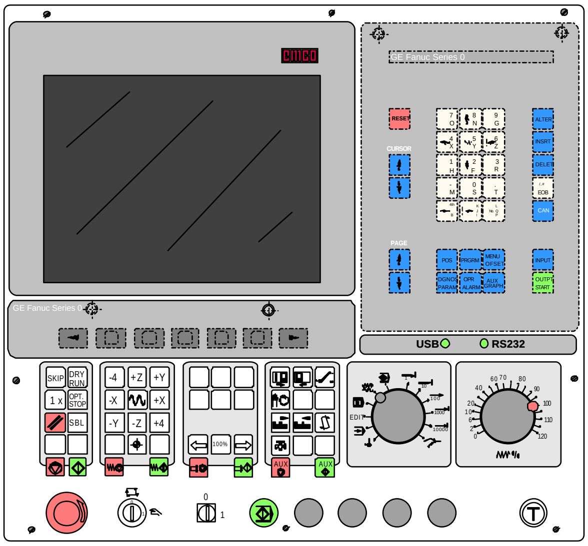
EMCO WinNC GE Series Fanuc 0-MC

Software description/ Software version from 13.76



Software description
EMCO WinNC Fanuc 0-MC

Ref.No. EN 1801 Edition I2003-7

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

Phone ++43-(0)62 45-891-0

Fax ++43-(0)62 45-869 65

Internet: www.emco.at

E-Mail: service@emco.co.at

innovative machine tools
industrial training systems

Preface

The software EMCO WinNC FANUC SERIES 0-MC Milling is a part of the EMCO education concept on PC basis.

Target of this concept is learning to operate and program the original control at the PC.

The milling machines of the EMCO PC MILL und CONCEPT MILL series can be directly controlled via PC by means of the EMCO WinNC for the EMCO MILL.

The operation is rendered very easy by the use of a digitizer or the control keyboard with TFT flat panel display (optional accessory), and it is didactically especially valuable since it remains very close to the original control.

This manual does not include the whole functionality of the control software FANUC SERIES 0-MC Milling, however emphasis was laid on the simple and clear illustration of the most important functions so as to achieve a most comprehensive learning success.

In case any questions or proposals for improving this manual should arise, please contact us directly:

EMCO MAIER Gesellschaft m. b. H.
Department Technical Documentation
A-5400 Hallein, Austria

Contents

G: Flexible NC programming

H: Alarms and Messages

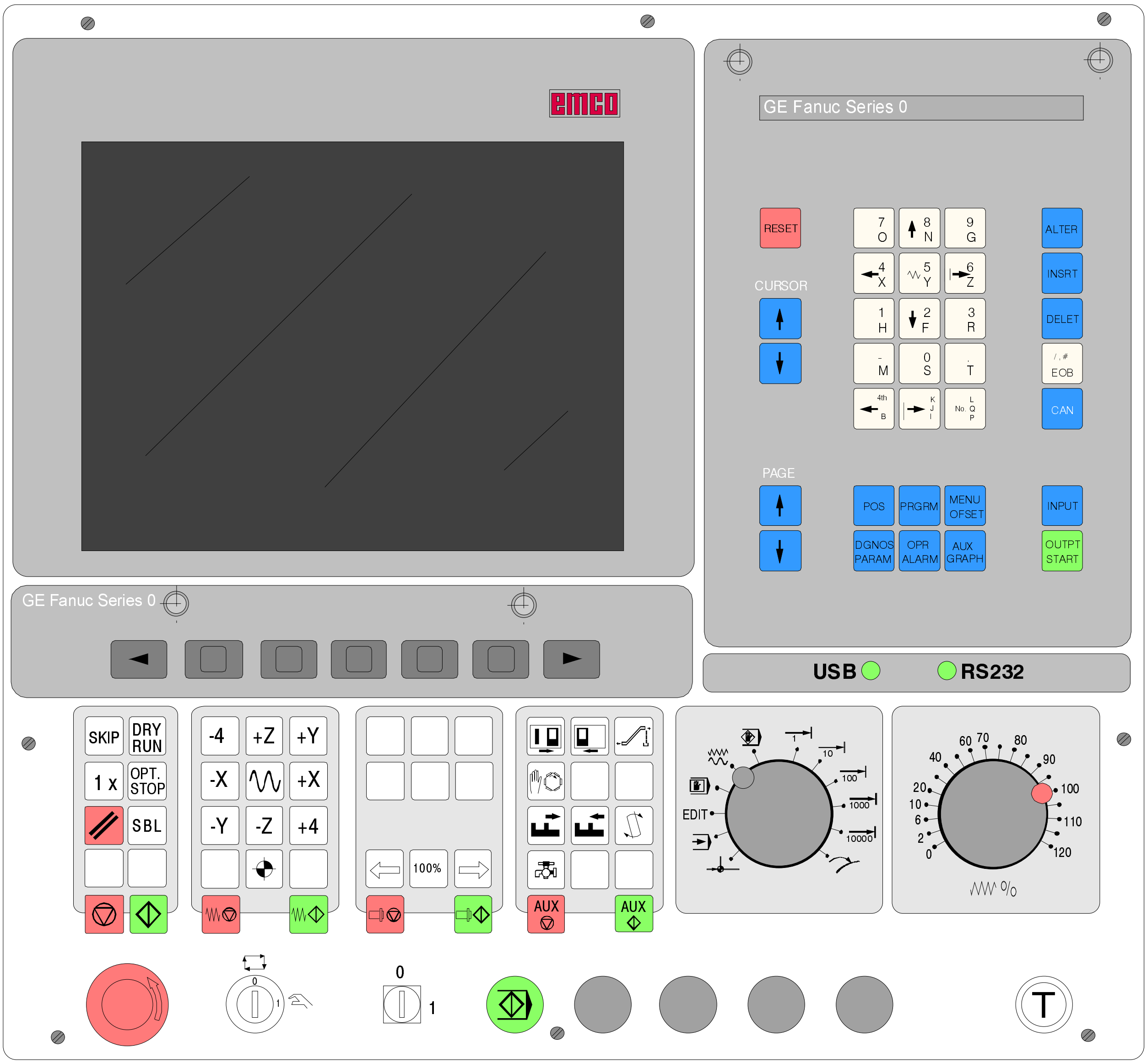
I: Control Alarms

Starting Information

see attachment

A: Key Description

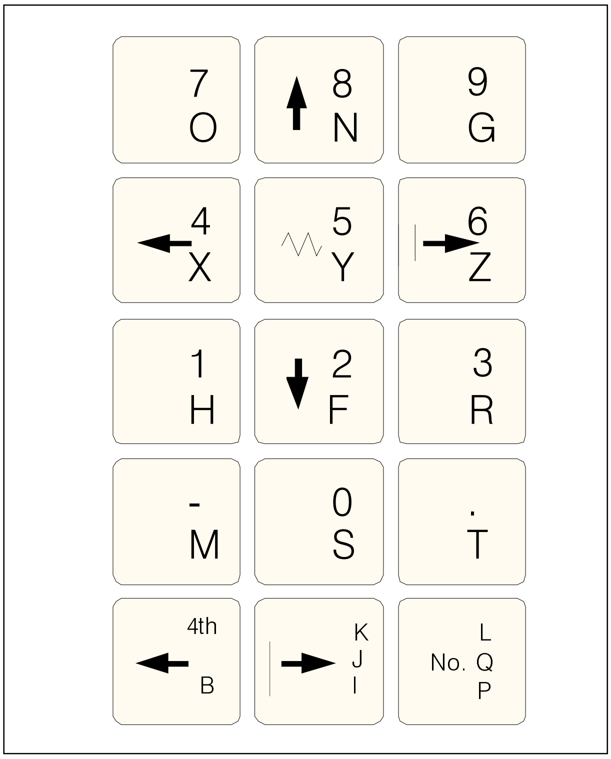
Control Keyboard, Digitizer Overlay



Key Functions

RESET .... Cancel an alarm, reset the CNC (e.g. interrupt a program), etc.
CURSOR .... Search function, line up/down
PAGE .... Page up/down
ALTER .... Alter word (replace)
INSRT .... Insert word, create new program
DELET .... Delete (program, block, word)
/,# EOB .... Skip block, **E**nd **O**f **B**lock
CAN .... Delete input
INPUT .... Word input, data input
OUTPT START .... Start data output

POS .... Indicates the current position
PRGRM .... Edit and display of the program, Input of the MDI data; Display of the command values in the automatic mode
MENU OFSET ...... Setting and display of offset values, tool and wear data, variables
DGNOS PARAM .. Setting and display of parameter and display of diagnostic data
OPR ALARM .... Alarm and message display
AUX GRAPH .... Graphic display

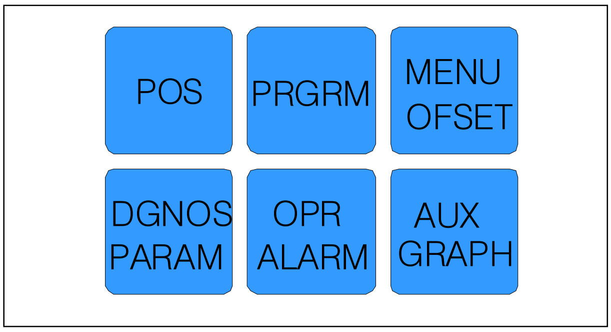


Data input keys

Data Input Keys

Note for the Data Input Keys

Each data input key runs several functions (numbers, address character(s)). Repeated pressing of the key switches to the next function automatically.



Function keys

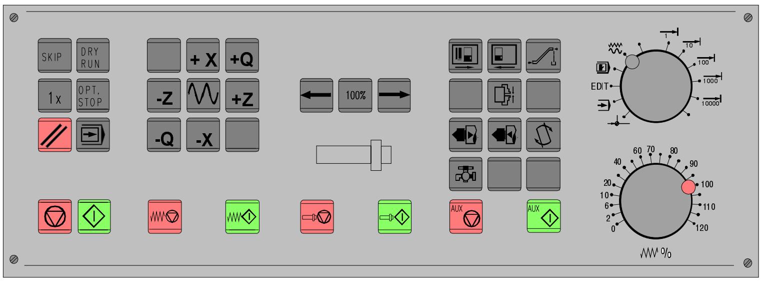
Function Keys

Note for Function Keys

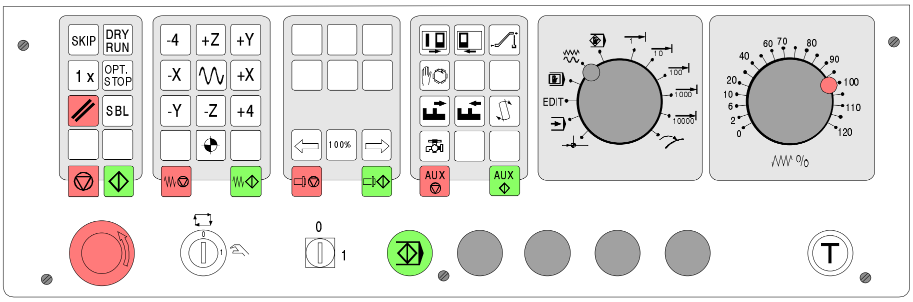
With the PC keyboard the function keys can be displayed as softkeys by pressing the key F12.

Machine Control Keys

The machine control keys are in the lower block of the control keyboard resp. the digitizer overlay.
Depending on the used machine and the used accessories not all functions may be active.



Machine control keyboard



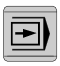
Machine control keyboard of the EMCO PC- Mill Serie

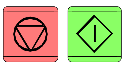
SKIP (skip blocks will not be executed)

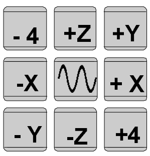
DRY RUN (test run of programs)

OPT STOP (program stop at M01)

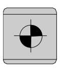

RESET

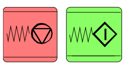

Single block machining

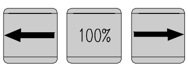

Program stop / program start



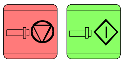
manual axis movement

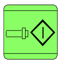
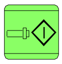

Approaching the reference point in all axes

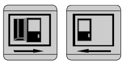

Feed stop / feed start



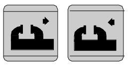
Spindle override lower / 100% / higher

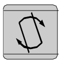

Spindel stop / spindle start; spindle start in JOG and INC1...INC10000 mode:

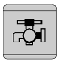
Clockwise: perss  key short, Counterclockwise: press  min. 1 sec.

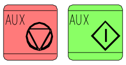

Open / close door


Swivel dividing head

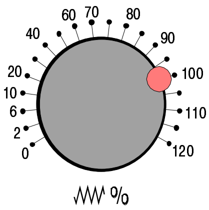

Open / close clamping device


Swivel tool turret

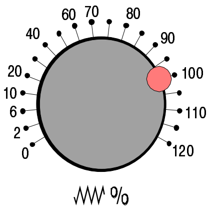

Coolant on/off



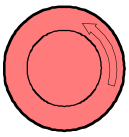
AUX OFF / AUX ON (auxiliary drives off / on)

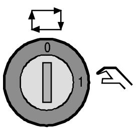


Vorschub- / Eilgangkorrekturschalter

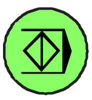


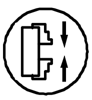
Feed / rapid feed override switch

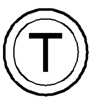

EMERGENCY OFF (Unlock: pull out button)

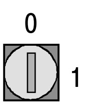


Key switch for special operations (siehe Maschinenbeschreibung)


Additional NC start key


Additional key clamping device


Consent key



No function

PC Keyboard

Some alarms will be acknowledged with the key ESC.

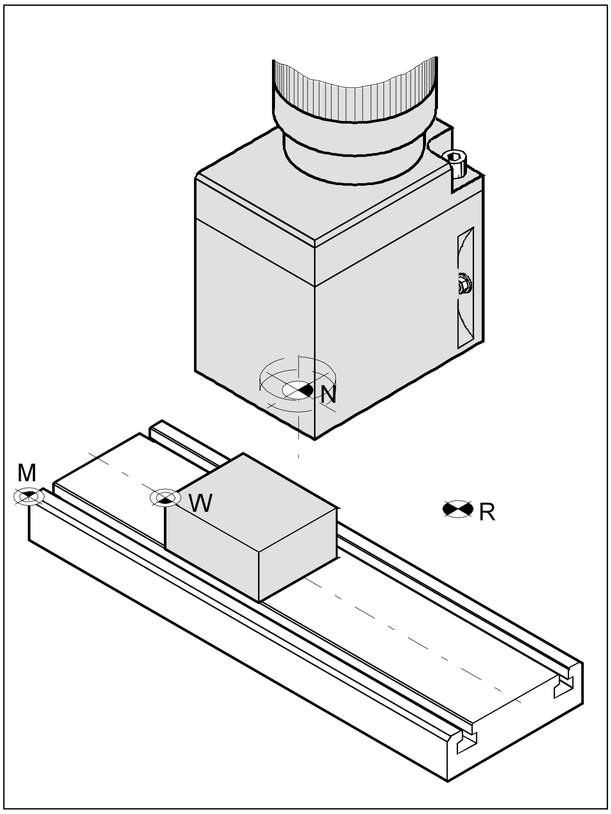
By pressing the key F1 the modes (MEM, EDIT, MDI,...) will be displayed in the softkey line.
The assignement of the accessory functions is described int the chapter "Accessory Functions".

The meaning of the key combination ctrl 2 depends on the machine:
EMCO PC MILL 50/55: Puff blowing ON/OFF
EMCO PC MILL 100/125/155: coolant ON/OFF

* With F12 the function keys POS, PROG, OFFSET SAETTING, SYSTEM, MESSAGES and GRAPH will be displayed in the softkey line.

The machine functions in the numeric key block are active only with active NUM lock.

B: Basics



Reference points in the working area

Reference Points of the EMCO Milling Machines

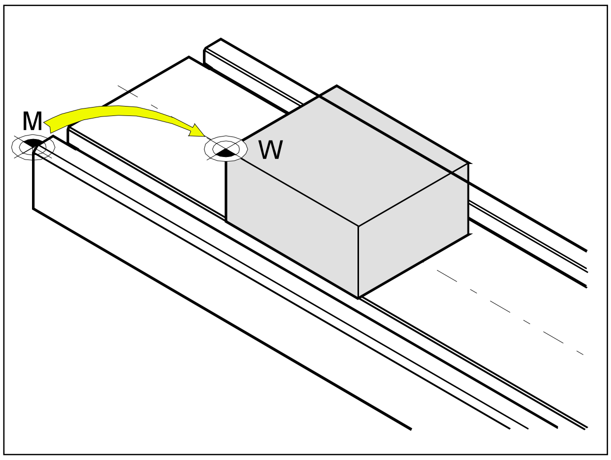
M = Machine zero point
An unchangeable reference point established by the machine manufacturer.
Proceeding from this point the entire machine is measured.
At the same time "M" is the origin of the coordinate system.

R = Reference point
A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the "R".
Required after every power failure.

N = Tool mount reference point
Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

W = Workpiece zero point
Starting point for the dimensions in the part program.
Can be freely established by the programmer and moved as desired within the part program.

Zero Offset



Zero offset from machine zero (M) to workpiece zero point (W)

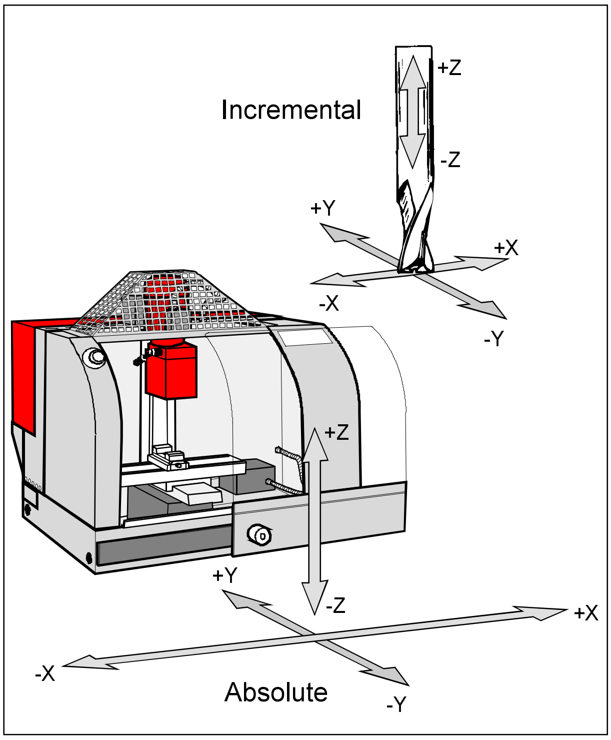
With EMCO milling machines the machine zero "M" lies at the left front edge of the machine table. This position is unsuitable as a starting point for dimensioning. With the so-called zero offset the coordinate system can be moved to a suitable point in the working area of the machine.

The offset register (WORK) offers seven adjustable zero offsets.

When you define a value in the offset register, this value will be considered with call in program (with G54 - G59) and the coordinate zero point will be shifted from the machine zero M to the workpiece zero W.

The workpiece zero point can be shifted within a program in any number.
More informations see in command description.

The Coordinate System

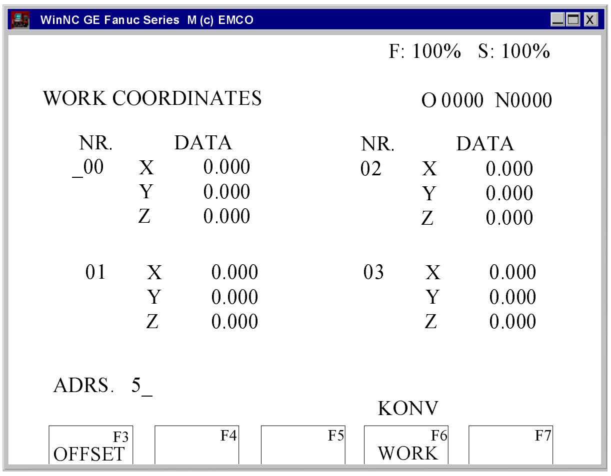


Absolute coordinates refer to a fixed point, incremental coordinates to the tool position

Coordinate system for absolute value programming
The origin of the coordinate system lies at the machine zero "M" or at the workpiece zero "W" following a programmed zero offset.
The X coordinate lies parallel to the machine table front edge, the Y coordinate parallel to the side edge and the Z coordinate vertical to the machine table.
Z Coordinate values in minus directions describe movements of the tool system towards the workpiece. Values in plus direction away from the workpiece,
All target points are described from the origin of the coordinate system by the indication of the respective X, Y and Z distances.

Coordinate system for incremental value programming
The origin of the coordinate system lies at the tool mount reference point "N" or at the cutting tip after tool length compensation.
Coordinate directions in plus and minus direction like with absolute programming. The plus and minus directions are the same as for absolute value programming.
With incremental value programming the actual paths of the tool (from point to point) are described.

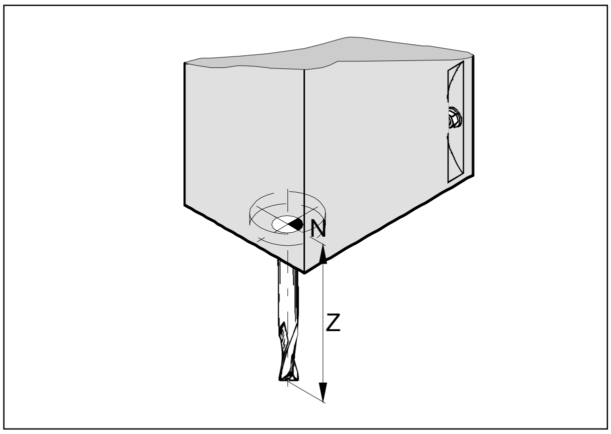
Input of the Zero Offset



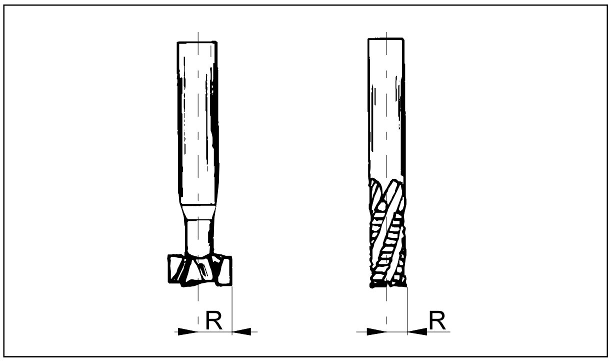
Input pattern for zero offsets

- Press the key [MENU OFSET]
- Select the softkey WORK
- The input pattern beside will be displayed
- You can enter the following offsets:
  00 .... basic offset 02 ..... G55
  01 .... G54 03 ..... G56
  The basic offset is always active, other offsets will be added to.
- By pressing the key [PAGE ↓] you get the next display page. Here you can enter the following offsets:
  04 .... G57 06...... G59
  05 .... G58
- Below X, Y, Z you can enter the distance **from the machine zero point to the workpiece zero point (pos. sign)**.
- Go with the cursor to the desired offset with the keys [CURSOR ↓] and [CURSOR ↑].
- Enter the desired offset (e.g.: X+30.5) and press the key [INPUT]
- Enter the desired offset values one by one.

Tool Data Measuring



Length correction



Cutter radius R

Aim of the tool data measuring:
The CNC should use the tool tip resp. the tool centre at the face end for positioning, not the tool mount reference point.

Every tool which is used for machining has to be measured. The distance between tool tip and tool mount reference point is to be measured.

To every of this distances a correction number in the offset register (OFFSET) is related to.

The correction number can be any register number, but has to be considered with tool call in program.

The length corrections can be measured half-automatically, the **cutter radius** has to be inserted manually.

Inserting the cutter radius is **only** necessary for using **cutter radius compensation** with this tool.

For G17 (XY plane active):
Tool data measuring occurs for
Z absolute from point "N"
R radius of the cutter

For all other active planes always the vertical axis to the plane is computed. In the following the normal case G17 is described.

Tool Data Measuring by Scraping

Procedure

- Clamp a workpiece in the working area. The measuring point has to be reachable with the tool mount reference point and with all tools to be measured.
  The tool mount reference point of the EMCO PC MILL 100 is on the reference tool (clamp before).
- Select the JOG mode
- Place a thin sheet of paper between work piece and milling spindle.
- Traverse with the tool mount reference point on the workpiece (standing spindle)
  Reduce feed to 1%
  Traverse with the spindle (tool mount reference point) down to the workpiece, so far that the paper still can be moved.
- Press the key POS and the softkey REL to show the relative position at the screen.
- Press the key 6 Z - the Z display flashes
- Reset Z value with CAN to 0
- Clamp tool to be measured
- Change to MDI mode
- Switch on the spindle (e.g. S1000 M3 NC-Start)
- Change to JOG mode
- Press the key MENU OFSET
- Scrap on the workpiece
- Now the screen shows the length difference between tool mount reference point and the tool tip (Z value relative)
- Select the corresponding parameter H in the offset register with the keys CURSOR ↓ CURSOR ↑
- Key in the displayed Z value as parameter H in the offset register and take it over with the INPUT key.
- Clamp next tool and scrap onto the workpiece surface etc.

C: Operating Sequences

Survey Operating Modes

ZRN

In this operating mode the reference point will be approached.
With reaching the reference point the actual position display is set to the value of the reference point coordinates. By that the control acknowledges the position of the slides in the working area.
With the following situations the reference point has to be approached::

- After switching on the machine
- After mains interruption
- After alarm "Approach reference point" or "Ref. point not reached"
- After collisions or if the slides stucked because of overload

AUTO

For working off a part program the control calls up block after block and interprets them.
The interpretation considers all correction which are called up by the program.
The so-handled blocks will be worked off one by one.

EDIT

In the EDIT mode you can enter part programs and transmit data.

MDI

In the MDI mode you can switch on the spindle and swivel the tool holder.
The control works off the entered block and deletes the intermediate store for new inputs..

JOG

With the JOG keys the slides can be traversed manually.

S1 ... S1000 1 ... 10000

In this operation mode the slides can be traversed for the desired increment (1...10000 in µm/10^{-4} inch) by means of the JOG keys -X + X - Y +Y -Z +Z.

The selected increment (1, 10, 100, ...) must be larger than the machine resolution (lowest possible traverse movement), otherwise no movement occurs.

REPOS

Repositioning, approach back to the contour in JOG mode.

Teach In

Making programs in dialogue with the machine in MDA mode.

Approach the Reference Point

By approaching the reference point the control will be synchronized to the machine.

- Change into ZRN mode
- Press as first the direction keys -Z or +Z , then -X or + X and - Y or +Y to approach the reference point in the respective direction.
- With the key REF ALL all axes will be approached automatically in the correct sequence (PC keyboard).

Danger of Collisions

Mind for obstacles in the working area (Clamping devices, clamped work pieces, etc.)

After reaching the reference point its position will be displayed as actual position. Now the machine is synchronized to the control.

Setting of Language and Workpiece Directory

- Press the key DGNOS PARAM.
- Press the key PAGE ↓ multiple, until the setting page (GENERAL) will be displayed.

Workpiece Directory

In the workpiece directory the CNC programs created by the operator will be stored.
The workpiece directory is a subdirectory of the program directory which was determined with installation.
Enter in the input field "PATH = ..." the name of the workpiece directory with the PC keyboard, max. 8 characters, no drives or pathes. Not existing directories will be created.

Active Language

Selection from installed languages, the selected language will be activated with restart of the software.
Enter the language sign in the input field "LANG = ..."

- DT for German
- EN for English
- FR for French
- SP for Spanish

Program Input

Part programs and subprograms can be entered in the EDIT mode.

Call Up a Program

- Change into EDIT mode
- Press the key PRGRM
- With the softkey LIB the existing programs will be displayed.
- Enter program number O...
- New program: Press the key INSRT
- Existing program: Press the key CURSOR ↓

Input of a block

Example:

N 8, Y 5, INSRT	Block number (not necessary)
G 9, H 1, INSRT	1. word
X 4, R 3, S 0, INSRT	2. word
... ...	
EOB, INSRT	EOB - End of block (on PC keyboard also ↵)
or INSRT	

Note:
With the parameter "SEQ" (SETTING 1) you can determine whether block numbering should occur automatically (1 = yes, 0 = no).

Insert a Word

Move the cursor before the word, that should be before the inserted word, enter the new word (address and value) and press the key INSRT.

Alter a Word

Move the cursor before the word that should be altered, enter the word and press the key ALTER.

Delete a Word

Move the cursor before the word, that should be deleted and press the key DELET.

Search a Word

Enter the address of the word to be searched (e.g.: X) and press the key CURSOR ↓.

Insert a Block

Move the cursor before the EOB sign ";" in that block which should be before the inserted block and enter the block to be inserted.

Delete a Block

Enter block number (if no block number exists: N0) and press the key DELET.

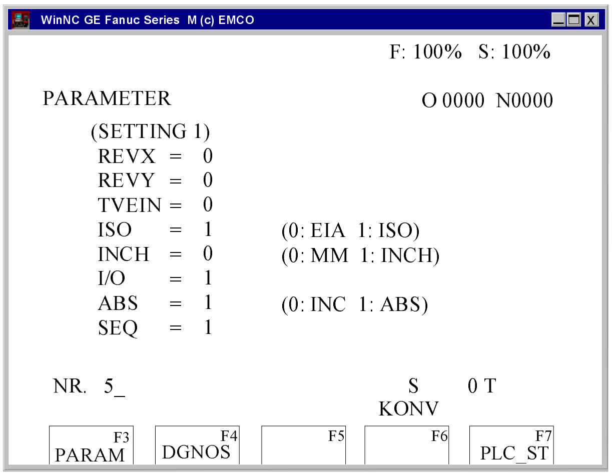
Delete a Program

EDIT mode
Enter the program number (e.g.: O22) and press the key DELET.

Delete All Programs

EDIT mode
Enter the program number O 0-9999 and press the key DELET.

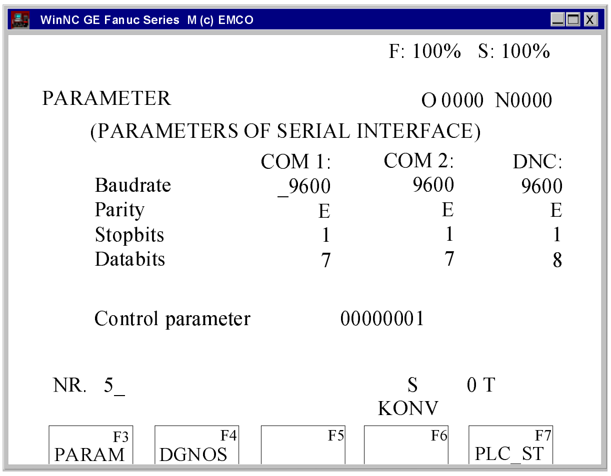
Data Input - Output



Selection of the input/output interface

- Press the key DGNOS PARAM.
  The screen shows (SETTING 1).
- Below "I/O" you can enter a serial interface (1 or 2) or a drive (A, B or C).
  1 serial interface COM1
  2 serial interface COM2
  A disk drive A
  B disk drive B
  C hard disk drive C, workpiece directory (Established with installation or in (GENERAL)).
  P Printer on LPT1.

Adjusting the Serial Interface



Adjusting the serial interface

- Press the key DGNOS PARAM.
- Press the key PAGE ↓, until (PARAMETERS OF SERIAL INTERFACE) is displayed.

Settings:
Baudrate 110, 150, 300, 600, 1200, 2400, 4800, 9600
Parity E, O, N
Stopbits 1, 2
Datenbits 7, 8
Data transmission from / to original control in ISO-Code only
ISO: 7 Datenbits, Parity even (=E)

Control parameter:
Bit 0: 1...Transmission will be cancelled with ETX (End of Text) code
0...Transmission will be cancelled with RESET
Bit 7: 1...Overwrite part program without message
0...Message, if a program already exists
ETX code: % (25H)

NOTE

When you use an interface expansion card (e.g. for COM 3 and COM 4), take care that for every interface a separate interrupt is used (e.g.: COM1 - IRQ4, COM2 - IRQ3, COM3 - IRQ11, COM4 - IRQ10).

Program Output

- EDIT mode
- Enter the receiver in (SETTING 1) below "I/O".
- Press the key PRGRM.
- Enter the program to be sent (e.g. O22)
- When you enter e.g. O5-15, all programs with the numbers 5 to inclusive 15 will be transmitted. When you enter O-9999 as program number, all programs will be transmitted.
- Press the key OUTPT START.

Program Input

- EDIT mode
- Enter the receiver in (SETTING 1) below "I/O".
- Press the key PRGRM.
- With input from disk or hard disk you have to enter a program number. Enter the program number when you want to read in one program (e.g.: O22). When you enter e.g. O5-15, all programs with the numbers 5 to inclusive 15 will be transmitted. When you enter O-9999 as program number, all programs will be transmitted.
- Press the key OUTPT START.

Tool Offset Output

- EDIT mode
- Enter the receiver in (SETTING 1) below "I/O".
- Press the keys MENU OFSET and OUTPT START.
- If a drive (A, B or C) is the receiver, the zero offsets will be transmitted additionally.

Tool Offset Input

- EDIT mode
- Enter the sender in (SETTING 1) below "I/O".
- Press the keys MENU OFSET and INPUT.

Print Programs

- The printer has to be connected to LPT1 and must be in ON LINE status.
- EDIT mode
- Enter P (printer) in (SETTING 1) below "I/O ".
- Press the key PRGRM.
- Enter the program to be print (e.g. O22)
- When you enter e.g. O5-15, all programs with the numbers 5 to inclusive 15 will be printed. When you enter O-9999 as program number, all programs will be printed.
- Press the key OUTPT START

Program Run

Start of a Part Program

Before starting a program the control and the machine must be ready for running the program.

- Select the EDIT mode.
- Press the key PRGRM
- Enter the desired part program number (e.g.: O79).
- Press the key CURSOR ↓.
- Change to AUTOMATIC mode.
- Press the key ◇.

Displays while Program Run

While program run different values can be shown.

- Press the softkey PRGRM (basic status). While program run the actual program block will be displayed.
- Press the softkey CHECK . While program run the actual program block, the actual positions, active G and M commands and speed, feed and tool will be displayed.
- Press the key POS. The positions will be shown enlarged at the screen.

Block Search

With this function you can start a program at any block.
While block search the same calculations will be proceeded as with normal program run but the slides do not move.

- EDIT mode
- Select the program to be machined.
- Move the cursor with the keys CURSOR ↓ and CURSOR ↑ on that block, with which machining should start.
- Change to AUTOMATIC mode.
- Start the program with the key ◇.

Program Influence

DRY RUN
DRY RUN is used for testing programs. The main spindle will not be switched on and all movements occur in rapid feed.
If DRY RUN is active, DRY will be displayed in the first line on the screen.

SKIP
With SKIP all program blocks which are marked with a "/" (e.g.: /N0120 G00 X...) will not be proceeded and the program will be continued with the next block without a "/" sign.
If SKIP is active, SKP will be displayed in the first line on the screen.

Program interruption

Single block mode
After every program block the program will be stopped.

Continue the program with the key ◇.

M00
After M00 (programmed stop) in the program the program will be stopped. Continue the program with the key ◇.

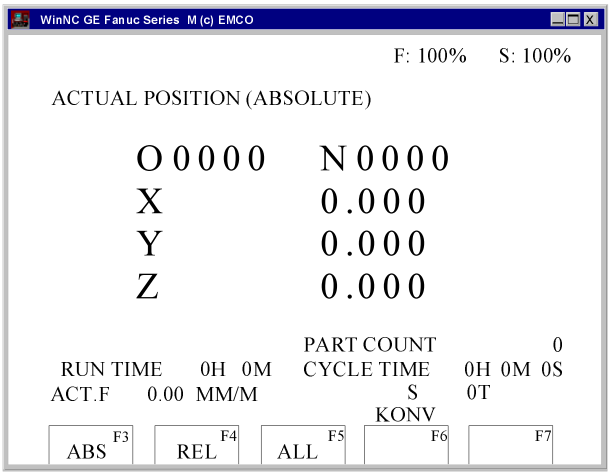
M01
If OPT. STOP is active, (display OPT in the first line of the screen) M01 works like M00, otherwise M01 has no effect.

Display of the Software Versions

- Press the key DGNOS PARAM
- Select softkey DGNOS

The software version of the control system and the eventually connected RS485 devices will be displayed.

Part Counter and Piece Time



Display of part counter and piece time

Below the position display the part counter and the piece time are displayed.

The part counter shows the number of program runs. Each M30 (or M02) increases the part counter for 1.

RUN TIME shows the complete running time of all program runs.

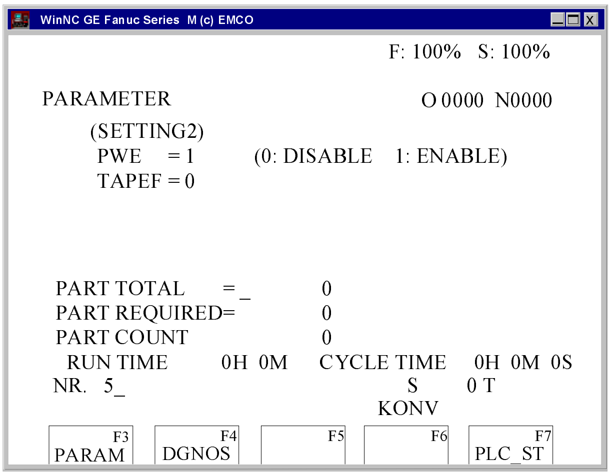
CYCLE TIME shows the running time of the actual program and will be reset to 0 with every program start.

Part Counter Reset

The part counter will be reset to 0 by pressing No. L O P (P) and CAN after.

RUN TIME Reset

The RUN TIME (total time) will be reset to 0 by pressing 3 R (R) and CAN after.



Presetting the piece counter

Preset of the Part Counter

The part counter can be preset in (SETTING 2). Therefore move the curor on the desired value and enter the new value.

PART TOTAL:
Each M30 increases this number by 1. Every program run of every program will be counted (= number of all program runs).

PART REQUIRED:
Preset part number. When this number is reached the program will be stopped and message 7043 PIECE COUNT REACHED will be displayed.
After that the program can be started only after resetting the part counter or increasing the preset part number.

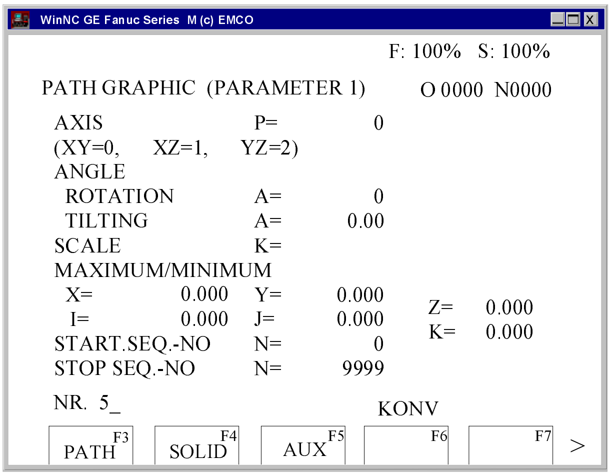
Graphic Simulation

NC-programs can be simulated graphically.

Press the key AUX GRAPH.

The screen shows the input pattern for graphic simulation.

The simulation area is a rectangular window, which is determined by the right upper and left lower edge.



Input pattern for graphic simulation

Inputs:

AXIS P

Enter the simulation plane here.

0 XY plane
1 XZ plane
2 YZ plane

MAXIMUM/MINIMUM

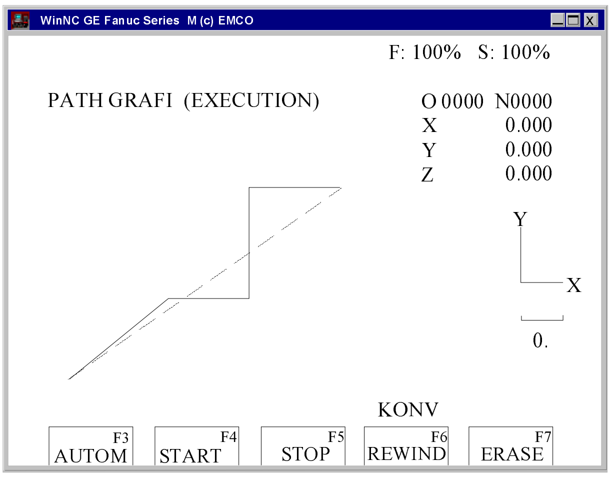
Enter here the right upper (X, Y, Z) and the left lower (I, J, K) edge of the simulation areaein.

All other inputs and the softkeys SOLID and AUX are not active.

After pressing the key ► the softkey 3DVIEW will be shown.

Win 3D View is an option and not included in the basic version of the software.

With the softkeys path and EXEC you will get into the simulation window.



Simulation window

The softkeys PARA-1, PARA-2, SCALE and POS are not active.

With the key ^ you will go back to the input pattern for graphic simulation.

With the softkey START the graphic simulation starts.

With the softkey STOP the graphic simulation stops.

With the softkey RESET the graphic simulation will be aborted.

Movements in rapid traverse will be displayed as dashed lines, movements in working traverse will be displayed as full lines.

D: Programming

Program Structure

CNC programming for machine tools according to DIN 66025 is used.
The CNC program is a sequence of program blocks which are stored in the control.
With machining of workpieces these blocks will be read and checked by the computer in the programmed order.
The corresponding control signals will be sent to the machine.

The CNC program consists of:
- Program number
- CNC blocks
- Words
- Addresses
- number combinations (for axis addresses partly with sign)

Command survey M-functions

M00 ...... Programmed stop
M01 ...... Programmed stop conditional
M02 ...... Program end
M03 ...... Spindle ON in clockwise direction
M04 ...... Spindle ON in counter-clockwise direction
M05[1] ...... Spindle OFF
M06 ...... Tool change
M07 ...... Minimum lubrication ON
M08 ...... Coolant ON
M09[1] ...... Coolant OFF
M10 ...... Clamp round axis
M11 ...... Round axis release clamping
M27 ...... Swivel dividing unit
M30 ...... Program end
M71 ...... Blow-out ON
M72[1] ...... Blow-out OFF
M98 ...... Subroutine call
M99 ...... Subroutine end

[1] .... Switch-on state

Used Addresses

C .... chamfer
F .... feed rate, thread pitch
G .... path function
H .... number of the correction value address in the offset register (OFFSET)
I, J, K .... circle parameter, scale factor, K also number of repetitions of a cycle, mirror axes
M .... miscellaneous function
N .... block number 1 to 9999
O .... Program number 1 to 9499
P .... dwell, subprogram call
Q .... cutting depth or shift value in cycle
R .... radius, retraction height with cycle
S .... spindle speed
T .... tool call
X, Y, Z .. position data (X also dwell)
; .... block end

Survey of G Commands

G00[1] .... Positioning (Rapid Traverse)
G01 .... Linear Interpolation
G02 .... Circular Interpolation Clockwise
G03 .... Circular Interpolation Counterclockwise
G04[2] .... Dwell
G09[2] .... Exact Stop
G10 .... Data Setting
G11 .... Data Setting Off
G15[1] .... End Polar Coordinate Interpolation
G16 .... Begin Polar Coordinate Interpolation
G17[1] .... Plane Selection XY
G18 .... Plane Selection ZX
G19 .... Plane Selection YZ
G20 .... Measuring in Inches
G21 .... Measuring in Millimeter
G28[2] .... Approach Reference Point
G33 .... Thread Cutting
G40[1] .... Cancel Cutter Radius Compensation
G41 .... Cutter Radius Compensation left
G42 .... Cutter Radius Compensation right
G43 .... Tool Length Compensation positive
G44 .... Tool Length Compensation negative
G49[1] .... Cancel Tool Length Compensation
G50[1] .... Cancel Scale Factor
G51 .... Scale Factor
G52[2] .... Local Coordinate System
G53[2] .... Machine Coordinate System
G54[1] .... Zero Offset 1
G55 .... Zero Offset 2
G56 .... Zero Offset 3
G57 .... Zero Offset 4
G58 .... Zero Offset 5
G59 .... Zero Offset 6
G61 .... Exact Stop Mode
G62 .... Automatic Corner Override
G64[1] .... Cutting mode
G68 .... Coordinate System Rotation ON
G69 .... Coordinate System Rotation OFF
G73 .... Chip Break Drilling Cycle
G74 .... Left Tapping Cycle
G76 .... Fine Drilling Cycle
G80[1] .... Cancel Drilling Cycles (G83 bis G85)
G81 .... Drilling Cycle
G82 .... Drilling Cycle with Dwell
G83 .... Withdrawal Drilling Cycle
G84 .... Tapping Cycle
G85 .... Reaming Cycle
G86 .... Drilling Cycle with Spindle Stop
G87 .... Back Pocket Drilling Cycle
G88 .... Drilling Cycle with Program Stop
G89 .... Reaming Cycle with Dwell
G90[1] .... Absolute Programming
G91 .... Incremental Programming
G92[2] .... Coordinate System Setting
G94[1] .... Feed per Minute
G95 .... Feed per Revolution
G97[1] .... Revolutions per Minute
G98[1] .... Retraction to Starting Plane (Drilling Cycles)
G99 .... Retraction to Withdrawal Plane

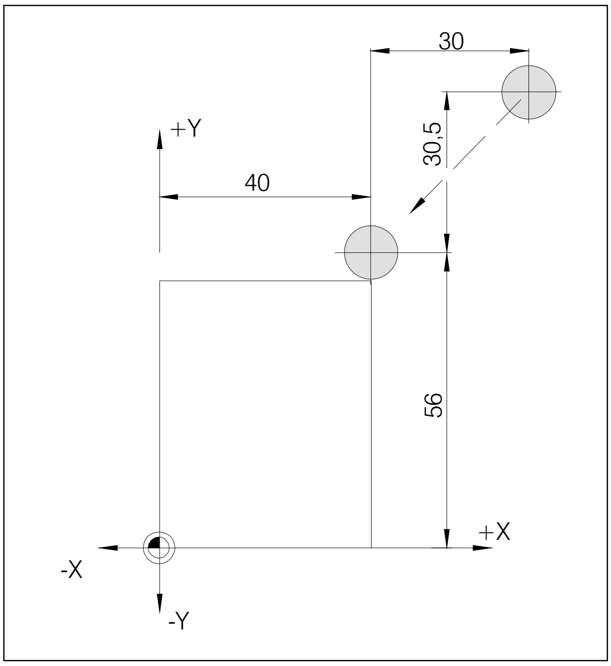
[1] .... Initial status
[2] .... Blockwise effective

Survey of G Commands for Command Definition C

Group	Command	Function
0	G04	Dwell
	G09	Exact Stop
	G10	Data Setting
	G11	Data Setting Off
	G28	Approach Reference Point
	G52	Local Coordinate System
	G53	Machine Coordinate System
	G92	Coordinate System Setting
1	G00	Positioning (Rapid Traverse)
	G01	Linear Interpolation
	G02	Circular Interpolation Clockwise
	G03	Circular Interpolation Counterclockwise
	G33	Thread Cutting
2	G17	Plane Selection XY
	G18	Plane Selection ZX
	G19	Plane Selection YZ
3	G90	Absolute Programming
	G91	Incremental Programming
5	G94	Feed per Minute
	G95	Feed per Revolution
6	G20	Measuring in Inches
	G21	Measuring in Millimeter
7	G40	Cancel Cutter Radius Compensation
	G41	Cutter Radius Compensation left
	G42	Cutter Radius Compensation right
8	G43	Tool Length Compensation positive
	G44	Tool Length Compensation negative
	G49	Cancel Tool Length Compensation
9	G73	Chip Break Drilling Cycle
	G74	Left Tapping Cycle
	G76	Fine Drilling Cycle
	G80	Cancel Drilling Cycles
	G81	Drilling Cycle
	G82	Drilling Cycle with Dwell
	G83	Withdrawal Drilling Cycle
	G84	Tapping Cycle
	G85	Reaming Cycle
	G86	Drilling Cycle with Spindle Stop
	G87	Back Pocket Drilling Cycle
	G88	Drilling Cycle with Program Stop
	G89	Reaming Cycle with Dwell
10	G98	Retraction to Starting Plane
	G99	Retraction to Withdrawal Plane
11	G50	Cancel Scale Factor
	G51	Scale Factor
13	G97	Revolutions per Minute
14	G54	Zero Offset 1
	G55	Zero Offset 2
	G56	Zero Offset 3
	G57	Zero Offset 4
	G58	Zero Offset 5
	G59	Zero Offset 6
15	G61	Exact Stop Mode
	G62	Automatic Corner Override
	G64	Cutting Mode
16	G68	Coordinate System Rotation ON
	G69	Coordinate System Rotation OFF
17	G15	End Polar Coordinate Interpolation
	G16	Begin Polar Coordinate Interpolation

Description of G Commands

G00 Positioning (Rapid Traverse)



Absolute and incremental measures

Format
N.... G00 X... Y... Z...

The slides are traversed at maximum speed to the programmed target point (tool change position, start point for a following machining routine)

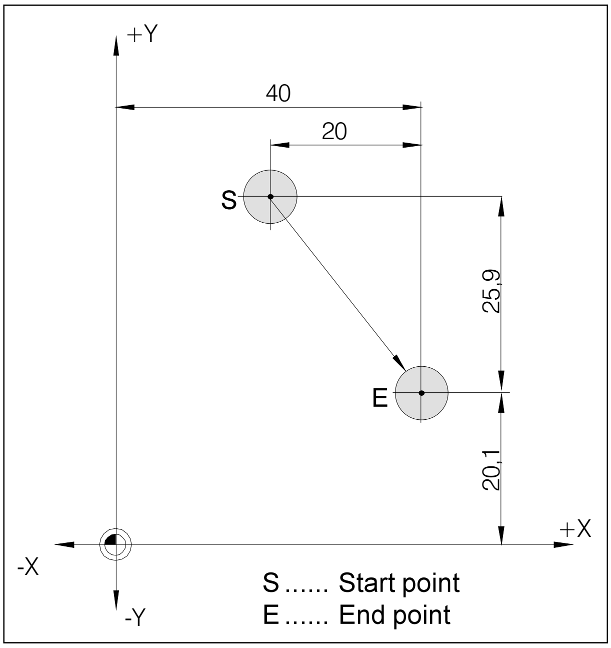
Notes
- A programmed feed F will be suppressed while G00
- The maximum speed is defined by the producer of the machine
- The feed override switch is active

Example

absolute G90
N50 G00 X40 Y56

incremental G91
N50 G00 X-30 Y-30.5

G01 Linear Interpolation



Absolute and incremental measures

Format
N... G01 X... Y... Z.... F....

Straight movements at the programmed feed rate.

Example

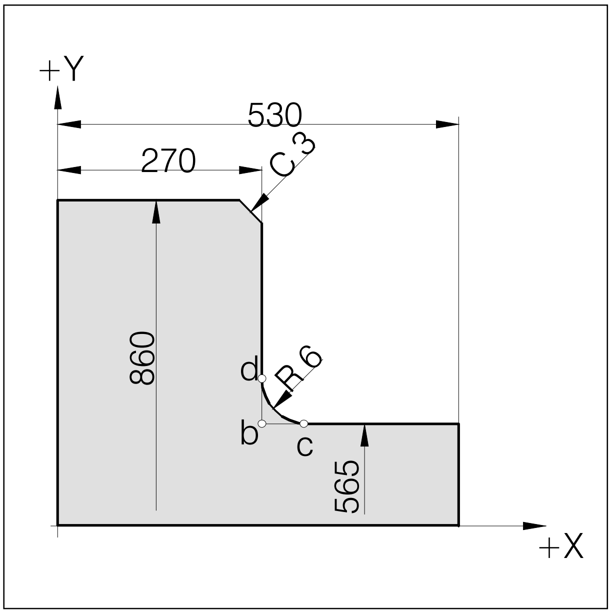
absolute G90
N.. G94
.....
N20 G01 X40 Y20.1 F500

incremental G91
N.. G94 F500
.....
N20 G01 X20 Y-25.9

Chamfers and Radius

By programming the parameter C or R a chamfer or a radius can be inserted between two G00 or G01 movements.

Format:
N.. G00/G01 X.. Y.. C/R
N.. G00/G01 X.. Y..



Chamfer and radius in a drawing

Programming of chamfers and radii is possible for the active plane only. Following the programming in the XY plane (G17) is described.

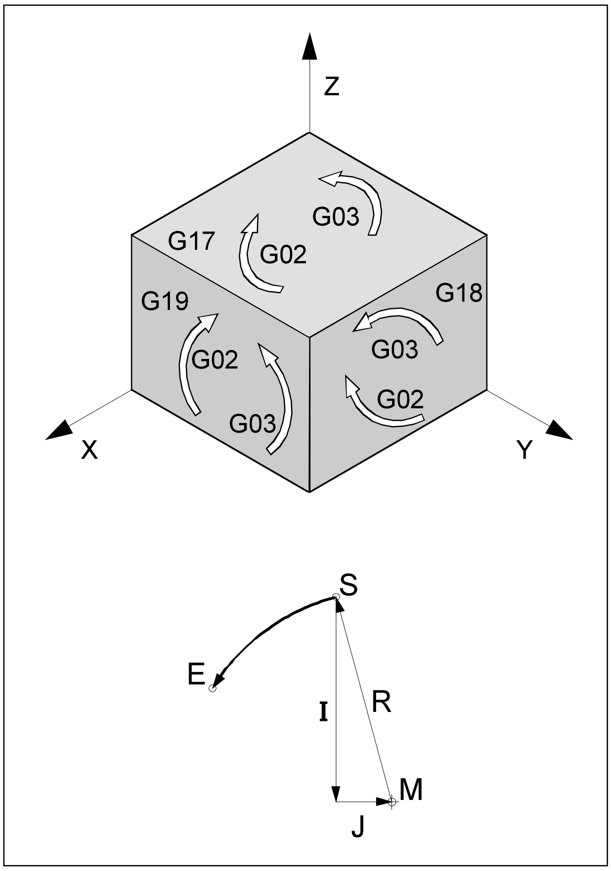
The movement which is programmed has to start at point b of the drawing.
With incremental programming the distance from point b must be programmed.

With single block mode the tool starts first at point c and then at point d.

The following situations cause an error message:
- If the traverse path in one of the two G00/G01 blocks is so short, that with inserting a chamfer or a radius no intersection point would be existing, error message no. 055 will appear.
- If in the second block no G00/G01 command is programmed, error message no. 51, 52 will appear.

G02 Circular Interpolation Clockwise

G03 Circular Interpolation Counterclockwise



Rotational directions of G02 and G03

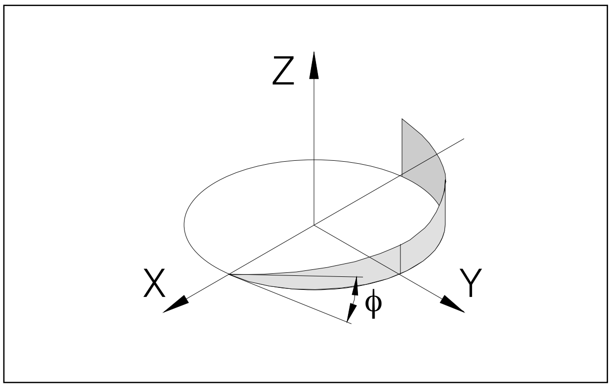
Format
N... G02/G03 X... Y... Z... I... J... K... F...
or
N... G02/G03 X... Y... Z... R... F...

X, Y, Z .. End point of the arc (abs. or incr.)
I, J, K .... Incremental circle parameter (distance from start point to the centre point, I is related to X, J to Y, K to Z)
R........... Radius of the arc (arc < semicircle with +R, > semicircle with -R), can be programmed instead of the circle parameter I, J, K

The tool will be traversed along the defined arc with the programmed feed F.

Notes
The circular interpolation can be proceeded in the active plane only.
Programming the value 0 for I, J or K can be omitted.
The observation of G02, G03 occurs always vertical to the active plane.

Helix Interpolation



Helix curve

Normally only two axes will be programmed for a circle. These axes determine also the active plane. If a third vertical axis will be programmed, the movements of the slides will be coupled in a way that a screw line results.
The programmed feed rate will not be hold at the real path, but on the circle path (projected). The third, linear traversed axis will be controlled in a way, that it reaches the end point at the same time as the circular traversed axes.

Limitations
- A helix interpolation is possible with G17 (XY plane) only.
- The gradient angle ϕ must be less than 45°.
- If the spatial tangents differ more than 2° with block transititions, an exact stop will be proceeded in every case before/after the helix.

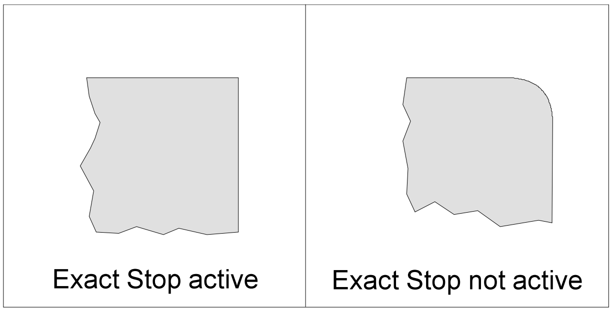
G04 Dwell

Format
N... G04 X... [sec]
or
N... G04 P... [msec]

The tool movement will be stopped for a time defined by X or P in the last reached position - sharp edges - transititions, cleaning drilling ground, exact stop

Notes
- With address P no decimal point can be used
- The dwell starts at the moment when the tool movement speed from the last movement becomes zero.
- t max. = 2000 sec
- Input resolution 100 msec (0.1 sec)

Examples
N75 G04 X2.5 (Dwell = 2.5 sec)
N95 G04 P1000 (Dwell = 1sec = 1000 msec)



G09 Exact Stop

Format
N... G09

A block will then be proceeded, when the slides are braked to 0 before. Therefore the edges will not be rounded and precise transititions will result.
G09 is effective blockwise.

G10 Data Setting

The command G10 allows to overwrite control data, programming parameters, writing tool data etc...
G10 is frequently used to program the workpiece zero point.

Zero point offset

Format
N... G10 L2 Pp IP...;

p=0	External workpiece zero point offset
p=1-6	Normal workpiece zero point offset corrresponding to the coordinatesystem 1 - 6
IP	Workpiece zero point offset for the several axis. At the programming IP become replaced by the axsletters (X,X,Z).

Tool Compensation

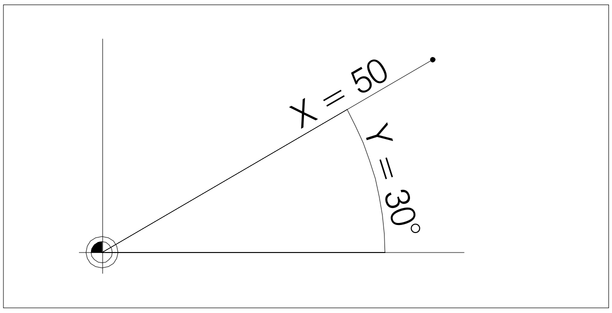
Format
N... G10 L11 P...R...;

P	Number of the toll compensation
R	Tool compensation value in the im absolute command- Mode (G90).

At the inkremental value programming (G91) the tool compensation value get add up to the existing value.

Note: By the reason of compatibility with older NC-programms the system allow the input of L1 instead of L11

G15 End Polar Coordinate Interpolation
G16 Begin Polar Coordinate Interpolation



A point determided by polar coordinates

Format

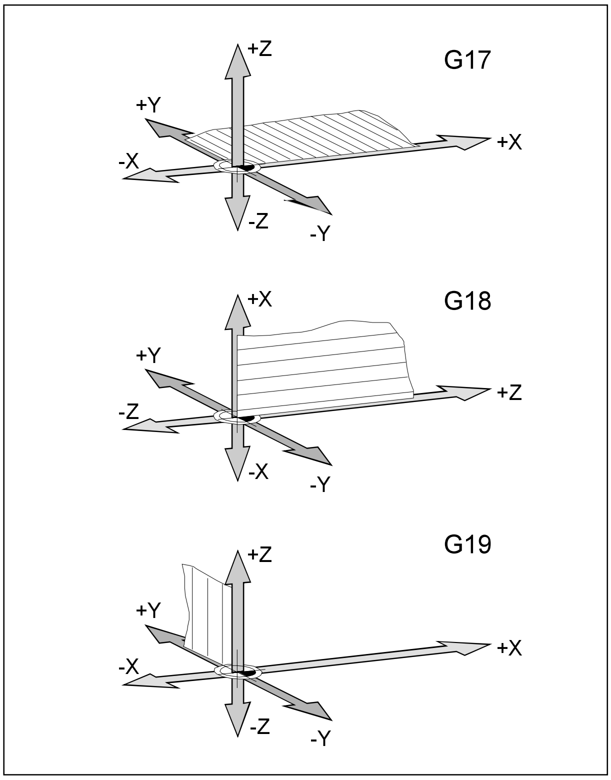
N... G15/G16

Between G16 and G15 points can be defined by polar coordinates.
The selection of the plane in which polar coordinates can be programmed occurs with G17 - G19.
With the address of the first axis the radius will be programmed, with the address of the second axis the angle will be programmed, both related to the workpiece zero point.

Example

N75 G17 G16
N80 G01 X50 Z30

first axis:	radius	X=50
second axis:	angle	Y=30



Definition of the main planes

G17-G19 Plane Selection

Format
N... G17/G18/G19

With G17 to G19 the plane will be defined, in which circular interpolation and polar coordinate interpolation can be proceeded and in which the cutter radius compensation will be calculated.
In the vertical axis to the active plane the tool length compensation will be proceeded.

G17 XY-Plane
G18 ZX-Plane
G19 YZ-Plane

G20 Measuring in Inches

Format
N... G20

By programming G20 the following values will be converted to the inch system:
- Feed F [mm/min, inch/min, mm/rev, inch/rev]
- Offset values (WORK, geometry and wear) [mm, inch]
- Traverse pathes [mm, inch]
- Display of the actual position [mm, inch]
- Cutting speed [m/min, feet/min]

Notes
- For clearness G20 should be programmed in the first block
- The last active measuring system will be hold - even with main switch off/on.
- To get back to the origin measuring system it is the best to use the MDI mode (e.g. MDI-G20-Cycle Start)

G21 Measuring in Millimeter

Format
N... G21

Comments and notes analogous to G20!

G28 Approach Reference Point

Format
N... G28 X... Y... Z...

X, Y, Z Coordinates of the intermediate point.

With G28 the reference point will be approached via an intermediate position (X, Y, Z).
First is the movement to X, Y and Z, then the reference point will be approached. Both movements occur with G00!

The shift G92 will be deleted.

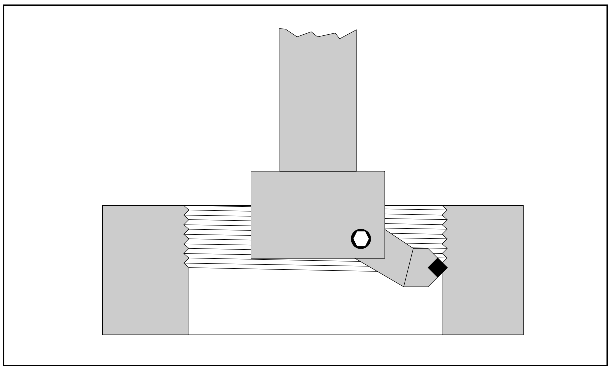
G33 Thread Cutting

Only for PC Mill 100
Format
N... G33 Z... F...

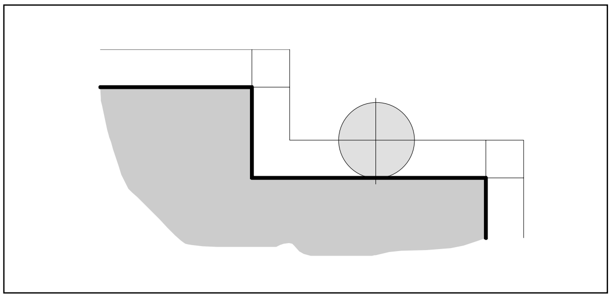
F .... Thread pitch [mm]
Z .... Thread depth

With a fitting tool (boring or facing head) threads can be cut.

Notes
- Feed and spindle override switch are not active while G33 (100%).
- G33 works only with the EMCO PC Mill 100, because the EMCO PC Mill 50 has no encoder on the milling spindle.



Application of thread cutting



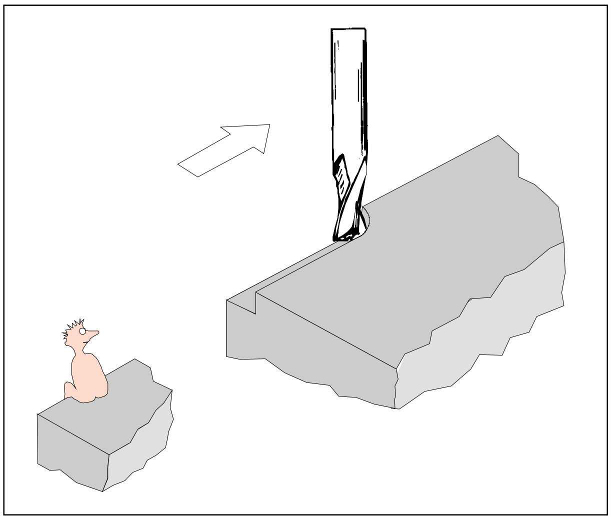
Radius compensated tool path

Cutter Radius Compensation

With the cutter radius compensation the control calculates automatically a path parallel to the programmed contour and compensates so the cutter radius.

G40 Cancel Cutter Radius Compensation

The cutter radius compensation will be cancelled by G40.
Cancellation is only permitted in combination with a linear traversing movement (G00, G01).
G40 can be programmed in the same block like G00 or G01 or in the previous block.
Usually G40 will be programmed with the retraction to the tool change point.



Definition ofG41 cutter radius compensation left

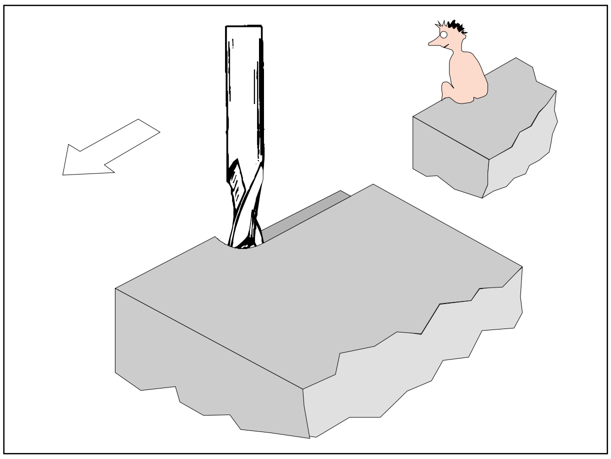
G41 Cutter Radius Compensation left

If the tool is (viewed in feed direction) at the **left** side of the contour to be worked, G41 has to be programmed.
For calculating a radius, an H parameter in the offset register (OFFSET) which represents the cutter radius must be programmed and called up with G41 e.g.:

N... G41 H..

Notes

- Direct change between G41 and G42 is not allowed - previous cancellation with G40.
- Selection in combination with G00 or G01 necessary
- Programming an H parameter is necessary unconditionally, the H parameter is effective modally.



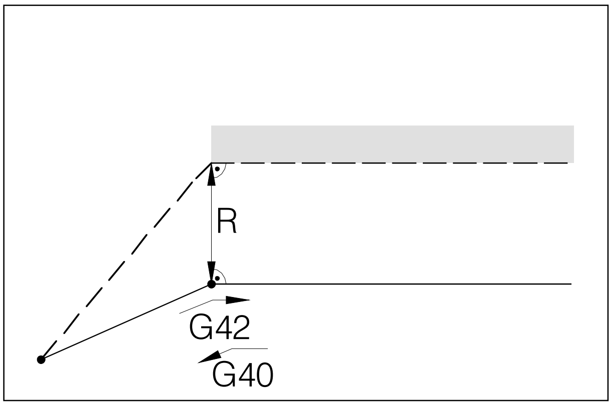
Definition ofG42 cutter radius compensation right

G42 Cutter Radius Compensation right

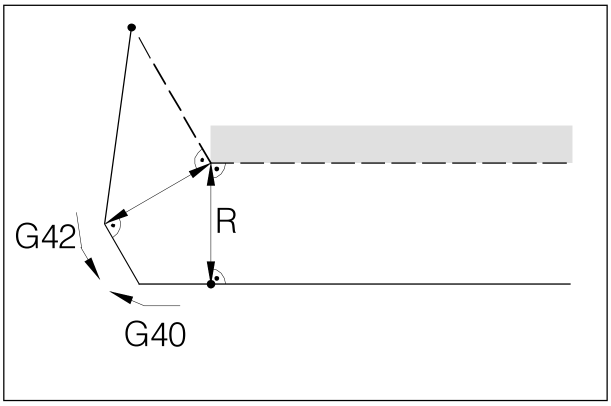
If the tool is (viewed in feed direction) at the **right** side of the contour to be worked, G42 has to be programmed.

Notes see G41!

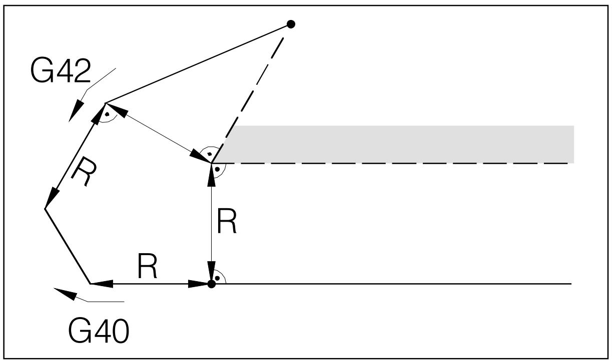
Tool pathes with selection / cancellation of the cutter radius compensation



Frontal approach or leaving of an edge point



Approach or leaving an edge point at side behind



Approach or leaving an edge point behind

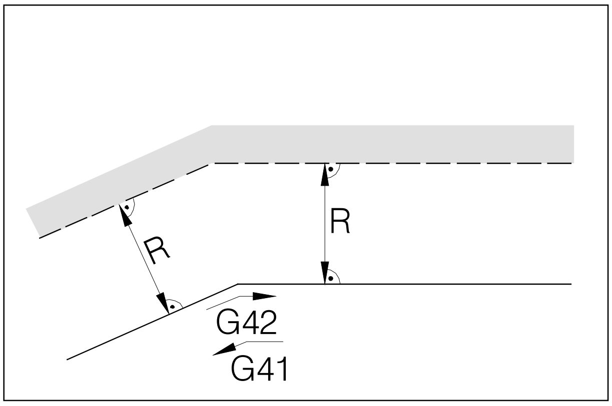
— — — programmed tool path
——— real traversed tool path

With arcs always the tangent of the end or start point of the arc will be approached.

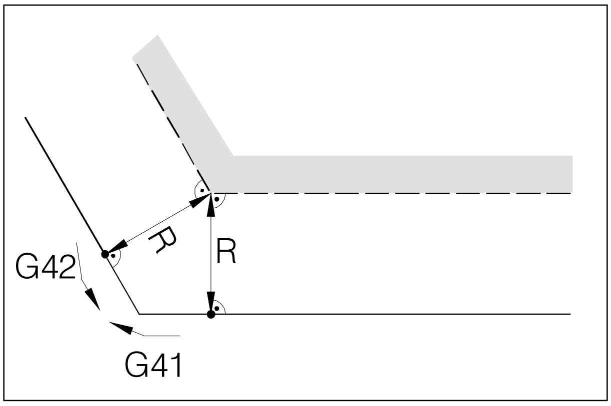
The approaching path to the contour and the leaving path from the contour must be larger than the tool radius R, otherwise program interruption with alarm.

If contour elements are smaller than the tool radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

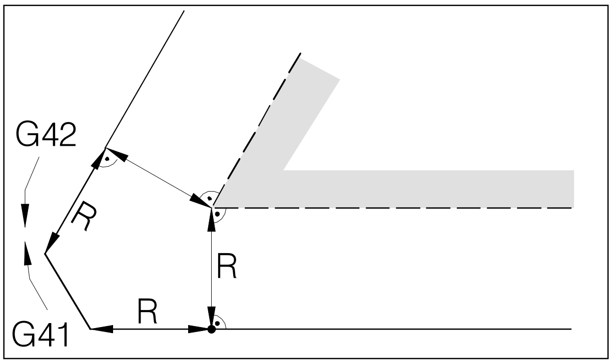
Tool pathes with program run with active cutter radius compensation



Tool path at an inner edge



Tool path at an outer edge > 90°



Tool path at an outer edge < 90°

— — — programmed tool path
——— real traversed tool path

With arcs always the tangent of the end or start point of the arc will be approached.

If contour elements are smaller than the cutter radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

G43 Tool Length Compensation positive
G44 Tool Length Compensation negative

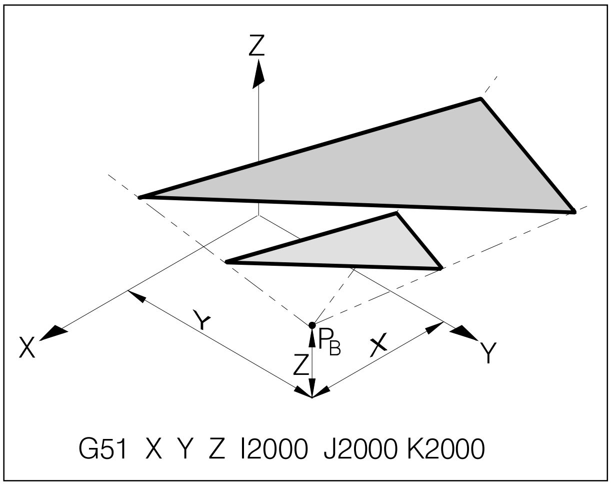
Format:
N... G43/G44 H..

With G43 and G44 a value from the offset register (OFFSET) can be called up and added to or subtracted from as tool length. To all following Z movements (with active XY plane - G17) in the program this value will be added to or subtracted from.

Example:
N... G43 H05
The value, which is written into the register under H05, will be added to all following Z movements as tool length.

G49 Cancel Tool Length Compensation

The positive (G43) or negative (G44) shift will be cancelled.

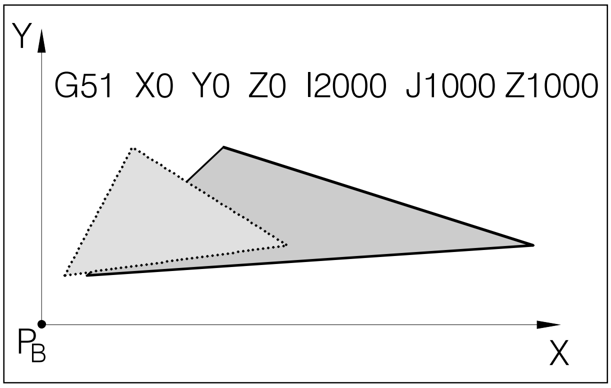


Enlarging a contour 1:2

G50 Cancel Scale Factor, Mirror
G51 Scale Factor, Mirror

Format:
N... G50
N... G51 X... Y... Z... I... J... K...

With G51 all position data will be calculated in a scale, until the scale will be deselected with G50.
With X, Y and Z a base point P_B will be defined, from this point all values will be calculated.
With I, J and K for every axis a scale factor (in 1/1000) can be defined.

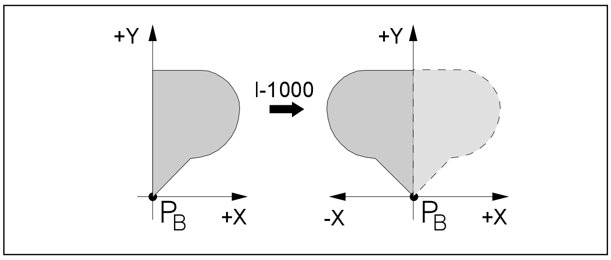


Distortion of a contour: X 1:2, Y,Z 1:1

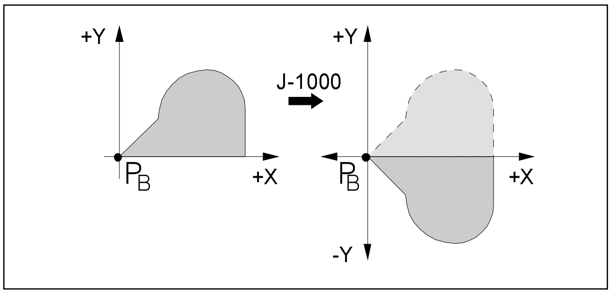
If different scale factors will be defined for the axes, the contour will be distorted.
Circular movements must not be distorted, otherwise alarm.

Mirroring a Contour

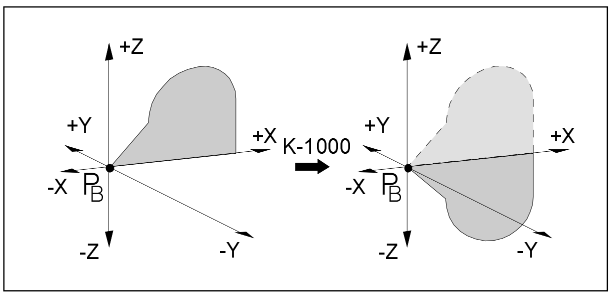
By programming a negative scale a contour will be mirrored around the base point P_B.



By programming I-1000 all X positions will be mirrored around the YZ plane.



By programming J-1000 all Y positions will be mirrored around the ZX plane.



By programming K-1000 all Z positions will be mirrored around the XY plane.

G52 Local Coordinate System

Format:
N... G52 X... Y... Z...

With G52 the actual coordinate zero point can be shifted for the values X, Y, Z.
With this function a sub coordinate system to the existing coordinate system can be created.
G52 is effective blockwise, the resulting shift will be holded, until another shift will be activated.

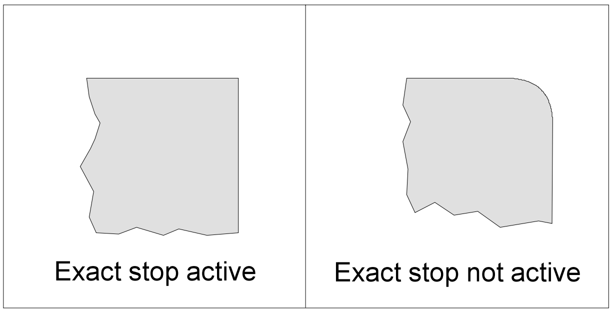
G53 Machine Coordinate System

Format:
N... G53

The machine zero point is determined by the machine manufacturer (EMCO milling machines: at the left front machine table corner).
Certain working sequences (tool change, measuring position...) always will be done at the same position in the working area.
With G53 the zero offset will be cancelled for one program block and the machine coordinate system is active for this block.

G54 - G59 Zero Offsets 1 - 6

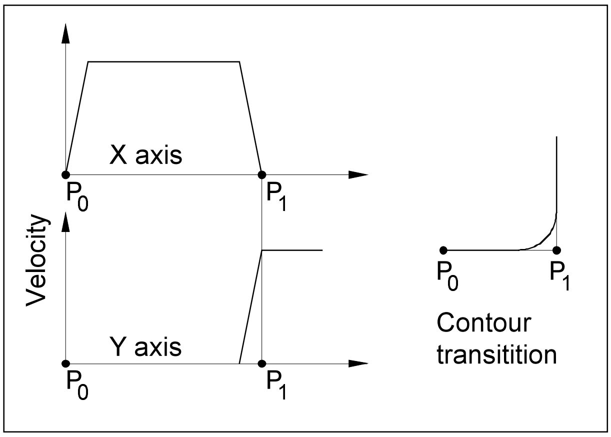
Six positions in the working area can be predetermined as zero points (e.g. points on fix mounted clamping devices). These zero points can be called up with G54 - G59.



G61 Exact Stop Mode

Format
N... G61

A block will then be proceeded, when the slides are braked to 0 before. Therefore the edges will not be rounded and precise transititions will result.
G61 is active, until it will be deselected with G62 or G64.



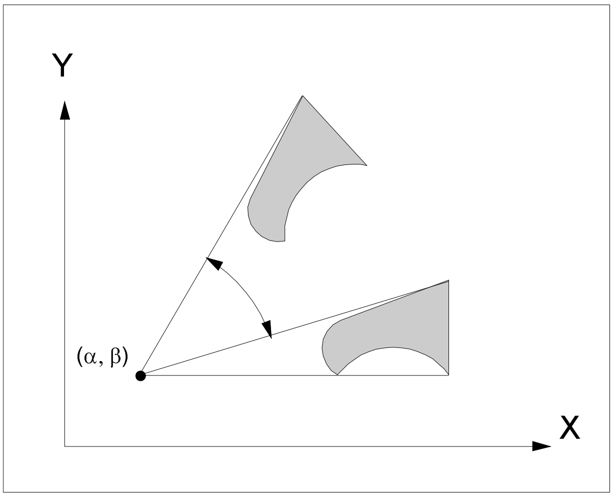
Speed reaction of the slides with G62 and G64

G62 Automatic Corner Override
G64 Cutting mode

Format
N... G62/64

G62 and G64 have the same effect.
Before reaching the target point in X direction the Y slide will already be accelerated. This causes a steady movement with contour transititions. The contour transitition is not exactly sharp-edged (parabola, hyperbola).
The size of the contour transititions is normally within the tolerance of the drawings.

G68 / G69 Coordinate System Rotation

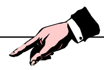


Coordinate System Rotation G68/ G69

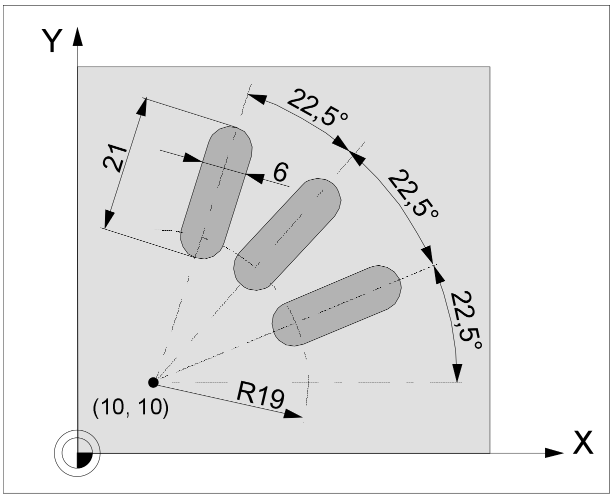
Format:
N... G68 a... b... R...
.
.
N... G69

G68 .... Coordinate System Rotation ON
G69 .... Coordinate System Rotation OFF
α / β .... Indicates the coordinates of the rotational center in the respective plane.
R............ Angel of rotation

For example, this function can be used to alter programs by using a rotational command.



The rotation occurs in the actual valid plane (G17, G18 or G19).



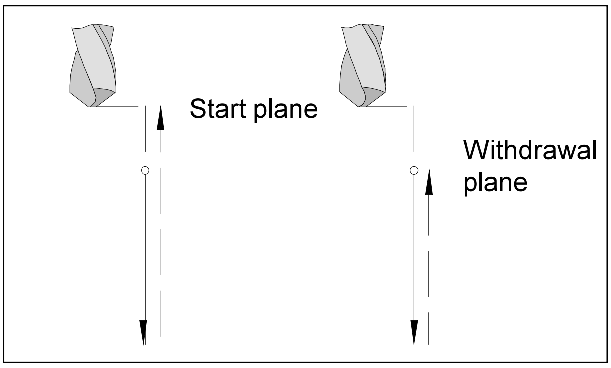
Example Coordinate System Rotation

Example:

```
N5  G54
N10  G43  T10  H10  M6
N15  S2000  M3  F300
N20  M98  P030100        ;Subprogram call
N25  G0  Z50
N30  M30
```

```
O0100 (Subprogram 0100)
N10  G91  G68  X10  Y10  R22.5
N15  G90  X30  Y10  Z5
N20  G1  Z-2
N25  X45
N30  G0  Z5
N35  M17
```

Drilling Cycles G73 - G89



Movements with G98 and G99

Systematic G98/G99

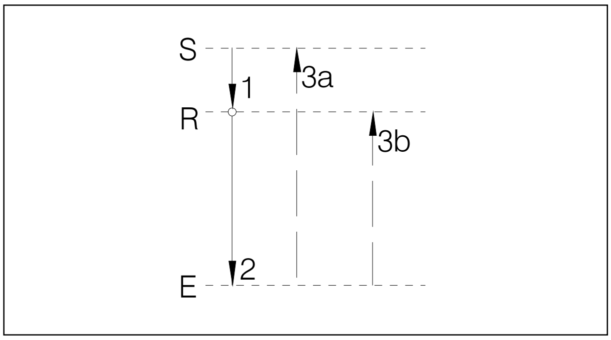
G98 .... After reaching the drilling depth the tool retracts to the start plane
G99 .... After reaching the drilling depth the tool retracts to the withdrawal plane- defined by the R parameter

Is no G98 or G99 active, the tool retracts to the start plane. If G99 (Withdrawal to the withdrawal plane) is programmed the address R must be programmed. With G98 R need not to be programmed.

The compution of the R parameter is different with incremental and absolute programming:

Absolute programming (G90):
R defines the height of the withdrawal plane over the actual workpiece zero point.

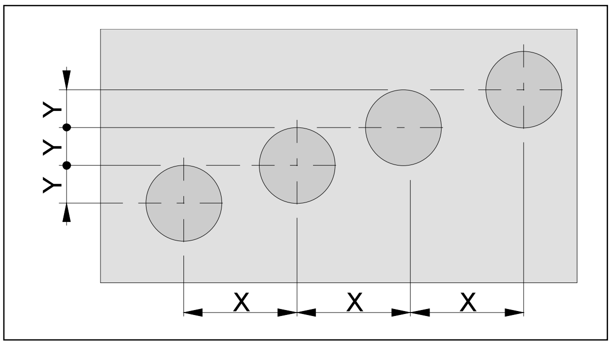
Incremental programming (G90):
R defines the height of the withdrawal plane related to the last Z position (start position of the drilling cycle). With a negative value for R the withdrawal plane will be below the start position, with a positive value the withdrawal plane will be over the start position



Sequence of movements G98, G99

Sequence of movements

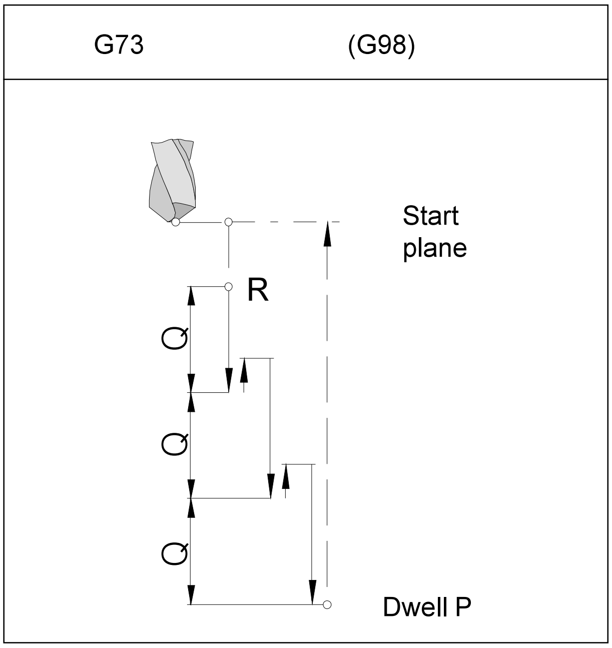
1: The tool traverses with rapid speed from the start position (S) to the plane defined by R (R).
2: Cycle-specific drill machining down to end deptht (E).
3: The withdrawal occurs a: with G98 to the start plane (S) and b: with G99 to the withdrawal plane.



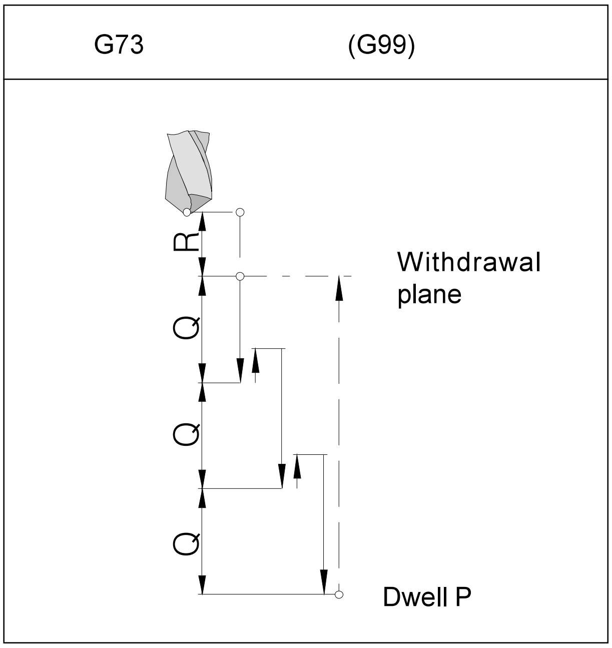
Cycle repetition for a row of holes

Number of repetitions

The K parameter defines the number of repetitions of the cycle.
With absolute programming (G90) it would make no sense to drill several times in the same hole.
With incremental programming (G91) the tool moves on each time for the distances X and Y. This is a simple way of programming rows of borings.



Movements of G73 with active G98



Movements of G73 with active G99

G73 Chip Break Drilling Cycle

Format

N... G98(G99) G73/G83 X... Y... Z... (R...) P... Q... F... K...

The tool dips into the work piece for the infeed Q, drives back 1 mm to break the chips, dips in again etc. until end depth is reached and retracts with rapid feed.

Applications

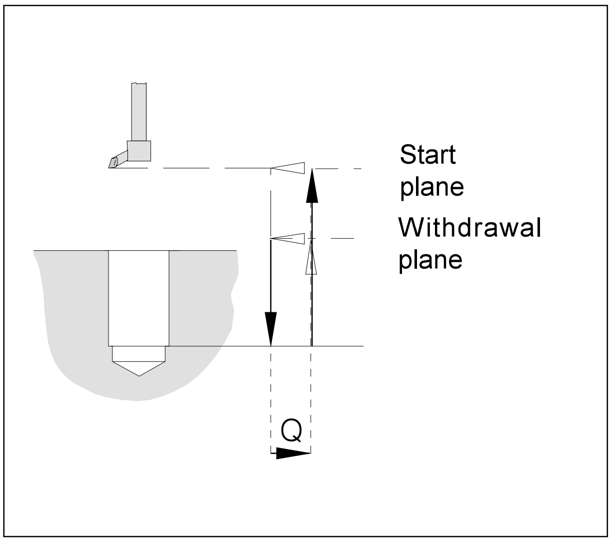
deep borings, material with bad cutting property

G98(G99) .. Return to starting plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
P [msec] .... Dwell at the hole bottom
P1000 = 1 sec
F .... Feed rate
Q [mm] ...... Cutting division - infeed per cut
K............... Number of repetitions

G74 Left Tapping Cycle

Only for PC Mill 100/125/155.
With this cycle left threads can be produced. The cycle G74 works like G84 but with reversed turning directions.
See Tapping Cycle G84.

G76 Fine Drilling Cycle



Movements of G76 - fine drilling cycle

Only for machines with oriented spindle stop.
Format
N...G98(G99) G76 X... Y... Z... (R...) F... Q... K...

This cycle is for enlarging borings with boring and facing heads.
The tool traverses with rapid feed to the withdrawal plane, with the programmed feed to the end depth, the milling spindle will be stopped oriented, the tool traverses with rapid speed horizontally (Q) off the surface (against stop direction) and traverses with rapid speed to the withdrawal plane (G99) or start plane (G98) and traverses back for the value Q to the original position.

G98(G99) .. Retraction to start plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
F .... Feed
Q .... Horizontal traverse-off value
K .... Number of repetitions

G80 Cancel Drilling Cycles

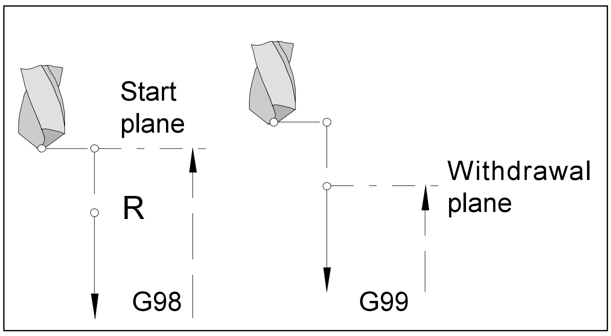
Format
N... G80

The drilling cycles are modal. They have to be cancelled by G80 or another group 1 command (G00, G01, ...).

G81 Drilling Cycle

Format
N...G98(G99) G81 X... Y... Z... (R...) F... K...

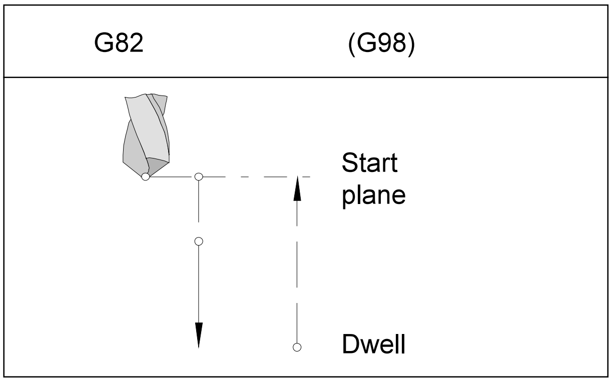
The tool traverses down to end depth with feed speed and retracts with rapid feed.



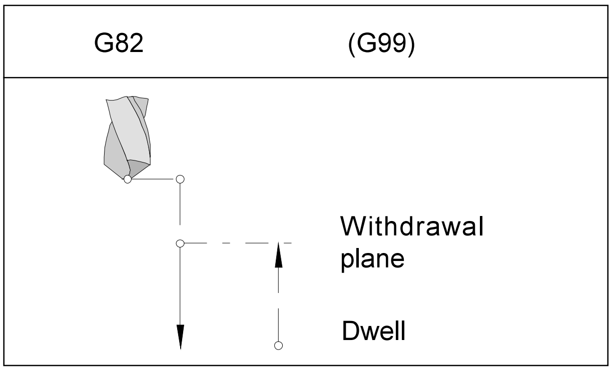
Movements of G81 - drilling cycle

Application:
Short drillings, material with good cutting properties

G98(G99) .. Retraction to start plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
F .... Feed
K .... Number of repetitions



Drilling cycle with dwell and retraction to the start plane



Drilling cycle with dwell and retraction to the withdrawal plane

G82 Drilling Cycle with Dwell

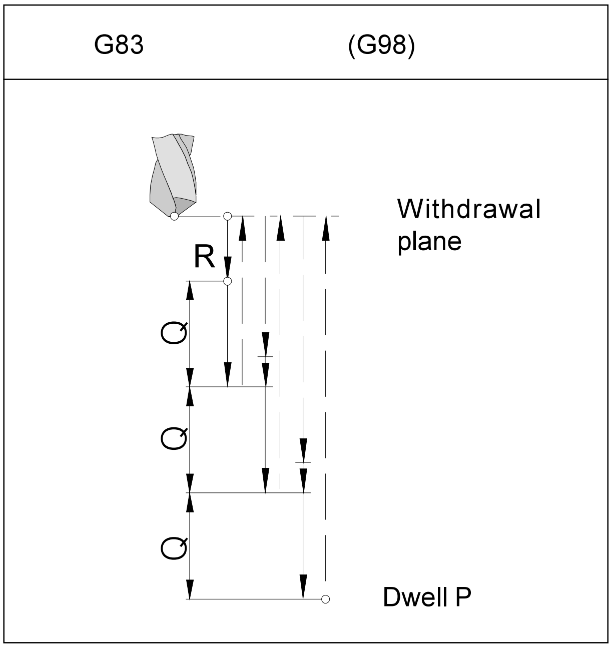
Format
N... G98(G99) G82 X... Y... Z... (R...) P... F... K...

The tool traverses down to end depth with feed speed, dwells turning to clean the hole ground and retracts with rapid feed.

Applications

Short borings, material with good cutting property

G98(G99) .. Return to starting plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
P [msec] .... Dwell at the hole bottom
P1000 = 1 sec
F .... Feed rate
K .... Number of repetitions



Movements of G83 with active G98

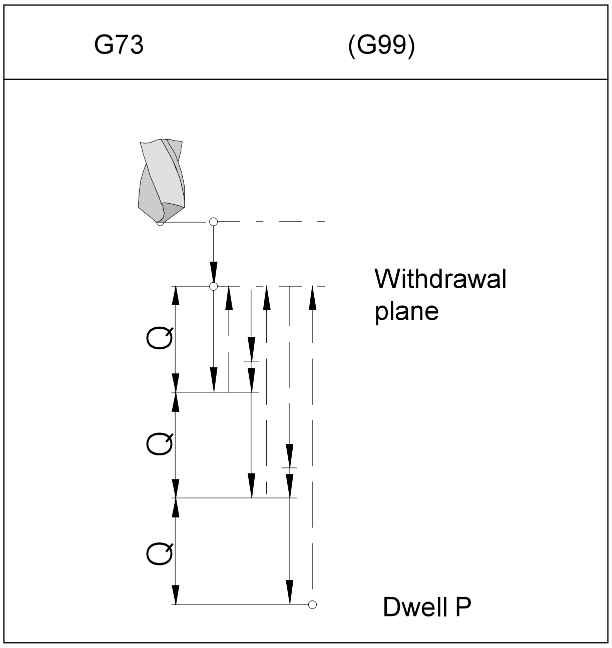
G83 Withdrawal Drilling Cycle

Format
N... G98(G99) G73/G83 X... Y... Z... (R...) P... Q... F... K...

The tool dips into the work piece for the infeed Q, drives back to the start plane (G98) or to the withdrawal plane (G99), to break the chips and remove it from the hole, traverses with rapid speed until 1 mm over the previous drilling depth, dips in again for the infeed Q etc. until end depth is reached and retracts with rapid feed.

Applications

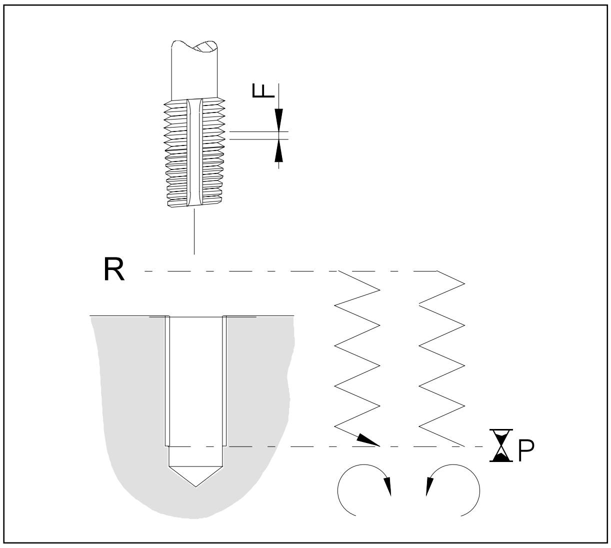
deep borings, (soft) material with long chips



Movements of G83 with active G99

G98(G99) .. Return to starting plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
P [msec] .... Dwell at the hole bottom
P1000 = 1 sec
F .... Feed rate
Q [mm] ...... Cutting division - infeed per cut
K .... Number of repetitions

G84 Tapping Cycle



Tapping cycle (with G99)

Only for PC Mill 100/125/155.
Format
N...G98(G99) G84 X... Y... Z... (R...) F... P... K...

A **tapping chuck with length compensation** must be used.
Spindle override and **feed override** will be set fix to **100 %** while machining.

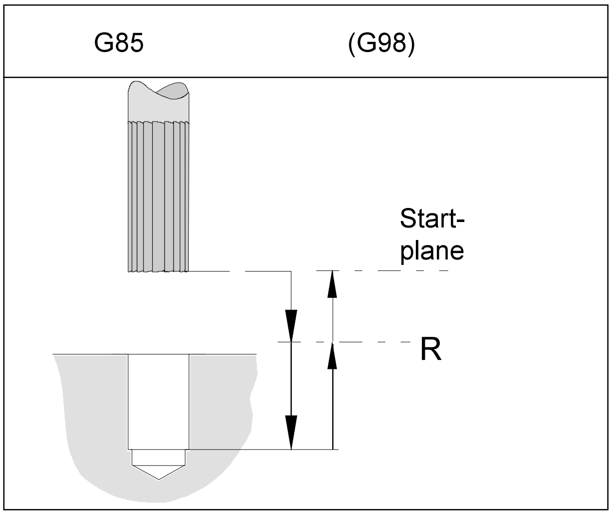
The tool moves turning clockwise with programmed feed into the workpiece down to drilling depth Z, dwells (P), switches to counterclockwise turning and retracts with feed.

G98(G99) .. Retraction to start plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) tapping depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
F .... Thread pitch (feed per revolution)
P .... Dwell at thread ground
K .... Number of repetitions

G85 Reaming Cycle

Format
N... G98 (G99) G85 X... Y... Z... (R...) F... K...

The tool traverses down to end depth with feed speed and retracts to the withdrawal plane with feed. Retraction to withdrawal plane with rapid feed depending on G98.



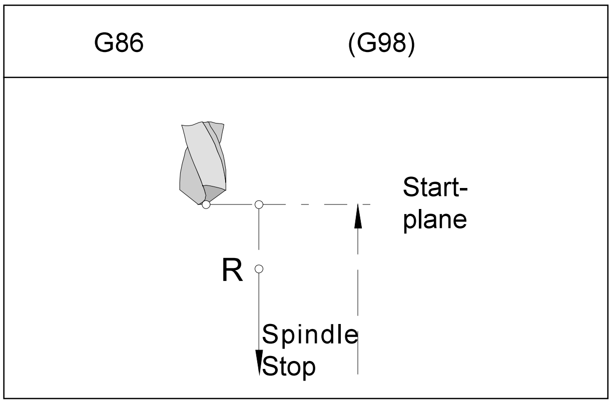
Reaming cycle with withdrawal to the start plane

G98(G99) .. Return to starting plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
F .... Feed rate
K................ Number of repetitions

G86 Drilling Cycle with Spindle Stop

Format
N... G98(G99) G86 X... Y... Z... (R...) F...

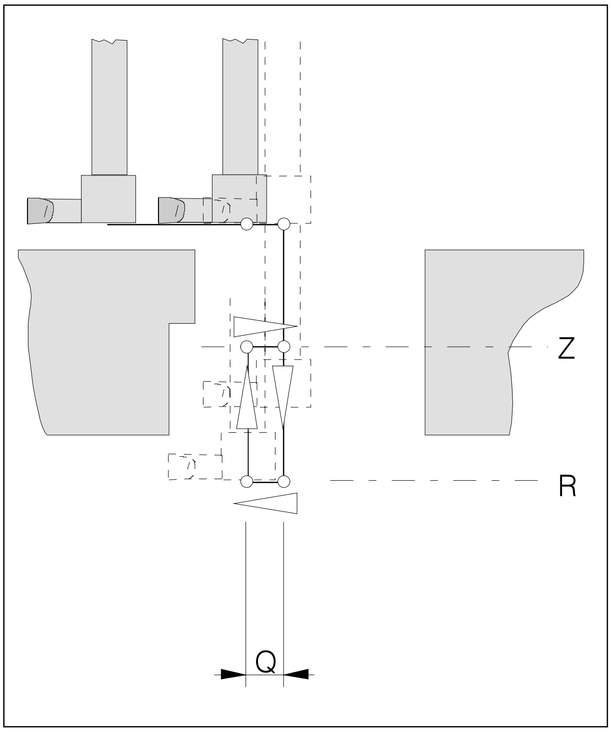
The tool traverses down to end depth with feed speed. At the hole ground the spindle stops and the tool retracts with rapid feed.



Drilling cycle with spindle stop and withdrawal to the start plane

G98(G99) .. Return to starting plane (withdrawal plane)
X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
F .... Feed rate
K................ Number of repetitions

G87 Back Pocket Drilling Cycle



Back pocket drilling cycle

Only for machines with oriented spindle stop

Format

N... G87 X... Y... Z... R... Q... F...

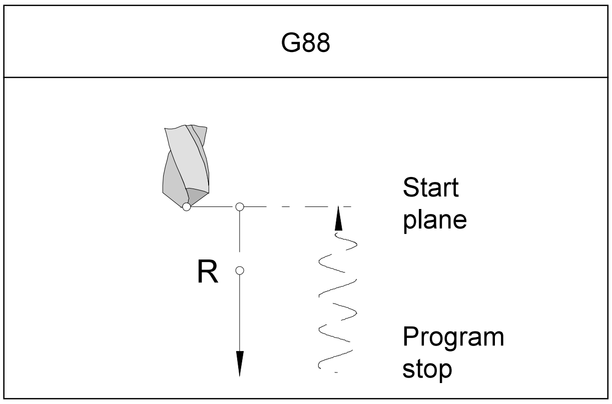
Existing drillings can be enlarged in one direction with a boring or facing head.

- The tool will be positioned in X and Y and stopped oriented.
- It will be traversed horizontally for the distance Q against the stop direction of the oriented stop. The value Q must be larger than the tool diameter to avoid collisions.
- The tool traverses to the depth R (no machining).
- The tool traverses back horizontally for the distance Q on the position X, Y (machining).
- The tool traverses vertical to the height Z (machining).
- At height Z the spindle stops oriented, traverses horizontally for the distance Q against the stop direction of the oriented stop (into the existing drilling) and with rapid feed out of the drilling.
- The tool traverses horizontally for the value Q back to the position X,Y.

G99 can not be programmed, the tool always retracts to the start plane.

X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Back drilling depth
F .... Feed rate

G88 Drilling Cycle with Program Stop



Drilling cycle with program stop

Format

N... G88 X... Y... Z... (R...) P... F... M...

The tool traverses with feed rate to the programmed end depth. At the end depth the program will be stopped after the programmed dwell, retraction occurs manually.

X, Y .... Hole position
Z .... Absolute (incremental) drilling depth
R [mm] ...... Absolute (with G91 incremental) value of the withdrawal plane
P [msec] .... Dwell at end depth: P1000 = 1 sec
F .... Feed rate

G89 Reaming Cycle with Dwell

See G85

The tool traverses with the programmed feed rate to the end depth and dwells (P). Retraction to the withdrawal plane occurs with feed rate, depending on G98 traverses the tool with rapid speed to the start plane.

G90 Absolute Programming

Format
N... G90

Notes
- A direct change between G90 and G91 is allowed also blockwise
- G90 (G91) can be programmed in combination with other G functions.
  (N... G90 G00 X... Y... Z...).

G91 Incremental Programming

Format
N... G91

Notes see G90.

G92 Coordinate System Setting

Format
N... G92 X... Z... (Coordinate System Setting)

Sometimes it is necessary to shift the zero point within a part program. This occurs with G92.

This zero offset is effective modally and will not be cancelled by M30 or RESET. Therefore it is necessary to activate the previous zero point before program end.

G94 Feed per Minute

With G94 all F (feed) values are in mm/min.

G95 Feed per Revolution

Only PC MILL 100
With G95 all F (feed) values are in mm/rev.

G97 Revolutions per Minute

With G97 all S values are in rev/min.

G98 Retraction to the Start Plane
G98 Retraction to the Withdrawal Plane

see "Drilling Cycles G73 - G89".

Description of M Commands

M00 Programmed Stop

This command effects a machining stop within a part program.
The milling spindle, feeds and coolant will be switched off.
The machine door can be opened without releasing an alarm.

With "NC START" the program run can be continued. After that the main drive will be switched on with all values which were valid before.

M01 Programmed Stop, Conditional

M01 works like M00, when OPT. STOP is active (display OPT in the first line at the screen). If OPT. STOP is not active, M01 has no effect.

With "NC START" the program run can be continued. After that the main drive will be switched on with all values which were valid before.

M02 Main Program End

M02 works like M30.

M03 Milling Spindle ON Clockwise

The spindle will be switched on provided that a cutting speed has been programmed, the machine doors are closed and a workpiece is correctly clamped.
M03 must be used for all right hand cutting tools.

M04 Milling Spindle ON Counterclockwise

The same conditions as described under M03 apply here.
M04 must be used for all left hand cutting tools.

M05 Milling Spindle OFF

The main drive is braked electrically.
At the program end the milling spindle is automatically switched off.

M06 Tool Change

Only for machines with tool turret.
The previously with the T word selected tool will be swivelled in.
The T word describes the tool turret station number.

Example:
N100 T04 M06
N110 G43 H4
In the block 100 the tool will be selected by T04 and swivelled in with M06. In the block 110 the length of the tool (entered in H4) will be considered for all following traverse movements (tool length compensation).
After that the main drive will be switched on with all values which were valid before.

M08 Coolant ON

Only for EMCO PC Mill 100/125/155.
The coolant will be switched on.

M09 Coolant OFF

Only for EMCO PC Mill 100/125/155.
The coolant will be switched off.

M27 Swivel Dividing Head

Only for accessory dividing head.
The dividing head will be swivelled for one step (step angle mechanically adjusted).

M30 Main Program End

With M30 all drives will be switched off and the control will be reset to program start.

M71 Puff blowing ON

Only for accessory puff blowing device.
The puff blowing device will be switched on.

M72 Puff blowing OFF

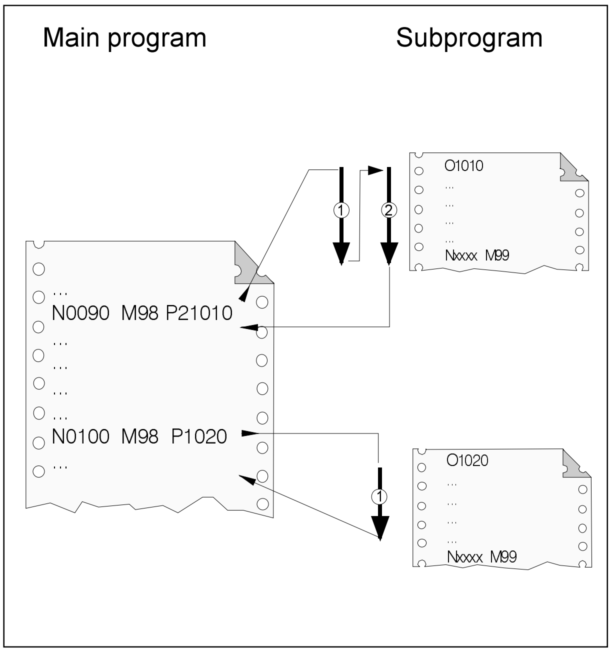
Only for accessory puff blowing device.
The puff blowing device will be switched off.

M98 Subprogram Call

Format
N... M98 P...

P .... The first four digits from the right determine the subprogram number, the other digits the number of repetitions.

Notes
- M98 can be designated in the same block as the movement command (e.g. G01 X25 M98 P1235001)
- When the count of repetitions is not specified, the subprogram is called once (M98 P5001)
- When the programmed subroutine does not exist an alarm occurs.
- A two loop subprogram call can be executed.



Sequence of program run

M99 Subprogram End, Jump Instruction

Format
N... M99 P...

M99 in the main program
Without jumping address:
Jump to the program start.
With jumping address Pxxxx:
Jump on block no. xxxx

M99 in the subprogram
Without jumping address:
Jump to the calling up program, on the next block after the calling up block (see drawing).
With jumping address Pxxxx:
Jump to the calling up program on block no. xxxx

Note
M99 must be the last command in the subprogram.

G: Flexible NC programming

Variable number	Variable type	Function
#0	Always zero system variable	This variable has always the value zero. Not changeable
#1-33	Local variable	At disposal for calculations in the program
#100-149	Global variables	At disposal for calculations in the program
#500-531	System variable	At disposal for calculations in the program
#1000	System variable	Loading magazine: bar end reached
#1001	System variable	Loading magazine: loader has advanced
#1002	System variable	Loading magazine: first part after bar change
#3901	System variable	Nominal piece number
#3901	System variable	Actual piece number

Variables and arithmetic parameters

By using variables instead of fixed values, a program can be configured more flexibly. Thus, you can react to signals, such as e.g. measuring values, or the same program can be used for different geometries by using variables as nominal value.
Together with variable calculation and program jumps you get the possibility to create a highly-flexible program archive and thus save programming time.

Local and global variables can be read and written. All other variables can only be read.

Local variables can only be used in that macro in which they are defined.
Global variables can be used in every macro irrespective of the macro in which they were defined.

Calculating with variables

Function	Example
=	#1=2
+	#1=#2+#3
-	#1=#2-#3
*	#1=#2*#3
/	#1=#2/#3

With the four basic arithmetic operations the usual mathematic notation is valid.

The term at the operator's right can contain constants and/or variables combined by functions.

Each variable can be replaced again by an arithemetic term in square brackets or by a constant.

Example

#1=#[#2]

During the calculation the limitation is valid that the execution of the calculation is carried out from left to right without observance of the calculation rule point before line.

Example

#1=#2*3+#5/2

Control structures

In programs the control sequence can be changed by IF and GOTO instructions. Three types of branchings are possible:

- IF[<condition>] THEN
- IF[<condition>] GOTO <n>
- GOTO <destination>

IF[<Condition>] THEN

After IF a provisory term must be indicated. If the provisory term applies, a determined macro instruction is carried out. Only one macro instruction can be carried out.

Example

With equal values of #1 and #2 the value 5 is allocated to #3.

IF [#1 EQ #2] THEN#3=5

IF[<Condition>] GOTO <n>

After IF a provisory term must be indicated. If the provisory term applies, the branching is carried out to block number n. Otherwise the subsequent block is carried out.

Example

If the value of the variable #1 is greater than 10, the branching is carried out to block number N4. Otherwise the subsequent block is carried out.

IF [#1 GT 10] GOTO 4

GOTO <n>

The jump command GOTO can also be programmed without condition. A variable or constant can be used as a branch destination. With a variable the number can be replaced again by a calculation term in square brackets.

Example

Jump to block number 3

GOTO 3

Example

Jump to variable #6

GOTO#6

Relational operators

Relational operators consist of two letters and are used to determine, in comparison with two values, if these are equal or if one value is greater and/or less than the other.

Operator	Meaning
EQ	Equal (=)
NE	Unequal (≠)
GT	Greater than (>)
GE	Greater than or equal (=)
LT	Less than (<)
LE	Less than or equal (=)

The expressions to be compared can be variable n or constants. A variable can be replaced again by a calculation term in square brackets.

Example

IF[#12 EQ 1] GOTO10

Comprising macro programming examples:

IF[#1000 EQ 1] GOTO10

IF[#[10]] NE #0] GOTO#[#1]

IF[1 EQ 1] THEN#2 =5

IF[#[#4+#[#2/2]] GT #20] THEN#[#10]] =#1*5+#7

H: Alarms and Messages

Missing digitizer calibration
Cause: A digitizer tablet has been installed but not calibrated
Remedy: Calibrate digitizer tablet (set corner points), see External Input Devices

6: CONVERTER ALREADY INITIALIZED
System error. Re-install the software.

7: MISSING SETUP CALL
System error. Re-install the software.

8: SETUP OF PROGRAM CONTROLLING FAILS
System error. Re-install the software.

9: PARSER SETUP FAILS
System error. Re-install the software.

10: REGISTRY SETUP FAILS
System error. Re-install the software.

11: SETUP OF WORKING POINTS FAILS
System error. Re-install the software.

12: SETUP OF WORKING OBJECTS FAILS
System error. Re-install the software.

13: SETUP OF COMMAND LIST FAILS
System error. Re-install the software.

14: SETUP OF START CONDITION FAILS
System error. Re-install the software.

15: SETUP OF EXPORT VARIABLE FAILS
System error. Re-install the software.

16: SETUP OF MAIN VARIABLE FAILS
System error. Re-install the software.

17: WAITING FOR AC INITIALIZIATION
System error. Re-install the software.

18: SETUP OF AC FAILS
System error. Re-install the software.

19: INVALID SWITCHTONEXTBLOCK ID
System error. Re-install the software.

20: NO PROGRAM
System error. Re-install the software.

21: PROGRAM NOT FOUND
System error. Re-install the software.

1000: PARSER - ERROR OT_FIRST
System error. Re-install the software.

1001: MARKING OF HEADER LINE NOT FOUND
System error. Re-install the software.

1002: MARKING OF MAIN PROGRAM NOT FOUND
System error. Re-install the software.

1005: INVALID LINE NUMBER
System error. Re-install the software.

1006: NO END OF FUNCTION FOUND
System error. Re-install the software.

1007: MODUL NAME ALREADY EXISTS
System error. Re-install the software.

1009: INVALID MODUL NAME
System error. Re-install the software.

1010: NO LINE NUMBER
System error. Re-install the software.

1018: WAIT FOR F OR S COMMAND
NC programming error. G4 was programmed without S or F address.

1020: NO MORE COMMANDS ALLOWED
NC programming error. G96 must be the only G command in a block.

1035: NO OR INVALID PARAMETER
System error. Re-install the software.

2016: INVALID S VALUE
NC programming error. Spindle index invalid. Allowed indexes are S[0] and S[1].

Input Device Alarms 3000 - 3999

These Alarms will be triggered by the control keyboard or digitizer.

Missing digitizer calibration
Cause: A digitizer tablet has been installed but not calibrated
Remedy: Calibrate digitizer tablet (set corner points), see External Input Devices

3001 General RS232 communication error
Remedy: Correct settings of serial interfacae.

3002 Control keyboard missing
Remedy: Connect control keyboard, switch on, ...

3003 Digitizer missing
Remedy: Connect digitizer, switch on, ...

3004 Check sum error in control keyboard

The keyboard tries an automatic re-initializing - when failed switch off / on keyboard.

3005 Error in control keyboard

The keyboard tries an automatic re-initializing - when failed switch off / on keyboard.

3006 Error with initializing control keyboard

The keyboard tries an automatic re-initializing - when failed switch off / on keyboard.

Machine Alarms 6000 - 7999

These alarms will be triggered by the machines. There are different alarms for the different machines. The alarms 6000 - 6999 normally must be confirmed with RESET. The alarms 7000 - 7999 are messages which normally will disappear when the releasing situation is finished.

PC MILL 50 / 55, PC TURN 50 / 55

The following alarms are valid for the turning and milling machines of the series 50 / 55.

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: CYCLE TIME EXCEEDS LIMIT
Contact EMCO Service.

6002: NO PLC PROGRAM LOADED
Contact EMCO Service.

6003: DB NOT EXISTENT
Contact EMCO Service.

6004: RAM ERROR ON PLC BOARD
Contact EMCO Service.

6009: FAILURE SAFETY CIRCUIT
Defective door limit switch or main contactor. Operating the machine is not possible.
Contact EMCO Service.

6010: X-AXIS NOT READY
Step motor board defective, 24 V or 30 V fuse defective. Check fuses and switch box fan filter.
Contact EMCO Service.

6011: Y-AXIS NOT READY
see alarm 6010.

6012: Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, cable defective, main drive overload.
Check fuse, reduce load.
Contact EMCO service.

6014: NO SPEED FOR MAIN SPINDLE
This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).

6019: VICE TIMEOUT
24 V fuse defective, hardware defective.
Contact EMCO service.

6020: VICE FAILURE
24 V fuse defective, hardware defective.
Contact EMCO service.

6024: DOOR NOT CLOSED
The door was opened while a machine movement. The program will be aborted.

6025: GEARBOX COVER NOT CLOSED
The gearbox cover was opened while a machine movement. A running CNC program will be aborted. Close the cover to continue.

6027: DOOR LIMIT SWITCH DEFECTIVE
The limit switch of the automatic door is displaced, defective, wrong cabled.
Contact EMCO service.

6028: DOOR TIMEOUT
The automatic door stucks, the pressured air supply is insufficient, the limit switch is displaced.
Check door, pressured air supply, limit switch or contact EMCO service.

6030: NO PART CLAMPED
No workpiece inserted, vice cheek displaced, control cam displaced, hardware defective.
Adjust or contact EMCO service.

6031: QUILL FAILURE

6037: CHUCK TIMEOUT

6039: CHUCK PRESSURE FAILURE

6041: TOOL CHANGE TIMEOUT
Tool turret stucks (collision?), 24 V fuse defective, hardware defective.
A running CNC program will be stopped.
Check for a collision or contact EMCO service.

6042: TOOL CHANGE TIMEOUT
see alarm 6041.

6043: TOOL CHANGE TIMEOUT
see alarm 6041.

6044: TOOL TURRET SYNC ERROR
Hardware defective.
Contact EMCO service.

6046: TOOL TURRET SYNC MISSING
Hardware defective.
Contact EMCO service.

6048: DIVIDING TIME EXCEEDED
Dividing head stucks, insufficient pressured air supply, hardware defective.
Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED
see alarm 6048

6050: FAILURE DIVIDING DEVICE
Hardware defective.
Contact EMCO service.

7000: INVALID TOOL NUMBER
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7007: FEED HOLD
In the robotic mode a HIGH signal is at input E3.7. Feed Stop is active until a low signal is at E3.7.

7017: GO FOR REFERENCE POINT
Approach the reference point.

7040: DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated.
Some accessories can be operated only with open machine door.
Close the machine to run a program.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7050: NO PART CLAMPED
After switching on or after an the vice is neither at the open position nor at the closed position.
NC-Start is locked.
Traverse the vice manually on a valid end position.

7051: DIVIDING DEVICE NOT INTERLOCKED
After switching on or after an the dividing head is not in a lock position. NC-Start is locked.

PC MILL 100/105/125/155

The following alarms are valid for the milling machines PC MILL 100/105/125/155.

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
Contact EMCO Service.

6003: PLC - NO DATA UNIT
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
Contact EMCO Service.

6005: OVERHEAT BRAKEMODUL
Main drive was braked too often, large changes of speed within a short time. E4.2 active

6006: OVERLOAD BRAKE RESISTOR
see 6005

6007: SAFETY CIRCUIT FAULT
Axis and main drive contactor with machine switched off not disabled. Contactor got stuck or contact error. E4.7 was not active during switch-on.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse or cabling is defective.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6011: DRIVE Y-AXIS NOT READY
see alarm 6010.

6012: DRIVE Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, main drive too hot, fuse defective.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxilliary drives will be stopped.

6024: MACHINE DOOR OPEN
The door was opened while a machine movement. The program will be aborted.

6040: TOOL TURRET INDEX FAILURE
After WZW procedure drum pressed down by Z-axis. Spindle position wrong or mechanical defect. E4.3=0 in lower state

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), main drive not ready, fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6043-6046: TOOL DISK POSITION FAULT
Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.
The Z axis could have been slipped out of the toothing while the machine was switched off.
A running CNC program will be stopped.
Contact EMCO service.

6047: TOOL DISK UNLOCKED
Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.
A running CNC program will be interrupted.
Contact EMCO service.
When the tool drum is turned out of locked position (no defect), act as following:
Turn the drum into locking position manually
Change into MANUAL (JOG) mode.
Turn the key switch. Traverse the Z slide upwards, until the alarm disappears.

6050: M25 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY
Cause: pressure failure automatic door
automatic door stucks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN
When opening the clamping pressure switch does not fall within 400ms. Pressure switch defective or mechanical problem. E22.3

6070: PRESSURE SWITCH FOR TANI MISSING
When closing the clamping pressure switch does not respond. No compressed air or mechanical problem. E22.3

6071: DIVIDING DEVICE NOT READY
Servo Ready Signal from frequency converter missing. Excess temperature drive TANI or frequency converter not ready for operation.

6072: VICE NOT READY
Attempt to start the spindle with an open vice or without clamped workpiece.
Vice stucks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.
Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY
Cause: locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device
lock the dividing device

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device

6075: M27 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 10.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives.

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is not possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: INITIALIZE TOOL TURRET
The tool turret operating was interrupted.
No traversing operation is possible.
Press tool turret key in JOG operation. Message occurs after alarm 6040.

7022: INITIALIZE TOOL TURRET !
see 7021

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated (except special operation mode)
Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7054: VICE OPEN
Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6073 (vice not ready) will be released.
Remedy: Clamp

7055: DIVIDING DEVICE NOT LOCKED
Cause: the dividing device is not locked
When switching on the main spindle with M3/M4 alarm 6073 (dividing device not ready) will be released.
Remedy: lock dividing device

7270: OFFSET COMPENSATION ACTIVE !
Only with PC-MILL 105
Offset compensation activated by the following operation sequence.
- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4

This must be carried out if prior to the tool change procedure spindle positioning is not completed (tolerance window too large)

7271: COMPENSATION FINISHED,DATA SAVED !
see 7270

PC TURN 105/120/125/155

The following alarms are valid for the lathes PC TURN 105/120/125/155.

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed.
The reference position will be lost, the auxiliary drives will be switched off.
Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
The auxiliary drives will be switched off.
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
The auxiliary drives will be switched off.
Contact EMCO Service.

6003: PLC - NO DATA UNIT
The auxiliary drives will be switched off.
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
The auxiliary drives will be switched off.
Contact EMCO Service.

6008: MISSING CAN SUBSCRIBER
Check fuses or contact EMCO Service.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6012: DRIVE Z-AXIS NOT READY
see 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This alarm will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxiliary drives will be switched off.

6015: NO DRIVEN TOOL SPINDLE SPEED
see 6014.

6024: MACHINE DOOR OPEN
The door was opened while a machine movement.
The program will be aborted.

6040: TOOL TURRET INDEX FAILURE
The tool turret is in no locked position, tool turret sensor board defective, cabling defective, fuse defective.
A running CNC program will be stopped.
Swivel the tool turret with the tool turret key, check fuses or contact EMCO service.

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6042: TOOL TURRET OVERHEAT
Tool turret motor too hot.
With the tool turret a max. of 14 swivel procedures a minute may be carried out.

6043: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6046: TOOL TURRET ENCODER FAULT
Fuse defective, hardware defective.
Check fuses or contact EMCO service.

6048: CHUCK NOT READY
Attempt to start the spindle with open chuck or without clamped workpiece.
Chuck stucks mechanically, insufficient pressured air supply, fuse defective, hardware defective.
Check fuses or contact EMCO service.

6049: COLLET NOT READY
see 6048

6050: M25 DURING SPINDLE ROTATION
With M25 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6055: NO PART CLAMPED
This alarm occurs when with rotating spindle the clamping device or the tailstock reach the end position.
The workpiece has been pushed out of the chuck or has been pushed into the chuck by the tailstock.
Check clamping device settings, clamping forces, alter cutting data.

6056: QUILL NOT READY
Attempt to start the spindle or to move an axis or to swivel the tool turret with undefined tailstock position.
Tailstock is locked mechanically (collision), insufficient pressured air supply, fuse defective, magnetic switch defective.
Check for collisions, check fuses or contact EMCO service.

6057: M20/M21 DURING SPINDLE ROTATION
With M20/M21 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6058: M25/M26 DURING QUILL FORWARD
To actuate the clamping device in an NC program with M25 or M26 the tailstock must be in back end position.

6059: C-AXIS SWING IN TIMEOUT
C-axis does not swivel in within 4 seconds.
Reason: not sufficient air pressure, and/or mechanics stuck.

6060: C-AXIS INDEX FAILURE
When swivelling in the C-axis the limit switch does not respond.
Check pneumatics, mechanics and limit switch.

6064: AUTOMATIC DOOR NOT READY
Door stucks mechanically (collision), insufficient pressured air supply, limit switch defective, fuse defective.
Check for collisions, check fuses or contact EMCO service.

6065: LOADER MAGAZINE FAILURE
Loader not ready.
Check if the loader is switched on, correctly connected and ready for operation and/or disable loader (WinConfig).

6066: CLAMPING DEVICE FAILURE
No compressed air at the clamping device
Check pneumatics and position of the clamping device proximity detectors.

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 8.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives (also a lubricating pulse will be released).

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7019: PNEUMATIC LUBRICATION MONITORING!
Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: TOOL TURRET NOT LOCKED
The tool turret operating was interrupted.
NC start and spindle start are locked. Press the tool turret key in the RESET status of the control.

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This alarm can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated (except special operation mode)
Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7048: CHUCK OPEN
This message shows that the chuck is open. It will disappear if a workpiece will be clamped.

7049: CHUCK - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7050: COLLET OPEN
This message shows that the collet is open. It will disappear if a workpiece will be clamped.

7051: COLLET - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7052: QUILL IN UNDEFINED POSITION
The tailstock is in no defined position.
All axis movements, the spindle and the tool turret are locked.
Drive the tailstock in back end position or clamp a workpiece with the tailstock.

7053: QUILL - NO PART CLAMPED
The tailstock reached the front end position. Traverse the tailstock back to the back end position to continue.

7054: NO PART CLAMPED
No part clamped, switch-on of the spindle is locked.

7055: CLAMPING DEVICE OPEN
This message indicates that the clamping device is not in clamping state. It disappears as soon as a part is clamped.

AC95 ALARMS

Axis Controller Alarms 8000 - 9999

8000 Fatal Error AC

8004 ORDxx Failure main-drive unit
8005 - 8009 ORDxx Internal error AC
Remedy: report to EMCO if repeatable

8010 ORDxx Syncr. error main drive
Cause: synchronisation mark missing for main drive
Remedy: report to service technician, if repeatable

8011 - 8013 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8014 ORDxx Decel.-time of axis too high
Remedy: report to service technician, if repeatable

8018 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8021 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8022 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8023 ORDxx Invalid Z value for helix
Cause: The Z value of the helix must be smaller than the length of the arc to be traversed
Remedy: Program correction

8100 Fatal init error AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8101 Fatal init error AC
see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

8106 No PC-COM card found
Cause: PC-COM board can not be accessed (ev. not mounted).
Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

8110 PC-COM init message missing
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

8117 Fatal init data error (pccom.hex)
see 8110.

8118 Fatal init error AC
see 8110, ev. insufficient RAM memory

8119 PC interrupt no. not valid
Cause: The PC interrupt number can not be used.
Remedy: Find out free interrupt number in the Windows95 system control (allowed: 5,7,10, 11, 12, 3, 4 und 5) and enter this number in WinConfig.

8120 PC interrupt no. unmaskable
see 8119

8121 Invalid command to PC-COM
Cause: Internal error or defective cable
Remedy: Check cables (screw it); Restart software or reinstall when necessary, report to EMCO, if repeatable.

8122 Internal AC mailbox overrun
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8123 Open error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8124 Write error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8125 Invalid memory for record buffer
Cause: Insufficient RAM, record time exceeding.
Remedy: Restart software, ev. remove drivers etc. to gain more RAM, reduce record time.

8126 AC Interpolation overrun
Cause: Ev. insufficient computer performance.
Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

8127 Insufficient memory
Cause: Insufficient RAM
Remedy: Close other programs, restart software, ev. remove drivers etc. to gain more RAM.

8128 Invalid message to AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8130 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The combination of linear and circular axes does not match.
Remedy: Program correction.

8140 Maschine (ACIF) not responding
Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

8141 Internal PC-COM error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8142 ACIF Program error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

8145 Fatal init data error (acif.hex)
see 8142.

8146 Multiple request for axis
see 8142.

8147 Invalid PC-COM state (DPRAM)
see 8142.

8148 Invalid PC-COM command (CNo)
see 8142.

8149 Invalid PC-COM command (Len)
see 8142.

8150 Fatal ACIF error
see 8142.

8151 AC Init Error (missing RPG file)
see 8142.

8152 AC Init Error (RPG file format)
see 8142.

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

8155 Invalid FPGA packet acknowledge
see 8142 or hardware error on ACIF board (contact EMCO Service).

8156 Sync within 1.5 revol. not found
see 8142 or Bero hardware error (contact EMCO Service).

8157 Data record done
see 8142.

8158 Bero width too large (referencing)
see 8142 or Bero hardware error (contact EMCO Service).

8159 Function not implemented
Bedeutung: In normal operation this function can not be executed

8160 Axis synchronization lost axis 3..7
Cause: Axis spins or slide is locked, axis synchronisation was lost
Remedy: Approach reference point

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:
- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm) or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost
see 8161

8163 Z-Axis synchronization lost
see 8161

8164 Software limit switch max axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8168 Software limit overtravel axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.
Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running

8174 INC not allowed

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

8182 Gear change not allowed

8183 Gear too high

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message

8191 Invalid jog feed unit

8192 Invalid axis in command

8193 Fatal PLC error
see 8175.

8194 Thread without length

8195 No thread slope in leading axis
Remedy: Program thread pitch

8196 Too manny axis for thread
Remedy: Program max. 2 axes for thread.

8197 Thread not long enough
Cause: Thread length too short.
With transition from one thread to the other the length of the second thread must be sufficient to produce a correct thread.
Remedy: Longer second thread or replace it by a linear interpolation (G1).

8198 Internal error (to manny threads)
see 8175.

8199 Internal error (thread state)
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

8203 Fatal AC error (0-ptr IPO)
see 8199.

8204 Fatal init error: PLC/IPO running
see 8199.

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8211 Feed too high (thread)
Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed
Remedy: Program correction, lower pitch or lower spindle speed for thread

8212 Rotation axis not allowed
8213 Circle and rotation axis can't be interpolated
8214 Thread and rotation axis cant't be interpolated
8215 Invalid state
8216 No rotation axis for rotation axis switch
8217 Axis type not valid!
8218 Referencing round axis without selected round axis!
8219 Thread not allowed without spindle encoder!
8220 Buffer length exceeded in PC send message!
8221 Spindle release although axis is no spindle!
8222 New master spindle is not valid
8223 Can't change master spindle (no M5)!
8224 Invalid stop mode
8225 Invalid parameter for BC_MOVE_TO_IO!
8226 Rotary axis switch not valid (MSD data)!
8227 Speed setting not allowed while rotary axis is active!
8228 Rotary axis switch not allowed while axis move!
8229 Spindle on not allowed while rotary axis is active!
8230 Program start not allowed due to active spindle rotation axis!
8231 Axis configuration (MSD) for TRANSMIT not valid!
8232 Axis configuration (MSD) for TRACYL not valid!
8233 Axis not available while TRANSMIT/TRACYL is active!
8234 Axis control grant removed by PLC while axis interpolates!
8235 Interpolation invalid while axis control grant is off by PLC!
8236 TRANSMIT/TRACYL activated while axis or spindle moves!
8237 Motion through pole in TRANSMIT!
8238 Speed limit in TRANSMIT exceeded!
8239 DAU exceeded 10V limit!
8240 Function not valid during active transformation (TRANSMIT/TRACYL)!
8241 TRANSMIT not enabled (MSD)!
8242 TRACYL not enabled (MSD)!
8243 Round axis invalid during active transformation!
8245 TRACYL radius = 0!
8246 Offset alignment not valid for this state!
8247 Offset alignment: MSD file write protected!
8248 Cyclic supervision failed!
8249 Axis motion check alarm!
8250 Spindle must be rotation axis !
8251 Lead for G331/G332 missing !
8252 Multiple or no linear axis programmed for G331/G332 !
8253 Speed value for G331/G332 and G96 missing !
8254 Value for thread starting point offset not valid!
8255 Reference point not in valid software limits!
8256 Spindle speed too low while executing G331/G332!

I: Control Alarms

Control Alarms

These alarm s can occur only with operating and programming the control functions or with running CNC programs.

1 RS232 parity error !
Cause: Data transmission error parity error, wrong RS 232 setting in external device
Remedy: Check data cables, set serial interface of the external device

2 RS232 transmission error !
Cause: Data transmission error character overflow Data transmission error invalid data frame
Remedy: Check data cables, set serial interface of the external device

10 Nxxxx Invalid G-code
Remedy: Program correction

11 ORDxx Feed wrong/missing
Cause: Attempt to start with feed = 0, also with G95/96, if S = 0 or M5
Remedy: Program correction

21 Nxxxx Circle: Wrong plane selected
Cause: The wrong plane (G17, 18, 19) is active for a circle
Remedy: Program correction

30 Nxxxx Invalid tool offset number
Cause: The lower 2 digits of the T number are to great
Remedy: Program correction

33 Nxxxx CRC can't be determined
Cause: Too much blocks without new position programmed, invalid contour element, programmed circle radius smaller than cutter radius, contour element to short.
Remedy: Program correction

34 Nxxxx Error on deactivating CRC
Remedy: Program correction

37 Nxxxx Plane change while CRC act.
Cause: Change of plane not permitted with active cutter radius compensation
Remedy: Program correction

41 Nxxxx Contour violation CRC
Cause: Invalid contour element, programmed circle radius smaller than cutter radius, contour element to short, contour violation with full circle.
Remedy: Program correction

51 Nxxxx Wrong chamfer/radius value
Cause: The contour elements between a chamfer / radius should be inserted are too short.
Remedy: Program correction

52 Nxxxx Invalid contour draft
Cause: From the programmed parameters no valid contour draft would result
Remedy: Program correction

53 Nxxxx Wrong parameter structure
Cause: From the programmed parameters no valid contour draft would result, wrong parameter programmed
Remedy: Program correction

56 Nxxxx Wrong angle value
Cause: With the programmed angle no intersection point would result
Remedy: Program correction

57 Nxxxx Error in contour draft
Cause: Invalid parameters programmed.
Remedy: Program correction

58 Nxxxx Contour draft not determinable
Cause: Too much blocks without new position programmed, program end while contour draft
Remedy: Program correction

60 Nxxxx Block number not found
Cause: Jump target not found
Remedy: Program correction

62 Nxxxx General cycle error
Cause: Call-up counter of subprogram call invalid, feed<=0, thread pitch missing/<=0, cutting depth missing/<=0/invalid, retraction height to small, block address P/Q missing, declaration pattern repetition missing/invalid, infeed for next cut missing/invalid, undercut at cycle ground <0, cycle end point missing/invalid, thread end point missing/invalid;
Remedy: Program correction

63 Nxxxx Wrong Cycle call
Cause: P/Q missing, wrong address
Remedy: Program correction

70 Insufficient memory
Cause: The PC has not enough memory
Remedy: Close all other Windows applications, remove resident programs from memory, restart the PC

71 Program not found
Cause: NC program not found
With program start no program was selected
Remedy: Correct call-up or create program, select program

73 File already exists !
Remedy: Select other file name.

77 Insufficient RAM for subroutine
Cause: Subprograms interlocked too deep
Remedy: Program correction

83 Nxxxx Circle not in active plane
Cause: Circle is not in active plane for CRC
Remedy: Program correction

142 Wrong simulation area
Cause: Wrong scale factor (e.g. 0) programmed
Remedy: Program correction

142 Invalid scale factor
Cause: No or an invalid simulation area was entered
Remedy: Enter correct simulation area

315 ORDxx Rotatory checking X
Cause: The step motor has fallen out of pace
Remedy: Reduce infeed and feed, check slides for smooth running, approach reference point

325 ORDxx Rotatory checking Y
see alarm 315

335 ORDxx Rotatory checking Z
see alarm 315

500 ORDxx Target point exceeds work.area
Cause: Target point, circle target point or circle out of working area limitation
Remedy: Program correction

501 ORDxx Target point exceeds SW limit
Cause: Target point, circle target point or circle out of working area limitation
Remedy: Program correction

510 ORDxx Software-limit switch X
Cause: Software limit switch in X exceeded (JOG)
Remedy: Traverse back manually

520 ORDxx Software-limit switch Y
see 510

530 ORDxx Software-limit switch Z
see 510

2501 ORDxx Synchronisation-error AC
Remedy: RESET, report to EMCO if reproducible

2502 ORDxx Synchronisation-error AC
see 2501

2503 ORDxx Synchronisation-error AC
see 2501

2504 ORDxx No memory for interpreter
Cause: Too less RAM memory, continueing the program is not possible
Remedy: Close all Windows application, close WinNC,remove resident programs from AUTOEXEC.BAT and CONFIG.SYS, restart the PC

2505 ORDxx No memory for interpreter
see 2504

2506 ORDxx Too less RAM
see 2504

2507 ORDxx Reference point not active
Remedy: Approach reference point

2508 ORDxx Internal error NC core
Remedy: RESET, report to EMCO if reproducible

2520 ORDxx RS485 device absent
Cause: With program start a RS485 device did not report, while program run a device got defective
AC Axis controller
SPS PLC
MT control keyboard
Remedy: Switch on RS485 device (machine, control keyboard), check cables and plugs, check terminator plug, report to EMCO if reproductible

2521 ORDxx RS485 communication error
Remedy: PC restart, report to EMCO if reproducible

2522 ORDxx RS485 communication error
Remedy: PC restart, report to EMCO if reproducible

2523 ORDxx INIT error on RS485 PC-board
See "Software Installation" , Mistakes with installation of the software

2524 ORDxx Gen.-Failure RS485 PC-board
Remedy: PC restart, report to EMCO if reproducible

2525 ORDxx Transmit error RS485
Cause: Transmission error by poor plug connections, missing terminator, external sources of electromagnetic interference
Remedy: Check the error sources above

2526 ORDxx Transmit error RS485
see 2525

2527 ORDxx Internal error AC
Remedy: Switch machine off/on, report to EMCO if reproducible

2528 ORDxx Operating system error PLC
Remedy: Switch machine off/on, report to EMCO if reproducible

2529 ORDxx External keyboard error
Remedy: The external keyboard always must be switched on after the PC. Restart the software, report to EMCO if reproducible

2540 ORDxx Error saving setting-data
Cause: Hard disk full, wrong path setting, no writing access
Remedy: Check hard disk space, check writing access, reinstallation of the software if reproducible

2545 ORDxx Drive / Device not ready
Remedy: Insert disk, lock drive, check disk drive, ...

2546 ORDxx Checksum error machine-data
Remedy: Restart, report to EMCO if reproductible

2550 ORDxx PLC simulation error
Remedy: Restart, report to EMCO if reproducible

2551 ORDxx PLC simulation error
Remedy: Restart, report to EMCO if reproducible

2562 Read error on CNC program
Cause: Defective program file, DOS read error (disk, hard disk)
Remedy: Solve problem on DOS level, eventually reinstallation of the software

2614 ORDxx Internal error MSD
Remedy: Report to EMCO if reproducible

2650 ORDxx Internal error cycle call up
Cause: Invalid cycle call when a cycle was called with a G command
Remedy: Program correction

2849 Internal error CRC
Remedy: Report to EMCO if reproducible

2904 Helix Z value too large
Cause: The pitch of the helix must not be larger than 45°
Remedy: Program correction