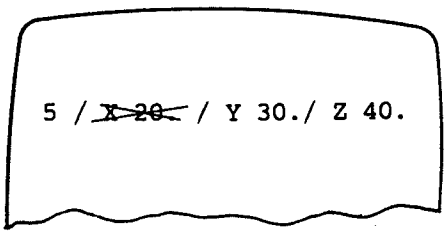
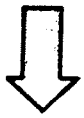
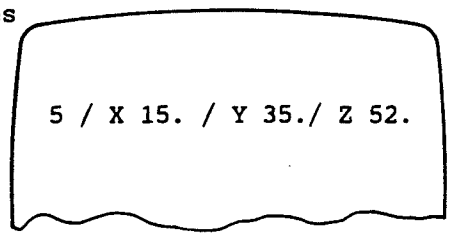
Old values





Active values



3. Mixed Values

If the measurements are indicated under G92 in a mixed up sequence, absolute with X,Y,Z and incremental with U,V,W then

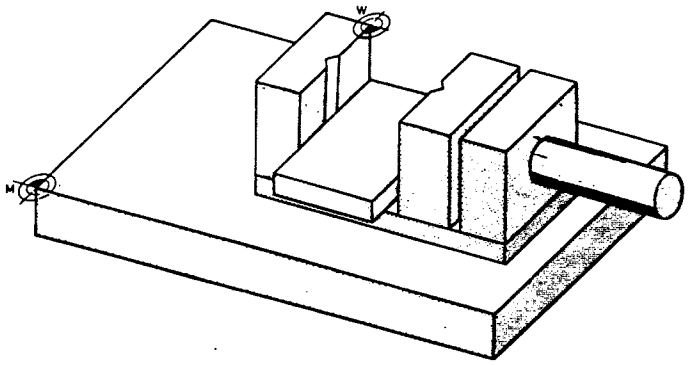
* the absolute G92 measurements are taken over into the register.
* The incremental G92 measurements are added to values of the position shift offset.

N..../G92/X15./V +5./W +12./

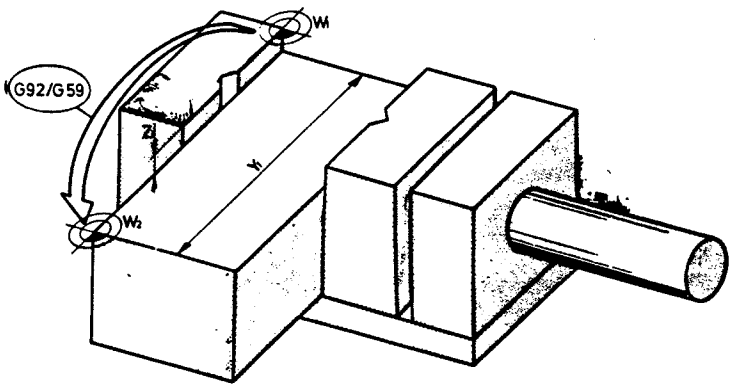
EXAMPLE 2:

2. Offset with Group 3 Instructions and Offset in Workpiece Program with G92 , G59

This proves to be quite a practical method:



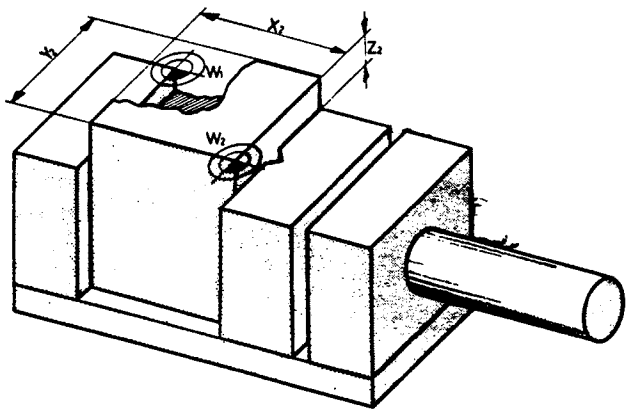
* The zeropoint is offset from M to W_1 using group 2 instruction.
* From this point the workpiece zeropoint of the respective workpiece W_2 is programmed using G92, G59.



Example 1:

N..../G54

N..../G92/X_1 /Y_1 /Z_1 /G59



Example 2:

N..../G54

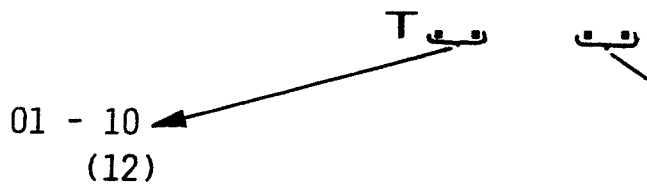
N..../G92/X_2 /Y_2 /Z_2 /G59

Tool Programming and Compensation

1. T-Address:

Tools are programmed under the T-address using a 4-digits number.

Code



Tool Number

The first two digits are the tool number (number of tool position on tool pallet with automatic tool change).

For VMC-100: 10 tool numbers admissible

For VMC-200: 12 tool numbers admissible

Tool Compensation Number

Number code for tool data (length, radius).
The tool compensation number is listed in the tool data memory.

2. Call-up

Every new T-address has to be called-up with a G00 block (otherwise Alarm sign).

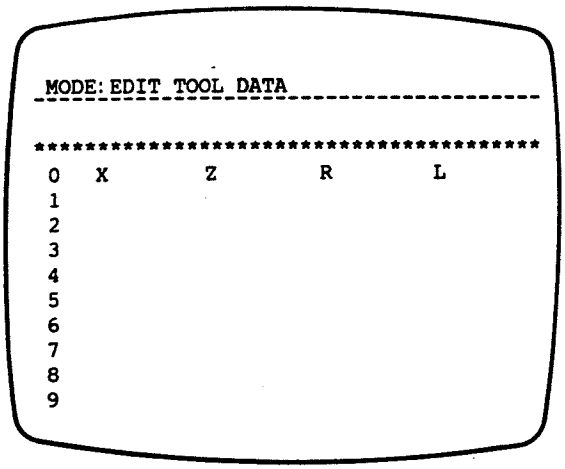
Example: Call-up in same block with G00

```
N  90 / M00
N 100 / G00 / X.../Y.../Z.../ T02 02
```

Example: After the T-call-up a G00 traverse instruction follows.

```
N 100 / T02 02
N 110 / G94 / F 130
N 120 / G54
N 130 / G00 / X.../Y.../Z.../
```

6. Input - Compensation:



The tool data are entered into the tool memory in mode EDIT.

1 corresponds to correction T.. 01
20 corresponds to correction T.. 20

Tool length: under Z-address
Radius: under R

Compensation:

If a tool is called-up in the program the computer fetches the data Z(length) and R (radius) which were put in under the code number.

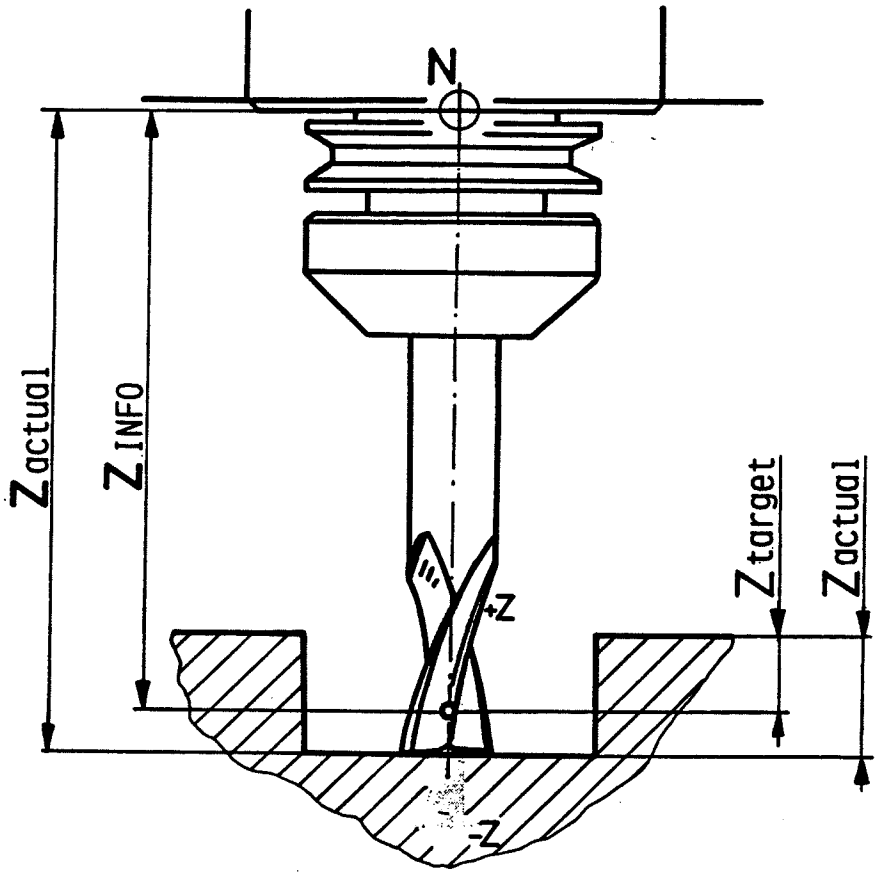
7. Programming Hint:

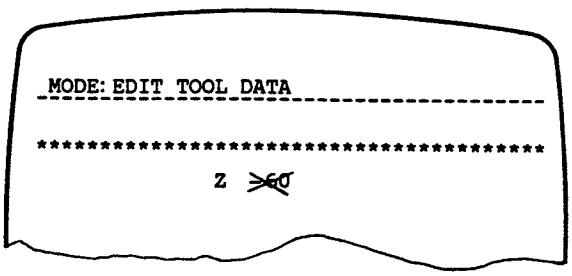
Number of compensation and of tool need not be the same; e.g. T 05 01. For a better overlook it is useful that compensation number and tool number are identical.

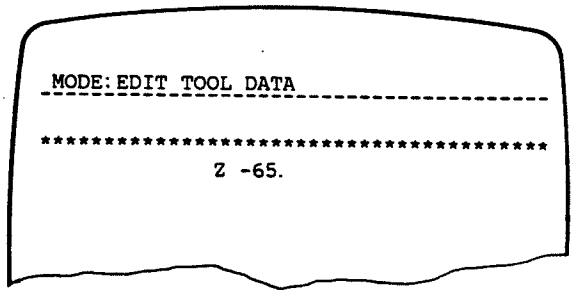
For VMC-100 the numbers 01 - 10 can be entered as tool numbers. If a tool number greater than 10 is programmed, an ALARM is given.

For VMC-200 the numbers 01 - 12 can be entered as tool numbers. If a tool number greater than 12 is programmed, an ALARM is given.

Correction of the Length Data





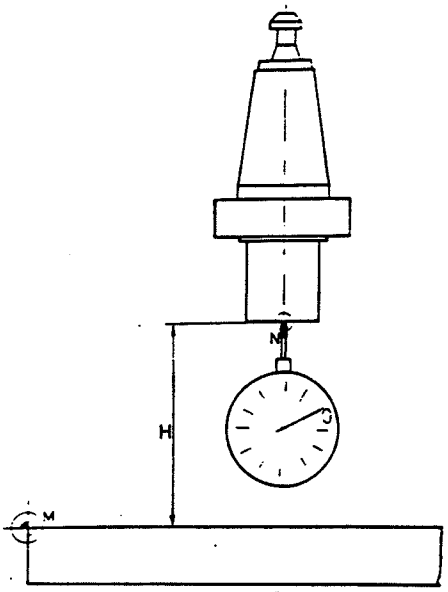


* Imagine the coordinates system in the target value.
* Measure difference between target value and actual value: Z = -5mm
* Add this value with the correct sign to the value in the tool data memory.

  Z - 60 mm + (-5mm) = -65 mm
* Write this value into the tool data memory.

Direct take-over of the tool lengths - VMC-100

MAN Mode



With EMCOTRONIC TM 02 it is possible to take over the tool lengths directly. This is a very accurate and comfortable method.

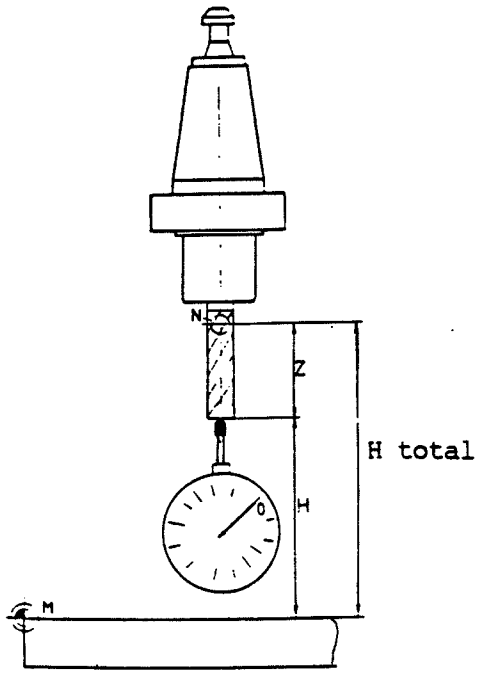
Option 1: Touching with dial gauge

* Select manual mode.
* Mount reference tool and swivel in.
* Touch dial gauge with reference tool. Set dial gauge to zero.

TO 0 0

ENTER

The height H is stored in the control by TO 00 ENTER.



* Swivel in the tool to be measured and touch dial gauge (gauge must show 0).

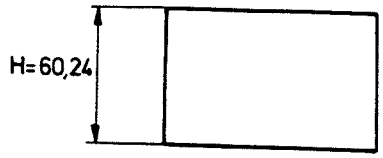
TO 0 1 Tool correction number wanted

ENTER

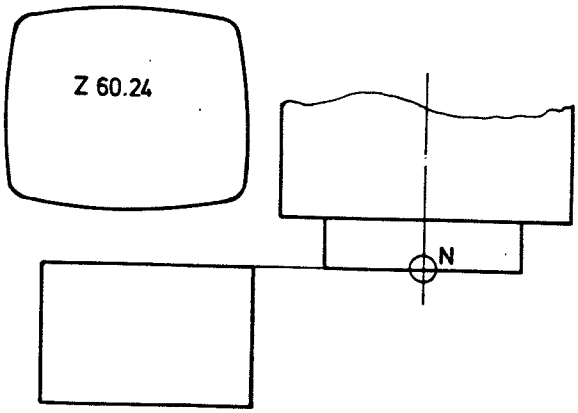
The tool length Z is stored under tool correction number T..01.

Scratching a trial workpiece

MAN Mode



1. Measure the height (H) of the trial workpiece to be scratched.

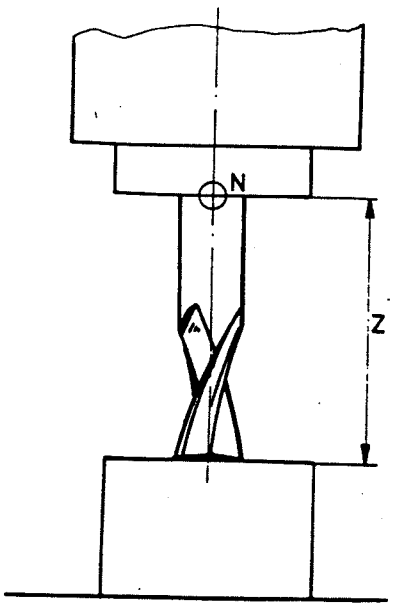


2. Move the point N to the Z-value H (screen display)

TO 0 0

ENTER

The reference measurement H is stored.



3. Scratch the workpiece surface.

TO 0 1

ENTER

current tool number

The measurement Z is stored under the compensation number T..01.

G40 Neutralization of the Cutter Tool Correction
G41 Cutter Path Correction left-hand
G42 Cutter Path Correction right-hand

Index G40/G41/G42

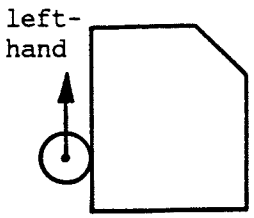
Terminology:

G40 Neutralization of the Cutter Tool Correction

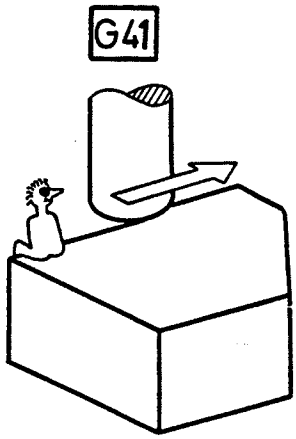
Cutter tool correction is the official DIN term. One means the cutter tool path correction and not the cutter tool length correction.

2. Definition G41,G42

G41 Cutter Path Correction left-hand



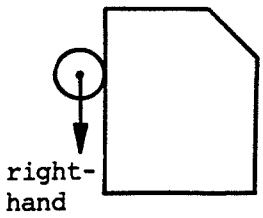
The tool is on the left-hand side of the workpiece, looked at in direction of the relative tool movement.



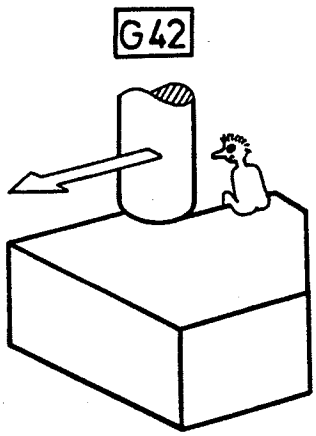
Rule of Memorisation:

Sit on the workpiece and look in direction of the feed.
If the tool is left-hand - G41.

G42 Cutter Path Correction right-hand



The tool is on the right-hand side of the workpiece. looked at in direction of relative tool traverse.



Rule of Memorisation:

Look after the cutter, the cutter is on the right-hand side of the workpiece - G42.

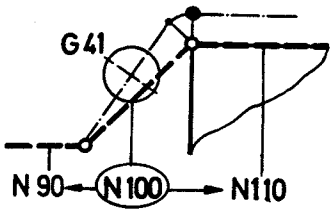
G40 Neutralization of the Cutter Correction

The programmed path is again the center path of the cutter.
M30 neutralizes also the cutter correction (G40 status).

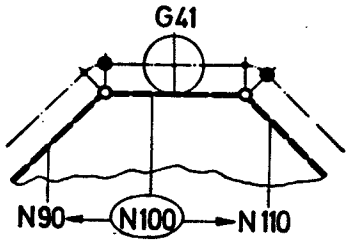
3.5 What Happens in the Computer

When a radius compensation is on, the computer knows the previous and the following traverse instruction in the plane of compensation.

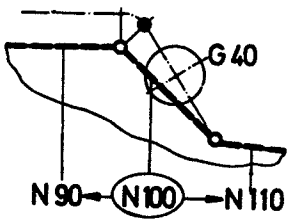
Activation Block



Whilst G41, G42 are active



When Disactivating



Pay Attention:

You must not program more than five "empty blocks" between the call and the first change of XY-value, or while G41, G42 is active.
"Empty blocks" are without a change in the XY-value. Pure "Z-blocks"are empty blocks, too.

Example: Activation

	N ...	G40
	N 90/	G00 / X_0 / Y_0
	N 100	G41
①	N 110	M03 / S 1000
②	N 120	M39
③	N 130	G94 / F 120
④	N 140	G00 / X_0 / Y_0 / Z_1
⑤	N 150	M08
⑥	N 160	F 180
	N 170	G00 / X_1 Y_1 Z_1

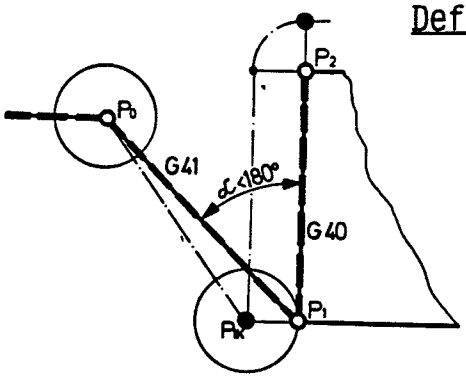
The computer indicates alarm 500

Example: 5 "Empty Blocks" with G41,G42 activ

N .../ G41
N 90 / G00 / X_0 Y_0 Z_0
N 100 / G00 / X_1 Y_1 Z_1
N 110 / M04
N 120 / G94 .....
N 130 .....
N 140 ..... Alarm 510
N 150 .....Z_2
N 160 .....
N 170 / G01 / X_2 / Y_2 / Z_2

4.1.2 Interior Corner $\alpha < 180°$

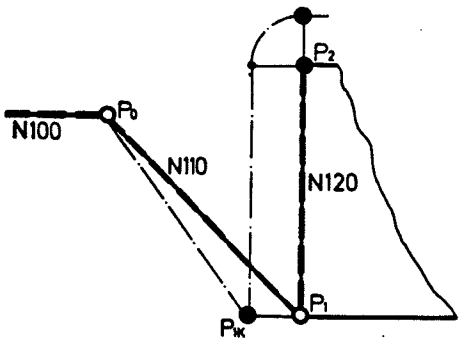
Definition:



The angle α between programmed approach path and P_1P_2 is smaller 180°

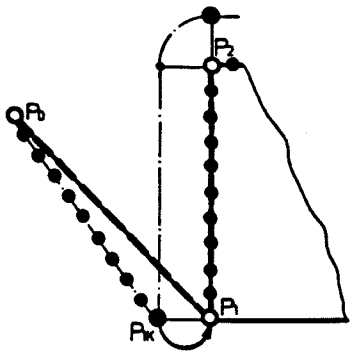
Programming:

N 100/G../XYZP$_0$/(G40)
N 110/G../XYZP$_1$/G41
N 120/G../XYZP$_2$/....



Programmed path: $\overline{P_0P_1}$
The point P_{1K} is directly approached, $\overline{P_1P_{1K}}$ is perpendicular to the programmed path $\overline{P_1P_2}$.

Position Read-out

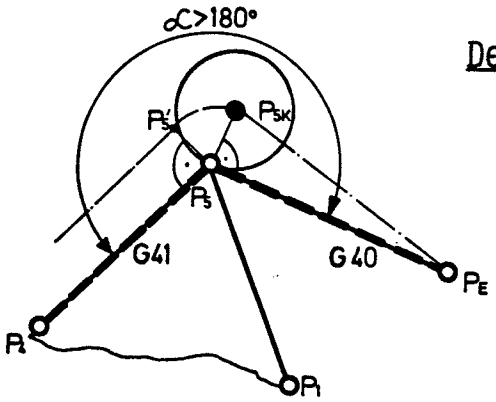


Center path of cutter $\overline{P_0\ P_{1K}}$
When P_{1K} is reached the read-out jumps to P_1.

- 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 Disactivation of Cutter Radius Compensation with G40

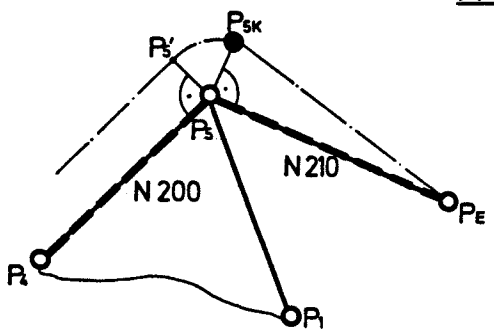
4.2.1 Exterior Corner: $\alpha > 180°$



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

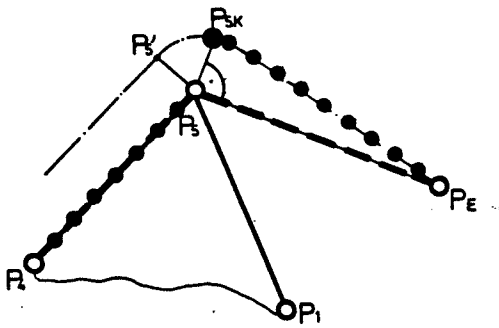
N 200/G../XYZP$_5$/

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:



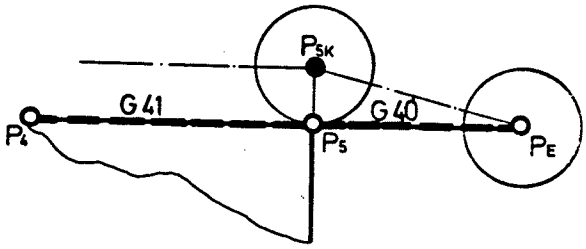
Block N 200: Programmed path up to N 200.

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}

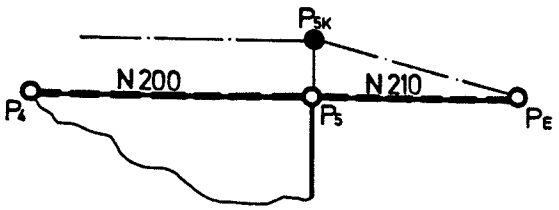
4.2.3 Neutral Disactivation: $\alpha = 180°$

Definition:



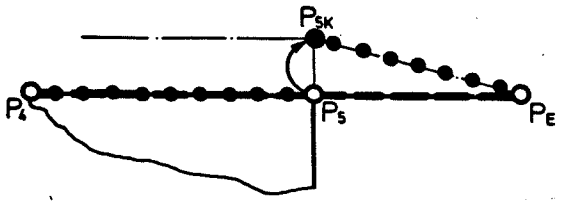
The angle α between $\overline{P_4P_5}$ and $\overline{P_5P_E}$ = 180°

Programming:



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

Position Read-Out:



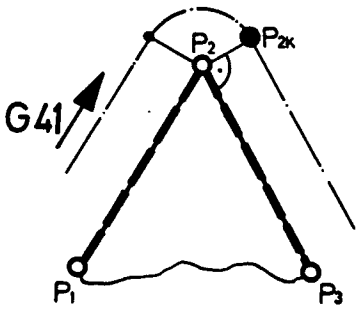
Block N 210: The position read-out jumps to the center point path $\overline{P_{5K}P_E}$ after point 5.

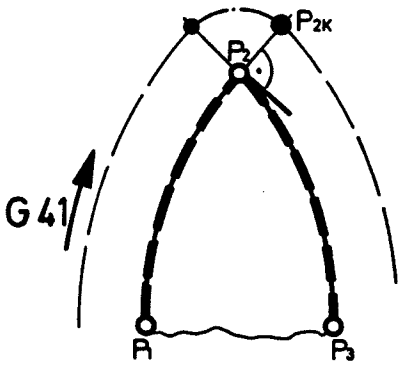
- Programmed path
- Center path of cutter
- Digital position read-out
- Programmed end of block P_X
- End of block center point of cutter P_{XK}

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.

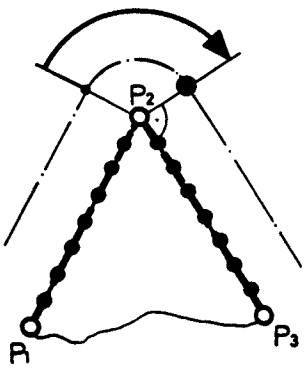




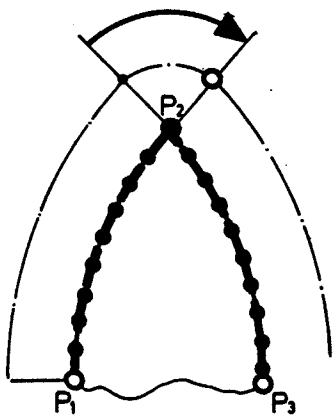
Position Read-Out:

During the transition radii the point P_2 is indicated.

P_2 Read-out



P_2 Read-out



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 XY must be programmed in comparison to the previous block.

If only 2 value is changed an alarm 524 appears.

aa) 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.

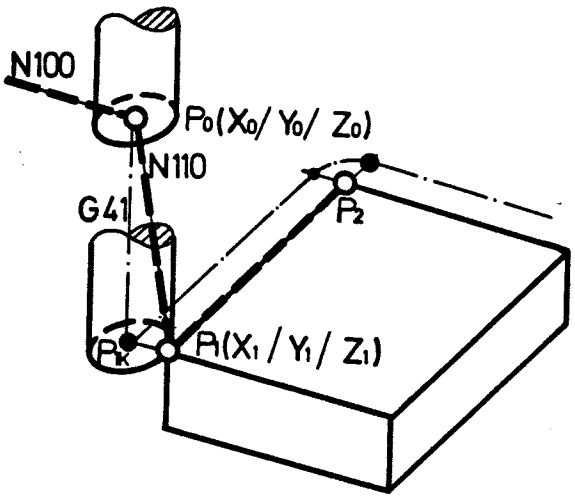
N 100/G00/X...Y...Z.../G41

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:

Please 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)

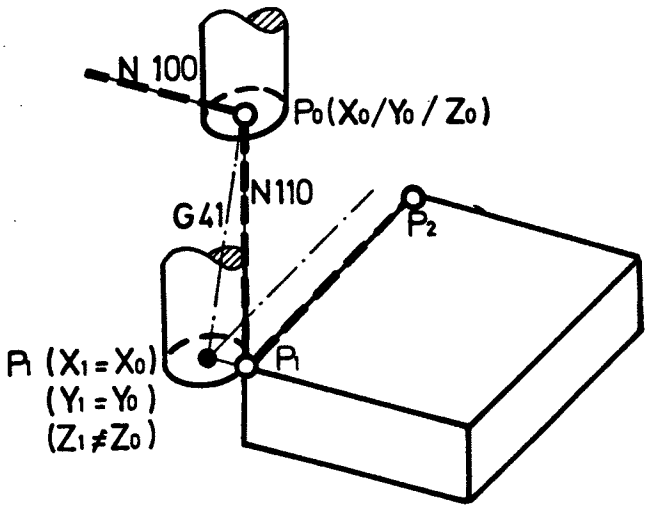
N 110/G01/P_1 ($X_1 \neq X_0$ /$Y_1 = Y_0$ /$Z_1 \neq Z_0$/G41

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.

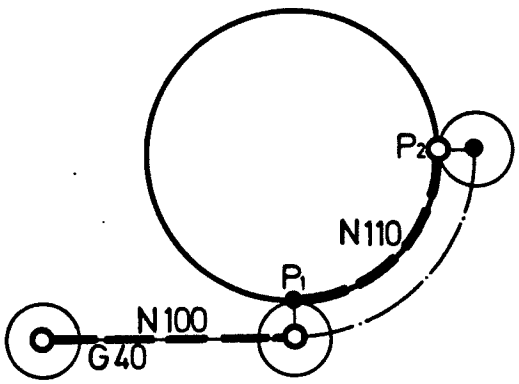
P_0 --> P_1

No change in XY value.

Alarm 520

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



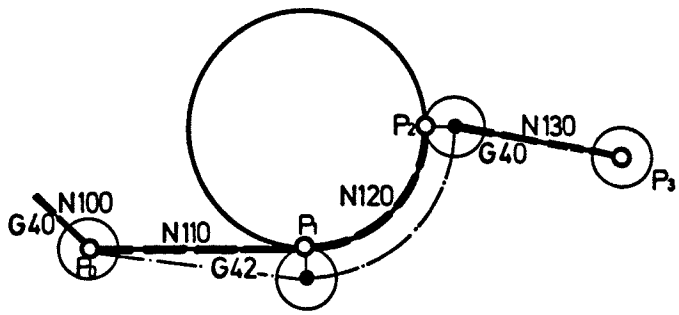
Alarm Situation

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)

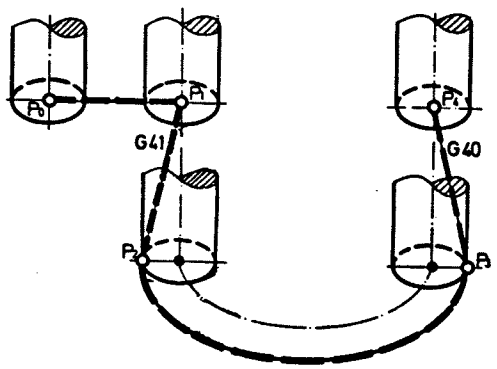


N ...,G40
N 100/G01/XYZP$_0$ /
N 110/G01/XYZP$_1$ /G42
N 120/G03/XYZP$_2$ /I.../J
N 130/G00/XYZP$_3$ /G40

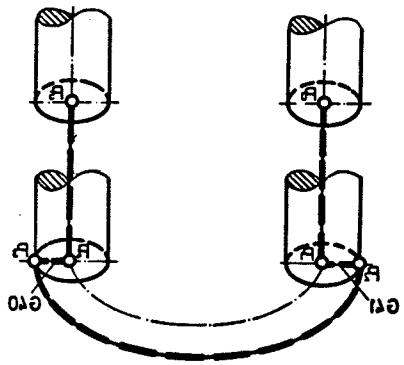
Examples with G02/G03

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

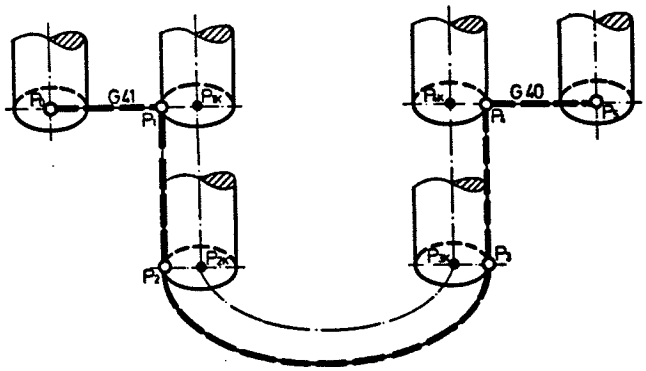
Possibility 1



Possibility 2



Possibility 3



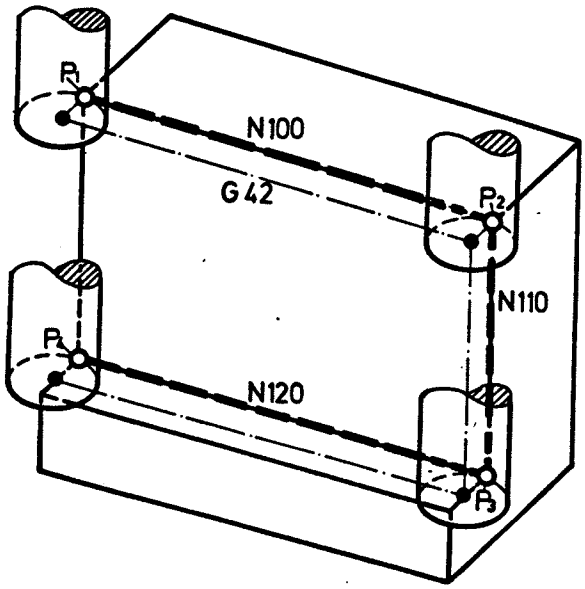
5.3 Direct Change from G41 to G42 --> Alarm 530

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

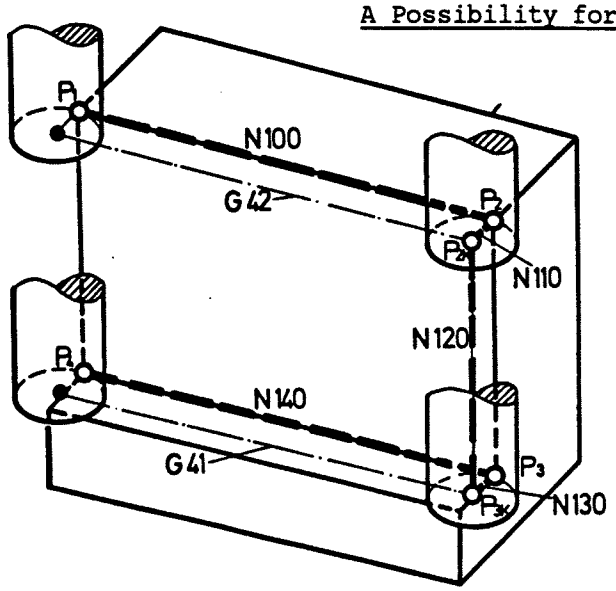
The depth of a shoulder is cut in various steps.



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.

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:

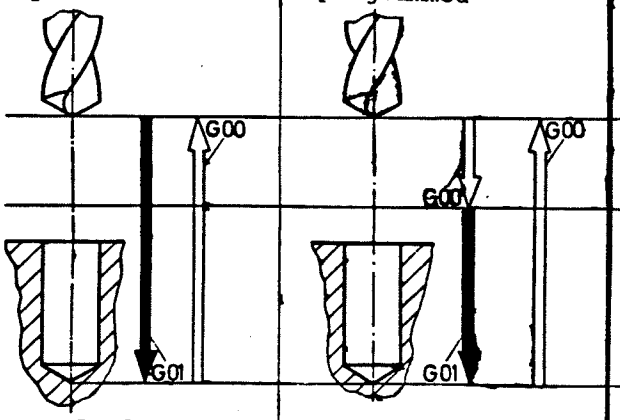
With cycles a withdrawal plane can be defined using P_3 resp. P_4. This has practical reasons; find a detailed description on the following pages.

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

G98

Withdrawal to starting plane

No P_3/P_4 programmed



In-feed from starting plane on with G01

Parameters D are effective from starting plane on.

P_3/P_4 programmed

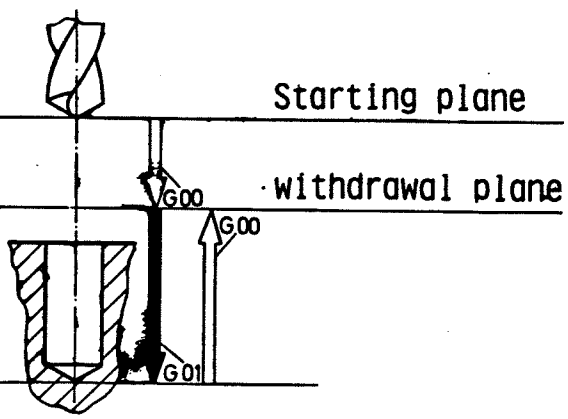
Up to P_3 resp. P_4 G00

Parameters D effective from P_3 resp. P_4 on.

G99

Withdrawal to withdrawal plane

P_3 resp. P_4 programmed



In-feed up to withdrawal plane with G00.

Parameters D effective from P_3 resp. P_4 on.

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.

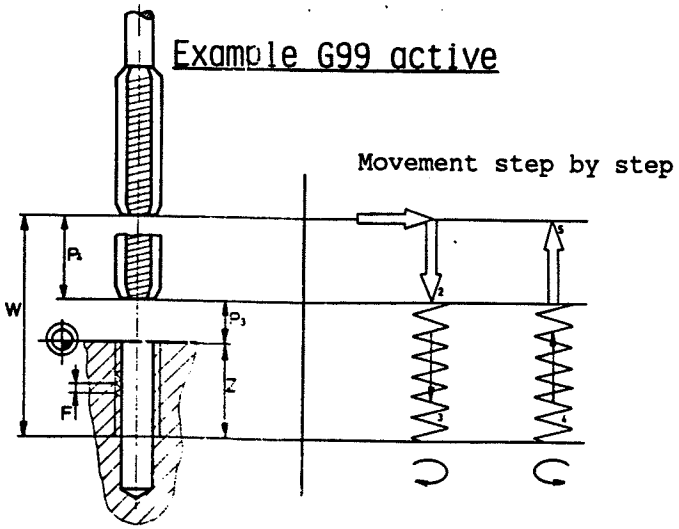
- G98 Withdrawal to starting plane
- G99 Withdrawal to withdrawal plane
- X,Y [mm] (U,V)[mm] Traverse movement in XY-plane
- Z(W) [mm] Drilling depth
- P_3 [mm] Absolute Z-measurement (from zero plane on)
- P_4 [mm] Incremental Z-measurement (from starting plane on)
- F [mm] Thread pitch

Sequence:

* Threading drill moves to programmed end point.
* Direction of revolution is switched over.
* Moving out to starting resp. withdrawal plane.
* Switching to programmed direction of revolution.

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..../G83/U±mm/W±mm/P_4±mm/Fµm/(G98/G99)

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

G99 being active:

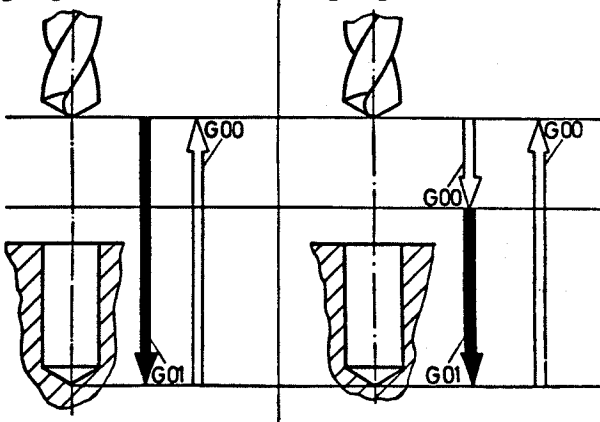
With P_3 resp. P_4 you can define a withdrawal plane.

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

G98

is withdrawal to starting plane

No P_3/P_4 programmed



Approach with G01 from starting plane onwards.

D Parameters are effective from starting plane onwards.

P_3/P_4 programmed

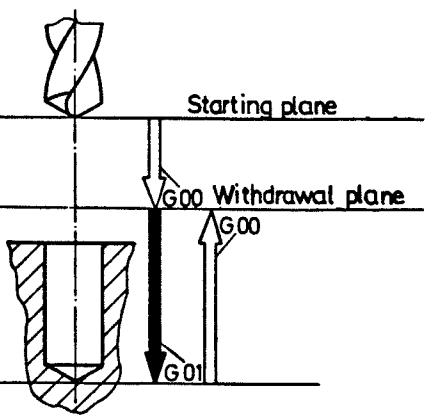
To P_3 resp P_4 with G00

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 !

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

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 (Y1A715000).

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 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:

* 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

G73: 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 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 no other traverse command than G01 (i.e. no cycle 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 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).

* 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

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

ALARM 731: PRINTER OFFLINE

It 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

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

Chapter 8

User Monitor

EMCOTRONIC TM 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

1. Call-up of the user monitor mode

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

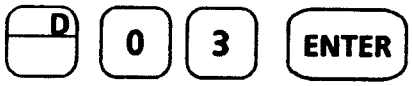
2. Selection of the parameters:

There are two possibilities:

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.



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!

* 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.

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.

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	
Status bit=0 LOW	EMCO internal	No ctrl			No parity check disable	Odd parity		
Value	0	0	0	0	0	0	0	0
Status bit=1 high	ISO	ctrl z			Parity check enable	Even parity		
Value	1	2	4	8	16	32	64	128

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

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

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	Like ASCII-character	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

Parameter 0_{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

0_{11} bit 3... Initial status direction logic - tool turret

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

M51 - select direction logic
0_{11} bit 3 = 1 (high) value 8

0_{11} bit 4 ... Initial status automatic door facility

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

0_{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).

Parameter 0_{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

0_{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).

0_{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.

0_{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.

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.

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	Like ASCII-character	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