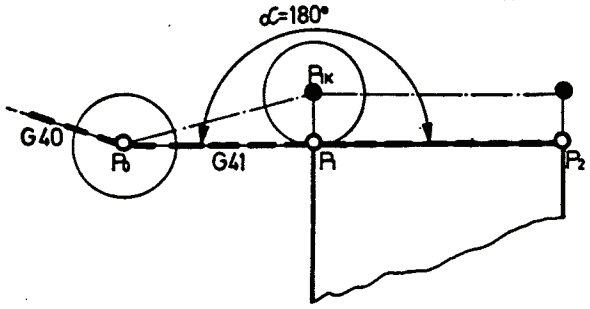
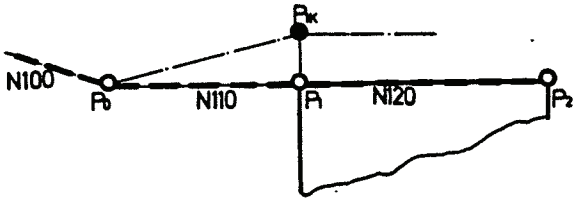
4.1.3 Neutral Approach $\alpha = 180°$

Definition:



The Angle α between programmed path $\overline{P_0P_1}$ and $\overline{P_1P_2}$ = 180°

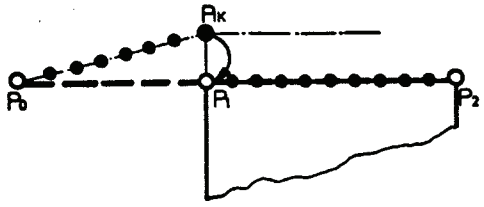
Programming:



Programmed path $\overline{P_0P_1}$.

→ P_0 N 100/G../XYZ$_{P0}$/
$\overline{P_0\ P_1}$ N 110/G../XYZ$_{P1}$/G41
$\overline{P_1\ P_2}$ N 120/G../XYZ$_{P2}$/

Position Read-Out:

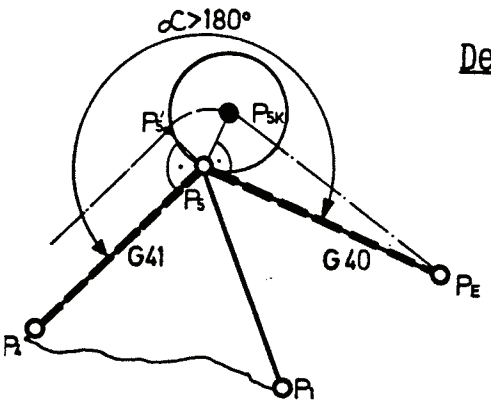


Center path of cutter $\overline{P_0P_{1K}}$
When P_{1K} is reached the read-out will jump.

- - - - Programmed path
——— Center path of cutter
●●● Digital position read-out
—o Programmed end of block P_X
—● End of block center point of cutter, P_{XK}

4.2 Disactivation of Cutter Radius Compensation with G40

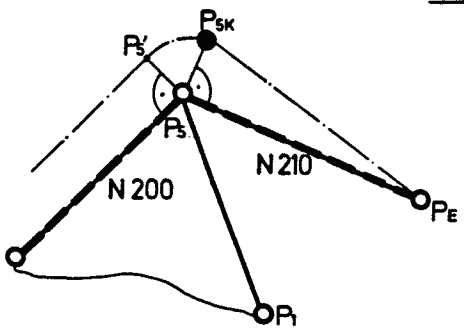
4.2.1 Exterior Corner: $\alpha > 180°$



Definition:

The angle α between the programmed paths $\overline{P_4P_5}$ and $\overline{P_5P_E}$ is larger 180°.

The point P_E cannot be approached directly from point P_5', otherwise contour damage.



Programming:

N.../G41

N 200/G../XYZP$_5$/

N 210/G../XYZP$_E$/G40

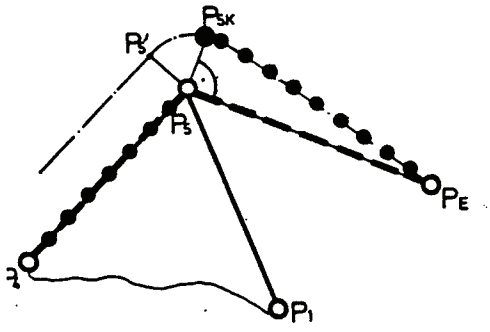
Block N 200: Cutter moves to normal line and the transition circle.

Block N 210: Approaching of P_E.

Read-Out:

Block N 200: Programmed path up to N 200.

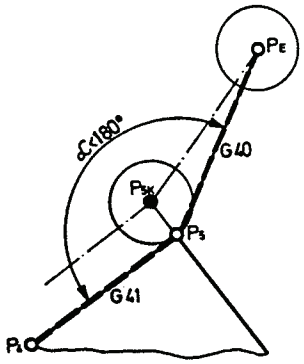
Block N 210: When the cutter center point has reached P_5' then the read-out jumps to P_{5K}.



- Programmed path
- Center path of cutter
- Digital position read-out
- Programmed end of block P_X
- End of block center point of cutter P_{XK}

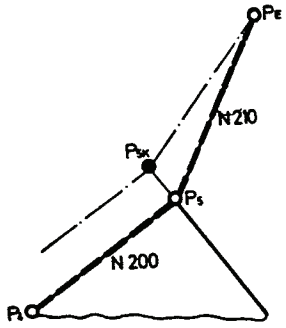
4.2.2 Interior Corner $\alpha < 180°$

Definition:



The angle α between the programmed paths $\overline{P_4P_5}$ and $\overline{P_5P_E}$ is smaller 180°.
The point P_E is approached directly since no contour damage occurs.

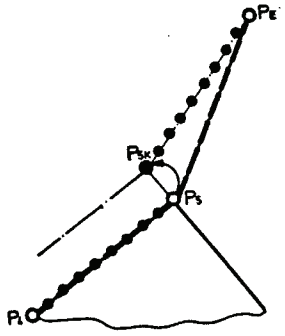
Programming:



N.../G41
N 200/G../XYZP5/
N 210/G../XYZPE/G40

P_E is approached directly.

Position Read-Out:

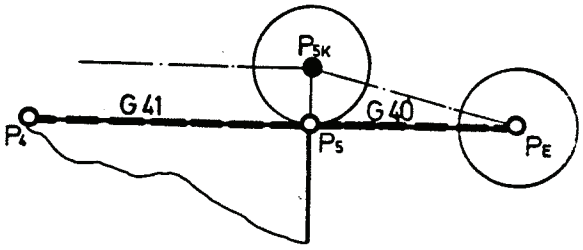


Block N 210: The position read-out jumps from P_5 to the center point path.

- - - - Programmed path
——— Center path of cutter
●●● Digital position read-out
—o Programmed end of block P_X
—● End of block center point of cutter P_{XK}

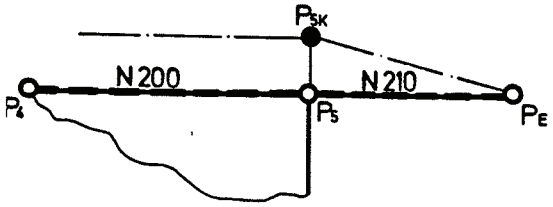
4.2.3 Neutral Disactivation: $\alpha = 180°$

Definition:



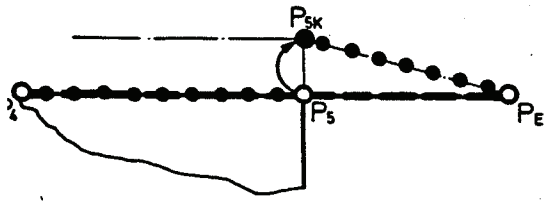
The angle α between $\overline{P_4P_5}$ and $\overline{P_5P_E}$ = 180°

Programming:



N..../G41
N 200/G../XYZP$_5$
N 210/G../XYZP$_E$/G40

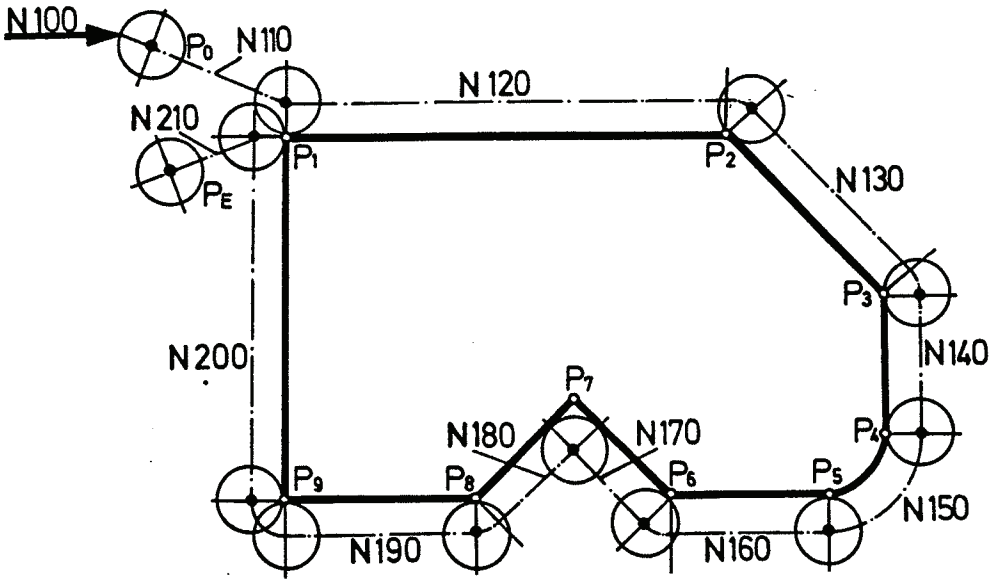
Position Read-Out:



Block N 210: The position read-out jumps to the center point path $\overline{P_{5K}P_E}$ after point 5.

- Programmed path
- Center path of cutter
- Digital position read-out
- Programmed end of block P_X
- End of block center point of cutter P_{XK}

4.3 The Cutter Path / Programmed Path with G41/G42



The picture shows the composition:
Programmed path and center point path
of the relative blocks.

Interior Corner:

The cutter moves to the equidistant points of intersection.

Exterior Corner:

The cutter traverses within the block also the transition movement up to the perpendicular line of the next contour.
(This can be seen in the program sequence when single block mode is on.)

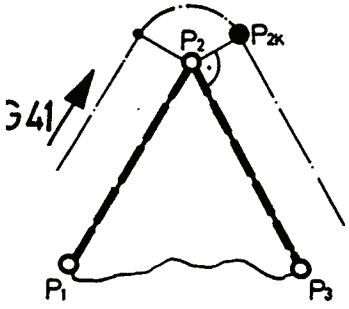
Program:

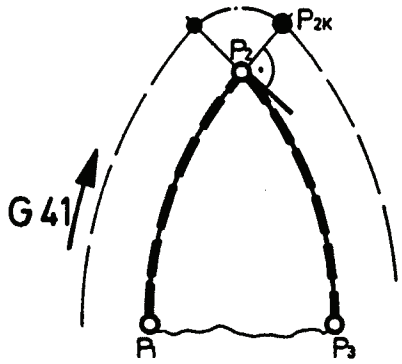
N 100	G00	X_0 Y_0 Z_0	(P_0)	(G40)
110	G01	X_1 Y_1 Z_1	(P_1)	G41
120			(P_2)	
130			(P_3)	
140			(P_4)	
150	G02	X_5 Y_5 Z_5 /I/(J)	(P_5)	
160	G01	X_6 Y_6 Z_6	(P_6)	
170			(P_7)	
180			(P_8)	
190			(P_9)	
200			(P_1)	
210	G00	X_E Y_E Z_E	(P_E)	G40

The Traverse (Movement) within the Block

Exterior Corners:

The Block finishes with a transition movement in the programmed point, up to the normal as to the programmed distance of the following block.

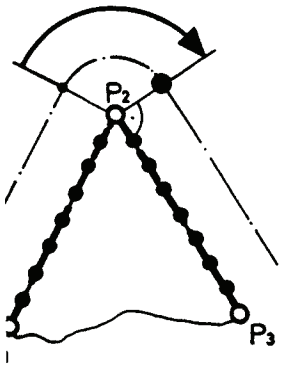




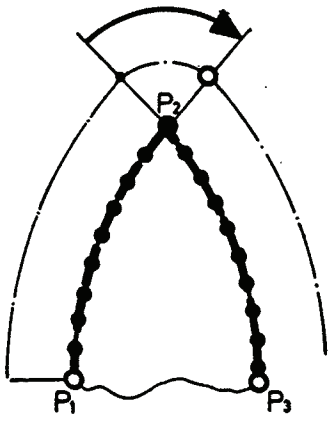
Position Read-Out:

During the transition radii the point P_2 is indicated.

P_2 Read-out

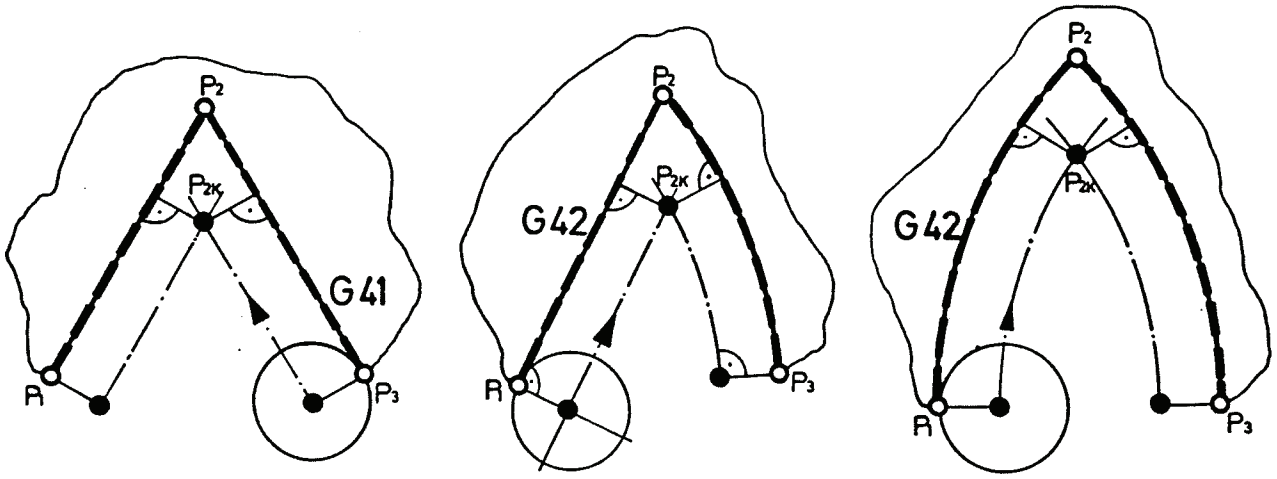


P_2 Read-out



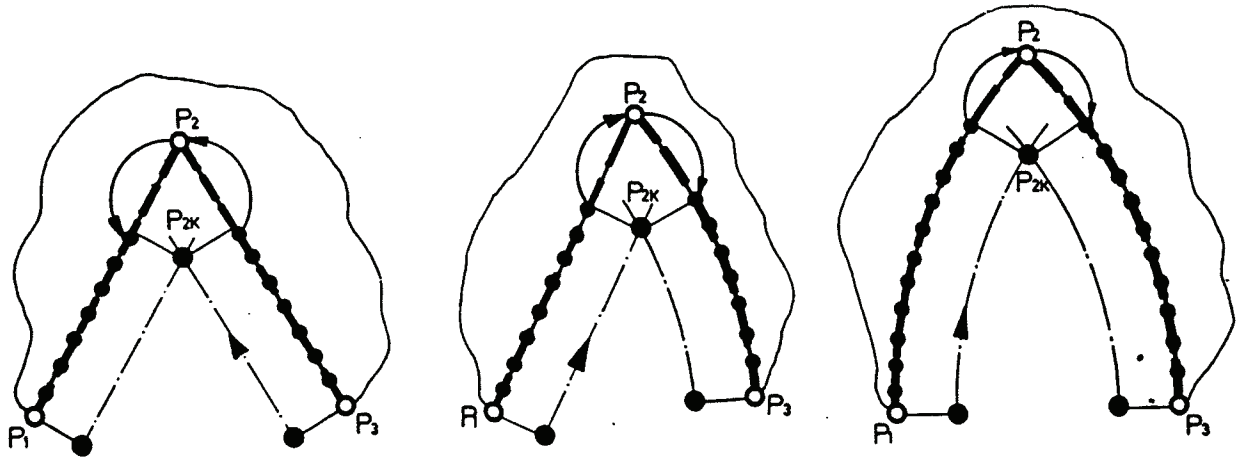
Interior corner:

The cutter moves up to the point of intersection of the equidistant P_{2K} (end of block and begin of block).



Position Read-Out:

At point of intersection the position read-out jumps to the programmed point P_2.



5. Further Syntax-Regulations, Special Cases, Exceptions, Alarms

Activation and Disactivation of cutter radius compensation G41/G42 active

Rule 1

G40 / G41 / G42 can only be activated or disactivated in conjunction with G00 or G01.

Rule 2

In the G00/G01 block a different value of X or Y or Z must be programmed in comparison to the previous block.

If only a value is changed or block G40 appears.

ad) Rule 1

Wrong examples:

Alarm A520

If in the activation/disactivation block another traverse instruction then G00/G01 is programmed.

Example:

G02,G03 in same block with G40, G41,G42.

N 100/G02/X..../Y..../Z..../I..../G41

Alarm A520

If in the following 5 blocks the first traverse instruction is no G00/G01 block.

Example:

N 100/G41
N 110/M03/S 1000
N 120/G94/F 120
N 130/G02/.....

Alarm A520

If the first G00/G01 block comes after 5 blocks

Right examples:

G40/G41/G42 in same blocks with G00/G01.

N 100/G00/X...Y...Z.../G41

In all following 5 blocks after Activation of G40,G41,G42.

N 100/G41
N 110/M03/S 1000
N 120/G94/F 120
N 130/G00/X.../Y.../Z...

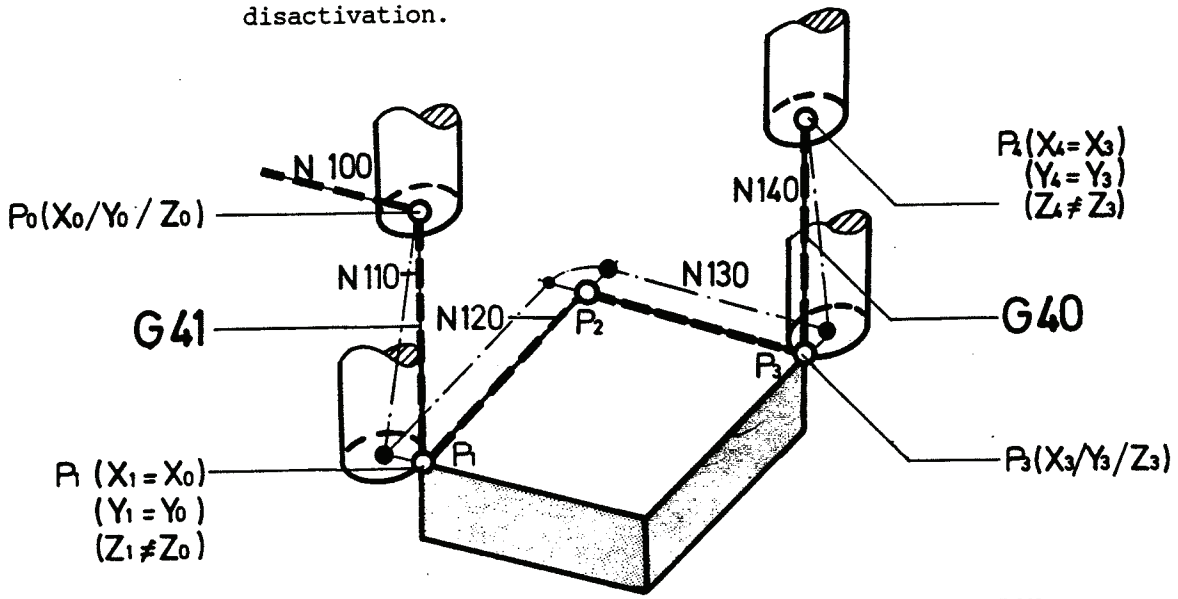
ad) Rule 2:

In the G00/G01 block a different value of X or Y or XY must be programmed in comparison to the previous block.

If only Z value is changed an alarm 520 appears.

Example: Alarm 520

No XY-value change with activation and disactivation.



ACTIVATION:

```
N ....G40
N 100/G00/P0(X0 ,Y0 ,Z0)
N 110/G01/P1(X1 =X0 /Y0/Z1)G41
```

Alarm A520

Because no change of X-or Y-or XY-values.

Notice:

If this program was possible, then the traverse would not only be in Z-direction but also in X-direction.

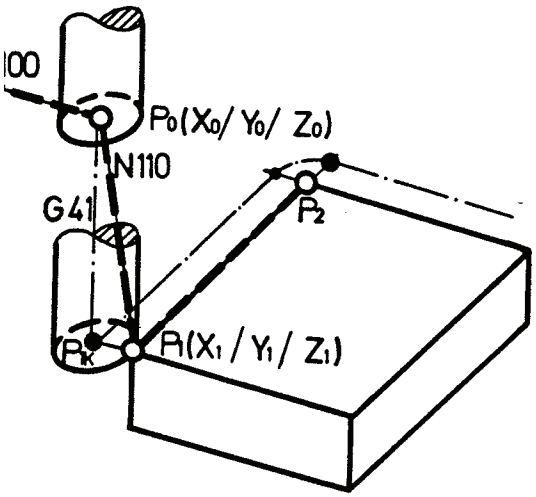
DISACTIVATION:

```
N ....G41
N 130/G01/P3(X3 /Y3 /Z3)/
N 140/G00/P4(X4 =X3 /Y4=Y3/Z4 /G40
```

Alarm because of same situation as above. No change in X,Y,XY values in the disactivation block.

ad) Rule 2:

ase compare the difference to the previous example with Alarm 520



P_0 = point in cutter axis

P_1 = circumferential point of cutter.

Programming:

N.... G40

N 100/G00/P_0 (X_0, Y_0, Z_0)

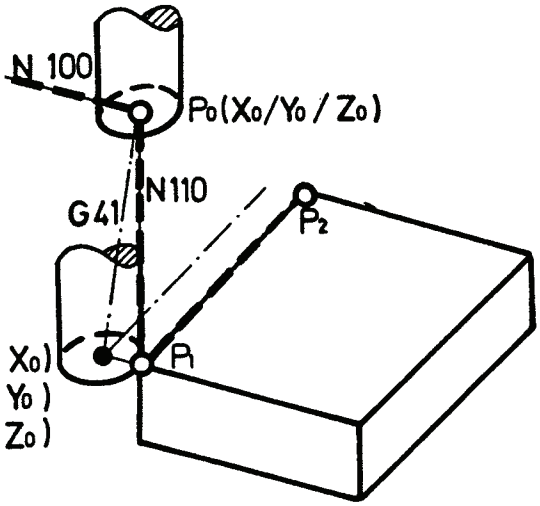
N 110/G01/P_1 ($X_1 \neq X_0$ /$Y_1 = Y_0$ /$Z_1 \neq Z_0$/G41

N 120/G01/P_2 .....

P_0 --> P_1

Only Z-traverse, no change in X-value

--> no alarm



Alarm 520

P_0 point in cutter axis

P_1 circumferential point of cutter.

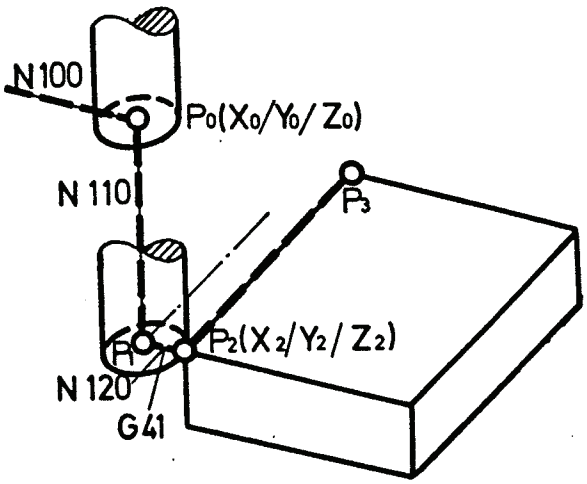
P_0 --> P_1

No change in XY value.

Alarm 520

ad) Rule 2

When activating resp. disactivating, there is no movement necessary only a value change in the plane of interpolation.



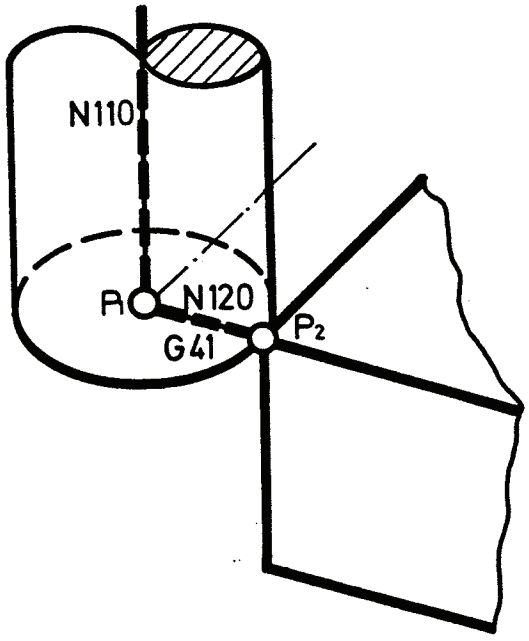
Example:

N G40
N 100/G00/P 0 (X_0, Y_0, Z_0)
N 110/G01/P 1 (X_1, Y_1, Z_1)
N 120/G01/P 2 ($X_2 \neq X_1$ / $Y_2 = Y_1$ / $Z_2 = Z_1$) G41

$X_2 \neq X_1$ --> value change thus no alarm

$Y_2 = Y_1$

$Z_2 = Z_1$

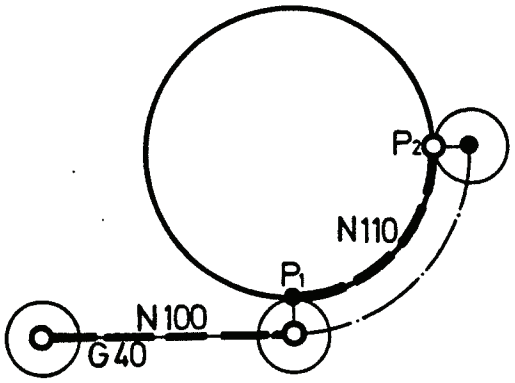


Block N 110: Cutter at point P_1 (G40 active)

Block N 120: - G41 activation
- No traverse movement but change of X value (value in plane of compensation)

Example: Activation and Disactivation in conjunction with G02/G03

The cutter radius compensation must not be activated or disactivated in one block with G02 or G03. --> Alarm 520



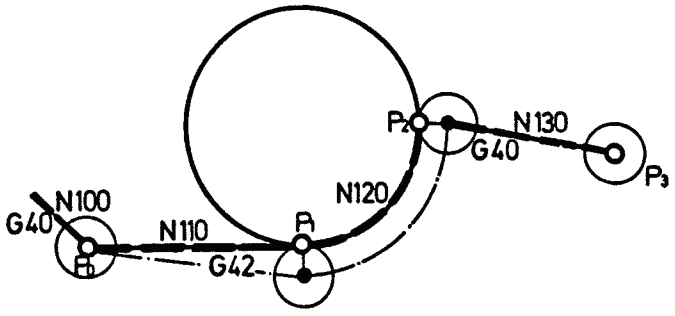
Alarm Situation

N 100 / G01 / XYZP$_1$ / G40
N 110 / G03 / XYZP$_2$ / G42 --> Alarm
N 120 / G40 --> Alarm

Activation in block with G03.
Disactivation without G00/G01 after G03.

Correct Possibilities of Activation/Disactivation

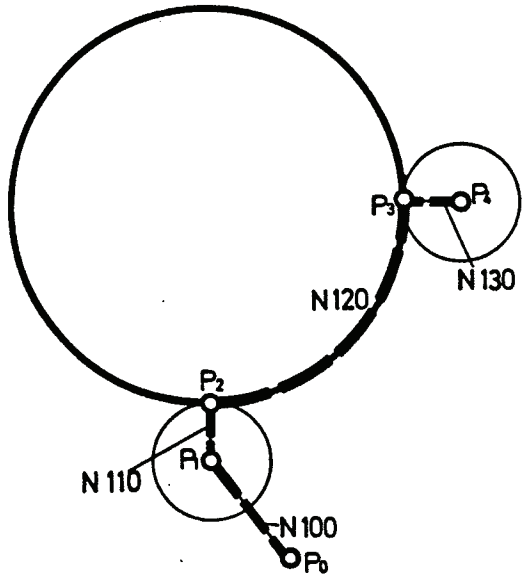
Possibility 1
Activation in approach-block (G00/G01)



N G40
N 100/G01/XYZP$_0$ /
N 110/G01/XYZP$_1$ /G42
N 120/G03/XYZP$_2$ /I.../J
N 130/G00/XYZP$_3$ /G40

Correct Activation/Disactivation with G02/G03

Possibility 2:



Block N 100: P_1 (= P_{2K}) is approached (G40)

Block N 110: Activation G42 - No traverse movement but change of values in the plane of compensation.

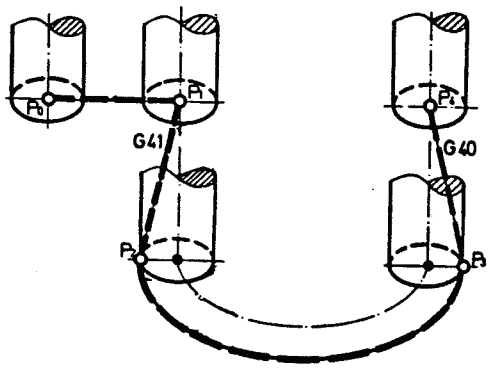
Block N 130: Disactivation - change of values in plane of compensation ($P_4 = P_{3K}$).

N.... G40
N 100/G01/P_1 (X_1,Y_1,Z_1)
N 110/G01/P_2 (X_2=X_1,$Y_2 \neq Y_1$,Z_2=Z_1)G42
N 120/G03/P_3 (X_3,Y_3,Z_3,I)
N 130/G01/P_4 ($X_4 \neq X_3$ /$Y_4 = Y_3$ /Z_4=Z_3)/G40

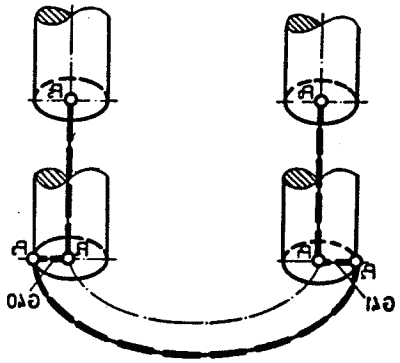
Examples with G02/G03

Assumption: Starting point of circle can only be approached in Z-direction.

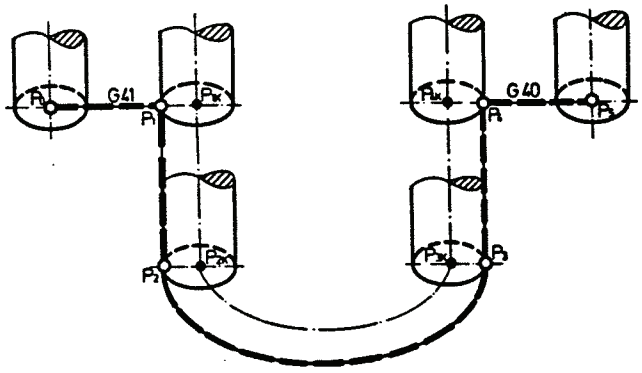
Possibility 1



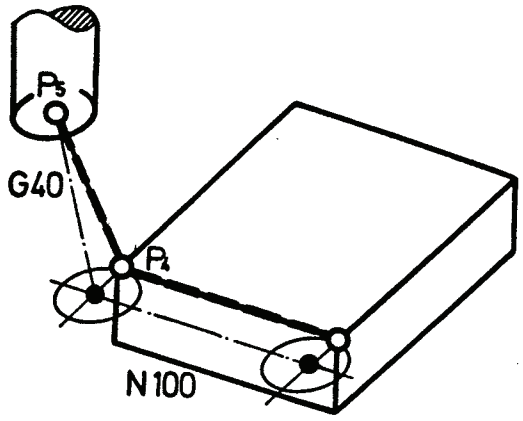
Possibility 2



Possibility 3



5.2 Tool Change active with G41/G42 --> Alarm 360



With a tool change (calling up a new T-address) the cutter radius compensation has to be disactivated. Otherwise alarm A360.

Alarm Situation A360

N G41
N 100 / G01 / P_4
N 110 / G00 / P_5.../T0303 A360

G41 is active with T-call.

Correct Programming

N G41
N 100 / G01 / P_4
N 110 / G00 / P_5 .... / G40 / T0303

or

N G41
N 100 / G01 / P_4
N 110 / G40
N 120 / G00/P_5 / T0303

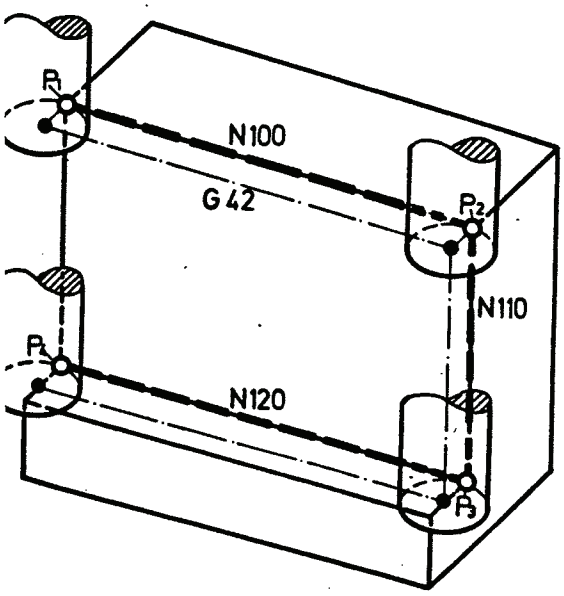
5.3 Direct Change from G41 to G42 --> Alarm 530

A direct change from G41 to G42 or vice versa causes alarm A530.

The cutter radius compensation has to be disactivated with G40 when changing from G41 to G42 and vice versa.
For activation and disactivation compare statements as per point 5.1.

Example: Alarm 530

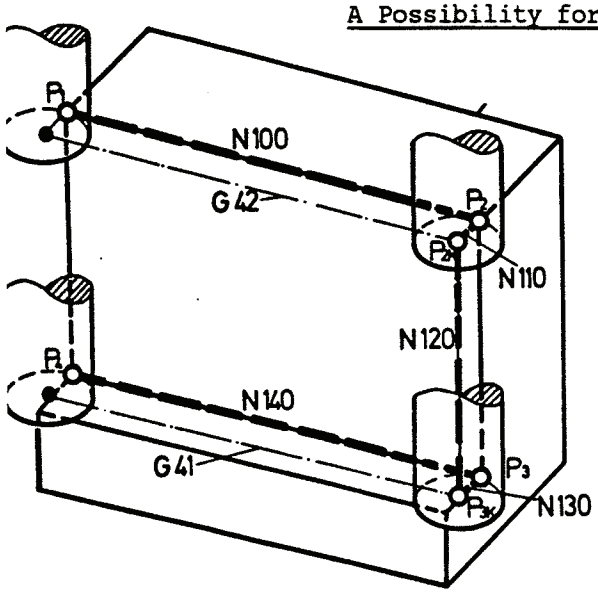
The depth of a shoulder is cut in various steps.



Alarm Situation

N G42
N 100 / G01 / $X_2/Y_2/Z_2$
N 110 / G01 / Z_3
N 120 / G01 / $X_4/Y_4/Z_4$ / G41
↑ Alarm 530

A Possibility for Correct Programming



N G42
N 100 / G01 / X_2 / Y_2 / Z_2 /
N 110 / G01 / X_{2K} / Y_{2K} / G40
N 120 / G01 / .....Z_{3K}
N 130 / G01 / X_3 / Y_3 / G41
N 140 / G01 / X_4 / Y_4

No traverse movements with block N 110 and N 130 but change in Y-value.

5.4 Number of Blocks when G41/G42 active (Alarm 510)

Minimum two blocks are necessary to program with XY-value change, otherwise alarm 51.

5.5 Disactivation of Cutter Radius Compensation (Alarm 510)

If the cutter radius compensation is disactivated with M30, then in block M30

1) G00 or G01 must be programmed
2) In block G00/G01 a change of the XY-value must be programmed.

5.6 Alarm 500

More than 5 Blocks without XY-Traverse Instruction

After activation of G41, G42 not more than five blocks without change of the XY-values must be programmed. Pure Z-traverse instructions do not count.

Example:

N 90 / G00 / X_0 /Y_0/Z_0
N 100 / G41
1 N 110 / M03 / S 1000
2 N 120 / M39 /
3 N 130 / G94 /
Alarm 500 4 N 140 / G00 / X_0 /Y_0/Z_1
5 N 150 / M08
6 N 160 / F 120

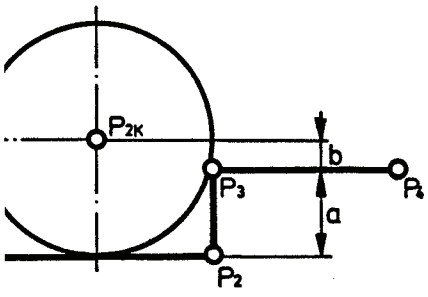
6. Geometry Alarm

The computer knows the contents of the previous and the following block in the plane of compensation. Thus it recognizes even contour damages which werde caused in a previous block or in a following block, and gives alarm.

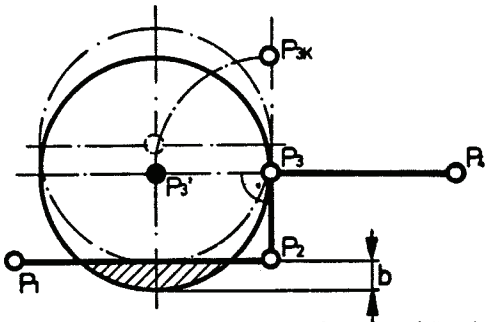
6.1 Shoulder smaller than Cutter Radius

Example 1: a < R

Approach of the contour normal at the target point results in contour damage in the previous block.



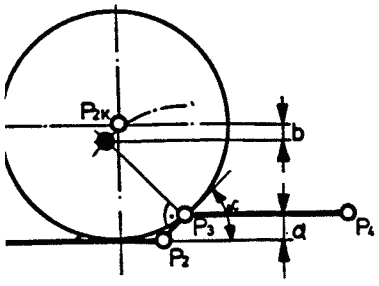
Cutter approaches P_{2K}.



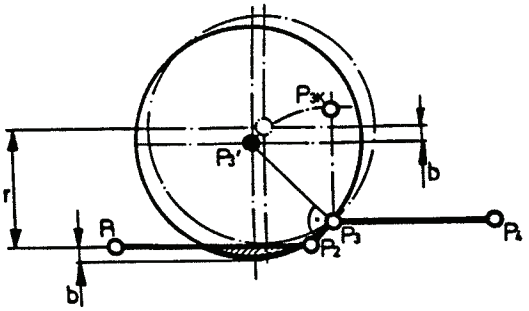
2: Cutter would approach P_3' (Contour normal to $\overline{P_2P_3}$ is $\overline{P_3P_3'}$) and would thereby damage the contour $\overline{P_1P_2}$.
Alarm 570

Example 2: a < R

a must be larger or equal to R (1-cosCC)



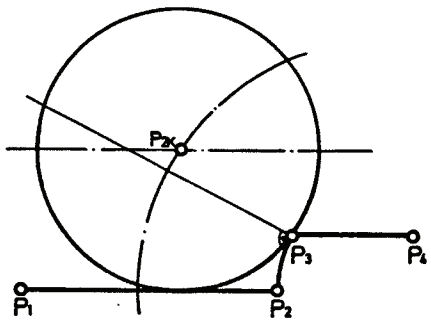
Cutter approaches P_{2K}.



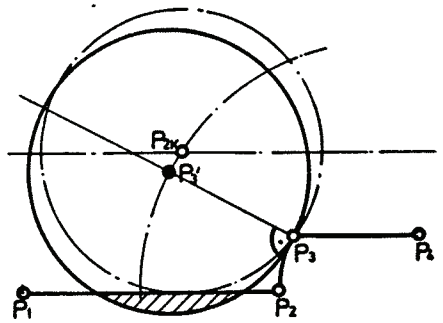
2: Cutter would approach normal $\overline{P_2P_3}$.
Contour damage in $\overline{P_1P_2}$ --> Alarm 570

6.2 Small Arcs in Comparison to Cutter Radius: ALARM 570

Example 1:



Cutter approaches intersection point of equidistants (P_{2K}).

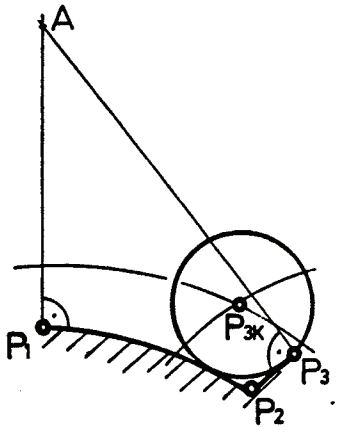


Cutter would approach point P_3' and would damage contour $\overline{P_1P_2}$.

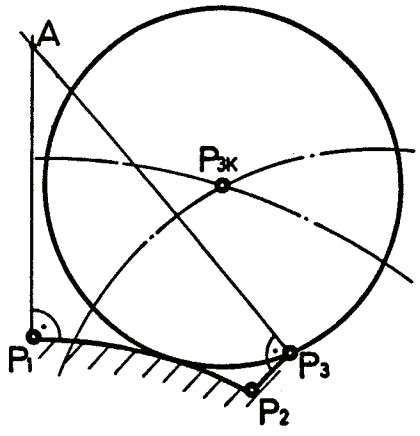
P_3' is on the normal line of the tangent in point 3.

Example 2:

Short Lengths of Arc compared to Cutter Radius.



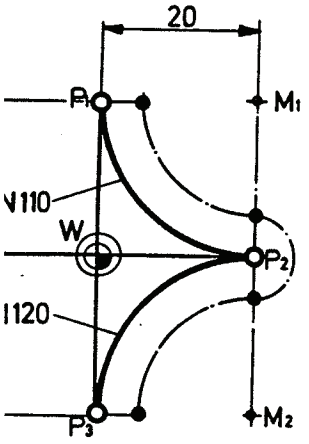
Intersection point of equidistants is positioned in the area $\overline{A\ P_1P_2P_3}$ --> NO ALARM 570.



Intersection point of equidistants is positioned outside the area $\overline{AP_1P_2P_3}$ --> ALARM 570.

6.3 Damages of Contour with Arcs

With unfavourable geometric configurations slight inaccuracies of center point data lead to contour damages.
Therefore the determining center point coordinate has to be put in as precise as possible, in marginal cases larger.



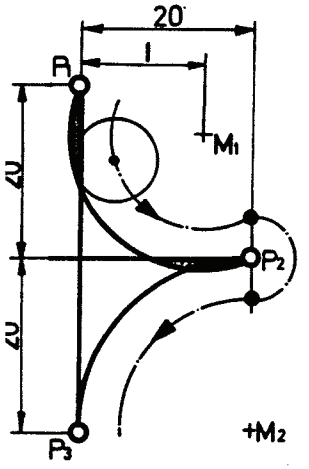
Example 1:

```
N..../G41
N 100/G01/X0/Y20./Z0
N 110/G03/X20./Y0/(Z0)/I20./J0
N 120/G03/X0/Y-20./(Z0)/I0/J-20.
N 130
```

The contour is cut correctly. Center point coordinate is precisely indicated.

Wrong center Point Values

G41

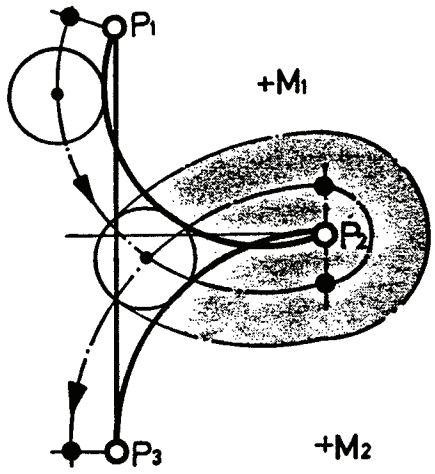


The description of the center point determines the size of the radius. If the center point coordinate is indicated too small, (e.g. 19,998 instead of 20,000 mm) it would lead to a contour damage. with P_1 - P_2.

Since the control knows the contents of the following block, Alarm 570 will appear.

If the same contour is programmed with G42, the interior corner would be transformed into an exterior one.
Severe contour damage!

G42

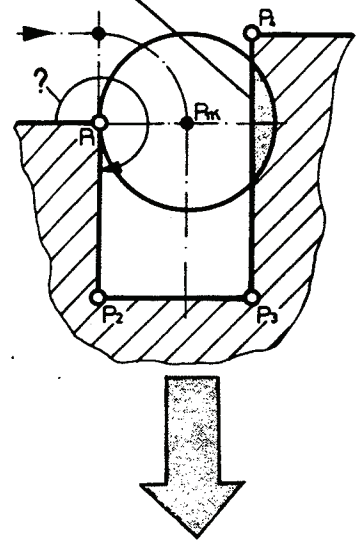


Therefore the determining center point coordinate has to be put in as precise as possible, in marginal cases larger.

6.4 Recognizeable and not recognizeable contour damages

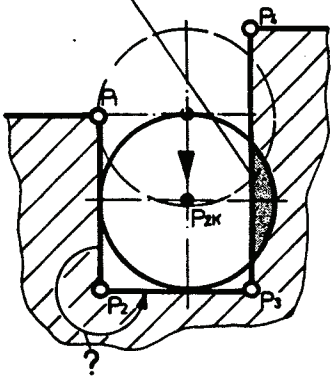
The cutter approaches the point of intersection of the equidistants.

Not recognizeable



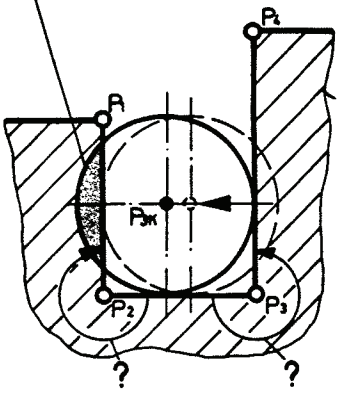
Cutter approaches point P_{1K}, contour damage $\overline{P_3P_4}$ not recognizeable, since two blocks ahead.

Not recognizeable



Cutter approaches point P_{2K}, contour damage $\overline{P_3P_4}$ not recognizeable.

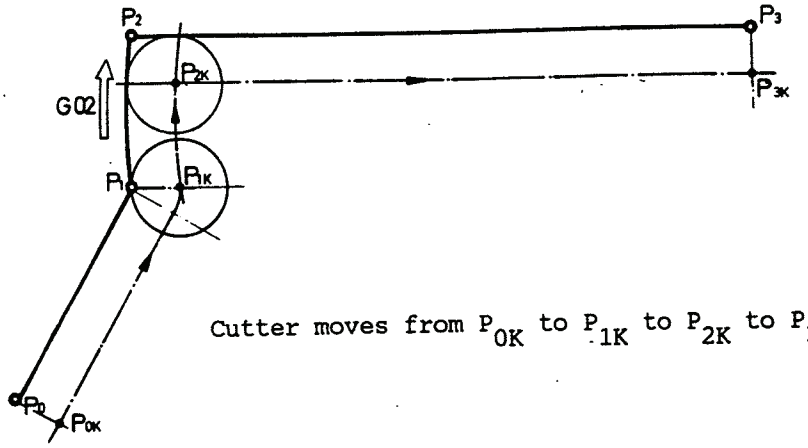
Recognizeable



Cutter would approach point P_{3K}, contour damages $\overline{P_1P_2}$ recognizeable therefore ALARM.

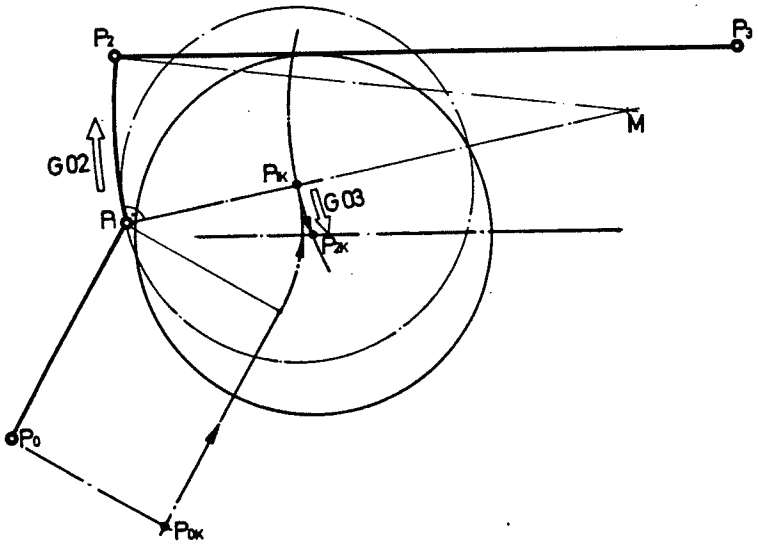
6.5 Different Cutter Radii - same Contour (theoretical alarm situations)

1. Cutter Path as shown in Illustration



Cutter moves from P_{0K} to P_{1K} to P_{2K} to P_{3K}.

2. Length of Arc short compared to cutter radius --> Change from G02 to G03

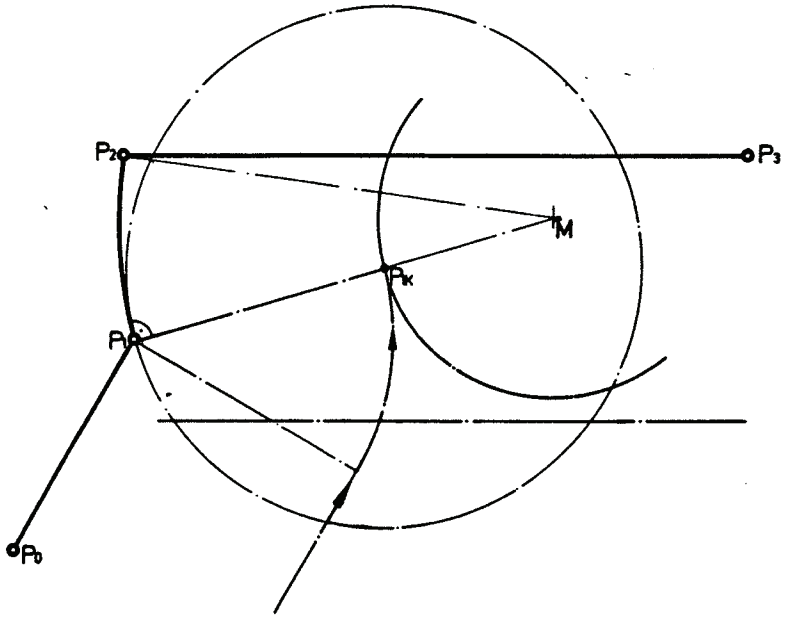


- Cutter moves from P_{0K} to P_{1K}.
  The cutter damages the contour P_2P_3 as this contour is more than one block ahead.

- P_{1K} to P_{2K}
  The cutter would go to the intersection point P_{2K} of equidistants. This would mean a change of direction of rotation from G02 to G03
  --> ALARM 180.

3. Length of Arc very short compared to Cutter Radius

--> No Intersection Point of Equidistants --> ALARM 570



G53 - G59 Zeropoint Offsets with Position Shift Offset

The offset values X,Y,Z are written into the position shift offset.

The coordinates system can be offset (displaced) from the machine zeropoint or from a selected zeropoint using G54, G55, G57, G58,G59.

With G53 the offset of G54, G55 is erased.

With G56 the offset of G57, G58, G59 is erased.

Structure

Group 3	G53	Erase G54, G55
	G54 ≙ 1 G55 ≙ 2	Call up position shift offsets 1, 2.
Group 5	G56	Erase G57, G58, G59
	G57 ≙ 3 G58 ≙ 4 G59 ≙ 5	Call up position shift offsets 3, 4, 5.

The PSO key on the control

PSO is the abbreviation of Position shift offset.

For details compare chapter Zeropoint Offset !

G70 Measurements in Inch
G71 Measurements in mm

Effect: At begin of program.

* Programs can be written in inch or mm.

* G70/G71 are self-retaining instructions with the same group.

Initial Status of Control EMCOTRONIC M1=G71

Even if no G71 is programmed all dimensions will be calculated as metric.

If you program G70 or G71, this has to stand in the first block of a main program (otherwise alarm 39)

Changing the Initial Status

The initial status (G71) can be changed by you in mode MON to G70.
Compare User Monitor (MON),chapter 8.

Note:

* G71 must be active during the loading of metric programs.

* G70 must be active for the loading of programs written in inches.

Boring patterns

Boring patterns

General explanations:

The program of a boring pattern consists of two parts:

- o definition section
- o execution section

Definition section:

In the definition section details are programmed on the position and number of elements of the boring pattern.

Notes:

- o The definition section must be before the execution section of the boring pattern in the NC program.
- o The definition section is self-holding. Therefore, as many program blocks as desired can lie between the definition and execution sections.
- o The definition section remains active until it is overwritten by a new definition.

Execution section:

The execution section consists of an execution call-up and description of the boring pattern element. All boring and milling cycles are admissible as boring pattern elements.

Note:

- o The boring pattern elements may only be programmed in Z(W)-direction (including all cycle parameters).

Classification of the boring patterns:

The boring patterns are classified into:

1. Circular boring patterns (G72/G73)
2. Rectangular boring patterns (G74/G75)

Preliminary explanations on parameter D_7:

Parameter D_7:

A definition must be programmed completely with all parameters. With parameter D_7 it is possible to take over parameters and parameter contents from a previous definition in the NC program.

Use of D_7:

By programming D_7 = 1 in the definition section, parameters and parameter contents not programmed are taken over into this definition section from a previous definition block.

Precondition:

A definition must already be made in the NC program.

Entry dimensions: [ ]

Notes:

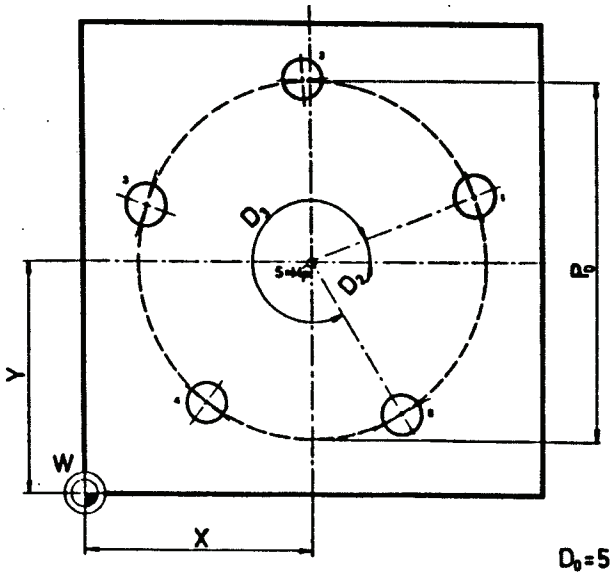
- With parameter D_7 only parameters can be taken over. The coordinate values X(U), Y(V), Z(W) cannot be taken over from previous definitions.
- If a parameter which is characterised with default option is programmed in a previous definition section, this parameter value is taken over when D_7 = 1 and not the default value of the parameter.
- If no D_7 or D_7 = 0 (default option) is programmed, no parameters are taken over.

1. Circular boring patterns (G72/G73)

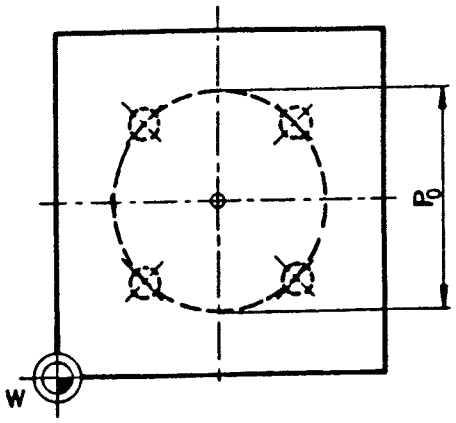
1.1 Definition section - circular boring pattern - G72

N 4	G72	X U ±43	Y V ±43	P_0 ±43	D_0 5	D_2 4	D_3 4	D_7 1

ADDRESSES	UNITS	DESCRIPTION	Default option: When the parameter is not programmed
X,U Y,V	[mm]	Coordinates of the boring pattern centre	
P_0	[mm]	Circle diameter	
D_0	[ ]	Number of boring pattern elements	
D_2	[Grad (°) x 10]	Initial angle D_2: 0 - 3600	0
D_3	[Grad (°) x 10]	Total angle: D_2: 0 - 3600	3600
D_7	[ ]	Take-over of parameters: D7 = 0 ... no take-over D7 = 1 ... take-over	0



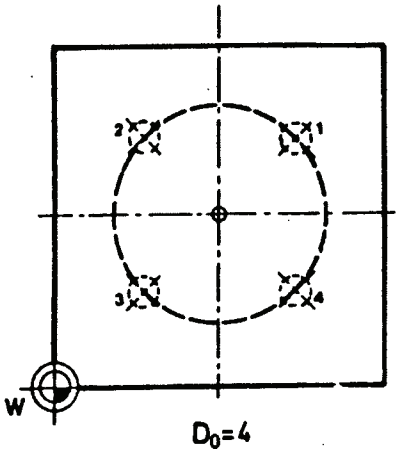
Explanations on the parameters of the definition section - circular boring pattern



Parameter P_0:

The diameter of the circular boring pattern (pitch circle diameter) is specified with parameter P_0.

Entry dimension: [mm]

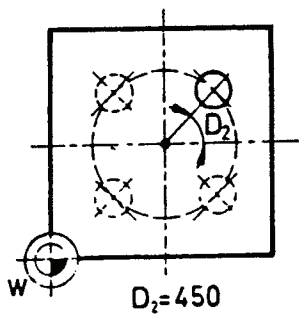


$D_0=4$

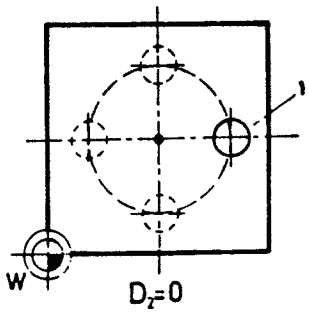
Parameter D_0:

The number of boring pattern elements is specified with parameter D_0.
All boring and milling cycles are admissible as boring pattern elements.

Entry dimension: [ ]



$D_2=450$



$D_2=0$

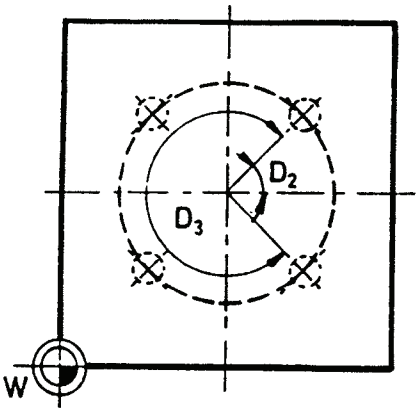
Parameter D_2:

The angle of the first boring pattern element is programmed with parameter D_2.

Entry range: 0 - 3600

Entry dimension: [degrees (°) x 10]

Note:
When no D_2 is programmed (default option), the first boring pattern element (1) is made on the horizontal symmetrical axis (0°).



Parameter D_3:

The total angle of the boring pattern is specified with parameter D_3 from the axis established with D_2.

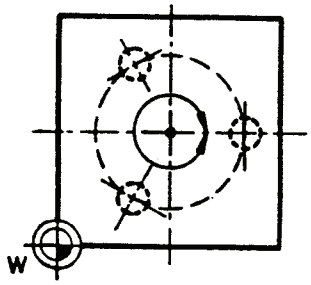
Entry range: 0 - 3600

Entry dimension: [degrees (°) x 10]

Note:
- When no D_3 is programmed (default option), the total angle of the boring pattern is 360°.
- The boring pattern elements are divided into equal parts between the total angle established with D_3.

Differentiation of the boring patterns:

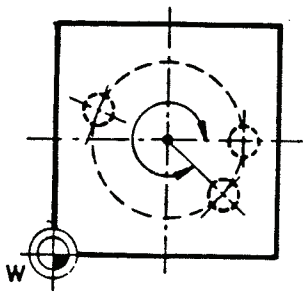
When subdividing the elements on the circle circumference, the following cases are to be differentiated:



D_3=3600

Closed boring pattern:

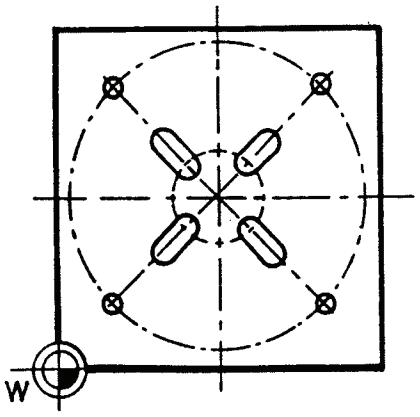
$D_3 = 3600$



D_3<3600

Open boring pattern:

$D_3 < 3600$



Parameter D_7:

With parameter D_7 parameters and parameter values can be taken over from previous definitions.
See preliminary explanations on parameter D_7.

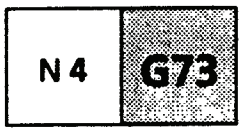
Entry dimension: [ ]

Example:

N..../....
N..../G72/X/Y/P_0/D_0/D_2/D_3
N..../....
N..../G73/G89/Z/.....
N..../....
N..../G72/X/Y/P_0/D_7 = 1
N..../.....
N..../G73/G82/Z/...
N...../.....

The parameters D_0, D_2 and D_3 established in the first definition block are taken over with D_7 = 1 in the second definition block.

1.2 Execution section - circular boring pattern G73

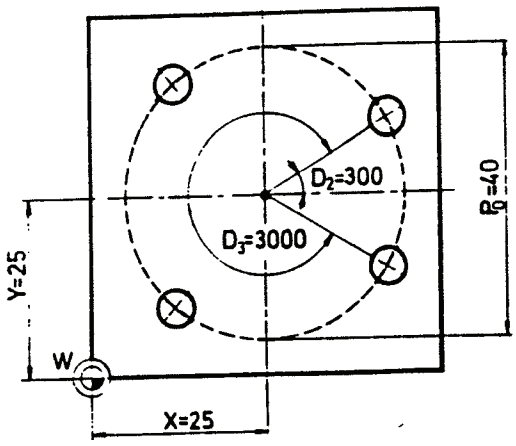


Machining run:

With the execution of the NC program machining commences at the boring pattern element which has been established in the definition section with the initial angle D_2.
Machining is continued in counterclockwise direction. After the machining of the last boring pattern element established by the total angle D_3, the tool remains stationary above the last pattern element. There is no return movement to the starting point.

Note:
The boring pattern elements may only be programmed in Z(W)-direction (including all cycle parameters).

Example G72/G73



boring depth: 10 mm

Absolute programming:

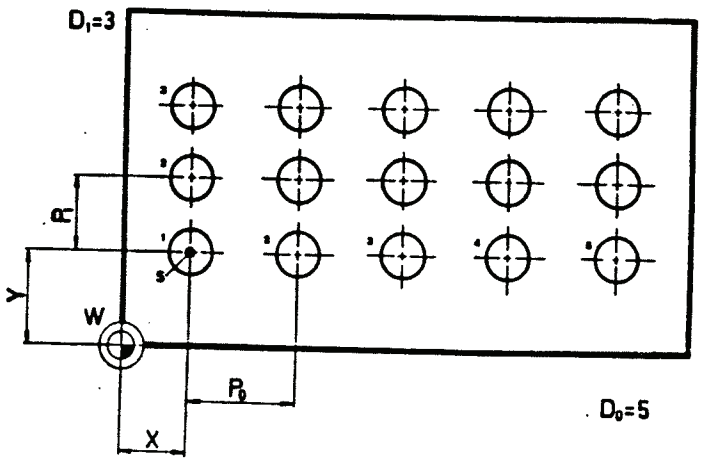
N..../....
N..../G00//Z5,000
N..../G72/X25,000/Y25,000/P_0 = 40,00(
D_0 = 4/D_2 = 300/D_3 = 3000/D_7 = 0
N..../....
N..../G73/G82/Z-10,000/D_4 = 50/F....

2. Rectangular boring patterns

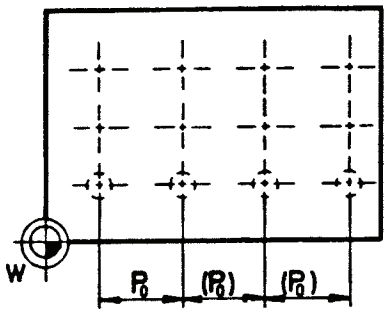
2.1 Definition section - rectangular boring patterns - G74

N 4	G74	X U ±43	Y V ±43	P_0 ±43	D_0 5	P_1 ±43	D_1 5	D_7 1

ADDRESSES	UNITS	DESCRIPTION	Default option: When the parameter is not programmed
X,U Y,V	[mm]	Coordinates of the first boring pattern element (pattern corner)	
P_0	[mm]	Horizontal distance	
D_0	[ ]	Horizontal number of boring pattern elements	
P_1	[mm]	Vertical distance	
D_1	[ ]	Number of vertical boring pattern elements	
D_7	[ ]	Take-over of parameters: D7 = 0 ... no take-over D7 = 1 ... take-over	0



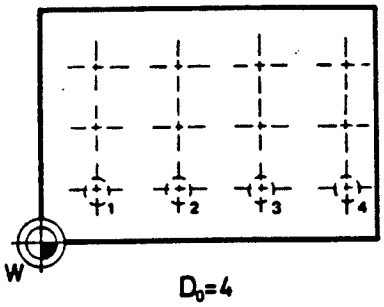
Explanations on the parameters of the definition section - rectangular boring pattern



Parameter P_0:

The same horizontal distance between the individual boring pattern elements is specified with parameter P_0.

Entry dimension: [mm]

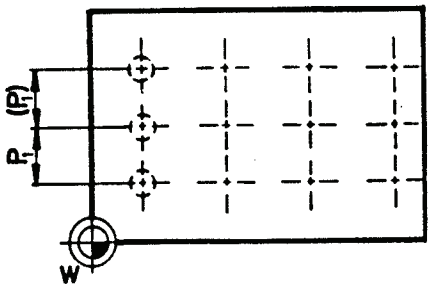


$D_0=4$

Parameter D_0:

The number of boring pattern elements in horizontal direction is specified with parameter D_0.

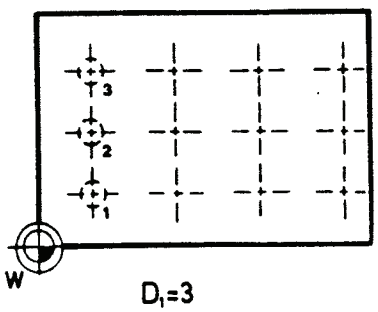
Entry dimension: []



Parameter P_1:

The same vertical distance between the individual boring pattern elements is specified with parameter P_1.

Entry dimension: [mm]



$D_1=3$

Parameter D_1:

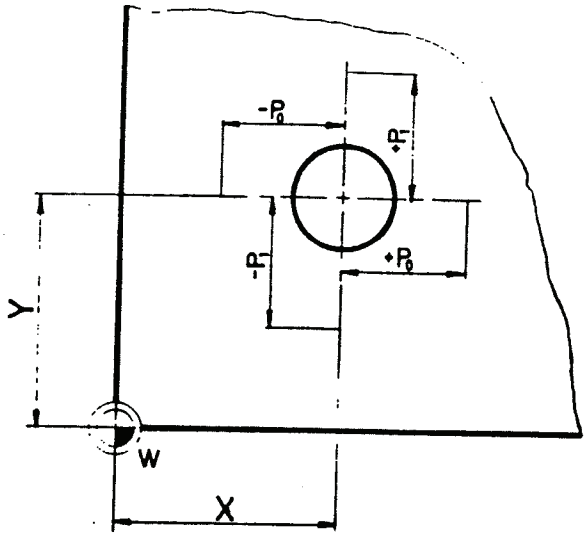
The number of boring pattern elements in vertical direction is specified with parameter D_1.

Entry dimension: []

Parameter D_7:

Parameters from previous definitions can be taken over with parameter D_7.
See preliminary explanations on parameter D_7.

Entry dimension: [ ]

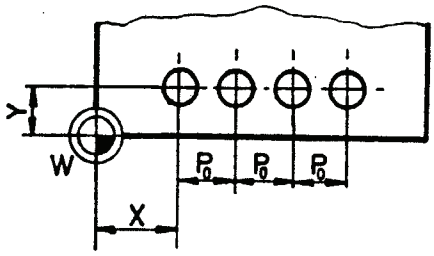


Position of the boring pattern elements:

The boring pattern corner is established with the aid of the addresses X(U) and Y(V) (1st boring pattern element).
The position of the other boring pattern elements is governed by the parameters P_0 and P_1 and their signs.

Special cases - rectangular boring patterns:

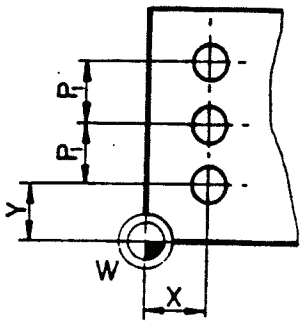
The rectangular boring pattern can be defined in one dimension with the selection of suitable parameters.



Example 1: horizontal boring pattern

N..../....
N..../G74/X/Y/P_0/D_0 = 4/D_1 = 1
N..../....

The parameter P_1 need not be programmed.

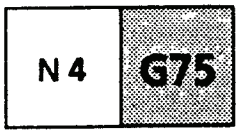


Example 2: vertical boring pattern

N..../....
N..../G74/X/Y/D_0 = 1/P_1/D_1 = 3
N..../....

The parameter P_0 need not be programmed.

2.2 Execution section - rectangular boring pattern G75

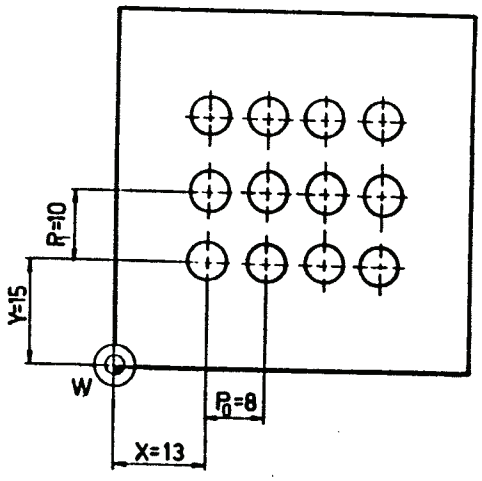


Machining run:

The execution commences with the boring pattern element which lies on the programmed pattern corner.
After the complete machining of the rectangular boring pattern the tool remains above the pattern element last executed. There is no return to the starting point.

Note:
The boring pattern elements may only be programmed in Z(W)-direction (including all cycle parameters).

Examples G74/G75



boring depth: 10 mm

Absolute programming:

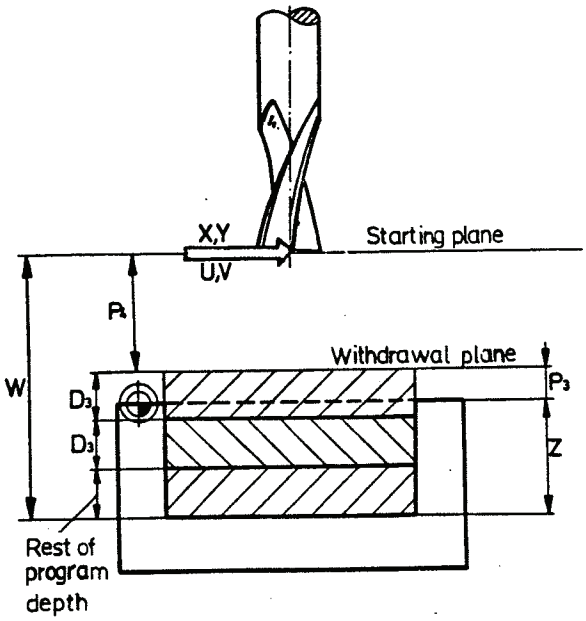
N..../....
N..../G00/Z5,000
N..../G74/X13,000/Y15,000/P_0 = 8,000/
P_1 = 10,000/D_0 = 4/D_1 = 3/D_7 = 0
N..../....
N..../G75/G82/Z-10,000/D_4 = 50/F....

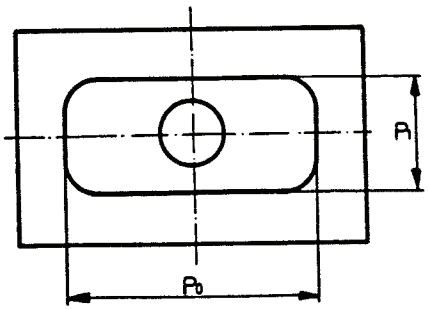
G87 Pocketing
Rectangular Pocket

N4	G87	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	$P_0$43	$P_1$43	$D_3$5	$D_5$2	$D_7$1

Sequence:

The cutter dips in in the middle of the pocket and has to be programmed in the XY-plane ,in the middle of the pocket, too.





Programming:

As with G81, but with additional parameters.

XY / UV [µm] Traverse path in XY plane

Z,W [mm] Depth of pocket

P_3/P_4 [mm] Withdrawal plane

Additional Parameters

P_0 [mm]	Size of pocket in X
P_1 [mm]	Size of pocket in Y
D_3 [µm]	In-feed in Z per cut with segmentation of cut
D_5=02	Normal milling
D_5=03	Counter milling
	(Initial status)
D_7	In-feed movement in Z
D_7=0	In-feed with G00
D_7=1	In-feed with half of the working feed (Initial status).

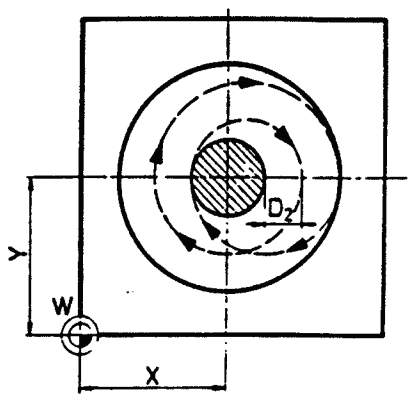
Absolute: N..../G87/X±mm/Y±mm/Z±mm/P_3±mm/P_0mm/P_1mm/D_3µm/D_5..../D_7..../F..../(G98/G99)

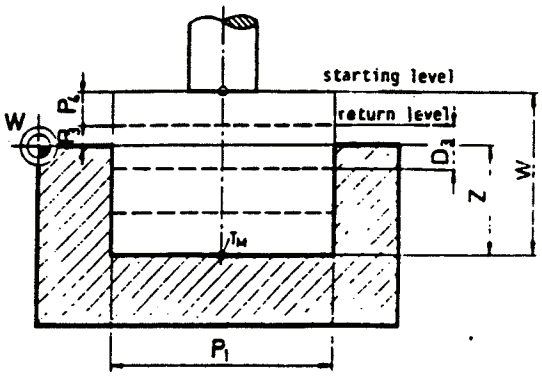
Incremental: N..../G87/U±mm/V±mm/W±mm/P_4±mm/P_0mm/P_1mm/D_3µm/D_5..../D_7..../F..../(G98/G99)

G88 - Circular pocket milling cycle

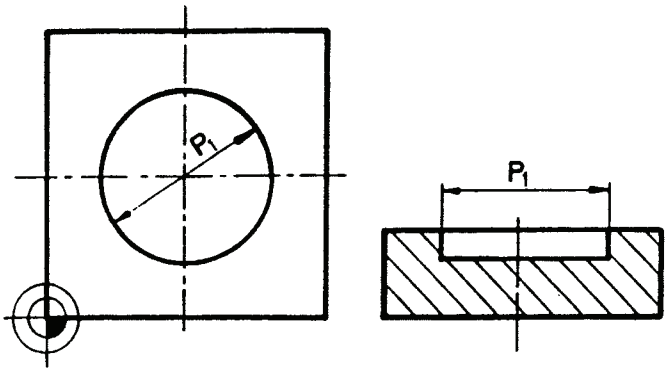
N4	G88	X/U ±43	Y/V ±43	Z/W ±43	P_1 43	P_3/P_4 ±43	D_2 5	D_3 5	D_4 1	D_5 1	D_7 1	F 4

ADDRESSES	UNIT	DESCRIPTION	Default option: When the parameter is not programmed
X,U Y,V Z,W	[mm]	Coordinates of the pocket centre at the deepest point T_M	
P_1	[mm]	Pocket diameter	
P_3	[mm]	Definition of the return level: Absolute dimension in Z-direction from the zero point level.	
P_4	[mm]	Definition of the return level: Incremental dimension in Z-direction from the starting level.	
D_2	[µm]	Horizontal infeed: D_2 mill diameter ALARM	D_2 = 1,7 x mill radius
D_3	[µm]	Vertical infeed	Machining takes place with one infeed
D_4		Finishing parameters: D_4 = 0 ... Full working feed at the outer radius of the circular pocket D_4 = 1 ... Half working feed at the outer radius of the circular pocket	D_4 = 0
D_5		Climb/opposed milling: D_5 = 2 ... climb milling D_5 = 3 ... opposed milling	D_5 = 3
D_7		Vertical feed: D_7 = 0 ... rapid feed D_7 = 1 ... half working speed	D_7 = 1
F	[mm/min] [mm/rev.]	Feed	





Explanations on the parameters:



Parameter P_1:
The pocket diameter is established with the parameter P_1.
An ALARM is given if a negative value is entered for P_1.
Entry dimension: (mm)

Parameter P_3:

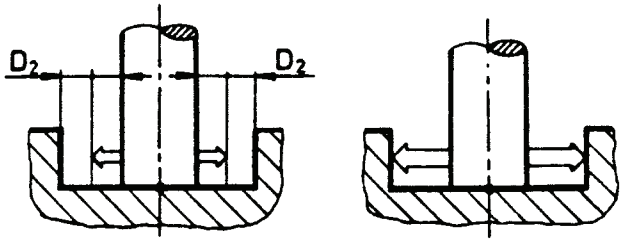
The withdrawal plane is defined with parameter P_3 with absolute programming (see G98, G99).

Entry dimension: (mm)

Parameter P_4:

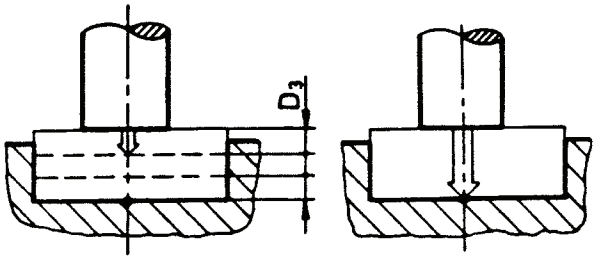
The withdrawal plane is defined with parameter P_4 with incremental programming (see G98, G99).

Entry dimension: (mm)



Parameter D_2:
- The horizontal infeed is specified with parameter D_2. The horizontal infeed must be smaller than the milling cutter diameter otherwise an ALARM is given.
- When the parameter D_2 is not programmed (default option), the control selects a value of 1.7 times the cutter radius as the horizontal infeed.

Entry dimension: (µm)



Parameter D_3:
- The vertical infeed is specified with parameter D_3.
- When the parameter D_3 is not programmed (default option), the infeed takes place in one operation.

Entry dimension: (µm)

Parameter D_4:
The following are established with D_4:

$D_4 = 1$ The outer radius of the circular pocket is milled at half the working feed.

$D_4 = 0$ The outer radius of the circular pocket is milled at full working feed.

When D_4 is not programmed (default option), the control assumes $D_4 = 0$.

Parameter D_5:
Climb or opposed milling is established with parameter D_5.

$D_5 = 2$ climb milling

$D_5 = 3$ opposed milling

When D_5 is not programmed (default option), the control assumes $D_5 = 3$ (opposed milling).

Parameter D_7:
The following are established with D_7:

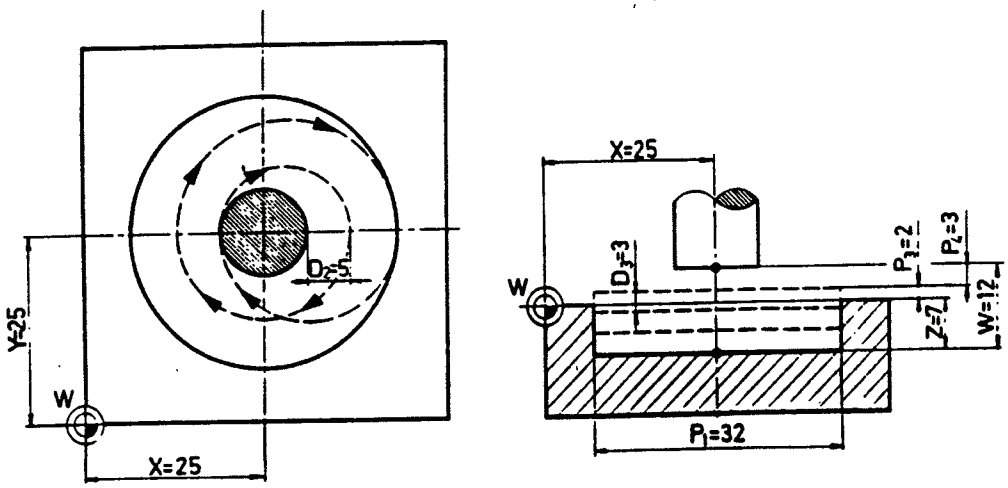
$D_7 = 0$ rapid feed with vertical infeed

$D_7 = 1$ vertical infeed at half working feed

When D_7 is not programmed (default option), the control assumes $D_7 = 1$.

Example G88:

mill diameter: 12 mm



Incremental programming:

N..../....
N..../G00/...
N..../G99
N..../G88/U00,000/V00,000/W-12,000/
P_1 = 32,000/P_4 = -3,000/D_2 = 5000/
D_3 = 3000/D_4 = 1/D_5 = 3/D_7 = 1/
F....
N..../....

Absolute programming:

N..../....
N..../G00/X25,000/Y25,000/Z5,000
N..../G99
N..../G88/X25,000/Y25,000/Z-7,000/
P_1 = 32,000/P_3 = 2,000/D_2 = 5000/
D_3 = 3000/D_4 = 1/D_5 = 3/D_7 = 1/
F....
N..../....

The Cycles G81 - G89

* Cycles are good for a simpler programming and are combinations of G00, G01, G04.
* According to the G-codes the movements are determined.
  Using the parameters P,D the movements can be specified.

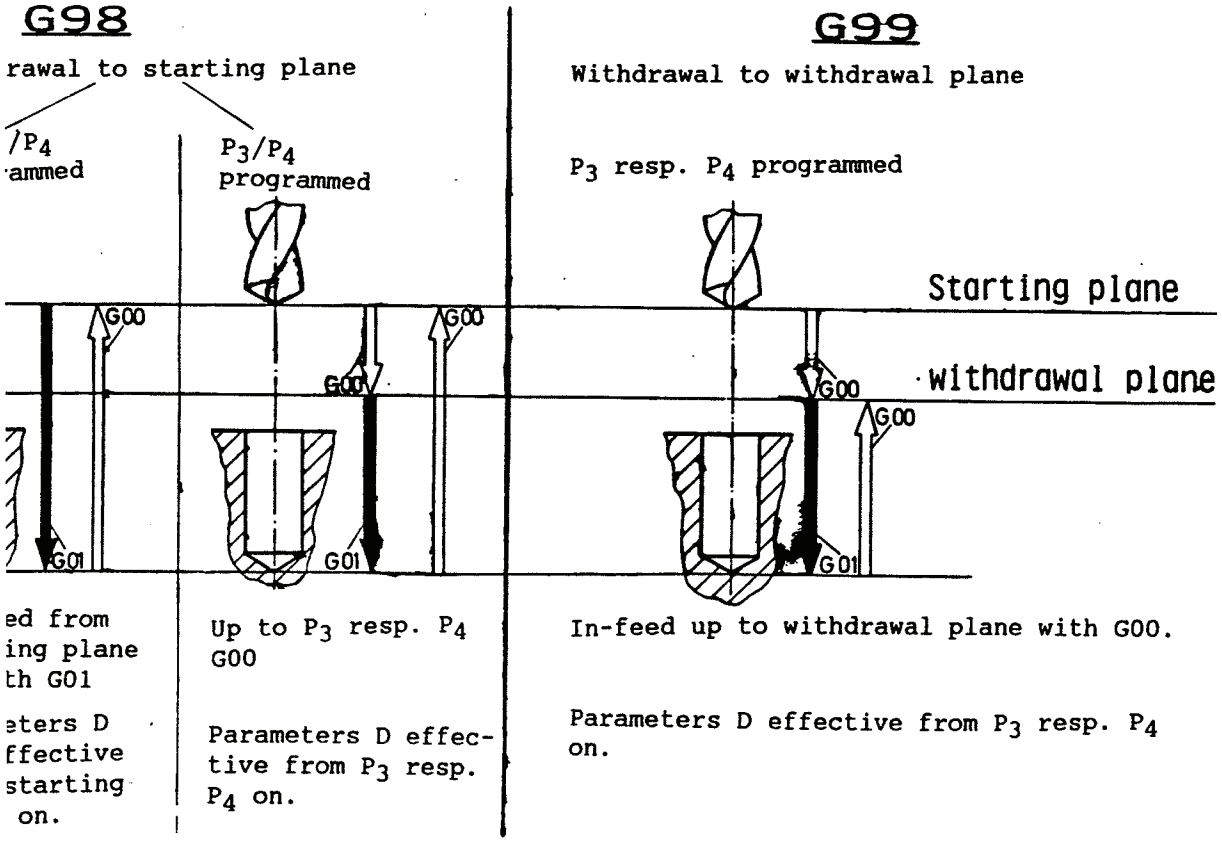
G98 Withdrawal to Starting Plane

G99 Withdrawal to Withdrawal Plane

With G99 being active:

With cycles a withdrawal plane can be defined using P_3 resp. P_4. This has practical reasons; find a detailed description on the following pages.

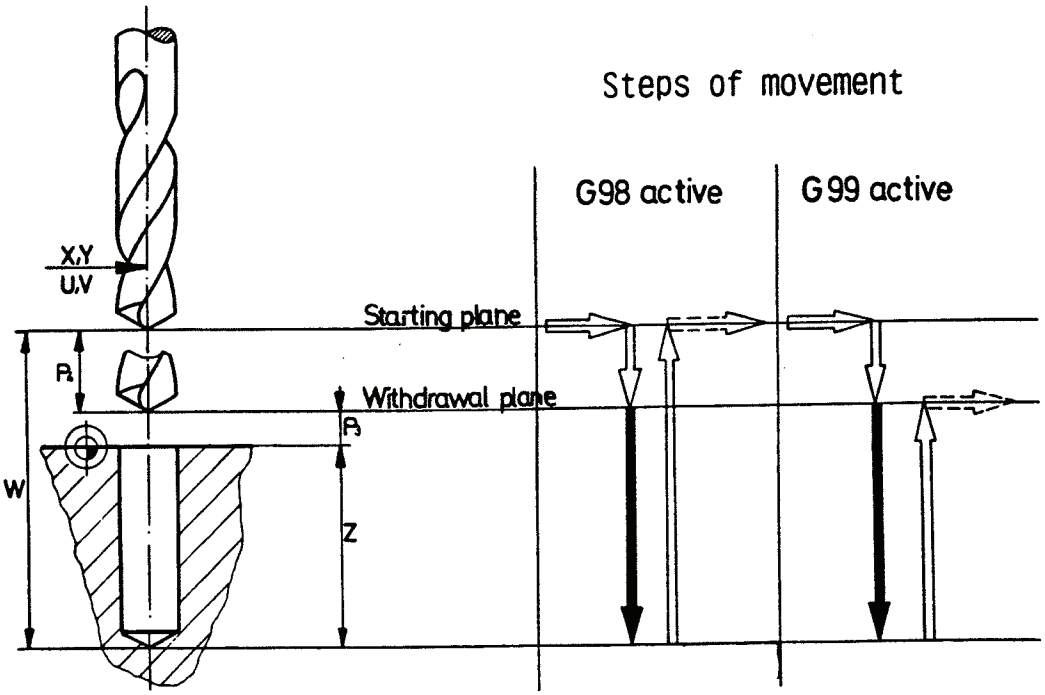
Parameter D_3 is only effective from the withdrawal plane onwards, after it's definition. (This is also valid with G98 active.)



G81 Drilling Cycle

N4	G81	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	F4

- G98 Withdrawal to starting plane
- G99 Withdrawal to withdrawal plane
- X,Y [mm] Traverse movement in XY-plane (U,V)[mm]
- Z(W) [mm] Drilling depth
- P_3 [mm] Absolute Z measurement (from zero plane on).
- P_4 [mm] Incremental Z measurement (from starting plane on).



G00 Movement

G01 Movement

Absolute: N..../G81/X±mm/Y±mm/Z±mm/P_3±mm/F..../(G98/G99)

Incremental: N..../G81/U±mm/V±mm/W±mm/P_4±mm/F..../(G98/G99)

Programming Alternatives with Cycles: Examples

Example 1: * G98 active
* No P_3/P_4 defined
* Tool positioned in XY-plane

Example 2: * G98 active
* P_3 resp. P_4 defined
* Tool positioned in XY-plane

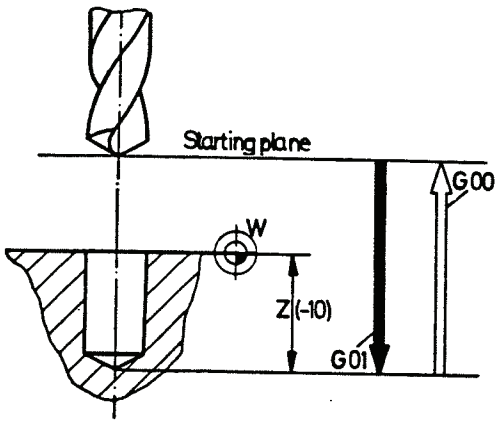
Example 3: * G99 active
* P_3 resp. P_4 defined
* Tool positioned in XY-plane

Example 4: Additional X,Y programming in cycle block.

G81/Example 1

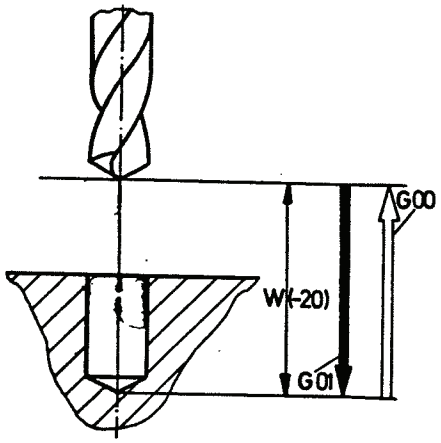
* G98 active
* No P_3/P_4 defined
* Tool positioned in XY-plane

Absolute



N..../G98
N 100/G81/Z-10./F....

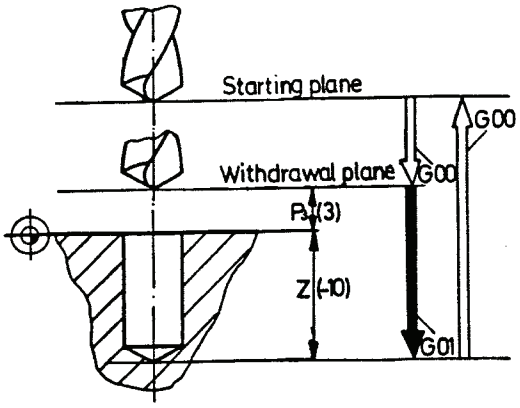
Incremental



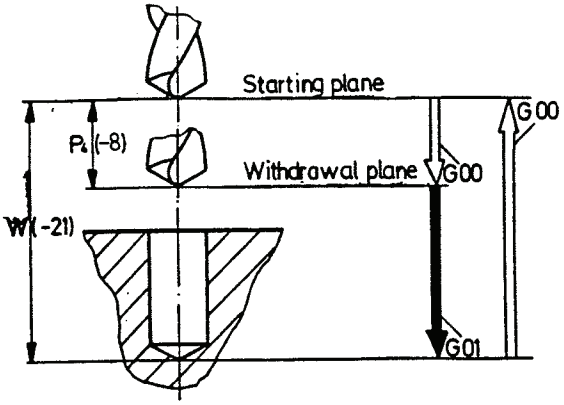
N..../G98
N 100/G81/ W-20./F....

G81 Example 2

* G98 active
* P_3 resp. P_4 defined
* Tool positioned in XY-plane



N..../G98
N 100/G81/Z-10./P3 3/F....

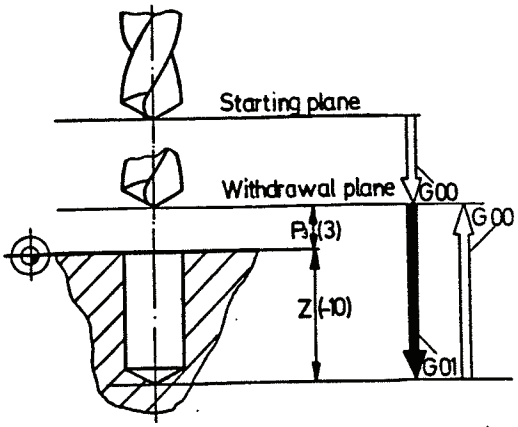


N..../G98
N 100/G81/W-21./P 4 -8./F....

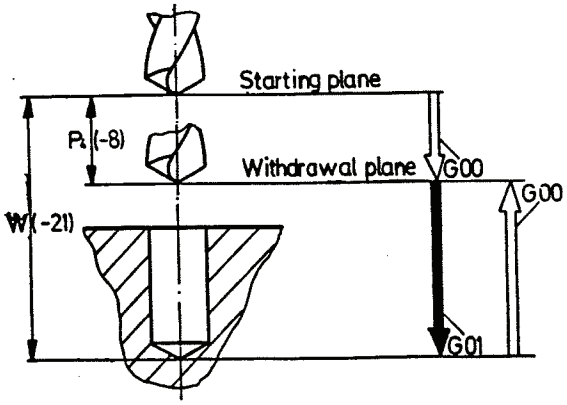
The withdrawal movement is done up to P_3 resp. P_4 with G00 withdrawal to starting plane.

G81 / Example 3

* G99 active
* Tool positioned in XY-plane
* P_3 resp. P_4 defined



N..../G99
N 100/G81/Z-10./P 3 3./F....



N..../G99
N 100/G81/W-21./P 4 -8./F....

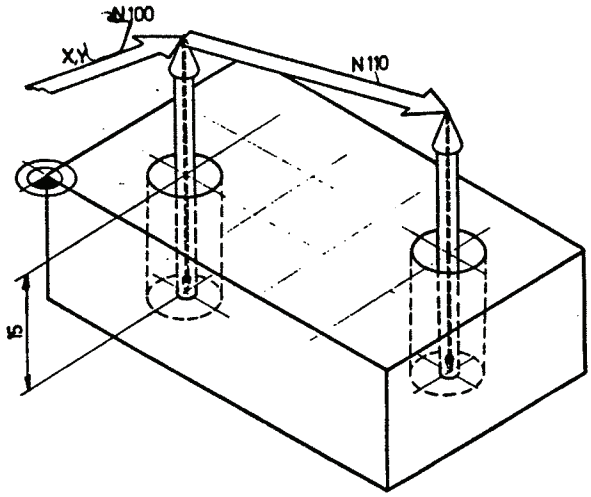
In-feed up to P_3 resp. P_4 with G00.
Withdrawal up to withdrawal plane as G99 is active.

G81/ Example 4

You can also program in the cycle block the traverse movement in the XY-plane until diving into Z-direction.

G81 / Example 4.1

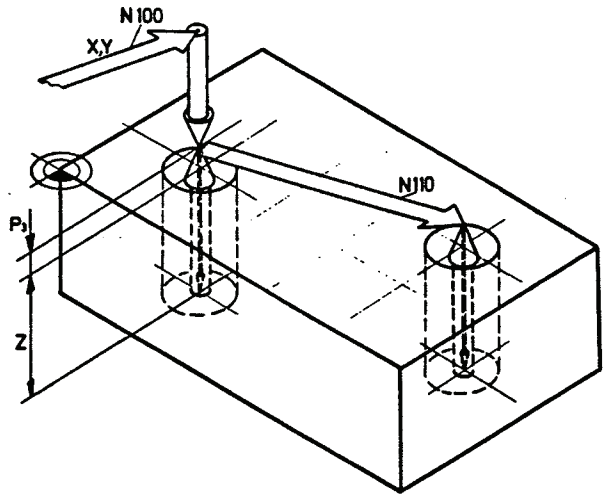
- G98 active
- No P_3/P_4 programmed



N..../G98
N 100/G81/X 10./Y 10./Z-15/F....
N 110 / /X 40./Y 20./

G81 / Example 4.2

G99 active



N ..../G99
N 100/G81/X 10./Y 10./Y 10./Z-15./$P_3$2,5/F....
N 110/ /X 40. /Y 20./

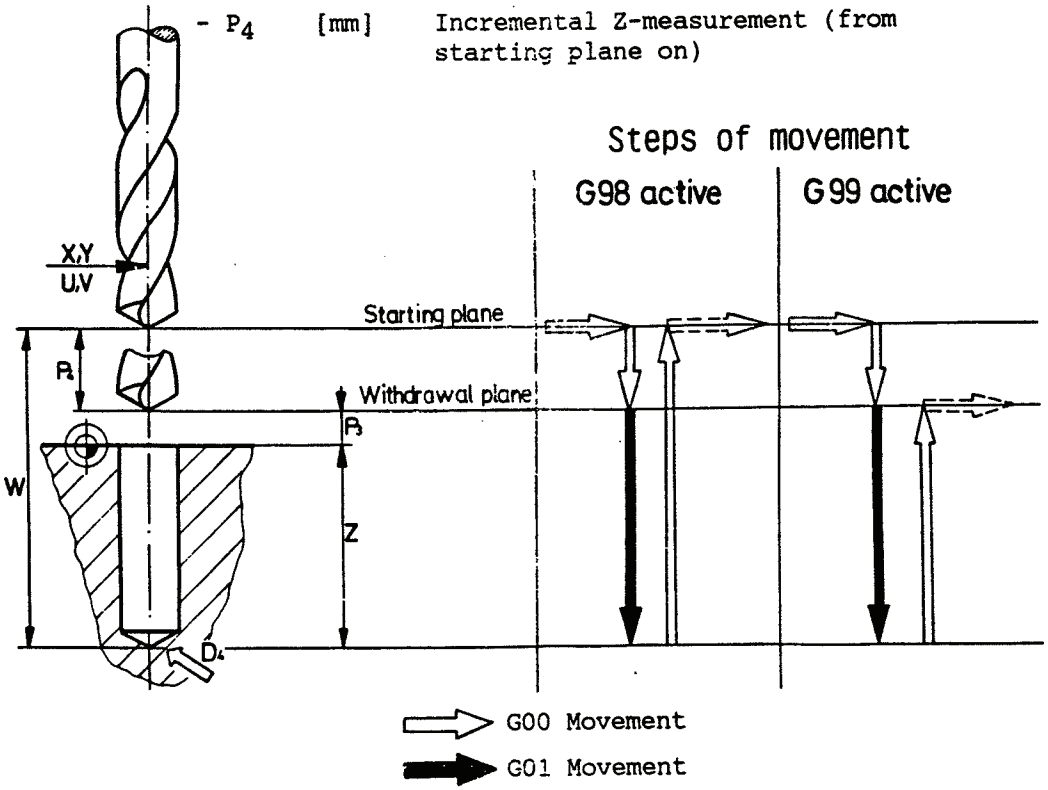
G82 Drilling Cycle with Dwell

N4	G82	X U ±43	Y V ±43	Z W ±43	$D_4$5	F4

Parameter and programming as with G81 but additionally D_4.

Under D_4 the Dwell at the End of the Z-Movement is programmed

- D_4 [$\frac{1}{10}$ sec.] Dwell
- G98 Withdrawal to starting plane
- G99 Withdrawal to withdrawal plane
- X,Y [mm] (U,V) [mm] Traverse movement in XY-plane
- Z(W) [mm] Drilling depth
- P_3 [mm] Absolute Z-measurement (from zero plane on)
- P_4 [mm] Incremental Z-measurement (from starting plane on)



Absolute: N..../G82/X±mm/Y±mm/Z±mm/$D_4$1/10 sec./F..../(G98/G99)

Incremental: N..../G82/U±mm/V±43/W±43/$D_4$1/10sec./F..../(G98/G99)

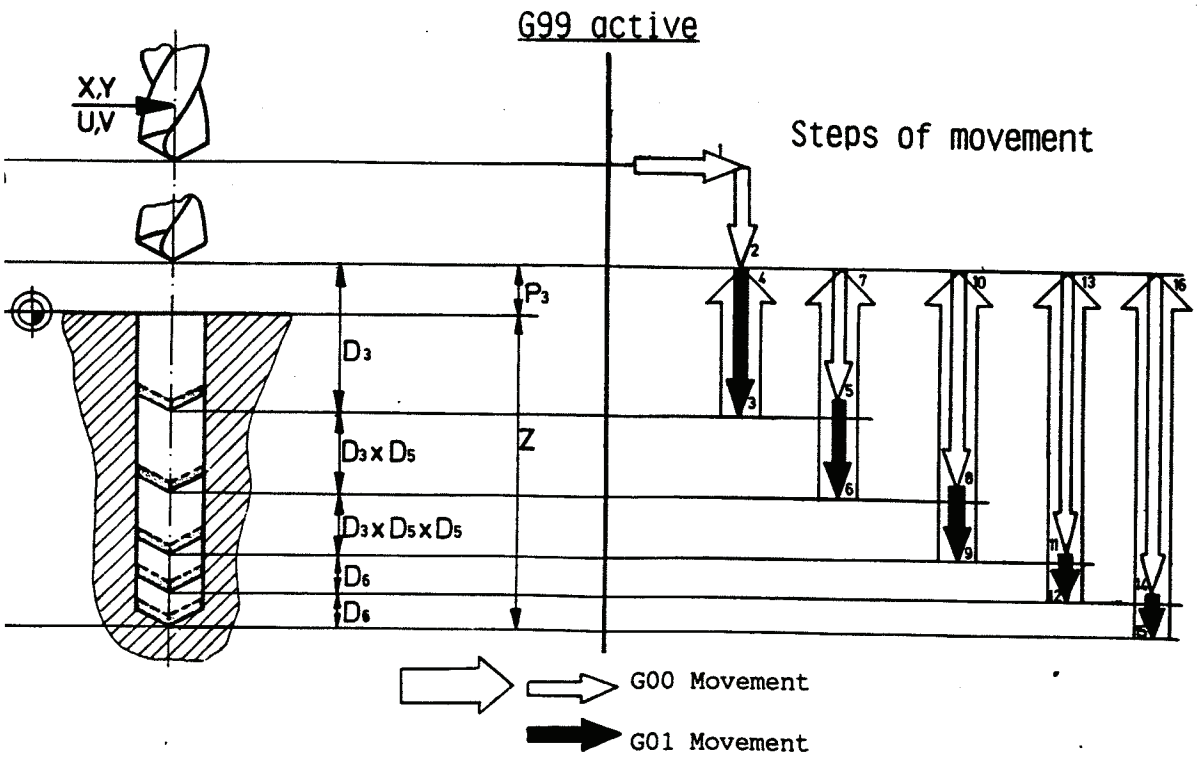
G83 Drilling Cycle

Deep Hole Drilling with Withdrawal

N4	G83	X ±43 U	Y ±43 V	Z ±43 W	P_3 ±43 P_4	$D_3$5	$D_5$3	$D_6$7	F4

Parameter and programming as with G81 but additionally $D_3/D_5/D_6$.

- G98 Withdrawal to starting plane
- G99 Withdrawal to withdrawal plane
- X,Y [mm] Traverse in XY-plane
  (U,V)[mm]
- Z(W)[mm] Drilling depth
- P_3 [mm] Absolute Z-measurement (from zero-plane on)
- P_4 [mm] Incremental Z-measurement (from starting plane on)
- D_3 [µm] Drilling depth for the first step
- D_4[$\frac{1}{10}$sec] Dwell
- D_5 [%] %value for decrease
- D_6 [µm] Minimum drilling depth



Absolute: N..../G83/X±mm/Y±mm/Z±mm/P_3±mm/D_3µm/D_5%/D_6µm/F..../(G98/G99)

Incremental: N..../G83/U±mm/V±mm/W±mm/P_4±mm/D_3µm/D_5%/D_6µm/F..../(G98/G99)

ad D_3: If no D_3 value is programmed, it will be drilled without withdrawal to full depth.

ad D_5: If no D_5 value is programmed, no decrease of D_3.

ad D_6: If no D_6 value is programmed, the minimum drilling depth, determined in the user monitor (MON) will be executed.

Note: If no P_3 resp. P_4 is programmed then D_3 will be calculated from the starting plane on.
If G98 was active in this example, the withdrawal would be up to the starting plane. (Movement .....4,7,10,13,16 up to starting plane).

G84 Tapping Cycle Threading

N4	G84	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	F5

Programming and Parameters as with G81.
Under F the thread pitch is programmed in µm.

- G98 Withdrawal to starting plane
- G99 Withdrawal to withdrawal plane
- X,Y [mm] (U,V)[mm] Traverse movement in XY-plane
- Z(W) [mm] Drilling depth
- P_3 [mm] Absolute Z-measurement (from zero plane on)
- P_4 [mm] Incremental Z-measurement (from starting plane on)
- F [mm] Thread pitch

ience:

ıreading drill moves to programmed ıd point.

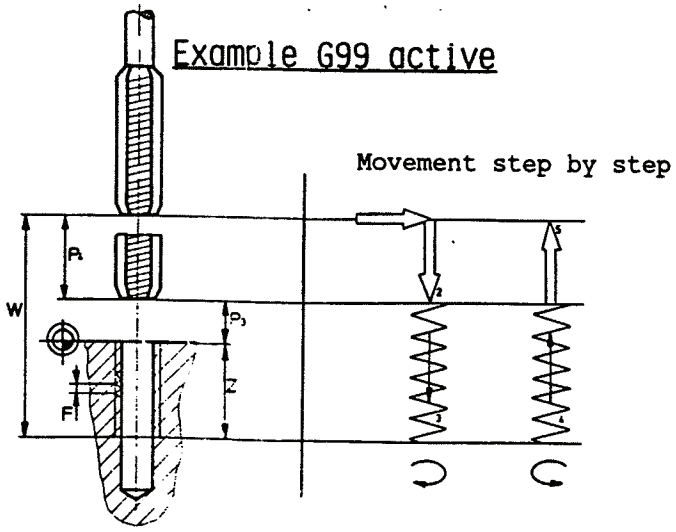
irection of revolution is switched ver.

›ving out to starting resp. with ·awal plane.

vitching to programmed direction of evolution.

Programming:

As with G81
Right hand thread M03
Left hand thread M04
F (µm) thread pitch.



Absolute: N..../G84/X±mm/Y±mm/Z±mm/P_3±mm/Fµm/(G98/G99)

Incremental: N..../G84/U±mm/W±mm/P_4±mm/Fµm/(G98/G99)

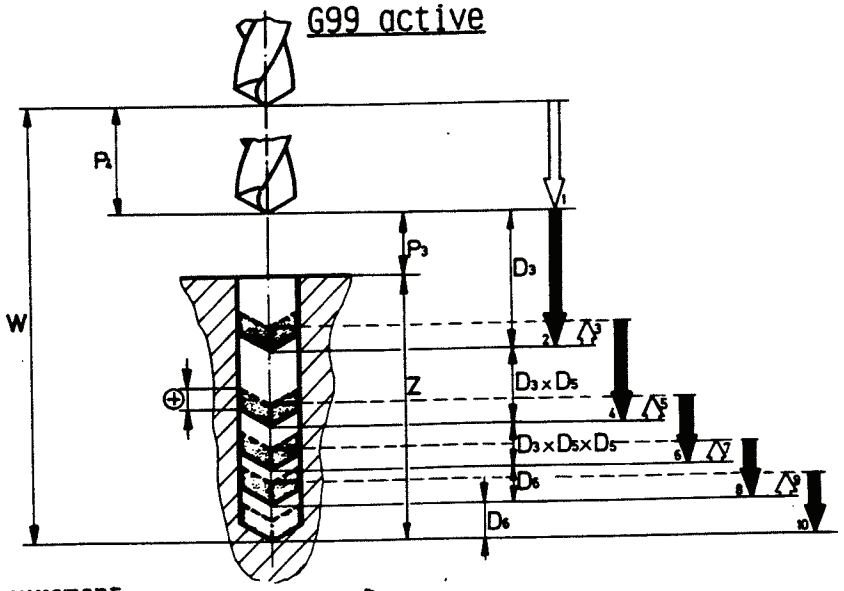
G86 Deep Hole Drilling with Chip Break

N4	G86	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	$D_3$5	$D_5$3	$D_6$7	F4

- G98 Withdrawal to starting plane
- G99 Withdrawal to withdrawal plane
- X,Y [mm] (U,V)[mm] Traverse movement in XY-plane
- Z(W)[mm] Drilling depth
- P_3 [mm] Absolute Z-measurement (from zero plane on)
- P_4 [mm] Incremental Z-measurement (from starting plane on)
- D_3 [µm] Drilling depth for first step
- D_4 [$\frac{1}{10}$sec] Dwell
- D_5 [%] % Value for decrease
- D_6 [µm] Minimum drilling depth

Sequence:

After each single drilling step (D_3, D_3 x D_5 etc.) the drill lifts off by a certain measurement. This measurement is fixed in the user monitor under D_3.



⊕ Withdrawal measurement is fixed in MON.

G00 Movement

G01 Movement

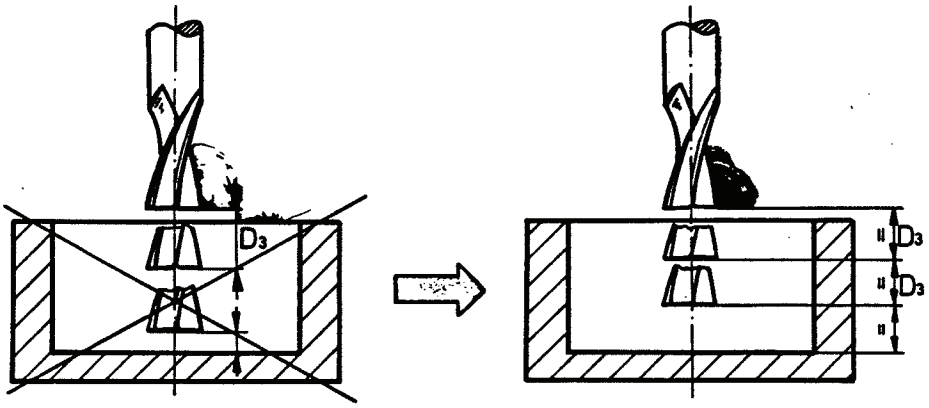
Absolute: N..../G86/X±mm/Y±mm/Z±mm/P_3±mm/D_3µm/D_5%/D_6µm/F..../(G98/G99)

Incremental: N..../G86/U±mm/V±mm/W±mm/P_4±mm/D_3µm/D_5%/D_6µm/F..../(G98/G99)

Parameter D_3

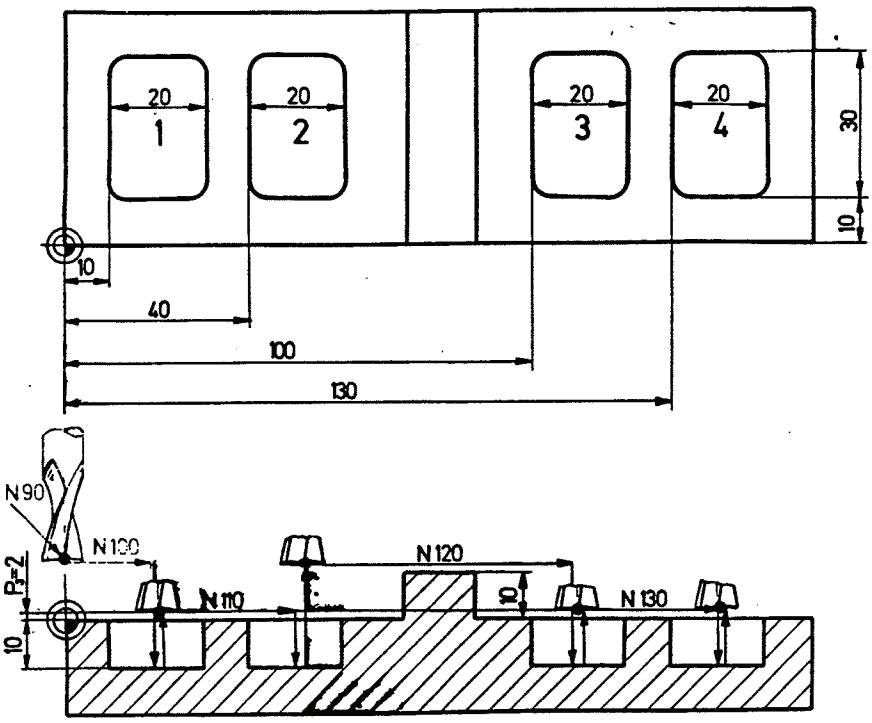
ad) D_3

The control calculates by itself in-feed depths of same size! These are at the most as large as the programmed D_3.



Example: Pocketing

The 4 pockets are separated from each other by a cross-piece.
Thus the starting plane is positioned higher than the cross-piece.



N 90/G00/Z 12./

N 100/G87/G99/X 20./Y 25./Z-10./$P_3$2/$P_0$20./$P_1$30./$D_3$6000/($D_5$03)/($D_7$1)/F....

N 110/X 50./G98

N 120/X 110./G99

N 130/X 140.

Block N 90: Positioning in /Z = 12.

N 100: Pocket 1 G99 active

N 110: Pocket 2 G98 active, cutter moves to the starting plane

N 120: Pocket 3 G99 active

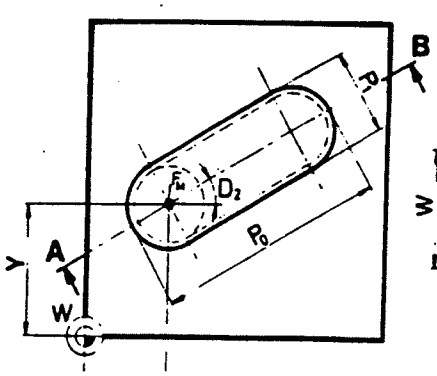
N 130: Pocket 4

The milling cutter is shown in the relative block end position.

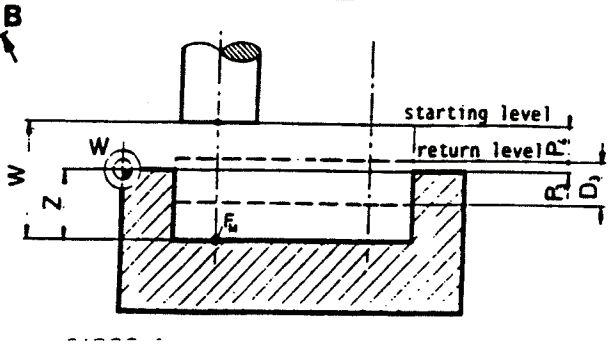
G89 - Slot milling cycle

G89	X U ±43	Y V ±43	Z W ±43	P_0 ±43	P_1 ±43	P3 P4 ±43	D_2 4	D_3 5	D_4 1	D_5 1	D_7 1	F 4

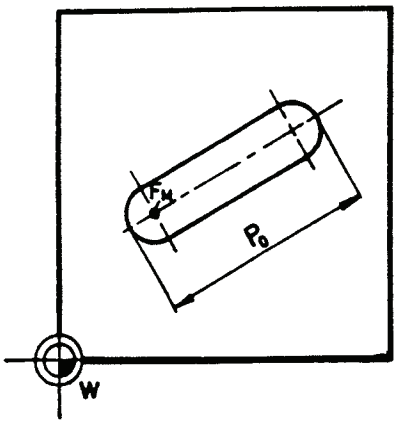
ADDRESSES	UNITS	DESCRIPTION	Default option: When the parameter is not programmed
X,U Y,V Z,W	[mm]	Coordinates of the mill diameter F_M at the deepest point of the slot	
P_0	[mm]	Length of the slot	
P_1	[mm]	Width of the slot	
P_3	[mm]	Definition of the return level Absolute dimension in Z-direction from the zero point level	
P_4	[mm]	Definition of the return level Incremental dimension in Z-direction from the starting level.	
D_2	[Grad (°) x 10]	Angle of the slot in relation to the X-axis	0
D_4	[ ]	Finishing parameters D_4 = 0 ... finishing infeed with working feed D_4 = 1 ... finishing infeed with half working feed	D_4 = 1
D_3	[ ]	Vertical infeed	Machining takes place with one infeed
D_5	[mm]	Climb/opposed milling D_5 = 2 ... climb milling D_5 = 3 ... opposed milling	D_5 = 3
D_7	[ ]	Type of vertical infeed D_7 = 1 ... vertical infeed D_7 = 2 ... inclined infeed	D_7 = 1



Section A-B:



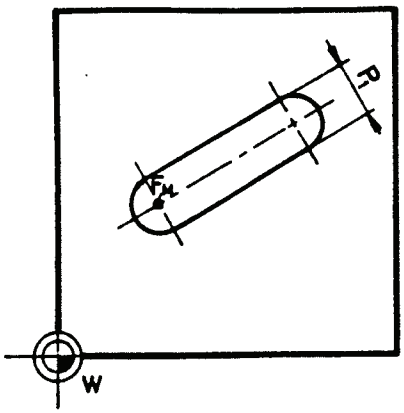
Explanations on the parameters:



Parameter P_0:

The length of the slot is established with the parameter P_0.

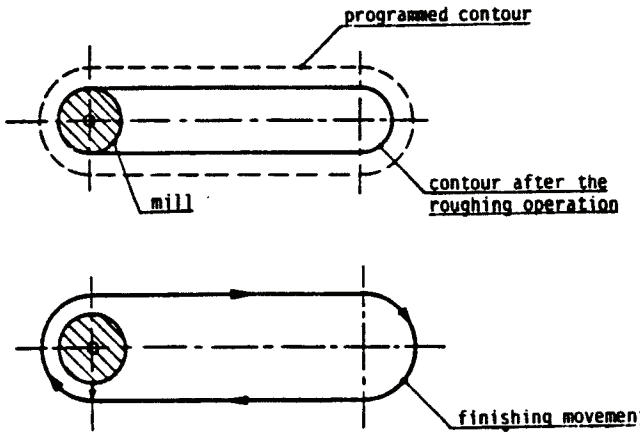
Entry dimension: [mm]



Parameter P_1:

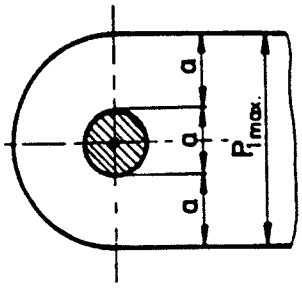
The width of the slot is established with the parameter P_1.

Entry dimension: [mm]



Explanations on P_1:

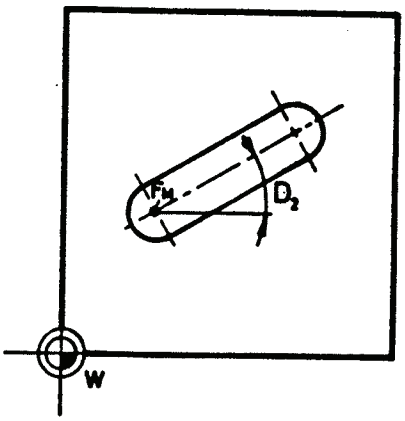
The slot milling cycle is a combined roughing/finishing cycle. The mill diameter is to be only slightly smaller than the width of the slot (P_1).
(Mill diameter = 2 x cutting edge radius)



Entry range of P_1:

a = 2 x cutting edge radius (mill radius)

P_1 min = 2 x cutting edge radius (a)
P_1 max = 6 x cutting edge radius (3xa)



Parameter P_3:

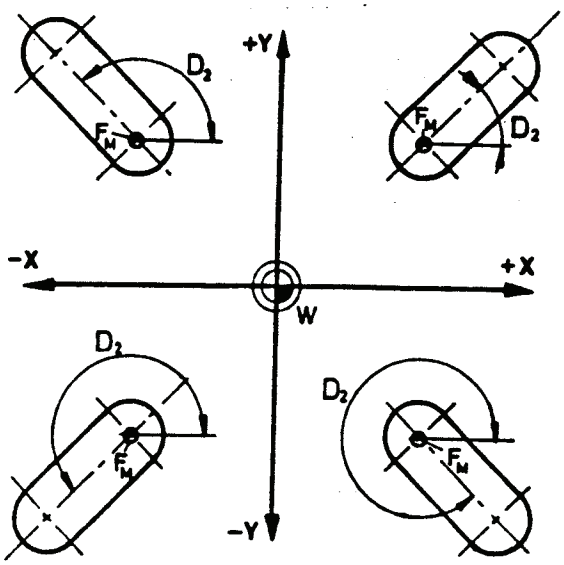
The withdrawal plane is defined with parameter P_3 with absolute programming (see G98, G99).

Entry dimension: [mm]

Parameter P_4:

The withdrawal plane is defined with parameter P_4 with incremental programming (see G98, G99).

Entry dimension: [mm]



Parameter D_2:

The angle of the slot to the X-axis is defined with parameter D_2.

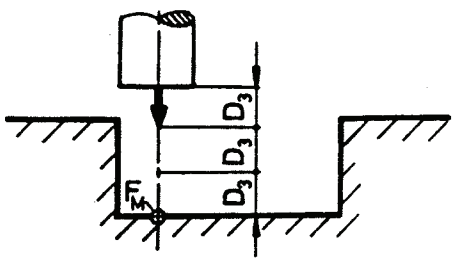
Note:
When parameter D_2 is not programmed (default option), the control executes $D_2 = 0$.

Entry dimension: [degrees (°) x 10]

Position of the slot in relation to point F_M:

F_M ... mill centre

The position of the slot depends on the selection of the parameter D_2. The parameters P_0, P_1 cannot influence the position as they can only be positive.

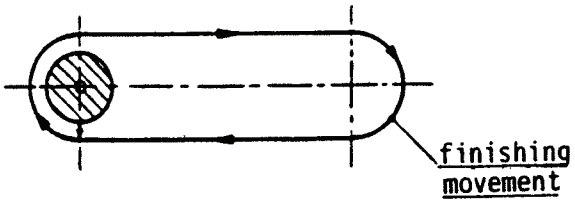


Parameter D_3:

The vertical infeed is specified with parameter D_3.

Note:
When D_3 is not programmed (default option), the infeed takes place in one operation.

Entry dimension: [mm]



Parameter D_4:

If parameter D_4 = 1 is programmed, horizontal finishing movement is executed at half the working feed.

Note:
When D4 = 0 or D4 is not programmed, a finishing movement is executed at the full working feed.

Entry dimension: []

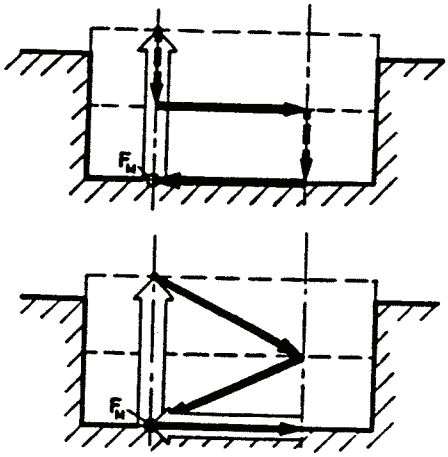
Parameter D_5:

Climb or opposed milling is established with parameter D_5.

D_5 = 2 climb milling
D_5 = 3 opposed milling

Note:
When D_5 is not programmed (default option), the control executes D_5 = 3 (opposed milling).

Entry dimension: []



Parameter D_7:

The type of infeed can be established with parameter D_7.
A difference is made between two types of infeed:

Possibility 1:
Vertical infeed $D_7 = 1$

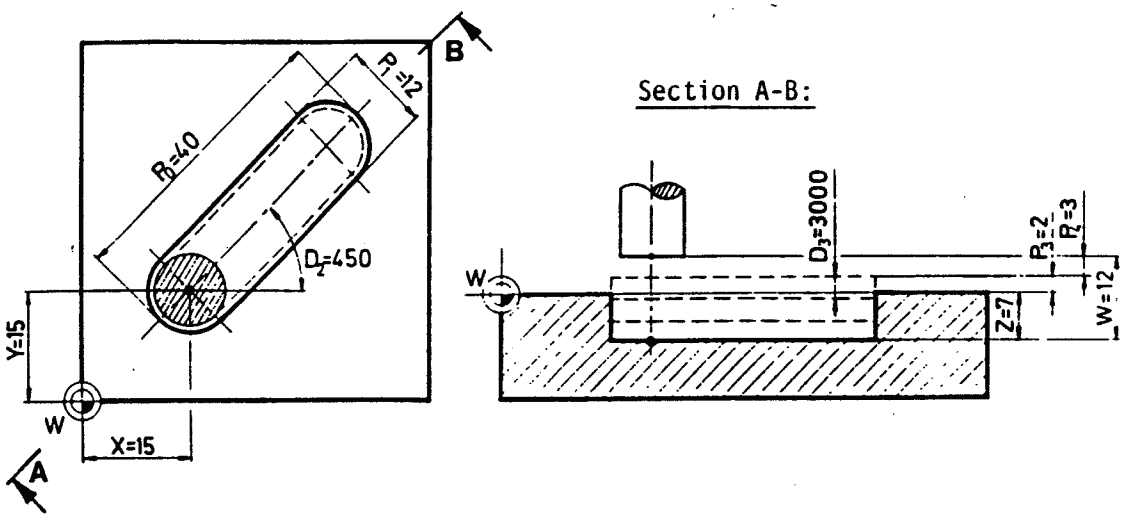
Possibility 2:
Inclined infeed $D_7 = 2$

Note:
When D_7 is not programmed (default option), the control executes $D_7 = 1$.

Entry dimension: [ ]

Example G89:

mill diameter: 10 mm



Incremental programming:

N..../....
N..../G00/....
N..../G99
N..../G89/U00,000/V00,000/W-12,000/
P_0 = 40,000/P_1 = 12,000/P_4 = -3,000/
D_2 = 450/D_3 = 3000/D_5 = 3/D_7 = 2/
F....
N..../.....

Absolute programming:

N..../....
N..../G00/X15,000/Y15,000/Z5,000
N..../G99
N..../G89/X15,000/Y15,000/Z-7,000/
P_0 = 40,000/P_1 = 12,000/P_3 = 2,000/
D_2 = 450/D_3 = 3000/D_5 = 3/D_7 = 2/
F....
N..../.....

G92 Set Register

General:

1) The measurements for the offset are written into the parts program with G92.

Example:

N..../G92/X -14.2/Y +13./Z +14./

During program run the values are written into the PSO number 5.

2) The offset is activated by the G59 instruction.

Example:

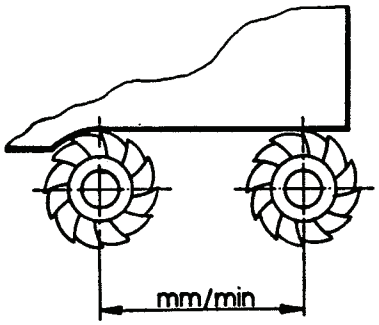
N..../G92/ X -14.2/Y +13./ + 14/

Notes:

* G59 cannot be programmed together with G92 in the same block but must be programmed in the following block.
* If G59 is programmed before a G92 block alarm will appear.
* If G59 follows a G-code of group 3, then both offsets will be added up.

For details compare chapter Zeropoint Offset!

G94 Feed Programming
Feed in mm/min (Inch/min)



G94 is the initial status of the control.

The feed speed is programmed.

Input Dimension:

Metric	Inch
mm/min	$\frac{1}{100}$ "/min

Example: metric

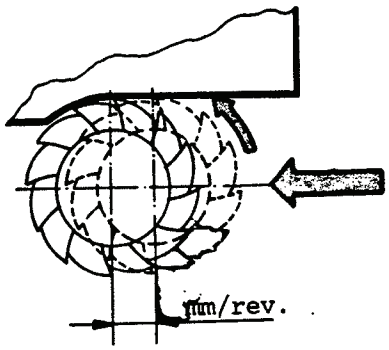
140 mm/min --> F = 140

Example: inch

2,4"/min --> F = 240

For maximum values compare "Technical Data of the CNC-Machine".

G95 Feed in mm/rev. (Inch/rev.)



Input Dimension:

Metric	Inch
$\frac{1}{1000}\ \frac{mm}{rev.}$	$\frac{1}{10000}$ "/rev.

Example: metric

0,15 mm/rev. --> F = 150

Example: Inch

0,013 "/rev. --> F = 130

G98 is Withdrawal to Starting Plane
G99 is Withdrawal to Withdrawal Plane

G99 being active:

With P_3 resp. P_4 you can define a withdrawal plane.

The parameters are effective only from the withdrawal plane onwards if it is defined. (This is also valid with G98 being active)

G98

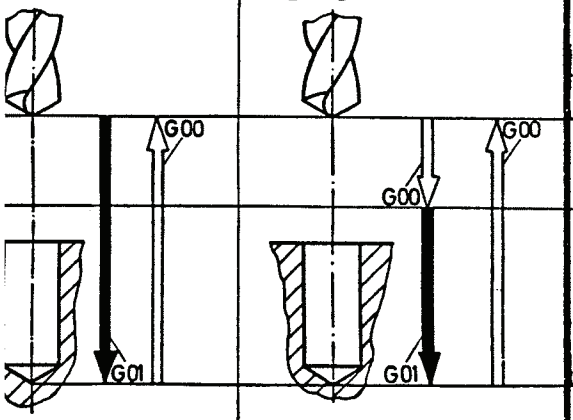
withdrawal to starting plane

P_3/P_4
grammed

proach with
1 from star-
ng plane on-
rds.

Parameters
e effective
om starting
ane onwards.

P_3/P_4
programmed



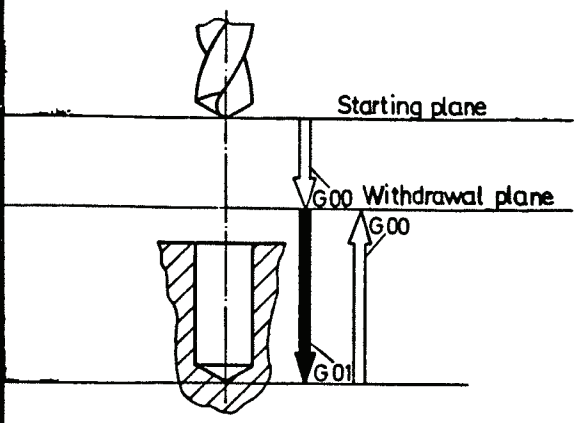
To P_3 resp P_4 with G00

D Parameter is effective from P_3 resp. P_4 onwards.

G99

is withdrawal to withdrawal plane

P_3 bzw. P_4 programmed



Approach to withdrawal plane with G00

D Parameters are effective from P_3 resp. P_4 onwards.

For Details compare G81 - G87 !

Chapter 7

Alarms

EMCOTRONIC TM 02 - Milling

Alarm messages
EMCOTRONIC TM 02 - Milling

(Version 6.00, status 91-4)

ALARM MESSAGES 000 - 029: AXIS CONTROLLER

ALARM 000: AC NOT READY

When switching on the control and during the data transmission from the data controller to the axis controller, it is checked whether the axis control unit correctly reads in the commands from the data controller within a certain time limit. If this limit is exceeded, the above alarm will be displayed which must the normally be removed via the hardware (axis controller!).

ALARM 001: X-AXIS: SOFTWARE LIMIT SWITCH OVERTRAVELLED

EXECUTE/AUTOMATIC mode: The programmed path is monitored by software limit switches which, in case of an overtravel, will output ALARM 001, 002, 003 (due to false data in the position-shift register, due to false tool data or a false circular path end point lies within the valid travelling range).
MANUAL: After positioning the axis at the reference point, the software limit switches are valid and will output an alarm and stop the axes when these limits are overtravelled.

ALARM 002: Y-AXIS: SOFTWARE LIMIT SWITCH OVERTRAVELLED

See alarm 001.

ALARM 003: Z-AXIS: SOFTWARE LIMIT SWITCH OVERTRAVELLED

See alarm 001.

ALARM 020: MAIN DRIVE NOT READY

This alarm will be displayed due to an error message output by the main drive at the following point of time:

* If, after switching on the control, no ready signal is output by the main drive.
* If the operator tries to switch on the main drive without a ready signal being present.
* If an error occurs at the main drive during the operation.
* If there is an error in the power supply of the main drive and/or the machine.

Eliminate the error cause and switch the main drive off and on to quit this alarm.

ALARM MESSAGES 030 - 080: PERIPHERY

ALARM 030: LUBRICANT PRESSURE FAILURE

Depending on the overall travel of the slides, the lubrication pump is switched on for an adjustable amount of time. At the end of this time, the built-up pressure in the lubrication lines is checked. Possible error causes:

* Lubrication pump does not operate.
* Operating time of the lubrication pump too short (false adjustment).
* Air in the lubrication system.
* Pressure switch at lubricant pump defective
* Lubricant line leakage
* Not enough oil in lubricant tank

The error cause should be eliminated immediately to avoid mechanical damage on the machine.

ALARM 031: LUBRICANT PUMP OVERLOAD

The thermal monitoring unit of the lubrication pump has been triggered (e.g. pump operating time too long, overload).
Check thermal protective switch in switch cabinet; no switch must be in "0" position.

ALARM 040: FRONT DOOR NOT CLOSED

This error occurs in the following cases with the maximum priority of the door limit switch:

* When actuating the "CYCLE START" key with the chip safety door open.
* In the MANUAL mode with the chip safety door open, if
  - the axes are to be travelled manually (jog keys)
  - the tool magazine is to be swivelled and
  - the main drive is to be switched on.
* Upon opening the door, if either main drive or CYCLE START are active.

Depending on the set priority for the door limit switch, ALARM 040 may also occur under the following conditions:

* Automatic chip safety door, M53 active: if, after a CYCLE START in AUTOMATIC mode, the door is not closed after 10 sec..

Independently of the actual priority of the door limit switch, ALARM 040 will occur under the following operating conditions:

* If a CYCLE START is initiated with the bar feed active and with the chip safety door and workholding tool open.
* If the door is opened with the bar feed activated and open workholding tools as long as CYCLE START is active.
* If the operator tries to operate the collection tray with the door open. On the operator monitor, different priority levels can be

selected for the door limit switch depending on the hardware condition of the axis controller (see description of the operator monitor).

ALARM 050: TOOL TURRET HARDWARE FAILURE

With the M1 tool magazine for the VMC-100 ALARM 050 occurs in the following situations (the significance of the limit switches E1 to E3 and an exact description of the M1 tool turret are given in the documents "Peripherals for EMCOTRONIC DC V3.0").

a) Reference position invalid
   * Indexing attempt as long as E1 and E2 are damped (the magazine must bemoved manually in a valid position so that E3 is undamped
   * Unsuccessful attempt to move the magazine into a valid position (time limits exceeded owing to defective E3, incorrect MSD or similar).

b) Reference position valid during normal indexing operation
   * Either E1 or E3 is damped after the indexing position in Z has been reached.
   * During indexing E3 does not switch within the time limit set under O 6
   * Error message of the position monitoring facility during indexing:
     E2 not damped if position 1 is expected or E2 damped if position 1 is not expected
   * After positioning of the magazine and expiry of the time element O 5, E3 is damped
   * During moving down from the indexing operation into the reference/machining position E3 is damped
   * On completion of the entire indexing operation (i.e after moving down into the reference position in Z) either E1 is not damped or E3 is damped

c) Reference position valid during error acknowledgement in the MANUAL mode:
   * E1 is damped although E3 is also damped (initially, a magazine position valid for the computer is to be set either by manual positioning of the magazine or eliminating a fault at E3
   * Unsuccessful attempt to move the magazine into a valid position (E3must be undamped after positioning of the magazine)
   * E1 or E3 is damped after the indexing position has been reached
   * Unsuccessful attempt to index to position 1:
     - When each new position has been reached, the limit switch E3 has to be undamped on expiry of the time element O 50.
     - A maximum number of as many consecutive magazine positions areindexed as is given in the MSD; ALARM 050 then occurs if the position has not been found (e.g. error at E2)

ALARM 051: MAINSPINDLE NOT IN POSITION

This alarm occurs only on the VMC-200 in conjunction with the tool-changing system.

* During the traversing movements between reference position and indexing point the main spindle must be in the sync. position as otherwise ALARM 051 is given.
* If the reference point is approached, the main spindle cannot be positioned because this would result in a collision with the retaining finger (E1 active).
* Indexing process can not be initiated because during positioning the main spindle would cause a collision with the retaining finger.

Remedy: In MANUAL mode the main spindle travels upwards (+ Z) until it stops automatically. Now the existing alarm can be eliminated and the reference point can be approached.

ALARM 060: TOOL TURRET NOT READY

With the M1 tool turret for the VMC-100 alarm 060 occurs in the following situations (the significance of the limit switche E1 to E3 and an exact description of the M1 tool turret are given in the documents "Peripherals for EMCOTRONIC DC V3.0"):

* On activation of cycle start:
  a) If a previous tool turret alarm has not been acknowledged by indexing once in the MANUAL mode
  b) If indexing once in the MANUAL mode has not been performed after abandoning the protected MONITOR.
* Response of the cycle monitoring facility of the tool turret limit switches: As long as neither a tool turret alarm is given nor an indexing operation active, E1 and E3 are called up cyclically every 100 ms. E1 must be damped, E3 must not be damped.
* Jog-Z in the MANUAL mode is only possible if E1 is damped.
* After the occurance of a tool turret alarm, working with the main drive is not permitted again until this alarm has been acknowledged in MANUAL mode by indexing once.

ALARM 080: COOLANT PUMP OVERLOAD

This error occurs in case of high power consumption of the coolant motor either caused through mechanical overload (coolant viscosity, chips) or due to a phase error (phase protection defect or coolant pump not connected).

After error elimination the thermal protective switch in the switch cabinet has to be switched on.

ALARM 090: AUXILIARY DRIVES NOT READY

The auxiliary drives are not switched on thus inhibiting the start of main drive, axes and additional drives (tool magazine, coolant / lubrication pump etc.).
After acknowledgement of the alarm message the auxiliary drives can be switched on with the AUX ON" key.

ALARM MESSAGES 100 - 190: AXIS CONTROLLER

ALARM 100: AC SYNTAX ERROR

Wrong format of a command to the axis control unit (AC). In normal operation, this error should not occur. After display of this alarm, the control must be re-initialized (switch off/on).

ALARM 101: X-AXIS: PROXIMITY DETECTOR ERROR

The inductive proximity switch for the stop check of the X axis is defect.

ALARM 102: Y-AXIS: PROXIMITY DETECTOR ERROR

The inductive proximity switch for the stop check of the Y axis is defect.

ALARM 103: Z-AXIS: PROXIMITY DETECTOR ERROR

The inductive proximity switch for the stop check of the Z axis is defect.

ALARM 104: X-AXIS: DEVICE NOT PRESENT

There is an error in the communication between processor and drive, i.e. the drive cannot be addressed by the processor.
Possible error causes:
The corresponding drive board is not in the right slot.
The corresponding drive board is defective.
If ALARMS 104-107 occur together there is an error in the 24V DC voltage supply unit. Check glass tube fuses on 24V supply unit (1A715000).

ALARM 105: Y-AXIS: DEVICE NOT PRESENT

see ALARM 104

ALARM 106: Z-AXIS: DEVICE NOT PRESENT

see ALARM 104

ALARM 107: MAIN DRIVE: DEVICE NOT PRESENT

see ALARM 104

ALARM 110: AC OUTPUT BUFFER OVERLOW

The processing of the status messages of the axis controller (AC) is too slow. In normal operation, this error should not occur. After display of the alarm, the control must be re-initialized (switching off/on).

ALARM 111: X-AXIS: ENCODER SUPPLY ERROR

There is an error in the area of the position encoder of the corresponding drive.
Possible error causes:
* Position encoder defective
* Interruption of cable connection between position encoder and drive unit.

This error can be eliminated by switching the control off and on.

ALARM 112: X-AXIS: SET SPEED NOT REACHED

There is an error within the driving unit which can only be acknowledeged through switching the control off and on again.
Possible error causes:
* Overload of axis drive
* Mechanical defect of axis drive
* Defect of control unit of axis drive unit

ALARM 113: X-AXIS: POWER SUPPLY NOT READY

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Defect of power supply unit on power board of drive unit
* Defect of control unit of drive unit.

ALARM 114: X-AXIS: THERMAL OVERLOAD

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Thermal overload on power board of drive unit due to an overload of the drive
* Heavy contamination of filters of switch cabinet ventilator

ALARM 115: X-AXIS: MOTOR HIGHLOAD

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.

ALARM 116: X-AXIS: POSITION OVERFLOW

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
The drive cannot reach the position preset by the computer.
Possible error causes:
* The drive is blocked mechanically
* The drive cannot travel due to lacking supply voltage
* There is a defect in the drive unit
* Due on an error of another drive unit the voltage supply of this drive was also interrupted
* The drive was moved manually with auxiliary drives switched off

ALARM 117: X-AXIS: OVERCURRENT

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Carbon brushes of drive motor are worn out
* The drive motor is defect
* The power board of the drive unit is defect
* The control unit of the drive unit is defect

ALARM 118: X-AXIS: MOTOR OVERLOAD

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Excessive stress of drive during chip removal
* Mechanical defect on drive (sluggish operation)

ALARM 119: X-AXIS: LIMIT SWITCH OVER-TRAVELLED

This alarm occurs if the slide is traversed over the limit of the working area. This is possible with existing ALARM 460.
To acknowledge this alarm the slide has to be removed to the working area with switched off auxiliary drives.
Subsequently the alarms that occur additionally have to be eliminated by switching off and on.

ALARM 121: Y-AXIS: ENCODER SUPPLY ERROR

see ALARM 111.

ALARM 122: Y-AXIS: SET SPEED NOT REACHED

see ALARM 112.

ALARM 123: Y-AXIS: POWER SUPPLY NOT READY

see ALARM 113.

ALARM 124: Y-AXIS: THERMAL OVERLOAD

see ALARM 114.

ALARM 125: Y-AXIS: MOTOR HIGHLOAD

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.

ALARM 126: Y-AXIS: POSITION OVERFLOW

see ALARM 116.

ALARM 127: Y-AXIS: OVERCURRENT

see ALARM 117.

ALARM 128: Y-AXIS: MOTOR OVERLOAD

see ALARM 118.

ALARM 129: Y-AXIS: LIMIT SWITCH OVER-TRAVELLED

see ALARM 119.

ALARM 130: VALUE OUT OF RANGE OR INVALID INPUT DATA

This error occurs if the axis controller reads-in setting data which cannot be processed. The causes are false machine status data (MSD).
Remedy: New setting of the machine status data (reading-in of the MSD cassette).

ALARM 131: Z-AXIS: ENCODER SUPPLY ERROR

see ALARM 111.

ALARM 132: Z-AXIS: SET SPEED NOT REACHED

see ALARM 112.

ALARM 133: Z-AXIS: POWER SUPPLY NOT READY

see ALARM 113.

ALARM 134: Z-AXIS: THERMAL OVERLOAD

see ALARM 114.

ALARM 135: Z-AXIS: MOTOR HIGHLOAD

There is an error within the driving unit which can only be quitted through switching the control off and on again.

ALARM 136: Z-AXIS: POSITION OVERFLOW

see ALARM 116.

ALARM 137: Z-AXIS: OVER CURRENT

see ALARM 117.

ALARM 138: Z-AXIS: MOTOR OVERLOAD

see ALARM 118.

ALARM 139: Z-AXIS: LIMIT SWITCH OVER-TRAVELLED

see ALARM 119.

ALARM 140: MAIN DRIVE SYNCHRONIZATION ERROR

The axis controller does not receive the correct signals in order to execute the rotation feed start command.

Error causes:
* A speed sensor of the main drive does not operate.
* Hardware error at the axis controller.
* Speed decrease due to main drive highload
* False, but plausible setting data otherwise ALARM 130).
* No synchronization pulse.

ALARM 141: MAINDRIVE: SET EPEED NOT REACHED

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Overload of main drive
* Shaft encoder belt defective
* The control unit of the main drive is defective
* The armature fuse of the DC main drive motor is defective

ALARM 142: MAINDRIVE: ENCODER SUPPLY ERROR

see ALARM 111.

ALARM 143: MAINDRIVE: EXITATION CURRENT EXCEEDED

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Bad zero conductor and/or earthing connection of the machine
* Voltage fluctuations in the power supply system
* Fuses on power unit of the main drive unit are defective
* Field winding in the main drive motor is interrupted

ALARM 144: MAINDRIVE: THERMAL OVERLOAD

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
a) The thermal protective switch of the main drive ventilator motor has actuated:
   Possible error causes:
   * Fuses on power unit of main drive unit are defect
   * Main drive ventilator motor is defective
   After elimination of the error cause the thermal protective switch in the switch cabinet has to be switched on.
b) The thermal protective switch of the main drive ventilator motor has not actuated:

* Overload of main drive motor
* Heavily contaminated air filter in main drive ventilator

ALARM 145: MAINDRIVE: MAXIMUM SPEED EXCEEDED

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Defect on main drive unit
* Potentiometer $n>n_{max}$ on control unit of main drive unit is adjusted wrongly
* Wrong machine status data
  Remedy: Read in MSD cassette

ALARM 146: MAINDRIVE: POSITION CONTROLLER ERROR

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Defect on main drive unit
* The main drive unit was overloaded for a short period
* The main drive is blocked mechanically

ALARM 147: MAINDRIVE: POWER SUPPLY ERROR

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Disturbances of power supply
* Defect of power unit on main drive unit

ALARM 148: MAINDRIVE: MAIN POWER SUPPLY ERROR

There is an error within the driving unit which can only be acknowledged through switching the control off and on again.
Possible error causes:
* Phase sequence error
  Remedy: Change phase sequence on terminals
* Phase failure in power supply system
* Fuse cartridges in switch cabinet are defective
* Asymmetries in power supply system
* W rong order of supply voltage
* Short-time failure influences of power supply system
* Bad zero conductor and/or earthing connection of machine to power supply system

ALARM 150: AXIS OUT OF SYNCHRONISATION REFERENCE POSITION LOST

This alarm occurs at pulse motor operated axis drives.
a) If the slide cannot be traversed after acknowledgement of the alarm and after new switch-on of the auxiliary drives, the exact error cause is indicated by luminous diodes at the pulse motor board.
   Possible error causes:
   * Thermal overload of the pulse motor board
   * Overload on-state current due to a defect pulse motor
   * Overvoltage or undervoltage due to bad electric connections
b) If the slide can be traversed without switching off and on the control the following error causes are possible:
   * Too high load of axis drive (e.g. collision)
   * Wrong machine setting data
     Remedy: Read in MSD cassette.
   * The slide is sluggish as regards mechanics (lubrication)
   * The inductive proximity switch is defect
   * The distance of the inductive proximity switch is too large

ALARM 151: X-AXIS OUT OF SYNCHRONISATION; REFERENCE POSITION LOST

The monitoring unit for the axis movements has detected an error in the X drive position caused by a feed motor overload.

ALARM 152: Y-AXIS OUT OF SYNCHRONISATION: REFERENCE POSITION LOST

The monitoring unit for the axis movements has detected an error in the Y drive position caused by a feed motor overload.

ALARM 153: Z-AXIS OUT OF SYNCHRONIZATION.REFERENCE POSITION LOST
The monitoring unit for the axis movements has detected an error in the Z drive position caused by a feed motor overload.

ALARM 160: BAD PARAMETER FOR G02 OR G03

In a circular command, a false parameter or a parameter with a false value was input. This alarm occurs under the following conditions:
* Centre coordinate input missing.
* Centre coordinate outside the numerical range of the machine (the second centre coordinate, which is not input, may be generated in this way).

* The second centre coordinate does not fit into a circle.

ALARM 170: TRIED TO START WITH FEED = 0

This alarm occurs if the operator tries to execute an axis movement which is not possible due to the following reasons:
- Straight feed: F = 0 active (F code missing)
- Rotational feed:
  a) F = 0 active (F code missing)
  b) Main drive not switched on
  c) S = 0 active (no speed was programmed)

Note: Resetting the feed rate override switch to zero will not trigger this alarm as long as the input of the feed command is correct.

ALARM 180: WRONG CENTER COORDINATE SPECIFIED

The centre coordinate of the axis with the shorter travelling distance between start point and target must be programmed.

ALARM 190: RADIUS TOO LARGE

ALARM MESSAGES 200 - 281: MACHINING CYCLES

ALARM 200: INVALID VALUE OF D OR P PARAMETER

G04: The maximum value for D4 (10000, i.e. dwell time of 1000 seconds) was exceeded.
G87: A negative value has been given for P0/P1An invalid value has been given for D5 (only 2 and 3 are valid)An invalid value has been given for D7 (only 0 and 1 are valid)

ALARM 201: P1 MUST BE POSITIVE

ALARM 202:NO ANGLE PARAMETER D2 ALLOWED

G89: If the slot milling cycle is used as a drilling pattern element of a circular drilling pattern (G73) only D2 = 0 allowed.

ALARM 203: NO MIRRORING ALLOWED

G83: A circular drilling pattern can only be mirrored, if it is closed. Set D3 to 360 degrees (D3 = 3600).

ALARM 204: G72/G74: M90 MUST BE ACTIVE

ALARM 208: P1 LARGER THAN 3x CUTTER DIAMETER

ALARM 209: D2 LARGER THAN CUTTER DIAMETER

ALARM 211: INVALID P0 OR P1

G72: P0 must be greater than 0
G74: P0 or P1 is zero although the relevant D0 or D1 parameter has not been programmed with 1.
P0 or P1 is not equal to 0 although the relevant D0 or D1 parameter has been specified as 1.
G89: Conditions for P0, P1:
P0, P1 must be greater than 0.
P0 must be equal to or greater than P1.

ALARM 221: INVALID D0 OR D1

G72: D0 has been programmed with 0 or 1.
G74: D0 or D1 has been programmed with 0.
D0 and D1 have both been programmed with 1.

ALARM 222: INVALID D2

G72: D2 must be less than 3600 (360 degrees)
G89: Maximum admissible value: D2 = 3600
G88: D2 must be selected greater than 1.

ALARM 223: INVALID D3

G72: D3 has been programmed with 0 or greater than 3600 (360 degrees).
G88, G89: D3 must be programmed with a value greater than 1.

ALARM 224: INVALID D4

G88, G89: D4 can only be 0 or 1.

ALARM 225: INVALID D5:

G88, G89: D5 can only be 2 or 3.

ALARM 227: INVALID D7

G72, G74, G88: D7 can only be 0 or 1.
G89: D7 can only be 1 or 2.

ALARM 230: INVALID CYCLE TARGET

* G87: Starting and target points in the Z-axis coincide or their distance is too large (maximum 1FFFFH (131 071) steps are permitted).
* G88, G89: The target point coincides with the starting point or lies above it.

ALARM 240: NO OR INVALID STEP DEPTH

G87, G88, G89: D3 has been specified larger than the distance starting point - target point in Z.

ALARM 250: D OR P PARAMETER FOR GIVEN CYCLE MISSING

* G72: P0 or D0 missing.
* G74: P0 or P1 has not been programmed although the relevant D0 or D1 parameter has not been given 1. - D0 or D1 missing.

NOTE: For G72 and G74 see also the description of the D7 parameter in the programming instruction.

* G87: P0 or P1 missing.
* G88: P1 missing.
* G89: P0 or P1 missing.

ALARM 270: NO OR INVALID WITHDRAWAL PLANE

* P3 and P4 have been programmed in one block
* The target point plane has been specified as the withdrawal plane.
* The withdrawal plane has been specified outside the starting and target points range (withdrawal plane = starting plane is still valid).

ALARM 280: CYCLES MUST START WITH G40 ACTIVE

ALARM 281: MIRROR START OR END ONLY WITH G40 ACTIVE

ALARM MESSAGES 290-340: ***PROGRAM SEQUENCE SUBROUTINES,G27***

ALARM 290: NO SIMULATION OUT FROM A SUBROUTINE

It is not allowed to start a simulation from a subroutine.

ALARM 300: MORE THAN 10 SUBROUTINES NESTED

ALARM 310: SUBROUTINE NOT IN MEMORY

* A subroutine called by G25 could not be found in the part program memory of the control.
* The called subroutine does not include any blocks.

ALARM 320: G25/G27 NOT ALLOWED IN EXECUTE-MODE

These jump instructions are not useful when processing individual blocks/codes from the block memory in the EXECUTE mode and are thus not executed.

ALARM 330: M17 WITHOUT G25 OR M30 IN A SUBROUTINE

* M17 was found in a part program which was started as main program.
* M 30 in a part prog. which was called by G25

ALARM 340: G25/G27 NOT ALLOWED IN LAST BLOCK OF PROGRAM

ALARM MESSAGES 350 - 440: ***PART PROGRAM INTERPRETER***

ALARM 350: INVALID CUTTER RADIUS

G41, G42: The radius of the active tool is zero. No tool offset is active.

ALARM 360: NO CHANGE OF T-WORD WITH G41/G42 ACTIVE

With the tool radius compensation active, no other tool offset function can be called.

ALARM 361: NO M65 WITH G41/G42 ACTIVE

With active cutter radius compensation no M65 must be programmed.

ALARM 370: NO CHANGE OF SCALE WHEN CYCLE ACTIVE

This alarm occurs if a new scaling factor is selected but a cycle is active (concerning milling) or if G51 is set active with a cycle in the same block.

ALARM 371: NO CHANGE OF PSO WHEN SCALE ACTIVE

This alarm occurs if with active scaling fator a PSO register is selected or deselected. A PSO register can only be changed if the scaling factor is not active.

ALARM 372: NO RELATIVE MOVES AFTER G51

After G51 an absolute move must ensue to make the starting point of the zoomed contour independent of the slide starting position (see programming instructions).

ALARM 373: NO NEGATIVE SCALE ALLOWED

Only positive values are allowed for the P7 parameter when indicating a scaling factor.

ALARM 374: INVALID PARAMETER FOR G51

When indicating the reference point for the scaling factor an invalid D or P parameter was indicated.

ALARM 375: SCALE CALCULATION OVERFLOW

The size of the resulting contour is too great. Check the reference point for the scaling factor and thescaling factor.

ALARM 380: BAD OR MISSING PARAMETER IN G25/G27

In a block with G25/G27 no L-code was programmed.
It was tried to call the part program already active.
An L-code belonging to G27 includes a block number not present in the active part program.

ALARM 381: AFTER CHAMFER/RADIUS ONLY G01 ALLOWED

After a block with programmed chamfer/radius only other traverse command than G01 (i.e. no circle and no G00) is allowed.

ALARM 382: MISSING POSITION PARAMETER FOR CHAMFER/RADIUS

The block after a programmed chamfer/radius must contain position parameters (absolute or incremental).

ALARM 383: THREE DIMENSIONAL CHAMFER/ RADIUS NOT ALLOWED

The programmed chamfer/radius must be in the same plane, inserting three-dimensional chamfers or radii is not possible.

ALARM 384: CHAMFER/RADIUS CALCULATION OVERFLOW

There was an overflow when computing the correction points for the chamferer/radius to be inserted.
Please check again the position data of the programmed chamfer/radius as well as the subsequent block.

ALARM 385: NO CHANGE OF PSO IF CHAMFER/ RADIUS ACTIVE

The PSO must not be changed in the block with the programmed chamfer/radius since otherwise the chamfer/radius that is to be inserted cannot be calculated correctly.

ALARM 386: NO CHANGE OF SCALE IF CHAMFER/RADIUS ACTIVE

The scaling factor must not be changed in the block with the programmed chamfer/radius, since otherwise the chamfer/radius that is to be inserted cannot be calculated correctly.

ALARM 387: NO CHANGE OF TOOL WHEN CHAMFER/RADIUS ACTIVE

Due to the calculation of the tool data during tool exchange no change of tool must be carried out in the block with the programmed chamfer/ radius, since otherwise the chamfer/radius that is to be inserted cannot be calculated correctly.

ALARM 388: CHAMFER/RADIUS IN EXECUTE MODE NOT ALLOWED

Programmed chamfers/radii cannot be executed in the EXECUTE mode because the subsequent block is necessary for a chamfer/radius that is to be inserted.

ALARM 389: PROGRAMMED CHAMFER/RADIUS TOO GREAT

The indicated chamfer/radius is to large. It must not be larger than the shorter of the two straight lines between which the chamfer/radius should be inserted.

ALARM 390: CHANGE OF G-CODE GROUP 7/9 ONLY IN FIRST BLOCK

ALARM 391: NEGATIVE CHAMFER/RADIUS NOT ALLOWED

Indicated chamfer/radius must not be negative.

ALARM 400: NO G-CODE FOR GIVEN PARAMETER ACTIVE

A selected parameter can be assigned to a G-code:

* No G-Code from group 0 for a selected position parameter is active (is also displayed, if a position parameter is programmed in a block with G04).
* An L-code was programmed without G25/27.

ALARM 401: NO G72 DEFINITION ACTIVE

G73: A circular drilling pattern can only be called up if it has already been defined. (G72 definition block)

ALARM 402: NO G74 DEFINITION ACTIVE

G75: A rectangular drilling pattern can only be called up if it has already been defined (G74 definition block)

ALARM 403: NOT ALLOWED G-CODE FOR G73 OR G75

The use of the specified G-code as a drilling pattern element is not allowed. No G-code has been specified for drilling pattern element.

ALARM 410: INVALID G-CODE

This alarm occurs, if a G-code which has been programmed is not by the control. The valid G-code block depends, inter alia, on the type of software the costumer desires for the machine (e.g. G41,42).

ALARM 416: BAD PARAMETER FOR G02 OR GO3

A false parameter or a parameter with a false value was input in a circular movement command. This alarm occurs under the following conditions:

* No centre coordinate
* Centre coordinate outside the numerical range of the machine (the second centre coordinate which was not input can be generated in this way).
* The second centre coordinate does not fit into a circle.

ALARM 418: WRONG CENTER COORDINATE SPECIFIED

The centre coordinate of the axis with the smaller travelling distance between start point and target must be programmed.

ALARM 419: RADIUS TOO LARGE

ALARM 420: INVALID M-CODE

This alarm occurs if an M-code which cannot be processed by the control is programmed. The set of valid M-codes depends among others on the customer-specific periphery of the machine (e.g. M20/21, M23/24, M25/26, M50/51).

ALARM 421: NO M03/M04 PROGRAMMED

G88: The control must know the main spindle direction of rotation in order to be able to process the D5 parameter (climb/conventional milling)

ALARM 430: INVALID T-WORD

If a tool offset is selected, the tool number must be input as well.

ALARM 440: TARGET LIMITS EXCEEDED

EXECUTE/AUTOMATIC mode: The programmed targets are monitored by softwar limit switches which trigger ALARM 440 if required. (Possibly false data in the position-shift register or false tool data.)

ALARM MESSAGES 450 - 490: ***GENERAL OPERATING ERRORS***

ALARM 450: ENTERED CAUTION ZONE

Manual mode: overtravel of the safety distance to the software limit switches. The control will switch to a lower feedrate in order to be able to stop when reaching the software limit switches wihout a considerable brake path length.

ALARM 460: REFERENCE POSITION NOT ACTIVE

The reference coordinate system of the machine is not active but after positioning the axis at the reference point; only this allows for the absolute positioning and display of absolute positions.

ALARM 470: RESTART MAIN DRIVE

* When switching off FEEDHOLD: The main drive was switched off during FEEDHOLD but was not switched on again.
* When switching off DRYRUN: If, at this point of time, M03 or M04 are active, the main spindle must be switched on again when DRYRUN is deselected.

ALARM 480: NO OR INVALID PARAMETER FOR G-GROUP 0

* An arc centre parameter was programmed although neither G02 nor G03 is active.
* In a cycle of the G-code group 0, an invalid D- or P-parameter was programmed.
* G04: Parameter D4 (dwell) was not programmed.
* G84/85/86: The target must be input for both axes.
* G87/88: The target must and may only be programmed in Z.

ALARM 481: CENTRE VALUES (X,Y) NOT DEFINED

G72,74: The centre point of a circular drilling pattern or the reference point must be defined in X and Y.

ALARM 482: Z (OR W) VALUE NOT ALLOWED

G72,74: The centre point of a circular drilling pattern or the reference point must be defined in X and Y but not in Z.

ALARM 483: ONLY Z (OR W) ALLOWED

G73,75: In the execution block of a circular drilling pattern position data are only permitted in Z.

ALARM 490: OFFSET CHANGED, GO WITH G00

After the changing of tool offset or position shift registe, the control will only accept G00 as travelling command.
E.g. T505 G01 U10. F500 = > ALARM 490

ALARM MESSAGES 500 - 580: TOOL RADIUS COMPENSATION

ALARM 500: TOO MANY BLOCKS WITHOUT SLIDE OPERATION

G41/42: More than five consecutive blocks without change of the XZ-value were programmed.

ALARM 510: TOO FEW POINTS PROGRAMMED

G41/42: Prior to deselecting the compensation with G40 or M30, at least two blocks with a change of the XZ-value must be programmed.This error will also occur when G41/42 are called in the EXECUTE mode.

ALARM 520: ERROR AT COMPENSATION START OR END

* The first movement after selecting or deselecting the tool compensation must be with G00 or G01.
* When selecting or deselecting the compensation, the XZ-values must have changed as against the subsequent or previous values. The change of only one value is also permissible.

ALARM 530: NO IMMEDIATE CHANGE OF G41/42

G41/42: In order to switch between G41 and G42, the compensation must first be deselected with G40 and a movement must be executed.
This requires a change of the XZ-values.

ALARM 540: BAD CIRCLE PARAMETER

G41/42: A circular movement command includes a false parameter or a false numerical value for a parameter. Possible error causes see ALARM 160.

ALARM 550: CIRCLE NOT ALLOWED WITH COMPENSATION

G41/42: No circular movement commands allowed in planes being vertical to the compensation plane

ALARM 560: RADIUS TOO LARGE

G41/42: The radius of a circular movement is too large (compare ALARM 190).

ALARM 570: RADIUS TOO LARGE FOR GIVEN CONTOUR

G41/42: The radius of the selected tool is too large for the programmed contour.
Possible causes of error:

* Programming of an arc with a smaller radius than the tool radius.
* Programming of small contour sections compared with the radius, thus causing a contour infringement in the last block executed.
* Programming of an internal corner which is limited by two arcs if special geometrical conditions apply (especially when the tool radius is considerably larger than the smallest programmed radius - cf. section on cutter radius compensation).
* G87: Pocket width smaller than the cutter diameter.
* G88: Pocket diameter smaller than the cutter diameter.
* G89: Slot width smaller than the cutter diameter.

Note: Contour violations in blocks more than one block after the presently processed block or blocks which will not be processed but after the violation cannot be detected.

ALARM 580: PART PROGRAM MUST END WITH G40 ACTIVE

G41/42: Part programs must be ended with the compensation function deselected (deselection with G40 or M30).

ALARM MESSAGES 600 - 710: EDITOR

ALARM 600: INCORRECT EDITING SEQUENCE

* Attempt to select a block although no part program is selected.
* Attempt to select a word although no block is selected (this is only possible in the EXECTUE mode).
* Input error for the functions "Erase program memory/erase offset register": the input sequence "PROGKILL/OFFSKILL" was not observed.
* Reading in of data from cassette/RS232: The data format input in MON parameter L4 for the reading process does not correspond to the data format used for writing.

ALARM 610: INVALID PARAMETER ENTERED

Another parameter than D, L or R was selected on the operator monitor.

ALARM 620: INPUT VALUE OUT OF NUMERICAL RANGE

* EDIT mode: Attempt to input a numerical value outside the limits fixed in the machine specifications.
* AUTOMATIC/EXECUTE: The limit for straight or rotational feed was exceeded.

Note: The numerical input limits are specific to the machine and thus to be obtain from the respective machine instructions.

ALARM 630: INVALID SUBROUTINE NUMBER

A part program can only be ended with M17 as subroutine if its 0 number lies within the numerical range valid for subroutines (this range is input in parameter L3 on the operator monitor (MON)).

ALARM 640: BLOCK NUMBER ALREADY EXISTS

Attempt to change the number of a block to the number of another block which is already present in the active part program.

ALARM 650: BLOCK MEMORY OVERFLOW

Attempt to input a too long block.

ALARM 660: USER MEMORY OVERFLOW

The memory capacity of the control for part programs is exhausted with the data already stored.

ALARM 670: CANNOT OPEN PROGRAM - TOO FEW MEMORY

To activate a program a certain freely available memory is necessary (corresponds to the program length of the program which is to be activated). If this memory is not available this alarm ensues.
Remedy: Create space by deleting programs that are not used any more.

ALARM 675: TOO MANY PROGRAMS IN THE MEMORY

ALARM 690: INVALID INDEX ENTERED

* EDIT and EXECUTE Mode: Attempt to input a P- or D-parameter with index > 7.

* Tool data or shift register selection: attempt to input a tool index > 99 or a shift register index > 5.

ALARM 700: NO CHANGE OF ACTIVE TOOL DATA/PSO

EDIT: Attempt to change the active selected tool offset or the active shift register. A change is only possible after deselecting the respective tool or register. This deselection is simply executed through pressing the RESET key or, in the EXECUTE mode, through processing a block with deselection function (other tool or T0 or other register or G53/56).
AUTOMATIC/EXECUTE: Attempt to change the shift register 5 with G92, although G59 is active.

ALARM 710: PROGRAM NUMBER ALREADY EXISTS

Attempt to change a program number to the number of a program which is already stored in the memory.

Note:
The following alarm messages 730 to 779 only occur in connection with the graphic simulation.

ALARM MESSAGES 730 - 760: GRAPHICIPP - INTERPRETER

ALARM 730: PRINTER NOT READY, HARDCOPY TERMINATED

was tried to print a screen content without the printer being ready for operation (e.g. printer turned off)

ALARM 731: PRINTER OFFLINE

was tried to print a screen content without the printer being ready for reception.
Help: turn ONLINE at the printer

ALARM 732: OUT OF PAPER

ALARM 733: PRINTER NOT CONNECTED

printing cable not connected properly or defective

ALARM 734: PRINTER ERROR

The printer sets his error line during printing. For error elimination the printer manual needs to be consulted.

ALARM 740: GRAPHIC LIMITS EXCEEDED
A position should be approached which exceeds the numerical limit of the graphics. (Check the actual tool shift data and the offset registers. It is also possible that too large a scale was chosen.

ALARM 741: TOOL NOT DEFINED

If a tool is called in the polygon program of which no marked line is programmed, alarm is effected.

ALARM 742: TOOL TOO LARGE

Occurs if the machining part of a tool is larger than the screen. This alarm can be eliminated by chosing another scale.

ALARM 743: ONLY 1 MB AVAILABLE

For the 3d-display and the zoom function a storage retrofit kit from 1 MB to 2 MB is required.

ALARM 744: 3D GRAPHIC NOT ACTIVATED

It was tried to display a 3d-picture without activating the 3d-graphics with the softkey 'AKTIVATE 3D'. To achieve a 3d-display this softkey needs to be activated and the programme has to be executed once again.

ALARM 745: WRONG MACHINE

It was tried to activate the 3d-graphics on a lathe.

ALARM 750: POLYGON PROGRAM NOT AVAILABLE

* It was attempted to call a polygon program with G26, which is not stored in the program memory.
* In case of a tool change (manually or via T-code): the polygon program corresponding to

the desired tool magazine position is not stored in the memory.

ALARM 751: INVALID POLYGON PROGRAM NUMBER

* It was attempted to call a polygon program with G26 with the program number in the L parameter being outside the numerical range reversed for polygon programs (7000 - 9999 with the exception of the number 8001 to 8899 reserved for tool profile programs).
* An invalid value was input in parameter L in program 8000 (for tool polygon programs only numbers 8001 to 8899 are valid).

ALARM 752: INVALID G-CODE IN POLYGON PROGRAM

* A G-code which is generally not permissible in polygon programs was detected in such a program.
* G61 is only valid in tool polygon programs (8001 - 8899).
* G63, G64 and G68 are only valid in blank polygon programs (7000 - 7499).
* G67 is only valid in blank (7000 - 7499), workholding tool (9000 - 9499) and sleeve polygon programs (9500 - 9999).

ALARM 753: INVALID PARAMETER IN POLYGON PROGRAM

* A parameter not permissible in polygon program was found in such a program.
* The parameters L and T are only valid in allocation programs (O 8000 and O 8900 - 8999).
* In O 8000 and O 8900 - 8999, only parameters L and T are valid.

ALARM 754: NO G-CODE FOR GIVEN PARAMETER IN POLYGON PROGRAM

A parameter in a block of a polygon program cannot be assigned to a G-code. This alarm e.g. occurs when programming a position parameter without previously or simultaneously programming a G-code.

ALARM 755: INVALID PARAMETER FOR GIVEN G-CODE IN POLYGON PROGRAM

* The parameter I, J, K are only valid for G02/03.
* G63 only accepts Z.
* G61, G63, G64: parameter X is not permissible.

ALARM 756: BAD PARAMETER FOR G02 OR G03 IN POLYGON PROGRAM

A false parameter or a parameter with a false numerical value was input in a circular movement command. This alarm occurs under the following conditions:

* Centre coordinate missing
* Too many target coordiantes (X, Y, Z - must all be programmed anew)
* Centre coordinates outside the numerical range of the machine (the second centre coordinate which is not indicated can be generated in this way).
* The second centre coordinate does not fit into a circle.

ALARM 757: PARAMETER FOR GIVEN G-CODE MISSING

* G60, G68: Either both or none of the parameters must be programmed.
* G67: At least one parameter must be programmed.

ALARM 758: WRONG CENTER COORDINATE SPECIFIED

The centre coordinate of the axis with the smaller travelling distance from start point to target must be programmed.

ALARM 759: RADIUS TOO LARGE IN POLYGON PROGRAM

ALARM 760: SYNTAX ERROR IN POLYGON PROGRAM

* T and L in O 8000 may only be programmed together.
* This error may also occur if the instructions for the graphic tool changer are missing or are insufficient or wrong.

ALARM MESSAGES 780 - 799: ***GENERAL OPERATING ERRORS***

ALARM 780: SAFETY LOCK ACTIVE

The control is in a locked state, where two priorities are to be distinguished: general lock of the control and memory lock. The lock can only be released by authorized operating personnel.

ALARM 785: WPC-PRESET = 0, CYCLE START IGNORED

If the number of pieces is set to zero (operator monitor) in the "workpiece presetting" mode, the cycle start key is ignored.

ALARM MESSAGES 800 - 870: DATA INTERFACE CASSETTE, RS 232

ALARM 800: CASSETTE DRIVE NOT READY

* No cassette was input.
* Hardware error of the cassette recorder

ALARM 810: WRITE PROTECTED CASSETTE IN USE

The black write protection button was removed from the cassette which is presently used.

ALARM 811: INTERFACE CONTROLLER NOT READY

The interface controller does not respond within a defined period of time. This alarm only occurs in case of a hardware error of the control. Initialize the control through switching it off and on again.

ALARM 820: BLOCK STRUCTURE ERROR

Use of a non-formatted cassette
Serious damages to a cassette due to mechanical or electrical causes (remedy: new formatting)
Hardware error of the cassette device
Starting from software version DC V3.0, it was attempted to write data on a cassette which was formatted with an older software version. If this is not possible, it may be necessary to read-in the data from the cassette into the control, re-format the cassette and then store the data from the control onto the newly formatted cassette.

ALARM 830: BLOCK CHECKSUM ERROR

Error in the data transmission between cassette and memory (countermeasure: new formatting)
Hardware error of the cassette device
Using an older software version, the user has tried to read-in a cassette onto which date were stored with software version 3.0 ff.

ALARM 840: INSUFFICIENT TAPE SPACE

The user has tried to store a program on a cassette; however, the program length exceeds the remaining memory capacity.

ALARM 850: PROGRAM NOT FOUND

* Attempt to read-in a program not yet stored.
* Attempt to read-out a non-existent program.
* Attempt to read machine data from other than MSD cassettes.
* False input sequence when reading-in the MSD cassette.

ALARM 860: INTERFACE OPERATING ERROR

False operating sequence when reading-in part programs via the serial interface (see description of the INTERFACE mode).

ALARM 870: WRONG BAUDRATE SELECTED

On the monitor, the baudrate for the data transmission via the serial interface can be set and input in D0. Only values between 150 - 4800 are permissible.

ALARM 880: INTERFACE ERROR

Occurs if e.g. during RS 232 read-in the baud rate or the configuration of the serial interface in control and PC do not correspond to each other.

ALARM MESSAGES 900 - 969: PERIPHERAL DEVICES

ALARM 905: DUST EXTRACTOR THERMAL OVERLOAD

The thermal control of the dust extractor has been actuated (e.g. too long operating time, overload).
Possible error causes:
* Overload
* Motor is defect
* Phase failure of power supply voltage of motor

ALARM 911: HYDRAULIC THERMAL OVERLOAD

The thermal control of the hydraulic system has been actuated (e.g. too long operating time, overload).
Possible error causes:
* Overload
* Motor is defect
* Phase failure of power supply voltage of motor

ALARM 915: CONVEYOR THERMAL OVERLOAD

The thermal control of the conveyor has been actuated (e.g. too long operating time, overload).
Possible error causes:
* Overload
* Motor is defect
* Phase failure of power supply voltage of motor
* The chip conveyor is blocked mechanically

ALARM 925: M27 TIMEOUT ERROR

Ensues if M27 ready message does not appear within the time limit selected in the machine data.

ALARM 926: M65 TIMEOUT ERROR

Ensues if M27 ready message does not appear within the time limit selected in the machine data.

ALARM MESSAGES 970 - 992: ***OPERATING SYSTEM ERRORS***

ALARM 970: FATAL ERROR, CONTACT EMCO!

ALARM 975: FATAL ERROR, CONTACT EMCO!

ALARM 976: FATAL ERROR, CONTACT EMCO!

ALARM 980: FATAL ERROR, CONTACT EMCO!

ALARM 981: FATAL ERROR, CONTACT EMCO!

ALARM 982: FATAL ERROR, CONTACT EMCO!

ALARM 983: FATAL ERROR, CONTACT EMCO!

ALARM 984: FATAL ERROR, CONTACT EMCO!

ALARM 985: FATAL ERROR, CONTACT EMCO!

ALARM 990: FATAL ERROR, CONTACT EMCO!

ALARM 991: FATAL ERROR, CONTACT EMCO!

ALARM 992: FATAL ERROR, CONTACT EMCO!

ALARM 993: FATAL ERROR, CONTACT EMCO!

ALARM 994: FATAL ERROR, CONTACT EMCO!

ALARM 995: FATAL ERROR, CONTACT EMCO!

These alarms with the numbers 970 to 999 should never occur! If they occur repeatedly, please contact your nearest EMCO agency.

Chapter 8

User Monitor

EMCOTRONIC TM 02

User Monitor EMCOTRONIC TM 02

In the user monitor (MON) machine and control statuses can be changed by the user.
The status is determined by the input of parameters.

Group structure of the parameters in the user monitor

Parameter	
D	General monitor data
G	Tool turret data
M	Main drive data
O	General setting data
R	Machine-specific position data
S	Setting data for cycles

Summary of the parameters in the user monitor

D	G	M	O	R	S
00 01 02 03	08	08	00 01 02 03 11 12 22 40	00 01 02 03 04 05 06 07 08 09	00 02

Call-up of user monitor

The user monitor (MON) is called up in the EDIT mode. If a workpiece program is active, it must be cancelled (RESET).

Data input

M O N ENTER

1. Call-up of the user monitor mode

Enter the letters M, O, N, ENTER, the control reports in the user monitor.

2. Selection of the parameters:

There are two possibilities:

D ENTER

2.1 Selection of a parameter group

Example:
If D is entered, the first parameter of this group (D_{00}) appears on the screen.

The parameters of this group can be selected with the ENTER key.

D 0 3 ENTER

2.2 Selection of an individual parameter

Example:
If D_{03} is entered, this parameter appears on the screen.

The parameters of this group can be selected with the ENTER key.

3. Entry and storage of a parameter

- Correction of the displayed value with the CLEAR ENTRY or CLEAR WORD key and entry of the desired value.
- ENTER, transfer into the memory, the value with the next index is displayed.

4. Abandoning the user monitor

The entry is completed by pressing any mode key or RESET. With RESET the EDIT mode remains selected.

Note:
Store the last value entered with ENTER!

D-parameters - General monitor data

D_{00} Entry of the baud rate for the standard interface

With D_{00} the speed (= baud rate) with which the data are loaded and read out via the V24/20mA interface is entered.
Input range: 150 - 4800 baud.

The baud rate to be set depends on the peripheral device connected (see description of peripherals). 300 baud has been set at the works.

D_{01} Priority of the door limit switch

In the normal condition of delivery it is not possible to switch on the main drive when the chip guard door is open.

If the chip guard door is opened with the door limit switch active, the main spindle, feed drives and the coolant are switched off (exception: coolant with D_{01} = 7).
D_{01} = 1 has been set at the works.

* D_{01} = 1
  Modes: MANUAL
  MANUAL/REFERNCE
  In these modes the door limit switch is not effective.

* D_{01} = 3
  Modes: MANUAL
  MANUAL/REFERENCE
  In these modes the door limit switch is not effective.

  Modes: AUTO/DRYRUN
  Work can be performed with the chip guard door open. It is not possible to switch on the main spindle.

* D_{01} = 7
  Modes: MANUAL
  MANUAL/REFERENCE
  AUTO/DRYRUN
  AUTO/SINGLE
  EXECUTE

* D_{01} = 8
  All the conditions regarding the door limit switch applicable to D_{01} = 7 also apply to D_{01} = 8. The only difference is the behaviour of the coolant when the door is opened when CYCLE START is active.

* Coolant behaviour:

o With M08 active (coolant ON) the coolant is switched off each time the mode is changed. The LED of the coolant key flashes to indicate that M08 is still active. By pressing the coolant key the coolant can be switched on again.

o Regardless of the door status, the coolant can be switched off and on in all modes with the exception of DRYRUN.

o CYCLE START key not pressed:
When the chip guard door is opened, the coolant is switched off.

o CYCLE START key pressed:
The chip guard door is opened with active coolant:

D_{01} = 1 alarm 400 and coolant OFF
D_{01} = 3 alarm 400 and coolant OFF.
D_{01} = 7 The coolant is not switched off.
D_{01} = 8 The coolant is switched off.

D_{02}/D_{03} (O_{40} bit 4, O_{40} bit 5) piece counter and counter presetting

1. Workpiece display

Activation of the display
Set O_{40} /bit 4 high

The workpiece counter can be activated in the user monitor MON by setting the parameter O_{40}/bit 4. In the automatic mode the number of program runs (workpieces) is then displayed (except with an alarm). The number increases by 1 with every M30 (program end).

Set the workpiece counter D_{03}

With the parameter D_{03} the number of the workpiece counter can be set (e.g.: reset to 0 by entering $D_{03} = 0$).

Note:
The number range of the counter covers values from 0 to 32,767; above this the counter is automatically reset to 0.

2. Presetting the counter O_{40}/bit 5

You can enter a specific number. By pressing the start key once, the program repeats itself according to the preset number automatically as CYCLE START is automatically activated when M30 is reached. By activating the key 1x (single workpiece) the automatic CYCLE START can be suppressed and program execution stops at every M30.

Activation:
Set parameter O_{40}/bit 5.

Entry of the number of automatic runs D_{02} (target number)

Enter the number under parameter D_{02}.

Example:
16 automatic runs

Entry: $D_{02} = 16$
After 16 runs the program stops.

G-parameters - Tool turret data

G_{08} bit 0,1 ... Activation of the tool turret

G_{08} bit 0 = 0 (low)
G_{08} bit 1 = 0 (low) all safety functions active.
G_{08} bit 0 = 1 (high)
G_{08} bit 1 = 1 (high) reduced safety functions.

If G_{08} bit 0 is set high:

Keying in MANUAL mode during indexing of the tool turret (delivery condition: G_{08} bit 0=0).

If G_{08} bit 1 is set high:

If the tool turret is moved into the indexing position, the FEED OVERRIDE switch become ineffective.
The feed rate is 100% for the duration of the indexing operation (delivery condition: G_{08} bit 1 = 1).

M-parameters - Main drive data

M_{08}... Establishing the spindle position

If a block with M19 is programmed in the NC program without the indication of an S-value, the control approaches the spindle position specified under M08 in the user monitor when this block is executed.
Input: (°)
M_{08} = 0 has been set at the works.

0-parameters - General setting data

Parameter 0_{00}

	Bit 0	Bit 1
	Interface data display	RS232: Output of an auto. leader/ trailer
Status with bit = 0 (low)	No display	No leader/ trailer
Value	0	
Status with bit = 1 (high)	Display	Output of an auto. leader/ trailer
Value	1	2

With the parameter 0_{00} bit 0 you can determine the display of the data during the loading process (MON).

0_{00} bit 0:

1) 0_{00} bit 0 low:
During loading no display of the loaded data.

2) 0_{00} bit 0 high:
This mode is used for direct editing via an external keyboard, e.g. teletype, PC. During the entry of 0 numbers (program numbers) no check is made to see whether a program of this number exists (no message "exists"). As a result it is possible to change a program in the memory or the current offsets. For detailed notes, see interface mode in the operating instructions EMCOTRONIC TM 02.

0_{00} bit 1:

If 0_{00} bit 1 = 1 (high) is set:
A leader and trailer of each 50 ASCII "NULL characters" are provided during reading out.

O_{01} Parameters

Establishing the data format for the standard interface

	Bit 0	Bit 1	Bit 2	Bit 3	Bit 4	Bit 5	Bit 6	Bit 7
	Data format	End work-piece prog.	Length of the individual character		Parity check	Parity Odd/ even	Number of stop bits	
atus t=0 V	EMCO internal	No ctrl			No parity check disable	Odd parity		
lue	0	0	0	0	0	0	0	0
itus :=1 jh	ISO	ctrl z			Parity check enable	Even parity		
ue	1	2	4	8	16	32	64	128

2	Bit 3	
ow) ue=0	0(low) value=0	invalid
igh) ue=4	1(high) value=0	invalid
ow) ue=0	1(high) value=8	7 bits
igh) ue=4	1(high) value=8	8 bits

Bit 6	Bit 7	
0(low) value=0	0(low) value=0	invalid
1(high) value=64	0(low)	1 stop bit
0(low) value=0	1(high) value=128	1 1/2 stop bits
1(high) value=0	1(high) value=128	2 stop bits

O_{00} bit 0:
Bit 0 = low
EMCO-internal data format, only for EMCO test purposes.
Bit 0 = high: ISO format (see also data formats EMCOTRONIC).

O_{01} bit 1:
Bit 1 = low
No "ctrl z" at the end of data transmission.
Bit 1 = high
A "ctrl z" control character is inserted at the end of data transmission.

O_{01} bit 2/bit 3:
Bit 2 and bit 3 are combined. The character length can be established with them.
Character lengths of 7 or 8 bits are usual.

O_{01} bit 4:
Establishing whether a parity check is to be performed or not

0_{01} bit6/bit7:
Establishing the number of stop bits. This number depends on the peripheral unit connected (see description of peripherals).

0_{01} bit 5:
Bit 5 is used to establish whether a check for an even or odd sum is to be made. This check is, of course, dispensed with if no parity check has been specified in bit 4.

Example parameter 0_{01}

		Value
Bit 0	ISO format	1
Bit 1	No ctrl z	0
Bit 2	Character	0
Bit 3	7 bits	8
Bit 4	Parity check	16
Bit 5	Even parity	32
Bit 6	1 stop bit	64
Bit 7		0
Input value 0_{01} = 121		

0_{02} Establishing the number of tool data to be stored

With 0_{02} the number of tool data which are to be stored on external data carriers can be established.

Input range: 0-99
0_{02} = 99 has been set at the works.
The 5 PSO registers are always stored even with 0_{02} = 0.
See also interface mode PSO and TO data.

Data Formats EMCOTRONIC

Input in EMCOTRONIC:

The program input to the interface RS 232C is basically done as with an input via the control board.

The sequence of characters sent must be in accordance with the exact sequence of operating keys on the EMCOTRONIC. Therefore it is necessary to know the data input procedure on the EMCOTRONIC - compare operating manual EMCOTRONIC.

The translation of the EMCOTRONIC instructions (e.g. ENTER, PREVIOUS) in ASCII codes you can find in the translation chart.

There are devices with which you can edit directly to the machine. The entered values can be seen on the monitor of the EMCOTRONIC. For this the interface read out has to be activated. (Parameter O_{00} Bit 0 has to be set to High; value for Bit 0 High = 1, compare user monitor).

Further Remarks:

- Programs can be started instead of % also with the letter "0". All characters before the first % or "0" are ignored.
- Commantaries can be written between round brackets on external devices. These contents in round brackets are not taken-over to the EMCOTRONIC when transfering data.
- The read-in procedure will be automatically finished by the EMCOTRONIC if there is a M30 instruction at the end of the block. If there is no M30 at the end of the block the transfer procedure will not be interruppted.
  (Purpose: Various programs can be entered one after the other)
- Automatic Start of the Read-in Operation:

  With O Zi Zi INP or with
  O INP (O flahes)
  The read-in procedure is automatically started.

Edit of EMCOTRONIC to External Devices:

Edit can be done in two formats.The edit mode can be determined in the user monitor.

ISO Format

User monitor:
Parameter O_{01} Bit 0 has to be set High (value = 1)

Program format:
% ZiZi ␣ crlf N ZiZiZiZi ␣ ␣ / ␣ GZiZi ␣
M ZiZi crlf N ZiZiZiZi ␣ ␣ PZi = ZiZi.ZiZiZi ␣
DZi = ZiZiZi crlf X ZiZi.ZiZiZi crlf

Zi = means number

Translating Chart

ASCII-character	Generation on external keyboard	Hex-Code	Interpretation by EMCOTRONIC	
			ISO-Format*	EMCO-Format*
NUL	ctrl Space Bar	ØØ	-	
SOH	ctrl A	Ø 1	-	
STX	ctrl B	Ø 2	C.B.	
ETX	ctrl C	Ø 3	-	
EOT	ctrl D	Ø 4	-	
ENQ	ctrl E	Ø 5	-	ENTER
ACK	ctrl F	Ø 6	-	
BEl	ctrl G	Ø 7	-	
BS	ctrl H/Backspace	Ø 8	SHIFT/ENTER	
HT	ctrl I/Tabulator	Ø 9	-	
LF	ctrl J/Line feed	Ø A	STORE/NEXT	-
VT	ctrl K	Ø B	-	
FF	ctrl L	Ø C	-	
CR	ctrl M/return	Ø D	ENTER	-
SO	ctrl N	Ø E	-	NEXT
SI	ctrl 0	Ø F	-	
DLE	ctrl P	1 Ø	PREVIOUS	
DC1	ctrl Q	11	-	
DC2	ctrl R	12	-	
DC3	ctrl S	13	SHIFT	
DC4	ctrl T	14	-	
NAK	ctrl U	15	-	
SYN	ctrl V	16	-	
ETB	ctrl W	17	C.W.	
CAN	ctrl X	18	-	
EM	ctrl Y	19	-	
SUB	ctrl Z	1A	-	
ESC	ctrl [/ESC	1B	"Escape" getting out of the interface mode	
FS	ctrl \	1C	-	
GS	ctrl]	1D	-	
RS	ctrl ~	1E	-	
US	ctrl ?	1F	-	
SP	Space bar	2 Ø	ENTER	
!	!	21	-	
"	"	22	-	
#	#	23	-	
$	$	24	-	
%	%	25	0	
&	&	26	-	
\	\	27	-	
(	(	28	(	
)	)	29	)	
*	*	2A	-	
+	+	2B	-	
,	,	2C	-	
-	-	2D	change sign ±	
.	.	2E	decpoint	
/	/	2F	/	
Ø	Ø	3 Ø	Ø	
1	1	31	1	
2	2	32	2	
3	3	33	3	

* Can be set in user monitor (MON) under L4: Bit Ø = Ø ...Emco
Bit Ø = 1 ...ISO

Chart Continuation

ASCII-character	Generation on external keyboard	Hex-Code	Interpretation by EMCOTRONIC (both Formats)
4		34	4
5		35	5
6		36	6
7		37	7
8		38	8
9		39	9
:		3A	-
;		3B	-
<		3C	-
=		3D	-
>		3E	-
?		3F	-
@		4Ø	-
,a		41,61	-
,b		42,62	-
,c		43,63	-
,d		44,64	D
,e		45,65	-
,f		46,66	F
,g		47,67	G
,h		48,68	-
,i		49,69	I
,j		4A,6A	J
,k		4B,6B	K
l		4C,6C	L
m		4D,6D	M
n		4E,6E	N
o		4F,6F	O
p		50,70	P
q		51,71	-
r		52,72	R
s		53,73	S
t		54,74	T
u		55,75	U
v		56,76	V
w		57,77	W
x		58,78	X
y		59,79	Y
z		5A,7A	Z
	Like ASCII-character	5B	-
		5C	-
		5D	-
		5E	-
		5F	-
		6Ø	-
		7B	-
		7C	-
		7D	-
		7E	-
	delete	7F	CE

O_{03} Establishing for the interface mode

O_{03} bit 0:
$I \neq O_{03}$ bit 0 = 1 (high) is set, the programs already existing in the control memory are overwritten when the INPUT ALL softkey is activated (no "ALREADY EXISTS" message appears on the screen).

O_{03} bit 1:
$I \neq O_{03}$ bit 1 = 1 (high) is set, the programs already exsiting on the cassette are overwritten. (No "ALREADY EXISTS" message appears on the screen).

Parameter O_{11}

	Bit 0	Bit 1	Bit 2	Bit 3	Bit 4
	NOT OCCUPIED	Initial status G70/G71	NOT OCCUPIED	Initial status tool turret	Initial status auto. door facility
Status with bit=0 (low)		metric (G70)		M50	M52
Value	0	0	0	0	0
Status with bit=1 (high)		inches (G70)		M51	M53
Value	1	2	4	8	16

O_{11} bit 3... Initial status direction logic - tool turret

With tool turret with direction logic:
M50 - cancel direction logic
O_{11} bit 3 = 0 (low) value 0

M51 - select direction logic
O_{11} bit 3 = 1 (high) value 8

O_{11} bit 4 ... Initial status automatic door facility

With automatic chip guard door:
O_{11} bit 4 = 0 (low) value 0
M52 = cancellation of automatic door facility.

O_{11} bit 4 = 1 (high) value 16
M53 = selection of automatic door facility.

- If M53 is active and CYCLE START is pressed, the automatic chip guard door is closed and then the NC program executed.
- M00 and M30 cause the automatic chip guard door to open.
  Exception: M30 with automatic CYCLE START (counter presetting).

O_{12}... Initial status of the G-functions of group 9

The initial status of the axis system (G17-G22) is established with parameter O_{12}.
O_{12} = 17 has been set at the works.

Note:
The axis system for the vertical milling machines VMC-100 and VMC-200 is established with G17. There are no other possible applications for other G-functions of the group 9 (G18-G22) with these machines.

O_{22} Establishing the lowest valid subroutine number

The lowest valid subroutine number is established with O_{22}.
The highest valid subroutine number is established with O 0255.
O_{22} = 80 has been set at the works.

Example:
O_{22} = 75
Subroutines can be entered from subroutine number O 0075 to O 0255.

Parameter O_{40}

	Bit 0	Bit 1	Bit 2	Bit 3	Bit 4	Bit 5	Bit 6	Bit 7
	Software limit switch MANUAL	NOT OCCUPIED	Memory lock	Control lock	Counter	Counter preset-ting	Contour infring. with active G41/G42	FMS mode
Status with bit = 0 (low)	active		No lock	No lock	not active	not active	ALARM 570	not active
Value	0	0	0	0	0	0		0
Status with but = 1 (high)	not active		Lock	Lock	active	active	No ALARM 570	active
Value	1	2	4	8	16	32	64	128

O_{40} bit 0: Value set to 1 (high)

The software limit switches are ignored in Manual mode. The bit is automatically cancelled again when RESET is activated again (exception: RESET in the T/PSO mode or in the monitor).

O_{40} bit 2: Memory lock

Bit 2 = 1 (high) input value 4

In the Edit mode it is not possible to select a program and change it; changing PSO and tool data is only incremental with the arrow keys.

Reason for memory lock:
Unauthorised persons should not be able to change the program and a mistake in changing offset data is to be avoided.

O_{40} bit 3: Control lock

Bit 3 = 1 (high) input value 8

The entire control is locked. The screen only displays EDIT. All functions are locked apart from the main switch and EMERGENCY-OFF.

O_{40} bit 4... Activate counter

See counter parameter D_{02}/D_{03}

O_{40} bit 5 ... Mode - counter presetting

See counter parameter D_{02}/D_{03}.

O_{40} bit 6:

If O_{40} bit 6 = 64 (high) is set:

If the control calculates a contour infringement with the radius compensation active, no ALARM 570 is given.
O_{40} bit 6 = 0 has been set at the works.

O_{40} bit 7 ... Selection of the FMS mode

O_{40} bit 7 = 0 has been set at the works.

S-parameters - Setting parameters for cycles

S_{00} Establishing the minimum infeed for G83, G86, (input in µm)

S_{00} = 100 µm has been set at the works.

S_{02} Establishing the withdrawal movement for G86 (input in µm)

S_{02} = 500 µm has been set at the works.

R-parameters - Machine-specific position data

The numerical values of the R-parameters depend on the machine version (for dimensions, see operating instructions of the machine in question).

R_{00} Reference point in X-direction

R_{01} Reference point in Y-direction

R_{02} Reference point in Z-direction

R_{03} Software limit switch in +X direction

R_{04} Software limit switch in +Y direction

R_{05} Software limit switch in +Z direction

R_{06} Software limit switch in -X direction

R_{07} Software limit switch in -Y direction

R_{08} Software limit switch in -Z direction

R_{09} Safety distance from the software limit switch in the MANUAL mode

Within this range a slow safety feed rate is automatically selected.

Chapter 9

Interface RS 232

EMCOTRONIC TM 02

The Serial Interface RS 232c of the EMCOTRONIC

1. Technical data

Function:

Data can be read out and in via the interface. The most frequent peripheral equipment is paper tape reader, paper tape punch, printer, computer. Computer specific software is required for computer connections, whilst the other equipment can be connected directly.

Data format EMCOTRONIC:

* 1 start bit
* 7 data bits
* 1 parity bit: for even parity, i.e. even number of holes on the paper tape for monitoring transmission errors.
* 1 stop bit

Baud rate EMCOTRONIC:

150 - 2400 BAUD
The baud rate is programmable in the MON operating mode under DØ.
Note: The baud rate of the EMCOTRONIC - Peripheral Equipment, must agree.

Electrical data V24:

Voltage: + 12 V = logic 0
- 12 V = logic 1

Electrical data 20 mA:

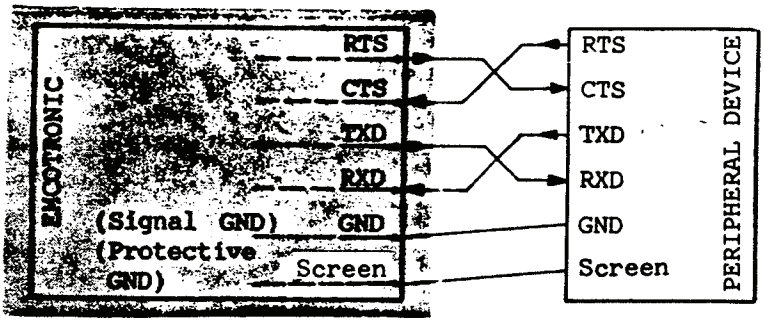
Current: 20 mA = logic 1
0 mA = logic 0

Plug type RS 232c:

EMCOTRONIC	Peripheral equipment
25 terminal RS 232c plug (male)	see equipment description

2. The general connection RS 232c of two devices via V24

2.1 No internal cross-bonding



Meaning of the individual pins:

RTS: request to send = Output : Device requests data

CTS: clear to send = Input = Device indicates readiness to send data

TXD : Transmit data = Data output

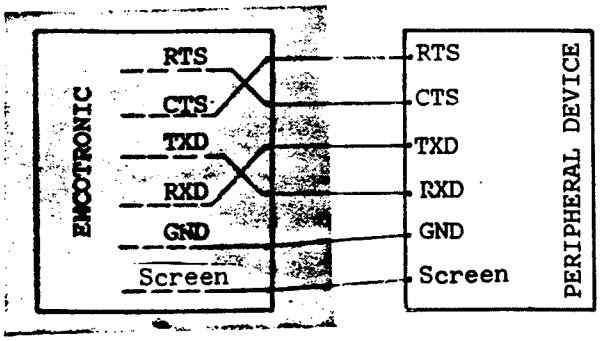
RXD: Receive data = Data input

GND: Ground = Earth

Screen = Cable screen

2.2 Internal cross-bonding 1 x

In the EMCOTRONIC V24 interface (not with 20 mA), the lines are already internally cross-bonded. For this reason, the following connection diagram must be observed



Precondition:

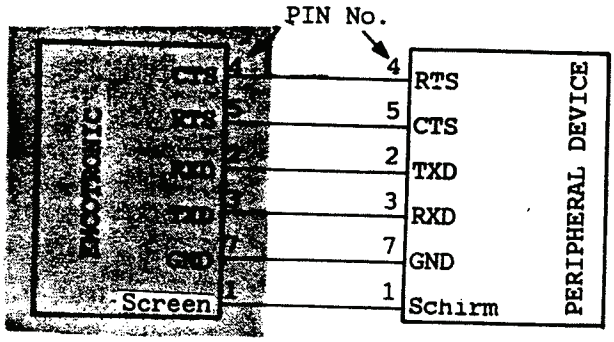
The manufacturer of the 2nd device has not already internally cross-bonded the lines (which is normally the case).

2.3 Internal cross-bonding 2 x

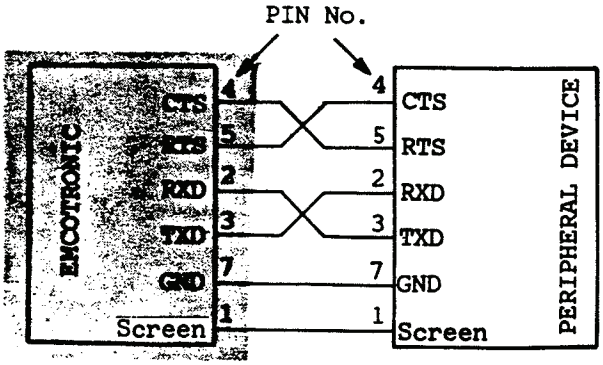
Where the manufacturer of the peripheral device has already made internal cross-bonding, the cable must be cross-bonded. Diagram, see above

3. RS 232c-V24 connection EMCOTRONIC

3.1 EMCOTRONIC is cross-bonded
Peripheral device is not cross-bonded



3.2 EMCOTRONIC is cross-bonded
Peripheral device is cross-bonded



4. 20 mA - Connection EMCOTRONIC

4.1 General

The 20 mA interface of the EMCOTRONIC is not cross-bonded internally.

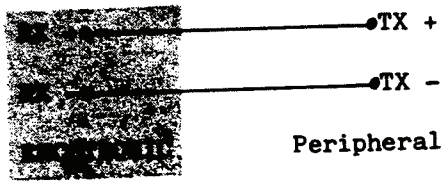
Two loops must be drawn due to the FULL DUPLEX operation of the EMCOTRONIC 20 mA interface.

* One for transmission
* One for reception

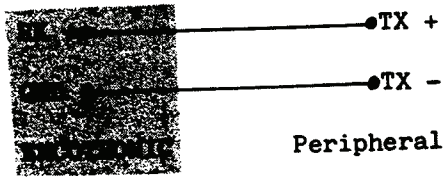
Transmission loop RS 232c

Reception loop RS 232c

Passive peripheral device



Active peripheral device



With the reception loop, there is a difference, depending on whether the peripheral device is active or passive in effect.

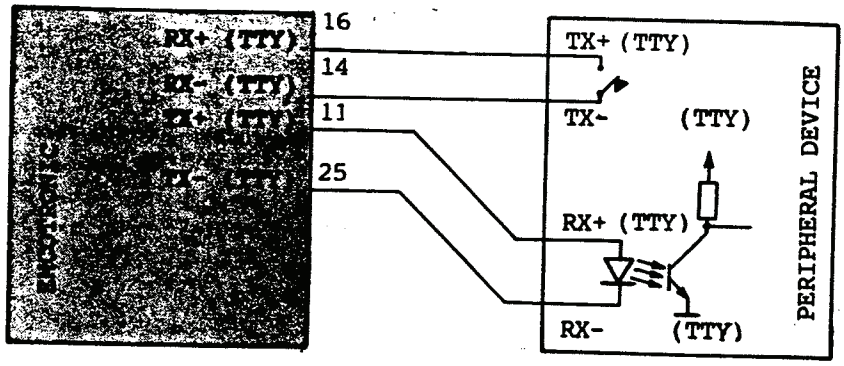
Active device: Itself supplies the signal current.

Passive device: Switches the signal current of the EMCOTRONIC On and Off (no own signal current present).

4.2 Circuit diagram 20mA RS 232c

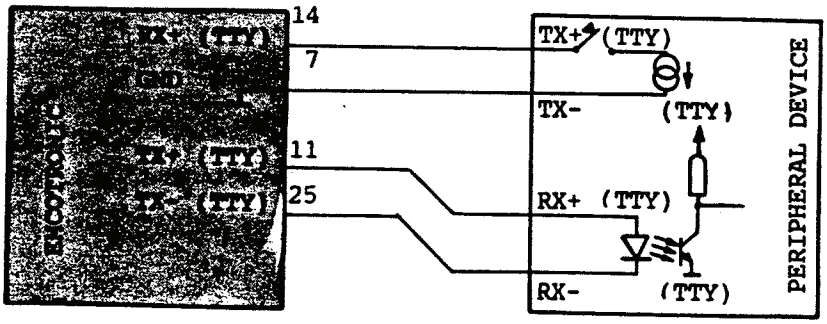
4.2.1 Passive transmitter device

(usual design of the peripheral equipment)



4.2.2 Active transmitter

(infrequent design of the peripheral equipment)

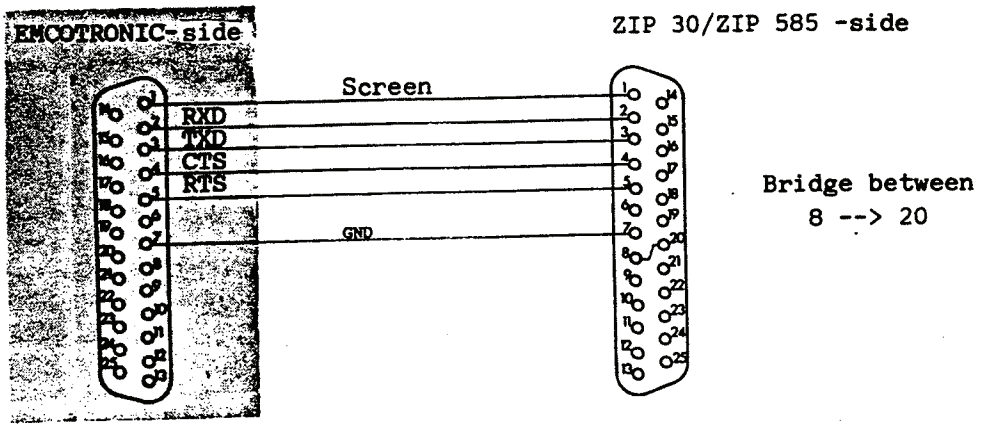


5. Connection occupancy of the equipment offered by EMCO

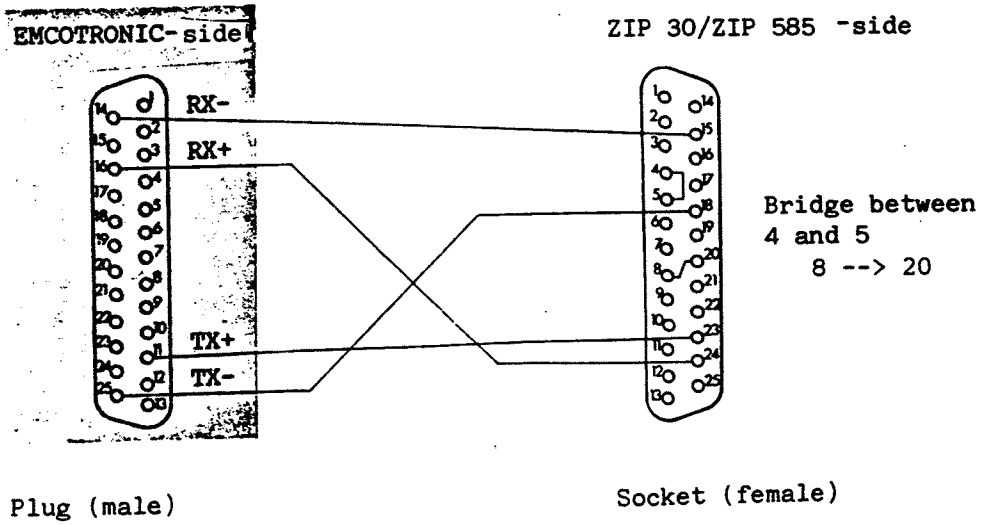
* Matrix printer with attached paper tape reader, punch; RS 232c (V24 or 20 mA interface)
  Teledynamics ZIP 30 KSR, ZIP 30 ASR, ZIP 30 ASR/ EDIT 1600
  Order No.: 573 470

* Portable paper tape reader, punch with RS 232c (V24 or 20 mA interface)
  Teledynamics ZIP 585
  Order No.: 573 480

5.1 Connection occupancy when using the V24 interface:



5.2 Connection occupancy when using the 20 mA interface:



Data Formats EMCOTRONIC

Input in EMCOTRONIC:

The program input to the interface RS 232C is basically done as with an input via the control board.

The sequence of characters sent must be in accordance with the exact sequence of operating keys on the EMCOTRONIC. Therefore it is necessary to know the data input procedure on the EMCOTRONIC - compare operating manual EMCOTRONIC.

The translation of the EMCOTRONIC instructions (e.g. ENTER, PREVIOUS) in ASCII codes you can find in the translation chart.

There are devices with which you can edit directly to the machine. The entered values can be seen on the monitor of the EMCOTRONIC. For this the interface read out has to be activated. (Parameter O_{00} Bit 0 has to be set to High; value for Bit 0 High = 1, compare user monitor).

Further Remarks:

- Programs can be started instead of % also with the letter "0". All characters before the first % or "0" are ignored.
- Commantaries can be written between round brackets on external devices. These contents in round brackets are not taken-over to the EMCOTRONIC when transfering data.
- The read-in procedure will be automatically finished by the EMCOTRONIC if there is a M30 instruction at the end of the block. If there is no M30 at the end of the block the transfer procedure will not be interruppted.
  (Purpose: Various programs can be entered one after the other)
- Automatic Start of the Read-in Operation:

  With O Zi Zi INP or with
  O INP (O flahes)
  The read-in procedure is automatically started.

Edit of EMCOTRONIC to External Devices:

Edit can be done in two formats.The edit mode can be determined in the user monitor.

ISO Format

User monitor:
Parameter O_{01} Bit 0 has to be set High (value = 1)

Program format:
% ZiZi ␣ crlf N ZiZiZiZi ␣ ␣ / ␣ GZiZi ␣
M ZiZi crlf N ZiZiZiZi ␣ ␣ PZi = ZiZi.ZiZiZi ␣
DZi = ZiZiZi crlf X ZiZi.ZiZiZi crlf

Translating Chart

ASCII-character	Generation on external keyboard	Hex-Code	Interpretation by EMCOTRONIC	
			ISO-Format*	EMCO-Format*
NUL	ctrl Space Bar	ØØ	-	
SOH	ctrl A	Ø 1	-	
STX	ctrl B	Ø 2	C.B.	
ETX	ctrl C	Ø 3	-	
EOT	ctrl D	Ø 4	-	
ENQ	ctrl E	Ø 5	-	ENTER
ACK	ctrl F	Ø 6	-	
BEl	ctrl G	Ø 7	-	
BS	ctrl H/Backspace	Ø 8	SHIFT/ENTER	
HT	ctrl I/Tabulator	Ø 9	-	
LF	ctrl J/Line feed	Ø A	STORE/NEXT	-
VT	ctrl K	Ø B	-	
FF	ctrl L	Ø C	-	
CR	ctrl M/return	Ø D	ENTER	-
SO	ctrl N	Ø E	-	NEXT
SI	ctrl O	Ø F	-	
DLE	ctrl P	1 Ø	PREVIOUS	
DC1	ctrl Q	11	-	
DC2	ctrl R	12	-	
DC3	ctrl S	13	SHIFT	
DC4	ctrl T	14	-	
NAK	ctrl U	15	-	
SYN	ctrl V	16	-	
ETB	ctrl W	17	C.W.	
CAN	ctrl X	18	-	
EM	ctrl Y	19	-	
SUB	ctrl Z	1A	-	
ESC	ctrl [/ESC	1B	"Escape" getting out of the interface mode	
FS	ctrl \	1C	-	
GS	ctrl]	1D	-	
RS	ctrl ~	1E	-	
US	ctrl ?	1F	-	
SP	Space bar	2 Ø	ENTER	
!	!	21	-	
"	"	22	-	
#	#	23	-	
$	$	24	-	
%	%	25	0	
&	&	26	-	
\	\	27	-	
(	(	28	(	
)	)	29	)	
*	*	2A	-	
+	+	2B	-	
,	,	2C	-	
-	-	2D	change sign ±	
.	.	2E	decpoint	
/	/	2F	/	
Ø	Ø	3 Ø	Ø	
1	1	31	1	
2	2	32	2	
3	3	33	3	

* Can be set in user monitor (MON) under L4: Bit Ø = Ø ...Emco
Bit Ø = 1 .:.ISO

Chart Continuation

ASCII-character	Generation on external keyboard	Hex-Code	Interpretation by EMCOTRONIC (both Formats)
4		34	4
5		35	5
6		36	5
7		37	7
8		38	8
9		39	9
:		3A	-
;		3B	-
<		3C	-
=		3D	-
>		3E	-
?		3F	-
@		4Ø	-
A,a		41,61	-
B,b		42,62	-
C,c		43,63	-
D,d		44,64	D
E,e		45,65	-
F,f		46,66	F
G,g		47,67	G
H,h		48,68	-
I,i		49,69	I
J,j		4A,6A	J
K,k		4B,6B	K
L,l		4C,6C	L
M,m		4D,6D	M
N,n		4E,6E	N
O,o		4F,6F	O
P,p		50,70	P
Q,q		51,71	-
R,r		52,72	R
S,s		53,73	S
T,t		54,74	T
U,u		55,75	U
V,v		56,76	V
W,w		57,77	W
X,x		58,78	X
Y,y		59,79	Y
Z,z		5A,7A	Z
[		5B	-
\		5C	-
]		5D	-
^		5E	-
_		5F	-
`		6Ø	-
{		7B	-
|		7C	-
}		7D	-
~		7E	-
DEL	delete	7F	CE

Like ASCII-character

6. Circuit diagram RS 232c