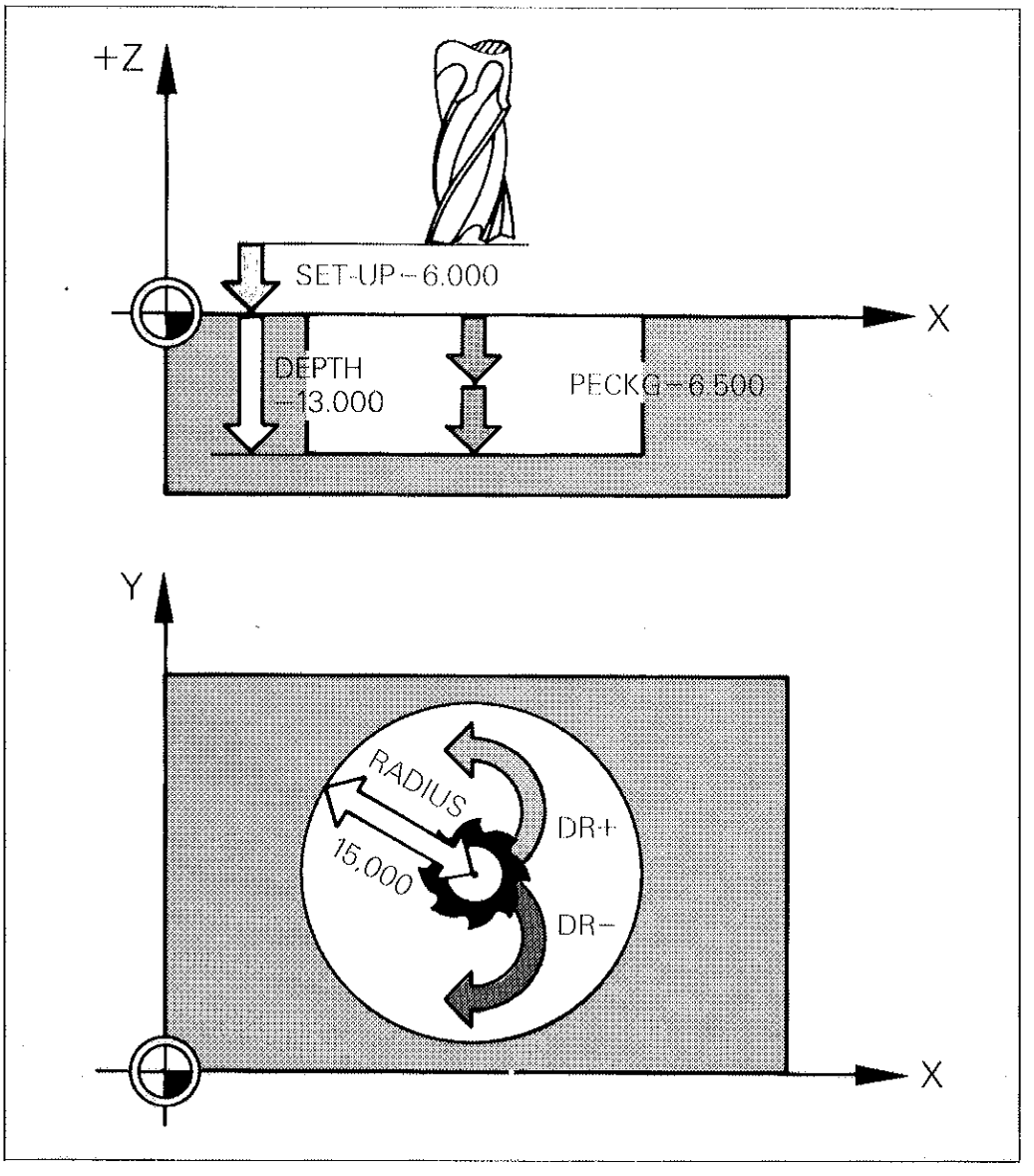
DR–: up-cut milling

Starting direction: positive axis direction of longer side.

CYCL DEF

Circular pocket milling

```
44  CYCL  DEF  5.0  CIRCULAR POCKET
45  CYCL  DEF  5.1  SET-UP -6,000
46  CYCL  DEF  5.2  DEPTH -13,000
47  CYCL  DEF  5.3  PECKG -6,500
    F60
48  CYCL  DEF  5.4  RADIUS 15,000
49  CYCL  DEF  5.5  F120    DR-
```



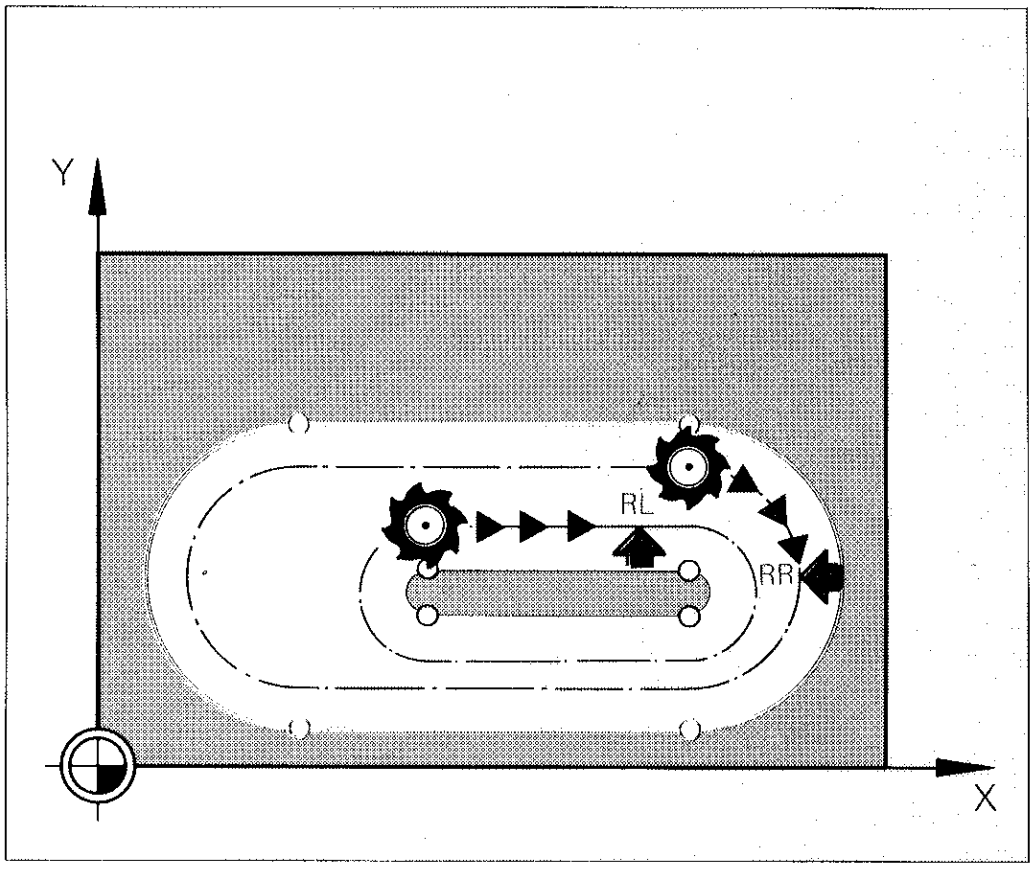
Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

DR+: down-cut milling

DR−: up-cut milling

Contour pocket

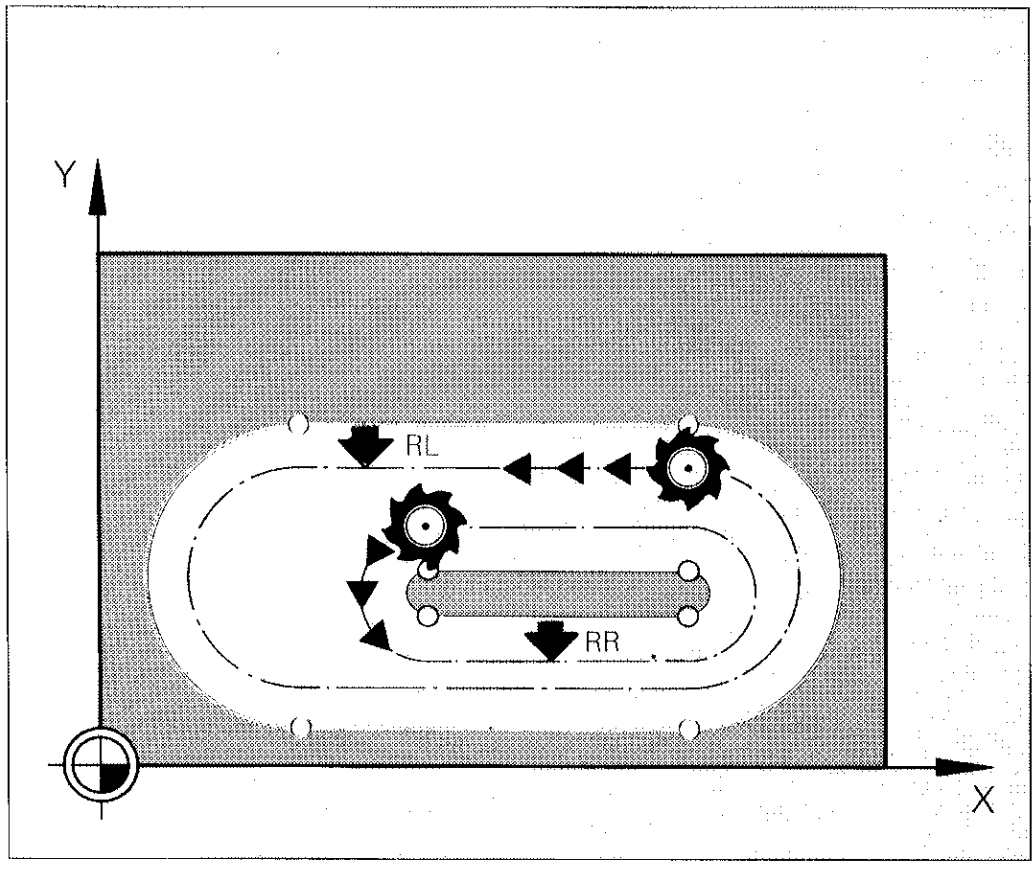
Contour definition: pocket and island



Programming of contour positions in clockwise direction

Pocket:
Radius compensation RR

Island:
Radius compensation RL



Programming of contour positions in counter-clockwise direction

Pocket:
Radius compensation RL

Island:
Radius compensation RR

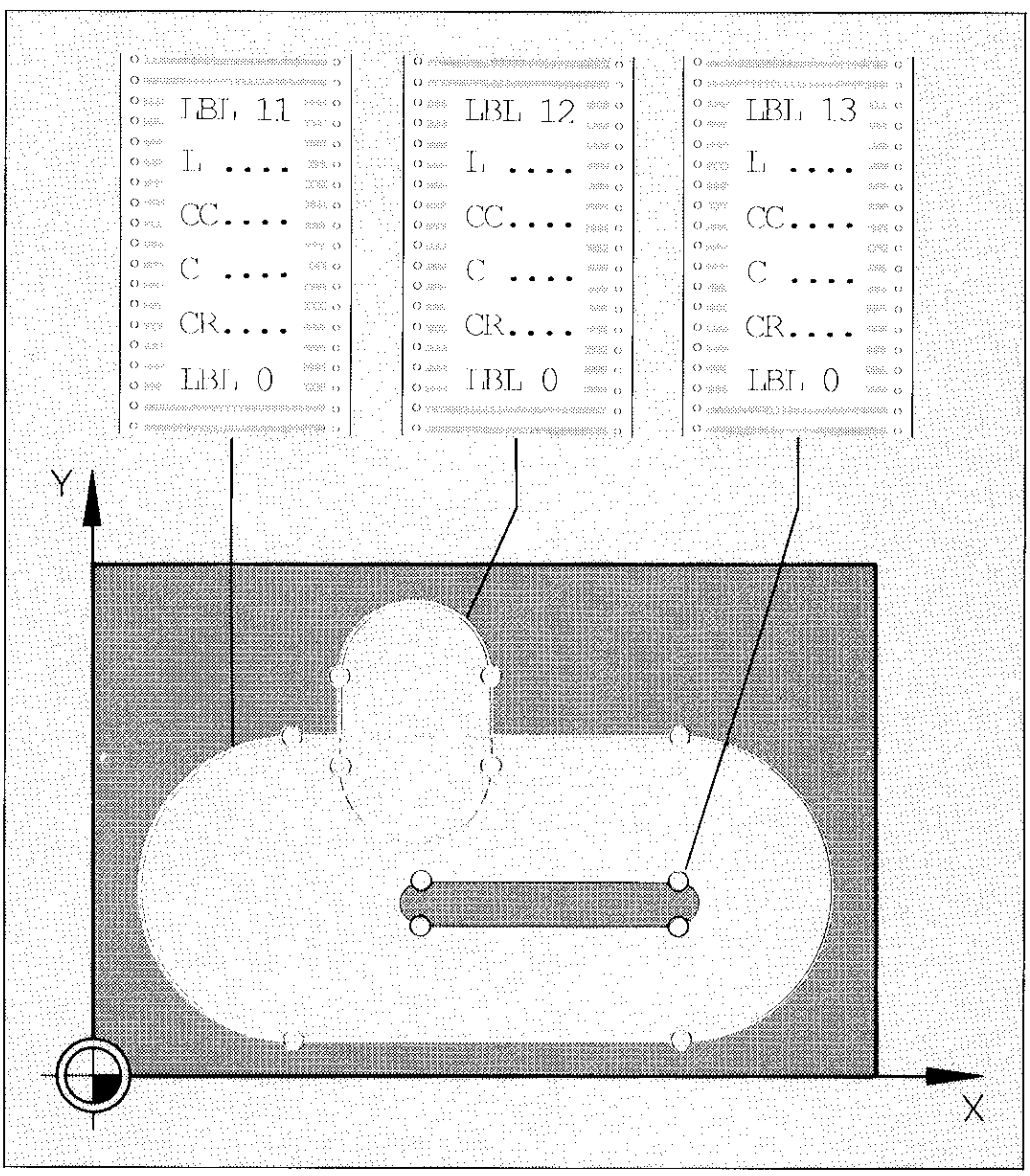
CYCL DEF

Contour pocket

Determination of subcontours

Label number of subcontours

```
18 CYCL DEF 14.0 CONTOUR
19 CYCL DEF 14.1 CONTOUR LABEL
   11 /12 /13
```



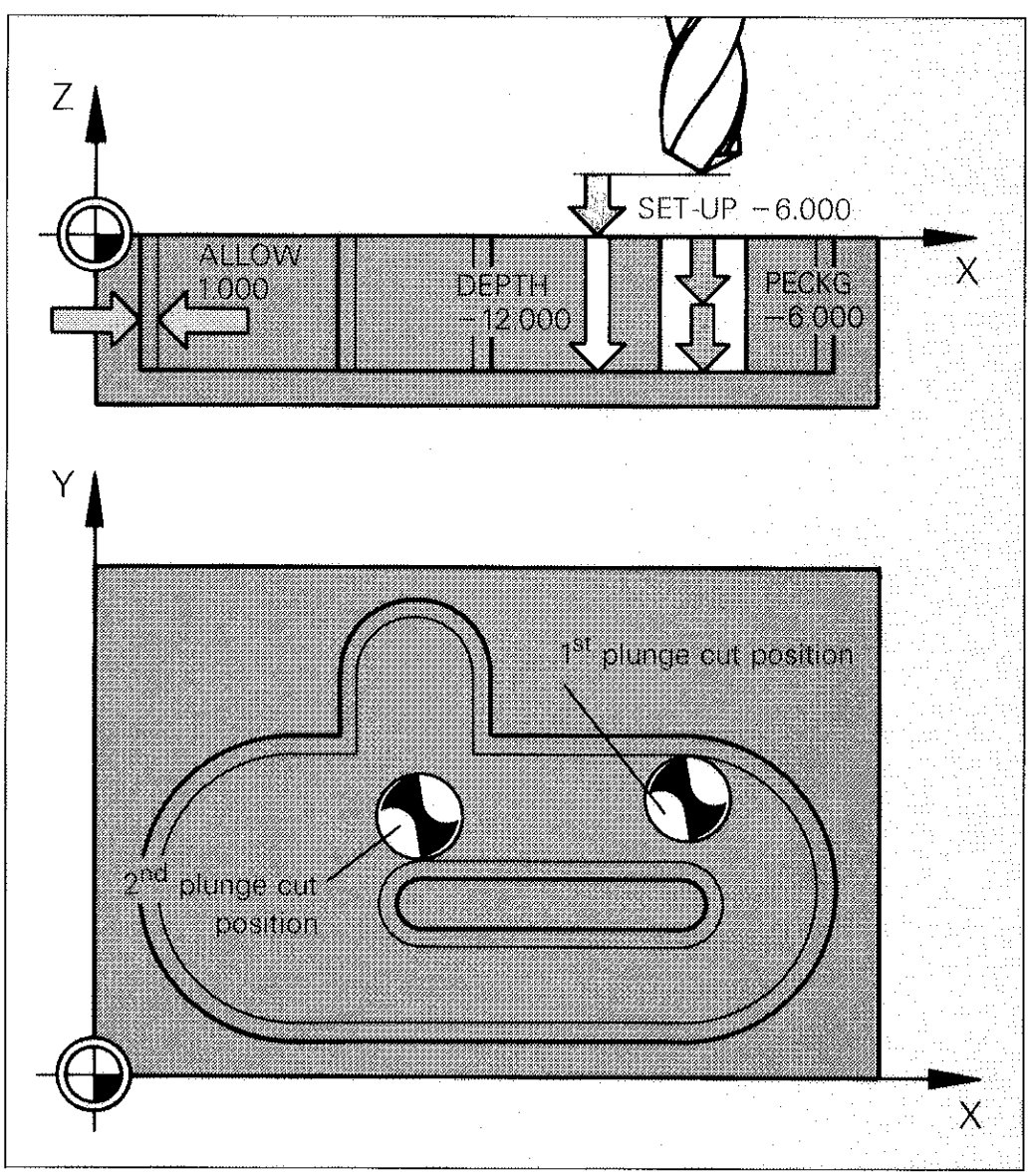
A pocket must be programmed in the first contour label.

Contour pocket

CYCL DEF

Pilot-drill

```
23  CYCL DEF 15.0 PILOT DRILL
24  CYCL DEF 15.1 SET-UP -6,000
                  DEPTH -12,000
25  CYCL DEF 15.2 PECKG -6,000
    F40           ALLOW+1,000
```



Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

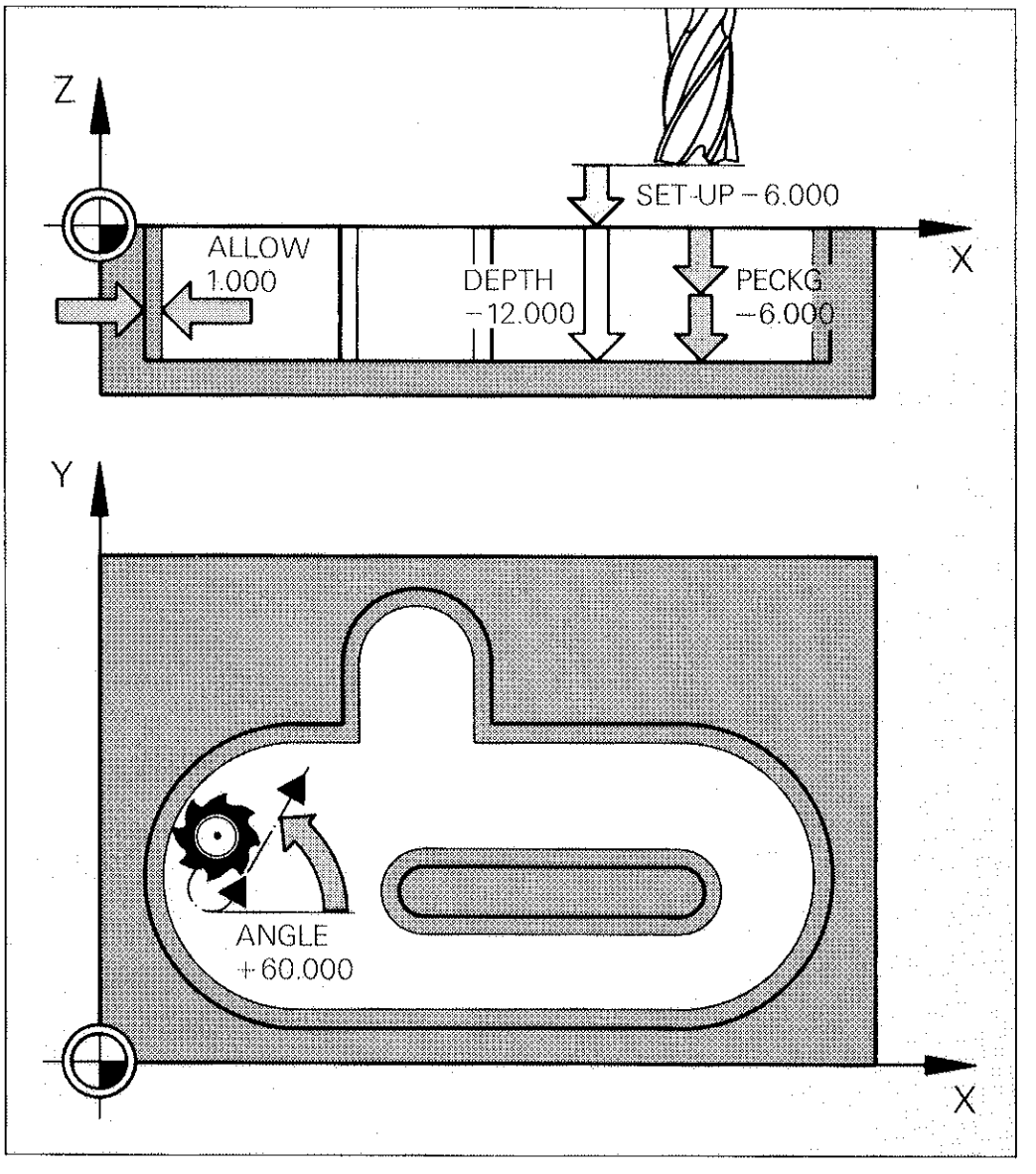
Plunge cuts:
First contour positions programmed in the subprograms.

CYCL DEF

Contour pocket

Rough-out

```
34  CYCL DEF  6.0  ROUGH-OUT
35  CYCL DEF  6.1  SET-UP -6,000
                   DEPTH -12,000
36  CYCL DEF  6.2  PECKG -6,000
    F40            ALLOW+1,000
37  CYCL DEF  6.3  ANGLE +60,000
                       F120
```



Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement.

Starting positions: plunge cuts of pilot drill-cycle.

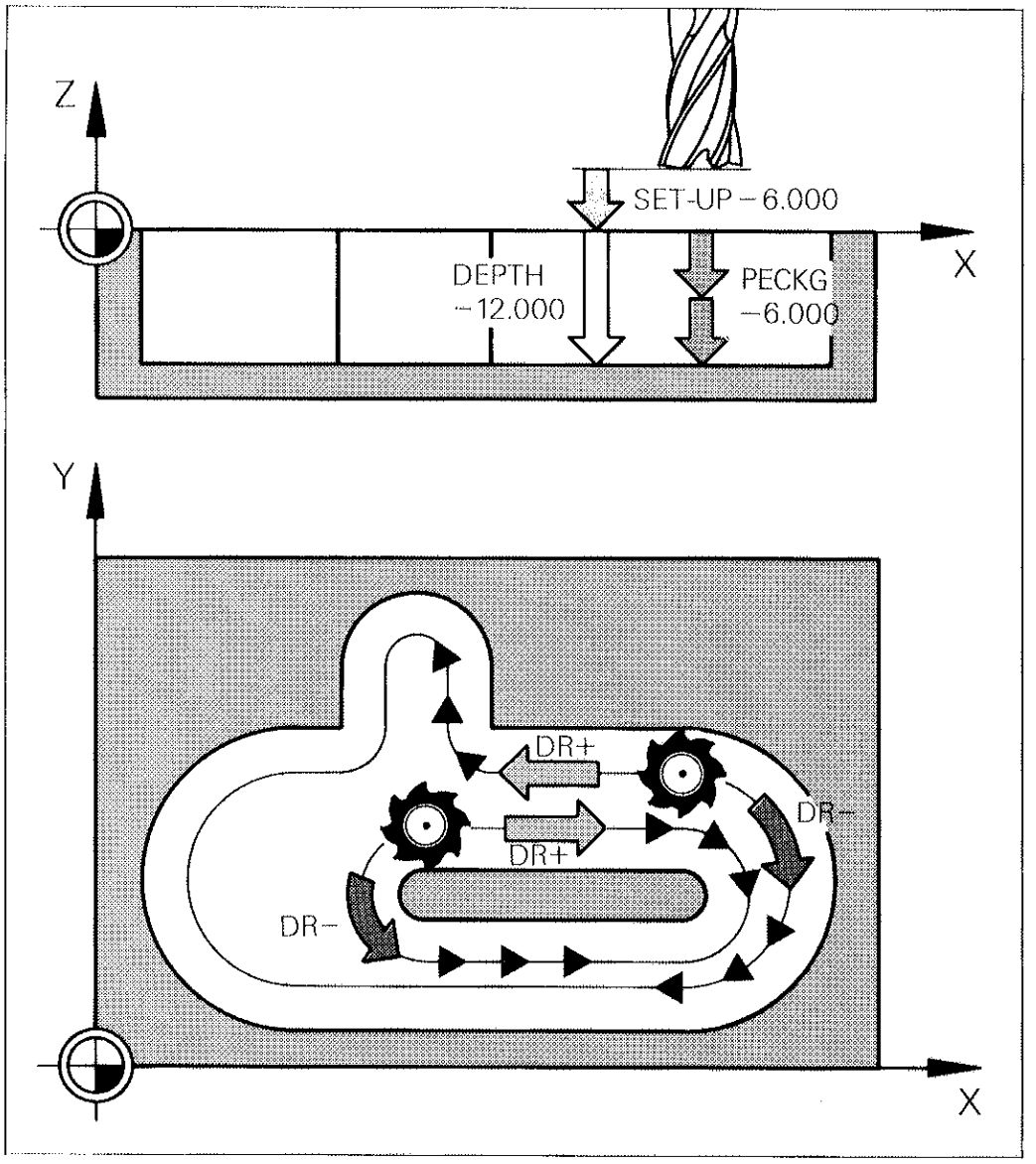
Contour pocket

CYCL DEF

Contour milling (finish cut)

```
44  CYCL DEF 16.0 CONTOUR MILLING
45  CYCL DEF 16.1 SET-UP -6,000
                  DEPTH -12,000
46  CYCL DEF 16.2 PECKG -6,000
    F60           DR-   F120
```

1st feed rate:
Pecking depth
2nd feed rate:
Contour milling



Arithmetical signs of cycle parameters are entered in accordance with the direction of tool movement

DR+: down-cut milling pocket and island

DR–: up-cut milling pocket and island

Starting positions: plunge cuts of pilot drill-cycle

CYCL DEF

Datum shift

```
14 CYCL DEF 7.0 DATUM SHIFT
15 CYCL DEF 7.1 X+31,000
16 CYCL DEF 7.2 Y+22,000
```

Coordinate specification referenced to the absolute zero datum.

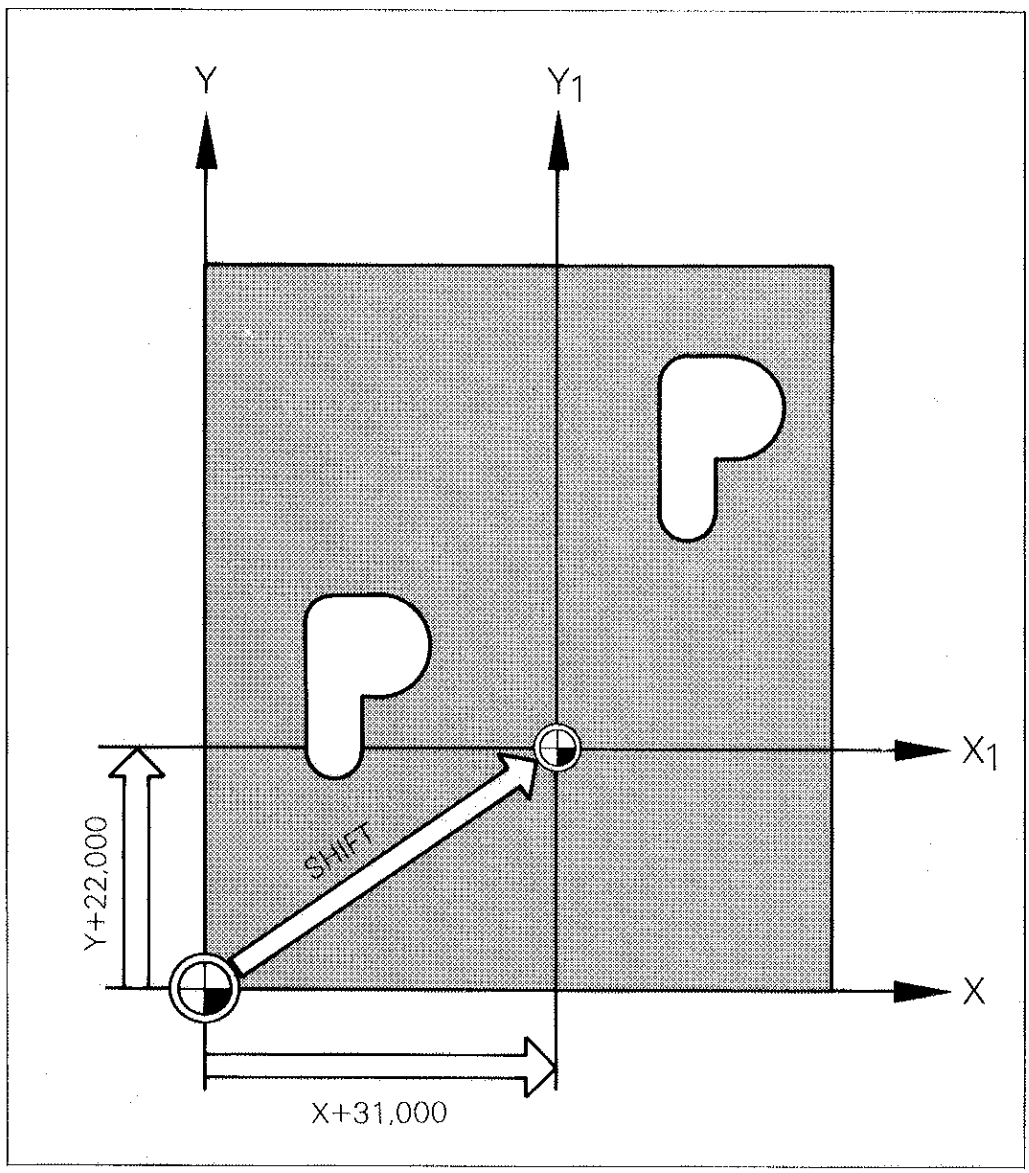
Datum shift is possible in all 4 axes.

Coordinate specification absolute: referenced to the absolute zero datum

incremental: referenced to the last datum set

Cancellation of cycle:
CYCL DEF 7 with X0.000/Y0.000 Z0.000/IV0.000

or M02 or M30 or END PGM.

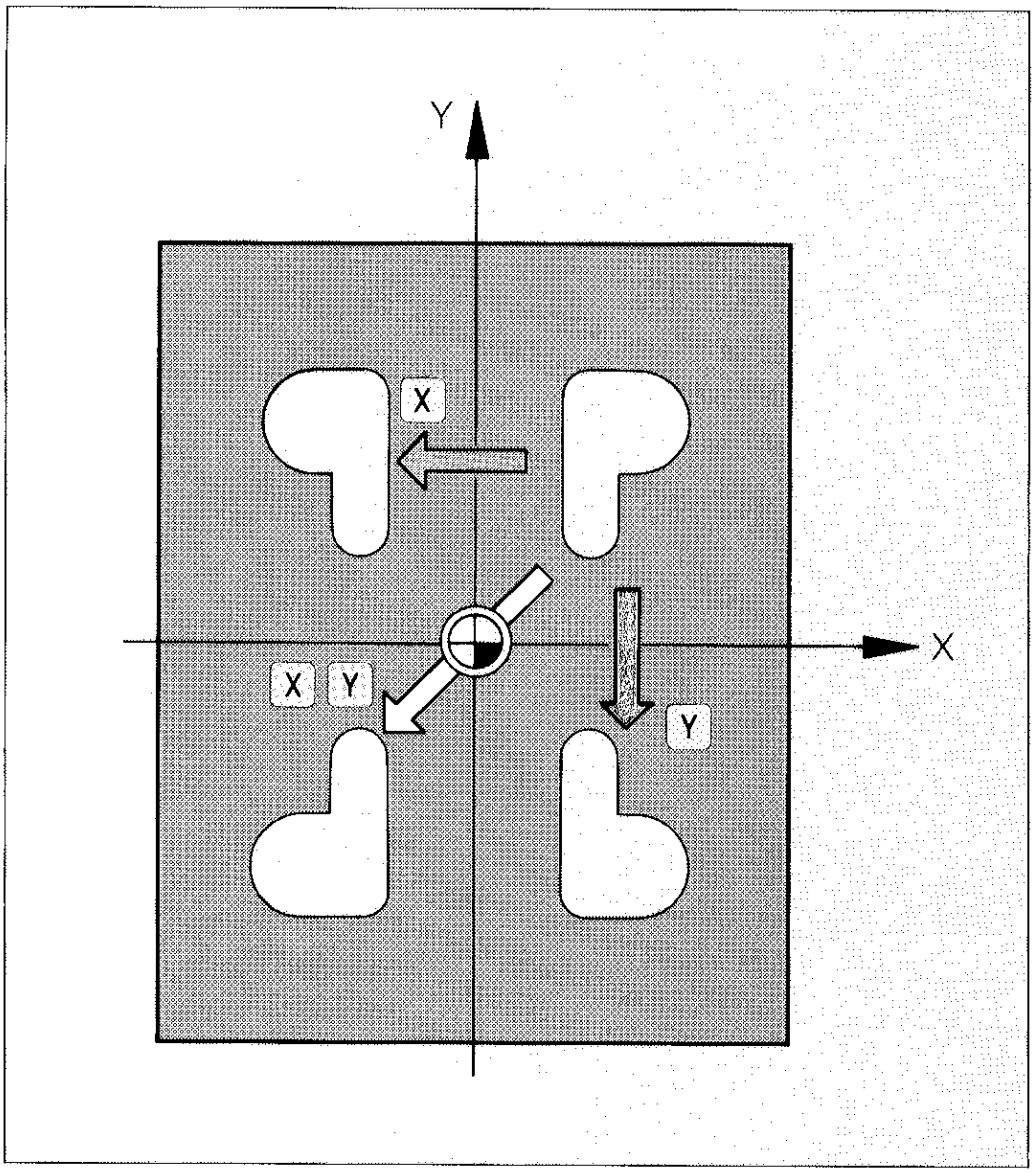


Mirror image

CYCL DEF

```
23 CYCL DEF 8.0 MIRROR IMAGE
24 CYCL DEF 8.1 X
```

Sign change of X-coordinates



In the working plane, mirror image is possible on max. 2 axes.

Cancellation of cycle:
CYCL DEF 8
Respond to mirror image with NO ENT

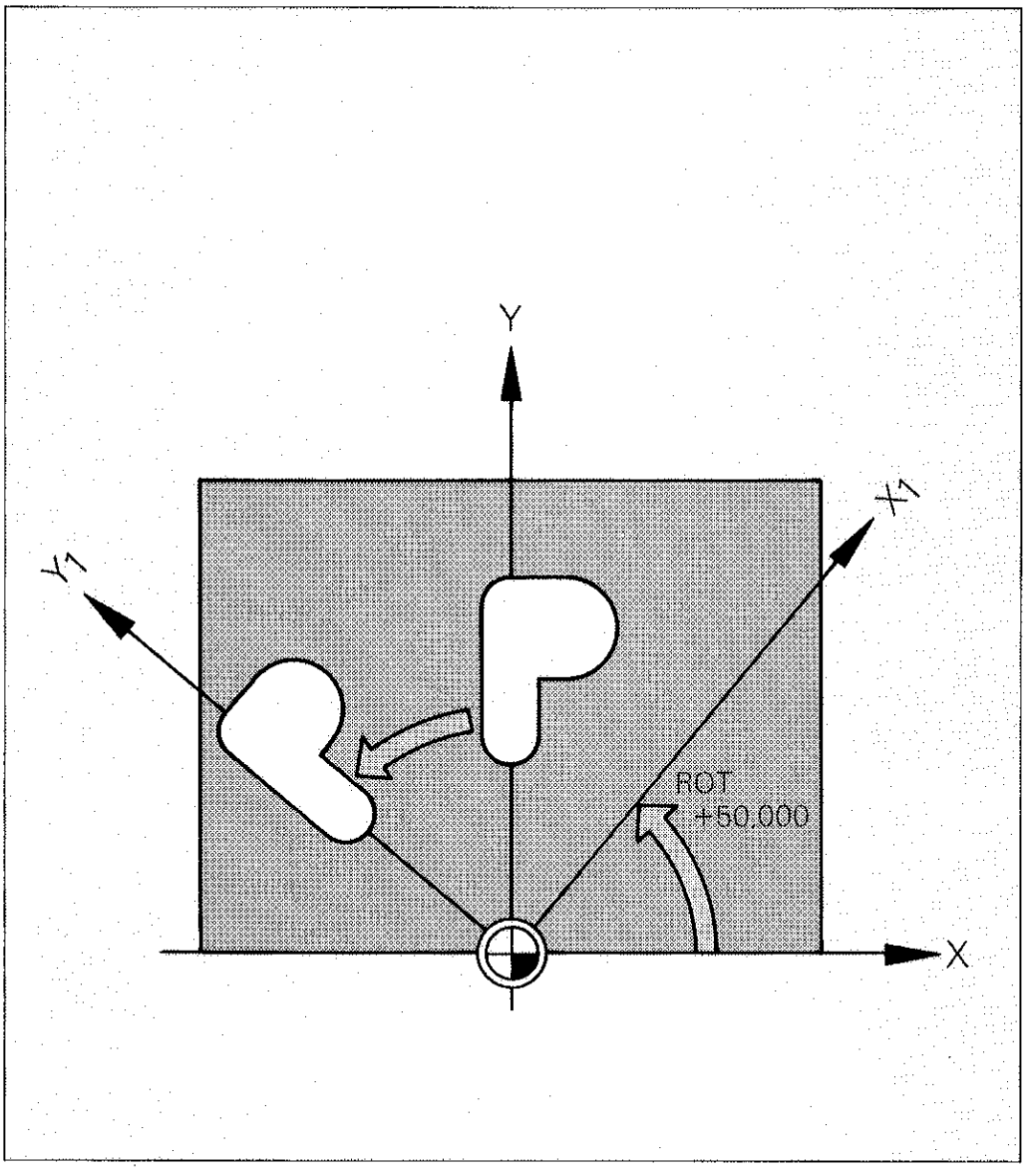
or M02 or M30
or END PGM.

CYCL DEF

Coordinate system rotation

Rotation angle

```
78 CYCL DEF 10.0 ROTATION
79 CYCL DEF 10.1 ROT+50,000
```



Coordinate system rotation takes place in the working plane.

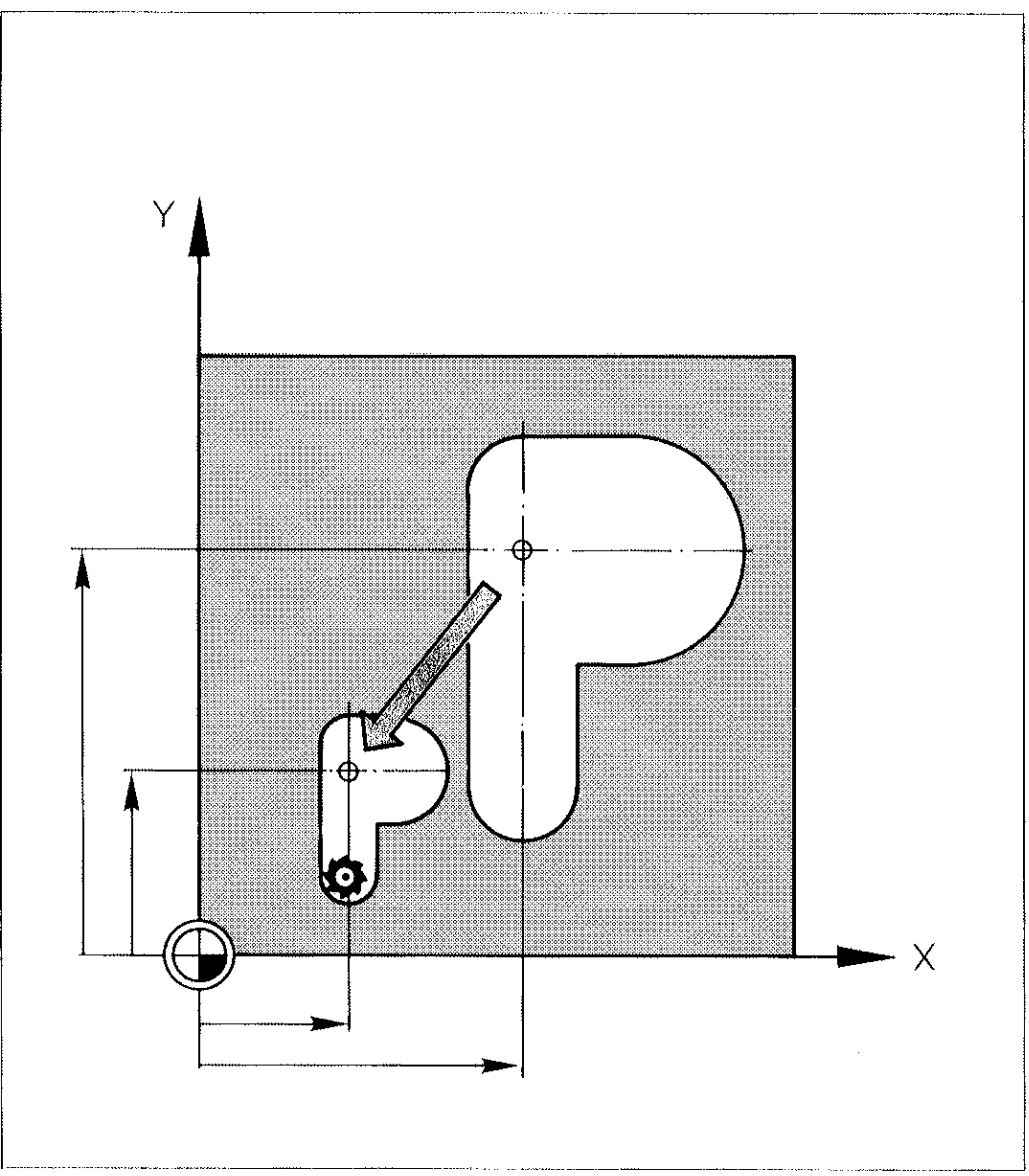
Cancellation of cycle:
CYCL DEF 10 with ROT 0.000

or M02 or M30
or END PGM.

Scaling

CYCL DEF

```
98 CYCL DEF 11.0 SCALING
99 CYCL DEF 11.1 SCL 0,500000
```



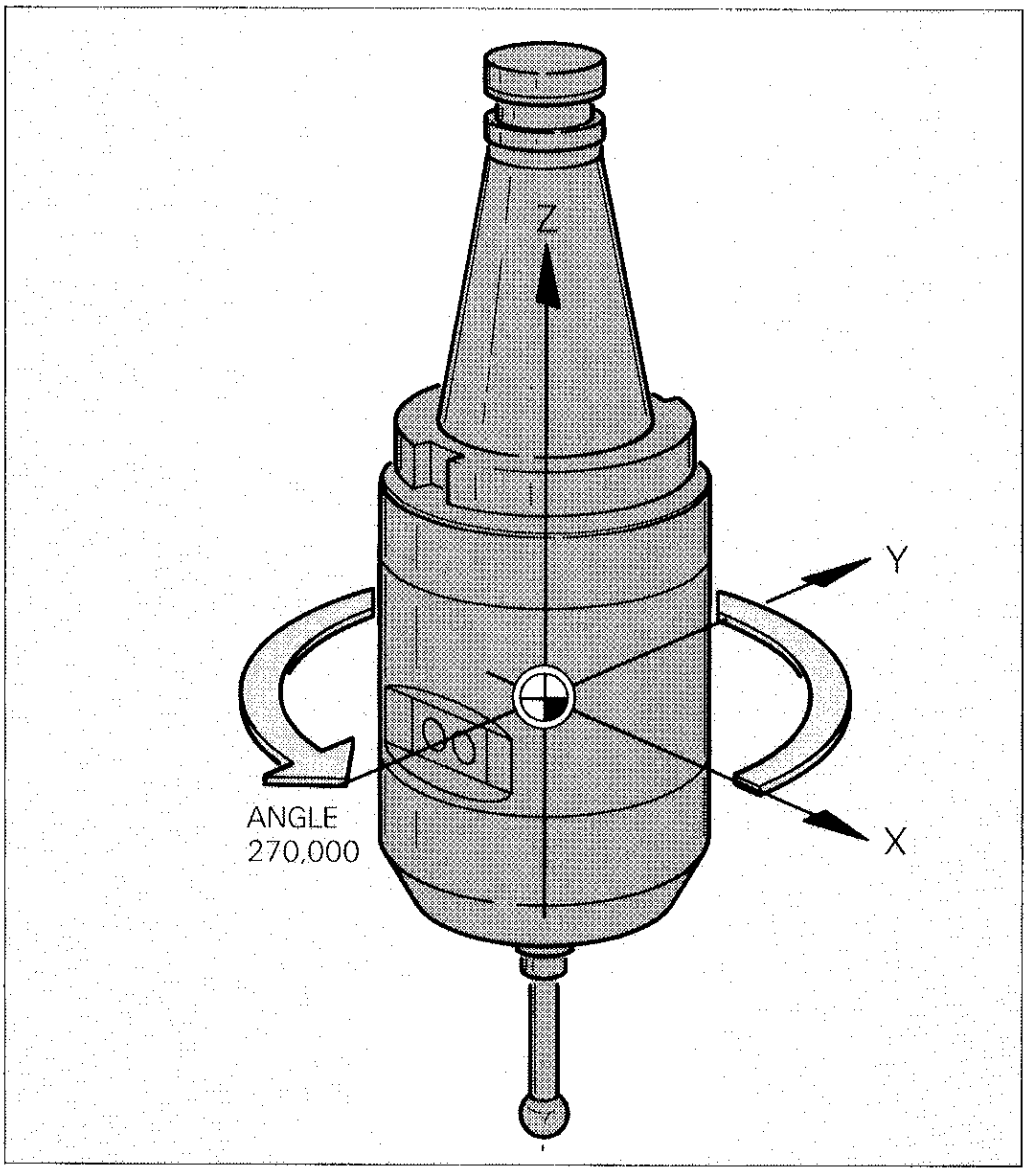
With the scaling factor SCL, all subsequent coordinates and radii within the working plane or the three axes X, Y and Z are multiplied by the scaling factor. Angle values remain the same.

Cancellation of cycle:
CYCL DEF 11 with SCL 1.0

or M02 or M30
or END PGM.

Spindle orientation

```
38 CYCL DEF 13.0 ORIENTATION
39 CYCL DEF 13.1 ANGLE 270,000
```



Cycle call is effected via a machine-specific M-function.

Notes

LBL SET

Subprogram

LBL CALL

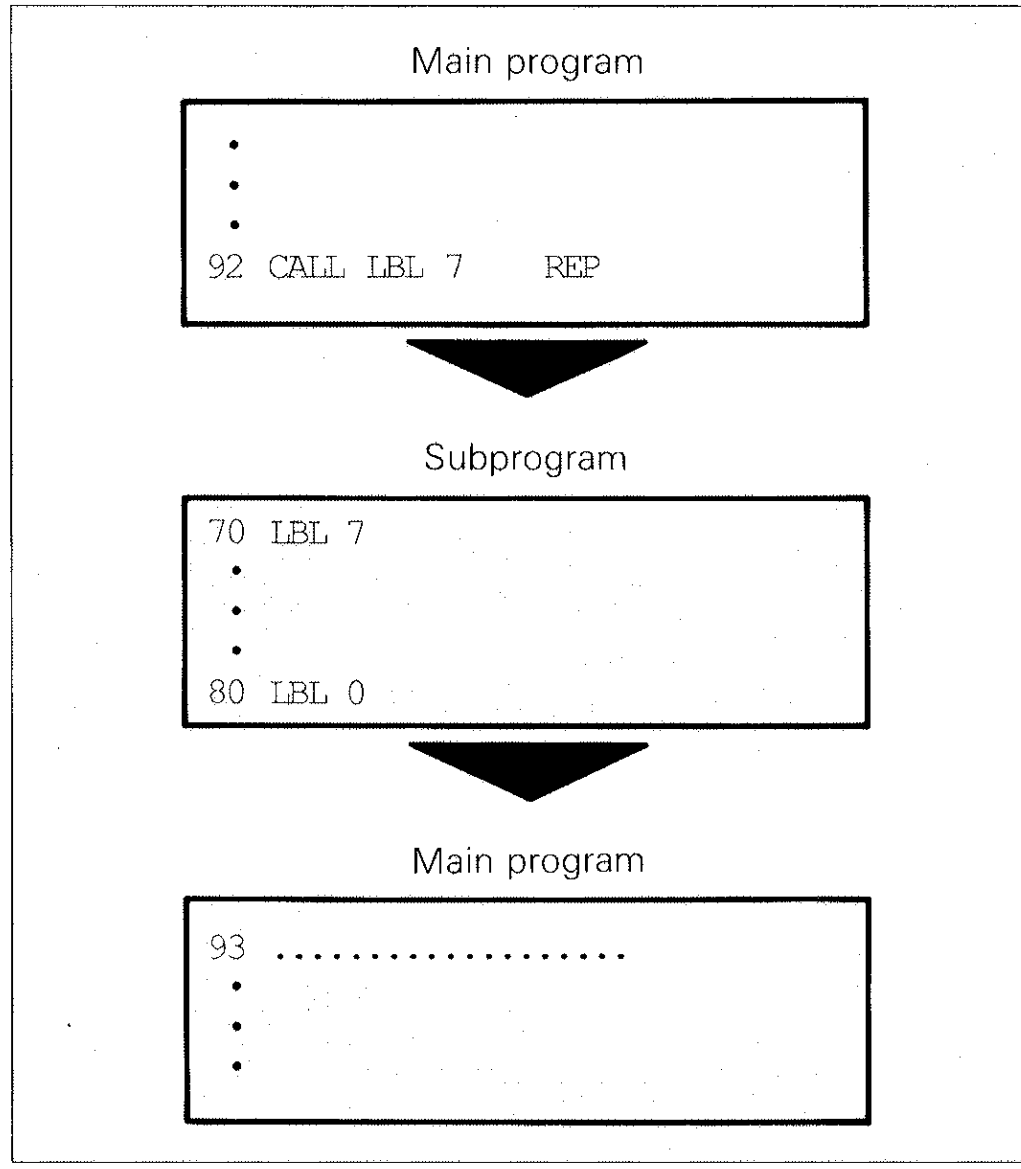
Subprogram call

Subprogram beginning

Subprogram end

Subprogram call

```
70 LBL7
 •
80 LBL 0
 •
92 CALL LBL 7     REP
```



Main program and subprogram form parts of a complete program.

With a subprogram call, repetitions (REP) must not be programmed.

Program part repeat

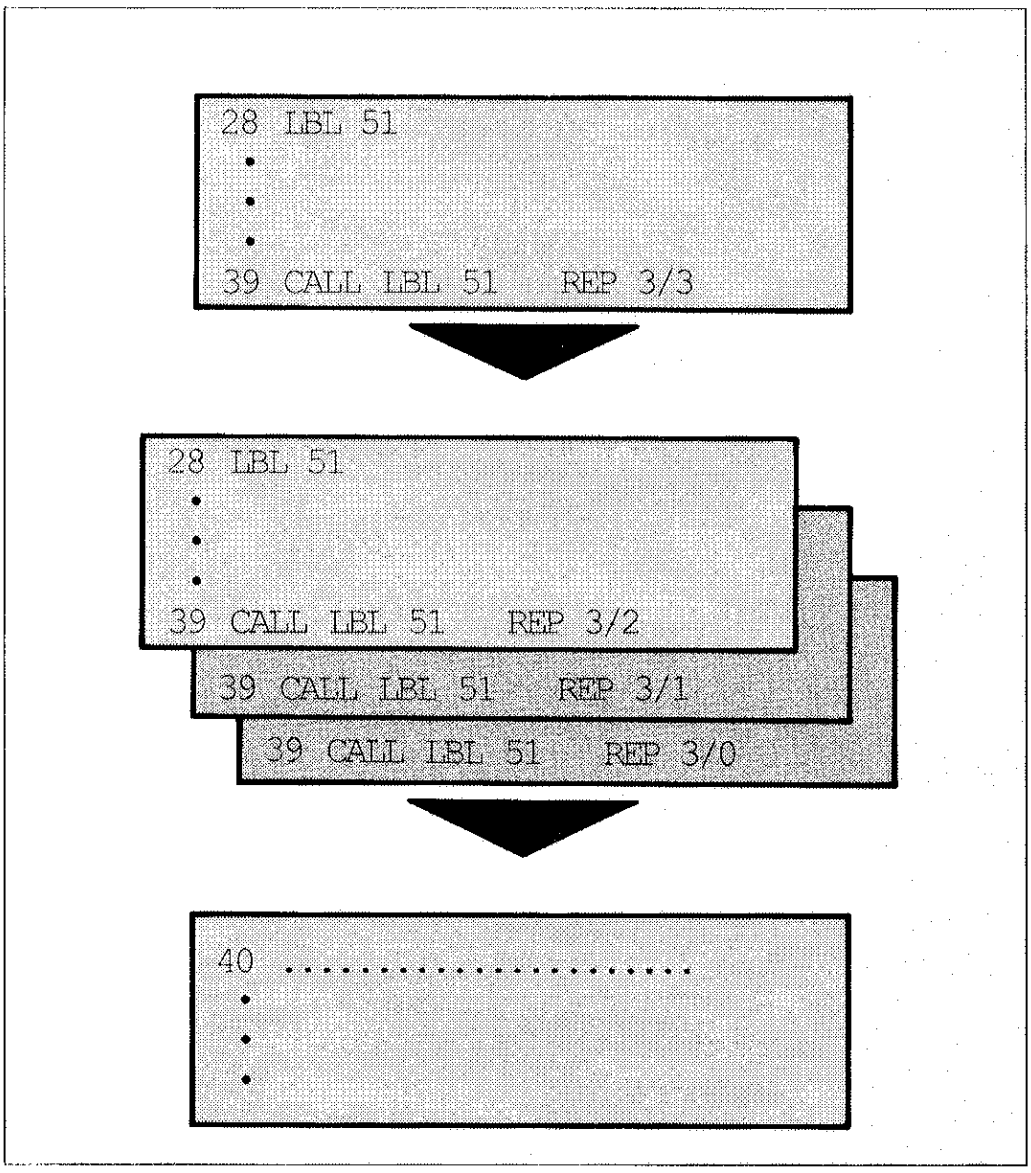
LBL SET

LBL CALL

```
28 LBL 51
 •
39 CALL LBL 51    REP 3/3
```

Beginning of part program

Program part end simultaneously a call-up for repetition



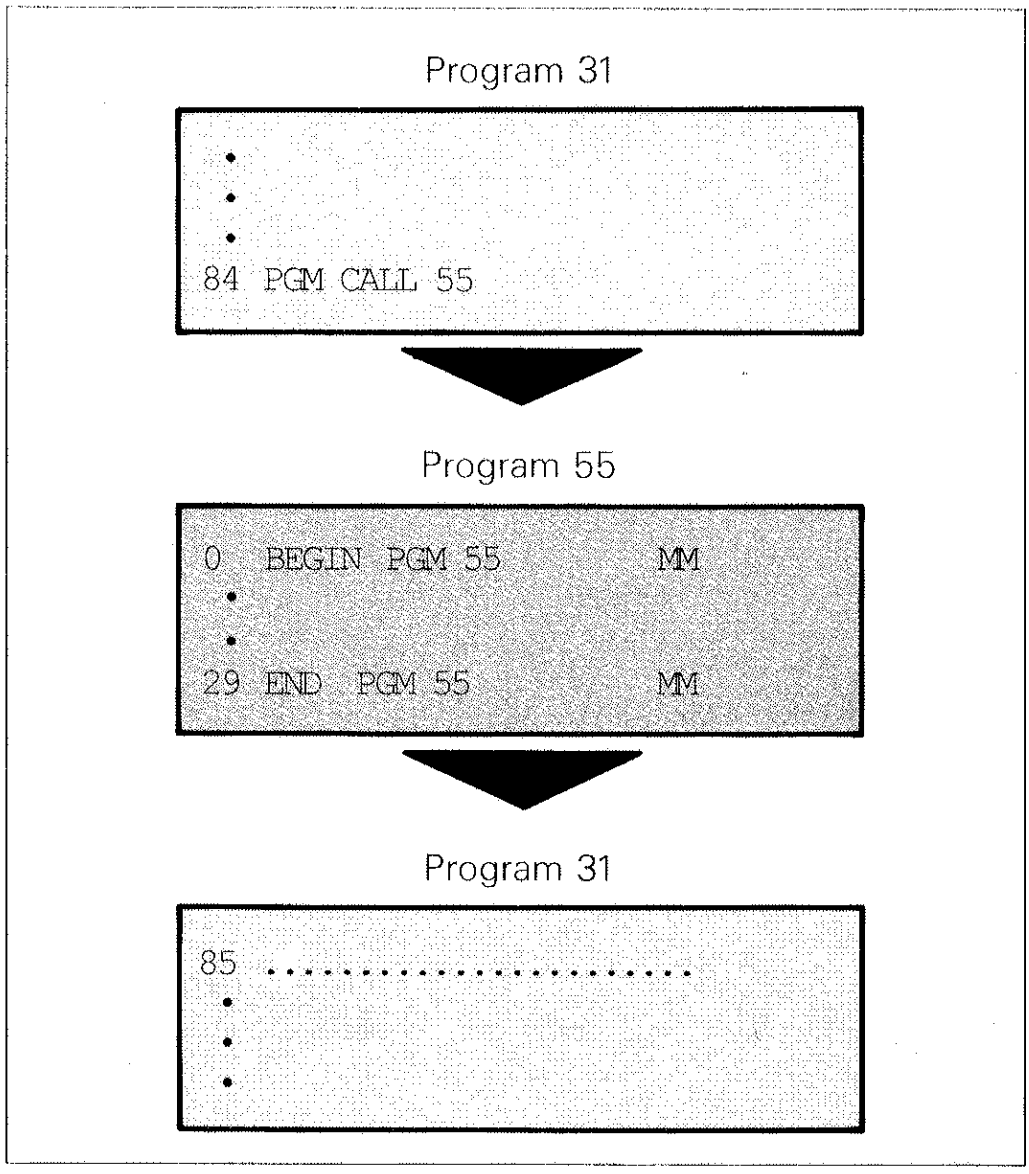
REP 3/3:

The countdown numeral after the stroke indicates the number of repetitions still to be executed. It is decreased by 1 on completion of each repetition.

PGM CALL

Program call

```
84 PGM CALL 55
```



In block 84 of program No. 31 a jump is made to the beginning of program No. 55.

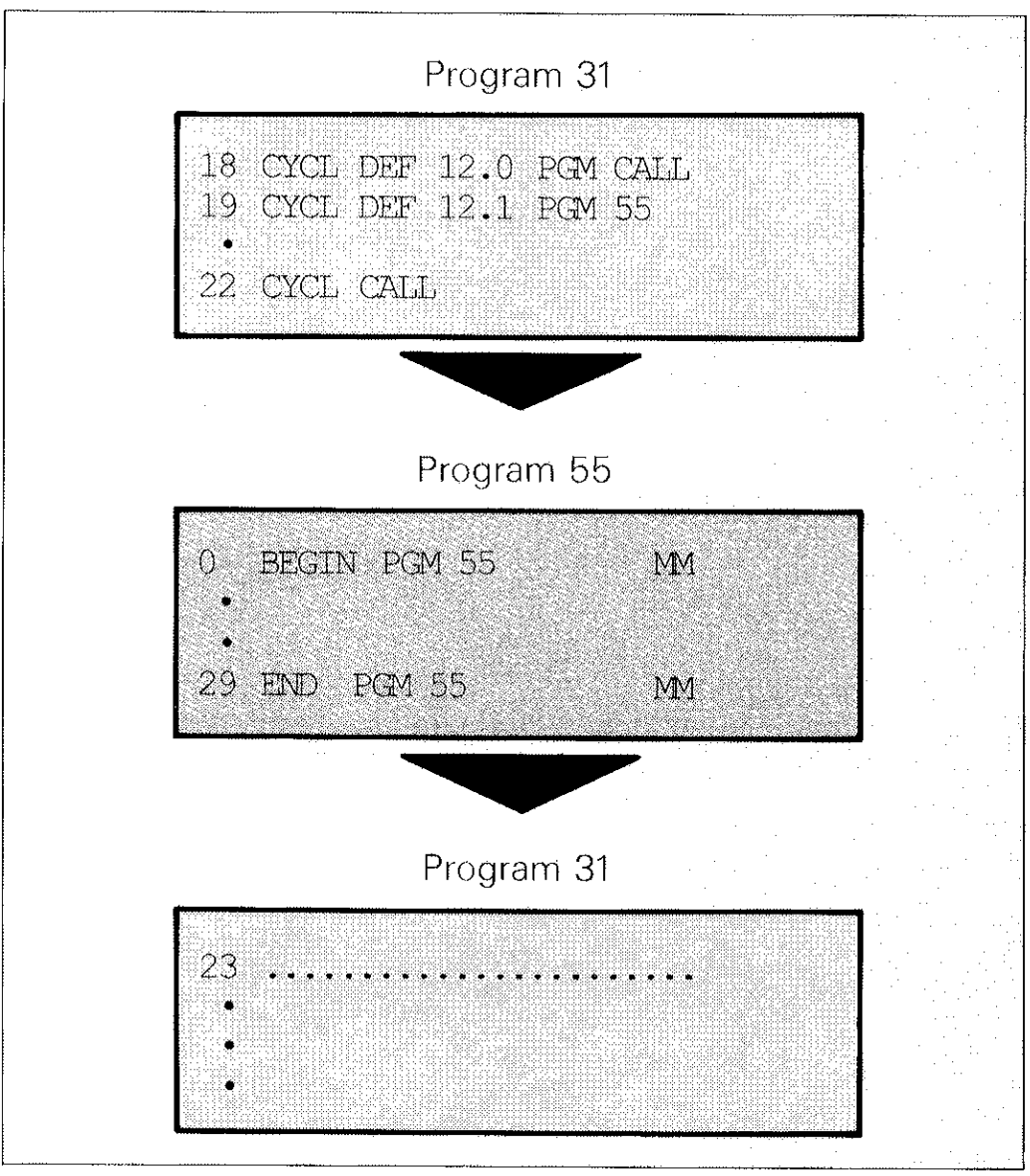
At the end of program No. 55 a return jump is made to program No. 31 continuing from block 85.

Program call cycle

CYCL DEF

```
18 CYCL DEF 12.0 PGM CALL
19 CYCL DEF 12.1 PGM 55
 •
22 CYCL CALL                    M
```

Call-up of CYCL 12



At block 22 of program No. 31 a jump is made to the beginning of program No. 55.

At the end of program No. 55 a return jump is made to program No. 31 continuing from block 23.

Cycle call can be initiated with either

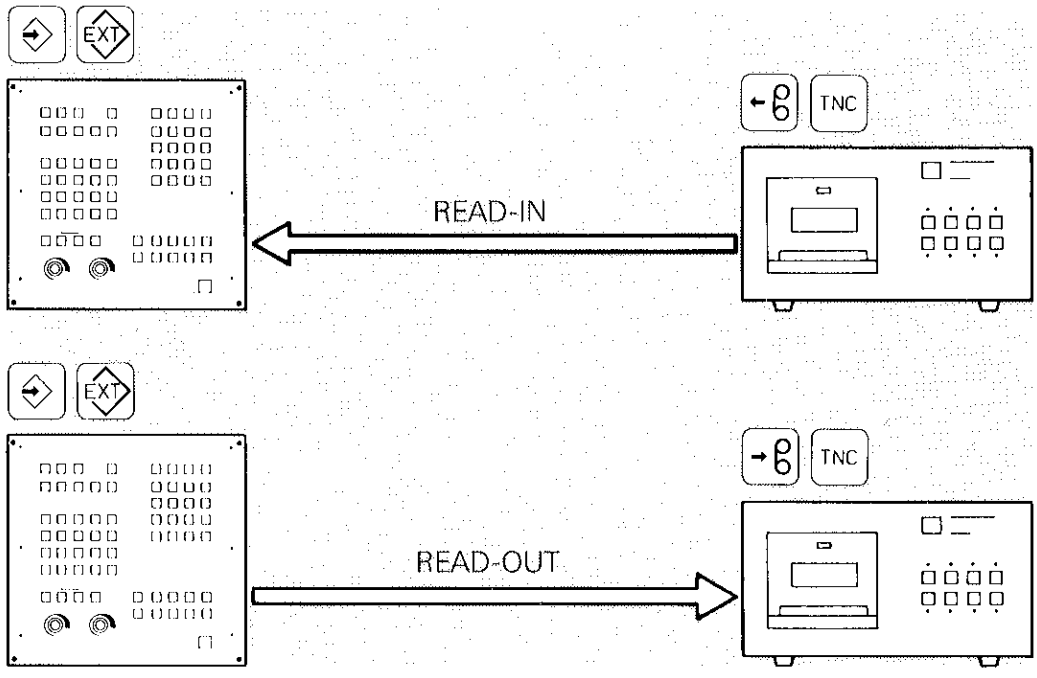
- CYCL CALL
- M99
- M89

External data transmission

V.24/RS-232-C data interface of TNC
Operating modes

V.24/RS-232-C INTERFACE = ME

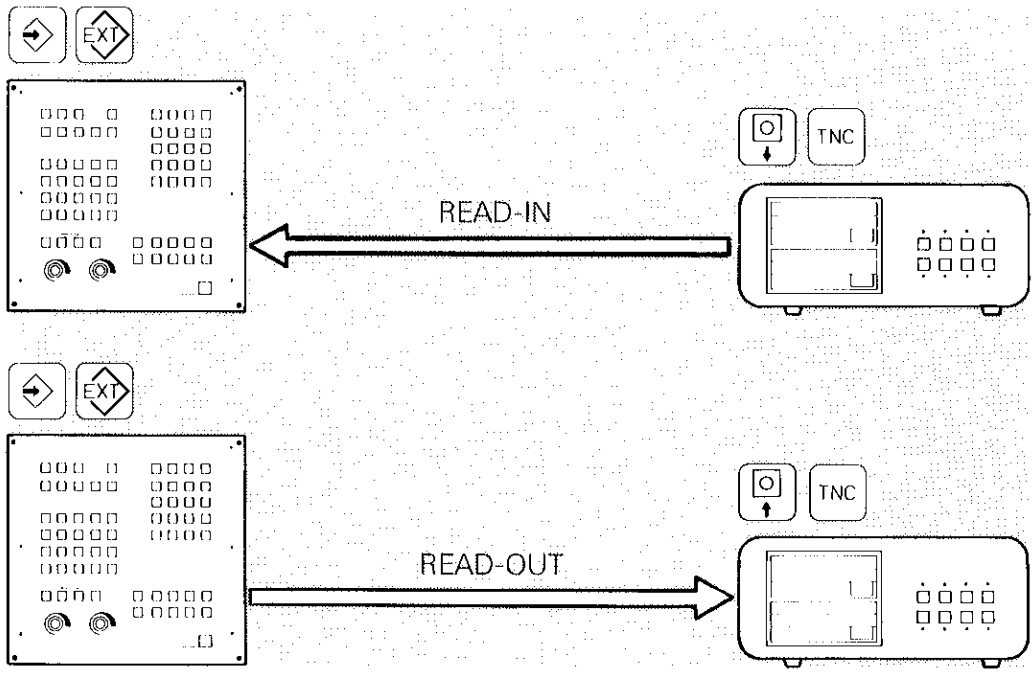
Transfer rate: 2400 Baud



Operation of the magnetic tape unit is only possible in the interface mode ME.

V.24/RS-232-C INTERFACE = ME

Transfer rate: 2400 Baud



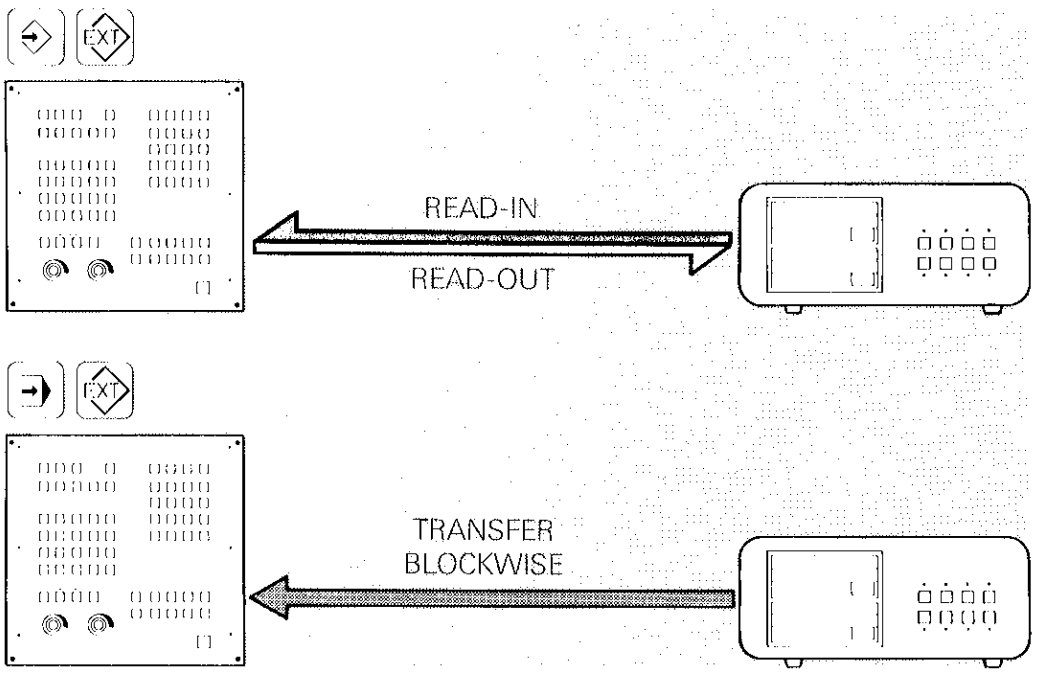
Operation of the disk unit is only possible in the interface modes ME and FE.

External data transmission

V.24/RS-232-C data interface of TNC
Operating modes

V.24/RS-232-C INTERFACE = FE

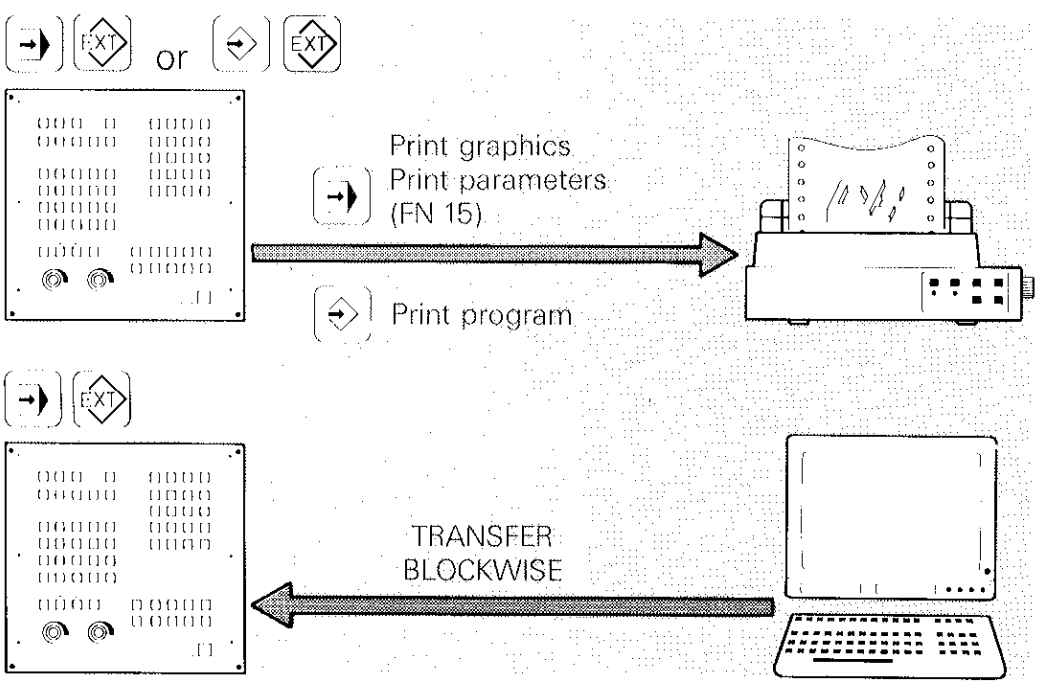
Transfer rate: 9600 Baud



In FE mode, operation of disk unit only via TNC keyboard.

V.24/RS-232-C INTERFACE = EXT

Transfer rate: 9600 Baud
other Baud rates programmable



Please observe the notes in the TNC interface description with the EXT-operation.

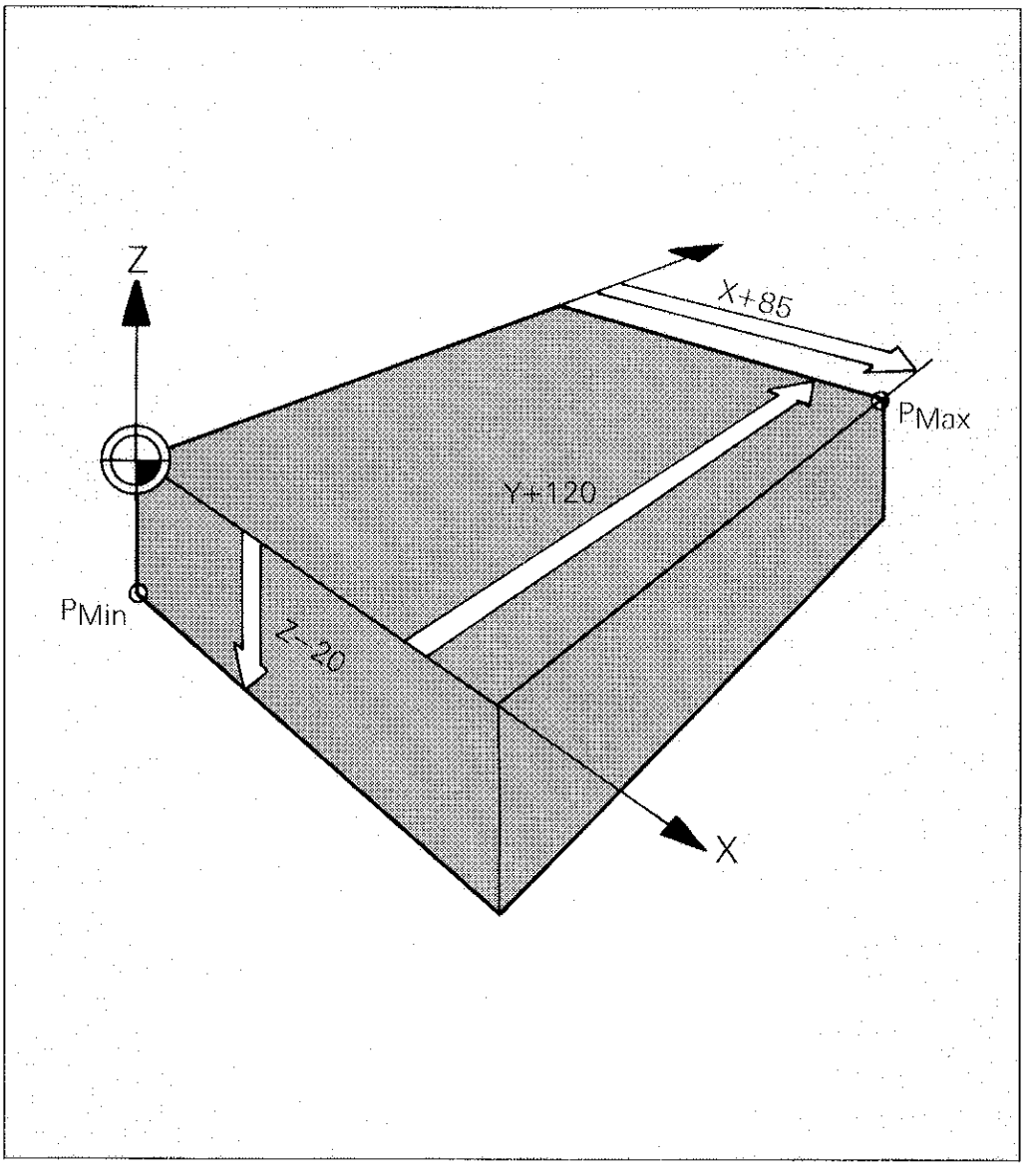
BLK FORM

Graphics

Spindle axis Z and Point P_{min}

Point P_{max}

```
1  BLK  FORM      0.1 Z  X+0
                  Y+0    Z-20
2  BLK  FORM      0.2    X+85
                  Y+120  Z+0
```



BLK FORM:
Blank form
= shape of blank

The working plane is always perpendicular to the spindle axis.

P_{min}: Only in absolute dimensions

P_{max}: Either absolute or incremental dimensions

Magnify

MAGN

Manual shift of sectional plane

In opposite direction

Automatic shift of sectional plane

In opposite direction

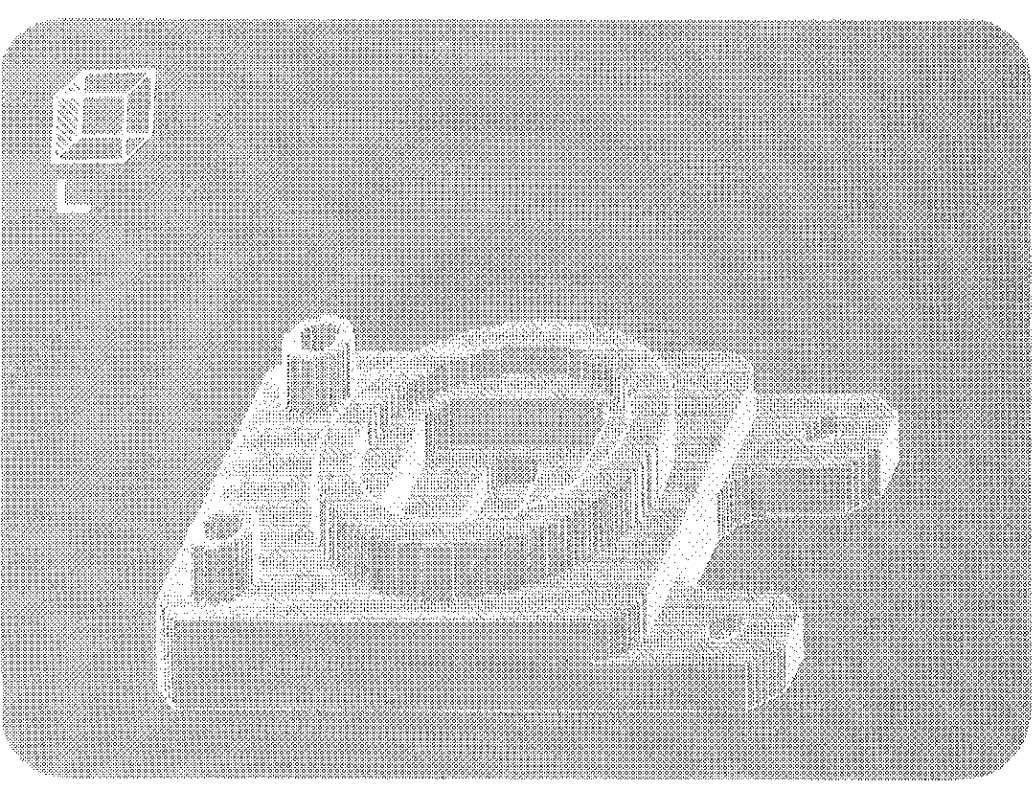
Shift stop

Select next sectional plane (forward paging)

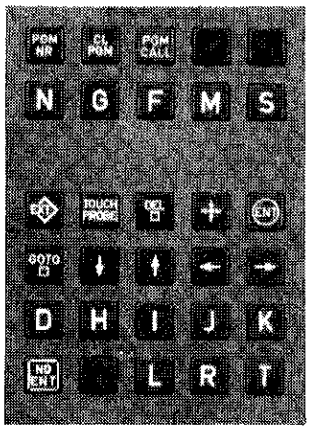
Select previous sectional plane (reverse paging)

TRANSFER GRAPHICS DETAIL = ENT

Start program run



Before selecting the magnify function, the control must be in the graphics mode "3D-view".



Program entry in ISO-format

Letter addresses

% Program beginning or program call
% Program call with G39

A Rotary axis about X-axis
B Rotary axis about Y-axis
C Rotary axis about Z-axis

D Parameter definition (Parameter Q)

F Feed code
F Dwell time with G04
F Scaling factor with G72

G Preparatory "GO" function

H Angle for polar co-ordinates
H Rotational angle with G73

I X-Coordinate of circle centre/pole
J Y-Coordinate of circle centre/pole
K Z-Coordinate of circle centre/pole

L Set label number with G98
L Jump to label number
L Tool length with G99

M Auxiliary (Miscellaneous) function

N Block number

P Cycle parameter for canned cycles
P Parameter in parameter definition

Q Q-parameter

R Radius for polar co-ordinates
R Circle radius with G02/G03/G05
R Round-off radius w. G25/G26/G27
R Chamfer with G24
R Tool radius with G99

S Spindle speed
S Spindle angle with G36

T Tool definition with G99
T Tool call
T Next tool with G51

U Additional linear axis parallel to X-axis
V Additional linear axis parallel to Y-axis
W Additional linear axis parallel to Z-axis

X X-Axis command
Y Y-Axis command
Z Z-Axis command

***** End of block (LF)

Program entry in ISO-format

D

Parameter definitions

D-code	Function
D00	Allocation
D01	Addition
D02	Subtraction
D03	Multiplication
D04	Division
D05	Root
D06	Sine
D07	Cosine
D08	Root of sum of squares
D09	if equal, jump
D10	if not equal, jump
D11	if greater, jump
D12	if less, jump
D13	Angle (r · sin α and r · cos α)
D14	Error number
D15	Print (output of parameters)

G

Program entry in ISO-format

G-codes

G00	Linear interpolation, Cartesian, in rapid
G01	Linear interpolation, Cartesian
G02	Circular interpolation, Cartesian CW
G03	Circular interpolation, Cartesian CCW
G05	Circular interpolation, Cartesian, without direction data
G06	Circular interpolation, Cartesian, tangential
● **G07**	Single axis block
G10	Linear interpolation, polar, in rapid
G11	Linear interpolation, polar
G12	Circular interpolation, polar CW
G13	Circular interpolation, polar CCW
G15	Circular interpolation, polar, without direction data
G16	Circular interpolation, polar, tangential contour approach
● **G04**	Dwell
G28	Mirror image
G36	Spindle orientation
G37	Contour geometry
● **G39**	Designates program for call-up via G79
G54	Datum shift
G56	Pilot drill (with G37)
G57	Rough-out (with G37)
G58	Contour mill CW (with G37)
G59	Contour mill CCW (with G37)
G72	Scaling
G73	Co-ordinate system (plane) rotation
G74	Slot milling
G75	Rectangular pocket milling CW
G76	Rectangular pocket milling CCW
G77	Circular pocket milling CW
G78	Circular pocket milling CCW
G83	Pecking
G84	Tapping
● **G79**	Cycle call

Program entry in ISO-format

G

G-codes

G17	XY-plane designation, Tool axis Z
G18	ZX-plane designation, Tool axis Y
G19	YZ-plane designation, Tool axis X
G20	FOURTH tool axis
● **G24**	Chamfer with R
● **G25**	Rounding of corners with R
● **G26**	Tangential contour approach (run-on) with R
● **G27**	Tangential contour departure (run-off) with R
● **G29**	Transfer of last nominal position value as pole
G30	Blank form definition for graphics – min. point
G31	Blank form definition for graphics – max. point
● **G38**	Program run STOP
G40	No tool compensation (R0)
G41	Tool radius compensation to contour, offset left (RL)
G42	Tool radius compensation to contour, offset right (RR)
G43	Paraxial compensation extension (R+)
G44	Paraxial compensation reduction (R–)
G50	Erase/Edit protection (at program beginning)
● **G51**	subsequent tool number (with central tool memory)
● **G55**	Touch probe function
G70	Dimensioning in inches (at program start)
G71	Dimensioning in millimetres (at program start)
G90	Absolute dimensioning
G91	Incremental dimensioning
● **G98**	Assign label number
● **G99**	Tool definition

● = G-codes which are effective blockwise only

Auxiliary functions M

M-functions which affect program run
(List of standard auxiliary functions. These may only be changed by the machine manufacturer).

M-code	Function	Effective at block begin.	Effective at block end
M00	Program run stop Spindle stop Coolant off		●
M02	Program run stop Spindle stop Coolant off Return jump to first program block		●
M03	Spindle on, clockwise	●	
M04	Spindle on, counter-clockwise	●	
M05	Spindle stop		●
M06	Tool change Program run stop (depends on machine parameters entered) Spindle stop		●
M08	Coolant on	●	
M09	Coolant off		●
M13	Spindle on, clockwise Coolant on	●	
M14	Spindle on, counter-clockwise Coolant on	●	
M30	As per M02		●

Auxiliary functions M

M-functions which affect program run (List of standard auxiliary functions. These may only be changed by the machine manufacturer).

M-code	Function	Effective at block begin.	Effective at block end
M89	Free auxiliary function or	●	
M89	Cycle call, modally effective (depending on the machine parameters entered		●
M90	Constant path feed rate on external and internal corners	●	
M91	**Within a positioning block:** Workpiece zero datum is replaced by reference point	●	
M92	**Within a positioning block:** The set workpiece zero datum is replaced by a position which is defined by the machine tool builder using a machine parameter, (e.g. tool change position).	●	
M94	Rotary table axis display reduction to a value below 360°	●	
M95	Change approach behaviour for internal corners: no calculation of point of intersect.		●
M96	Change approach behaviour for external corners: insertion of a transitional circle		●
M97	Compensation of path intersection on external corners: point of intersection instead of transitional circle		●
M98	End comp. of path intersection blockwise		●
M99	Cycle call blockwise		●

Notes

Operating modes
"Manual",
"Electronic handwheel",
"Positioning with MDI",
"Automatic" and
"Program run in single block",
auxiliary modes, parameter programming and single axis machining via the axis keys are not dealt with in this PILOT.
Detailed information on these topics is provided in the TNC 355 Operating Manual.

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
D-8225 Traunreut · Tel. (08669) 31-0

Canada
HEIDENHAIN CORPORATION
Canadian Regional Office
1075 Meyerside Drive, Unit 5
Mississauga, Ontario L5T 1M3, Canada
☎ (416) 670-8900
FAX (416) 670-4426

USA
HEIDENHAIN CORPORATION
115 Commerce Drive
Schaumburg, IL 60173
☎ (312) 490-1191
FAX (312) 490-3931

U.K. and Ireland
HEIDENHAIN (G.B.) Limited
200 London Road, Burgess Hill
West Sussex RH15 9RD
☎ (0444) 247711
FAX (0444) 870024
Tx 877125

Singapore
HEIDENHAIN PACIFIC PTE LTD
2, Leng Kee Road No. 03-05
Thye Hong Centre
Singapore, 0315
☎ 4722222
FAX 4728916

Conception, Text and Layout: Schwenkert, Kastenhuber and Partners GmbH, Grünwald Munich, Bavaria
242 287 20 · 20 · 8/89 · H · Printed in West Germany · Rights of alteration reserved