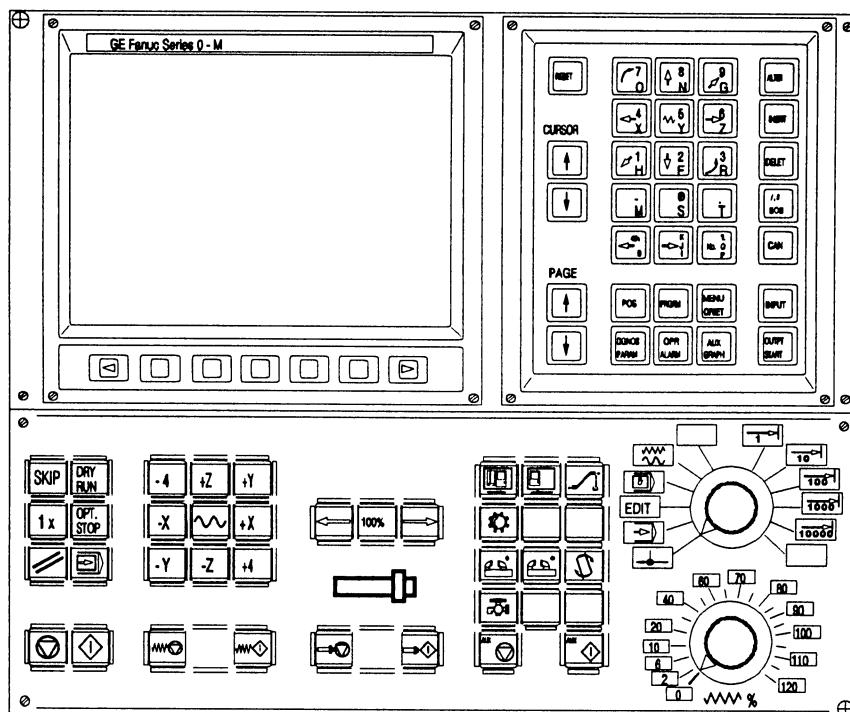


Software Description

EMCO WinNC

GE FANUC Series 0-MC

Software Version from 3.34



Edition 1996

Ref. No. EN 1801

**Software Description
EMCO WinNC
GE FANUC Series 0-MC
F96-11 EN 1801**



EMCO MAIER Gesellschaft m.b.H. • P.O. Box 131 • A-5400 Hallein/Austria • Tel. (06245) 891 • Fax (06245) 86965

Preface

WinNC GE FANUC Series 0-MC is a part of the EMCO educational concept on PC basis.

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

With the machine version of the EMCO WinNC the milling machines EMCO PC MILL 50 and EMCO PC MILL 100 can be controlled directly.

By using a digitizer or the control keyboard, operating the software will be much more easier and by the similarity to the original control didactical more effective.

Additional to this software description the following educational material is in preparation:

- Instruction manual
- Trainers guide manual
- Overhead slides

Contents

A: Key Description 5

Control Keyboard, Digitizer Overlay	5
Key Functions	5
Data Input Keys	6
Function Keys	6
Machine Control Keys	7
PC Keyboard	8

B: Basics 9

Reference Points of the EMCO Milling Machines	9
Zero Offset	10
The Coordinate System	10
Input of the Zero Offset	11
Tool Data Measuring	12
Tool Data Measuring by Scraping	13

C: Operating Sequences 15

Survey Operating Modes	15
Approach the Reference Point	16
Setting of Language and Workpiece Directory	16
Program Input	17
Call Up a Program	17
Input of a block	17
Search a Word	17
Insert a Word	17
Alter a Word	17
Delete a Word	17
Insert a Block	17
Delete a Block	17
Delete a Program	18
Delete All Programs	18
Data Input - Output	18
Adjusting the Serial Interface	18
Program Output	19
Program Input	19
Tool Offset Output	19
Tool Offset Input	19
Print Programs	19
Program Run	20
Start of a Part Program	20
Displays while Program Run	20
Block Search	20
Program Influence	20
Program interruption	20
Display of the Software Versions	20
Part Counter and Piece Time	21
Graphic Simulation	22

D: Programming 23

Program Structure	23
Used Addresses	23
Survey of M Commands	23
Description of G Commands	25
G00 Positioning (Rapid Traverse)	25
G01 Linear Interpolation	25
G02 Circular Interpolation Clockwise	27
G03 Circular Interpolation Counterclockwise	27
Helix Interpolation	27
G04 Dwell	28
G09 Exact Stop	28
G15 End Polar Coordinate Interpolation	28
G16 Begin Polar Coordinate Interpolation	28
G17-G19 Plane Selection	29
G20 Measuring in Inches	29
G21 Measuring in Millimeter	29
G28 Approach Reference Point	30
G33 Thread Cutting	30
Cutter Radius Compensation	31
G40 Cancel Cutter Radius Compensation	31
G41 Cutter Radius Compensation left	31
G42 Cutter Radius Compensation right	31
G43 Tool Length Compensation positive	33
G44 Tool Length Compensation negative	33
G49 Cancel Tool Length Compensation	33
G50 Cancel Scale Factor, Mirror	33
G51 Scale Factor, Mirror	33
G52 Local Coordinate System	35
G53 Machine Coordinate System	35
G54 - G59 Zero Offsets 1 - 6	35
G61 Exact Stop Mode	36
G62 Automatic Corner Override	36
G64 Cutting mode	36
Drilling Cycles G73 - G89	37
Systematic G98/G99	37
Number of repetitions	37
G73 Chip Break Drilling Cycle	38
G74 Left Tapping Cycle	38
G76 Fine Drilling Cycle	39
G80 Cancel Drilling Cycles	39
G81 Drilling Cycle	39
G82 Drilling Cycle with Dwell	40
G83 Withdrawal Drilling Cycle	40

G84	Tapping Cycle	41
G85	Reaming Cycle	42
G86	Drilling Cycle with Spindle Stop	42
G87	Back Pocket Drilling Cycle	43
G88	Drilling Cycle with Program Stop.....	43
G89	Reaming Cycle with Dwell.....	44
G90	Absolute Programming.....	44
G91	Incremental Programming	44
G92	Coordinate System Setting	44
G94	Feed per Minute	44
G95	Feed per Revolution	44
G97	Revolutions per Minute	44
G98	Retraction to the Start Plane	44
G98	Retraction to the Withdrawal Plane.....	44
	Description of M Commands.....	45
M00	Programmed Stop	45
M01	Programmed Stop, Conditional	45
M02	Main Program End	45
M03	Milling Spindle ON Clockwise	45
M04	Milling Spindle ON Counterclockwise	45
M05	Milling Spindle OFF	45
M06	Tool Change.....	45
M08	Coolant ON	45
M09	Coolant OFF	45
M27	Swivel Dividing Head	45
M30	Main Program End	45
M71	Puff blowing ON	45
M72	Puff blowing OFF	45
M98	Subprogram Call	46
M99	Subprogram End, Jump Instruction.....	46

E:	Alarms and Messages	47
	Startup Alarms	47
	Control Alarms	48
	Machine Alarms	51

F:	Accessory Functions	55
	Activate Accessory Functions	55
	Robotic Interface PC MILL 50	56
	Robotic Interface PC MILL 100.....	57
	Automatic Vice	58
	Door Automatic	58
	Puff Blowing Device	58
	DNC Interface	58
	Activate Tool Turret	58

WinConfig WinConfig 1

General	WinConfig 1	
1.	Start WinConfig	WinConfig 1
2.	Select Program path of WinNC	WinConfig 2
3.	Basic Settings for WinConfig	WinConfig 2
4.	Change Ini Data of WinNC	WinConfig 3
4.1	Alter Directories	WinConfig 4
4.2	System of Measurement	WinConfig 4
4.3	Interface Selection	WinConfig 5
4.4	Test Options - Error Recording	WinConfig 6
4.5	Language Selection	WinConfig 7
4.6	Screen Settings	WinConfig 7
4.7	WinCTS Settings	WinConfig 8
5.	Change Msd Data of WinNC	WinConfig 9
5.1	RS485 Device List.....	WinConfig 10
5.2	PLC Machine Data	WinConfig 11
5.3	Activate Accessories	WinConfig 12
6.	Store Changes	WinConfig 12

Software Installation I1

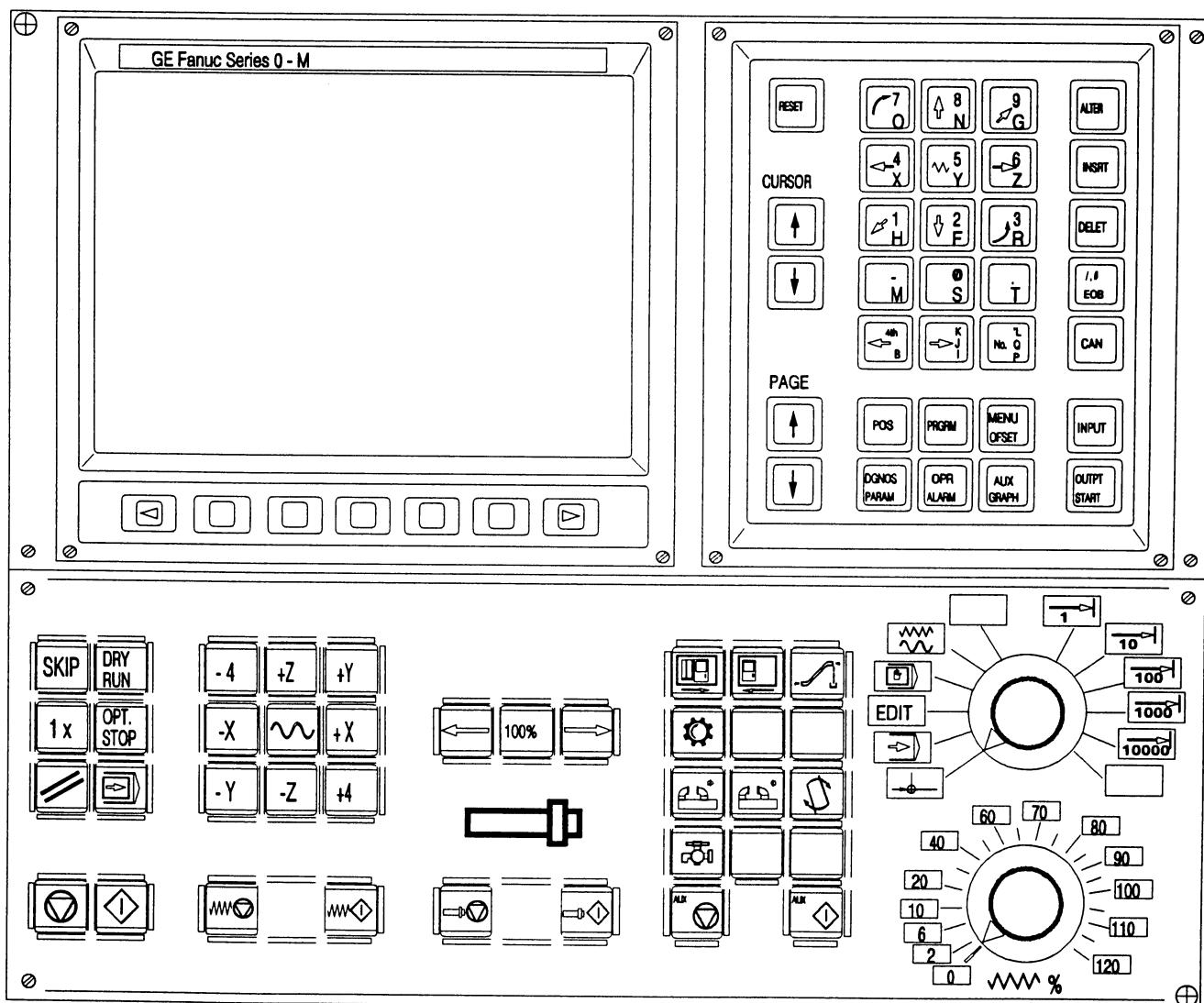
General	I1
Software Update	I1
Sequence of Installation	I2
Notes for Network Installation	I9
Starting WinNC	I10
Closing WinNC	I10
Mistakes with Installation of the Software	I11

External Input Devices E1

Digitizer Tablet	E1
EMCO Control Keyboard	E2

A: Key Description

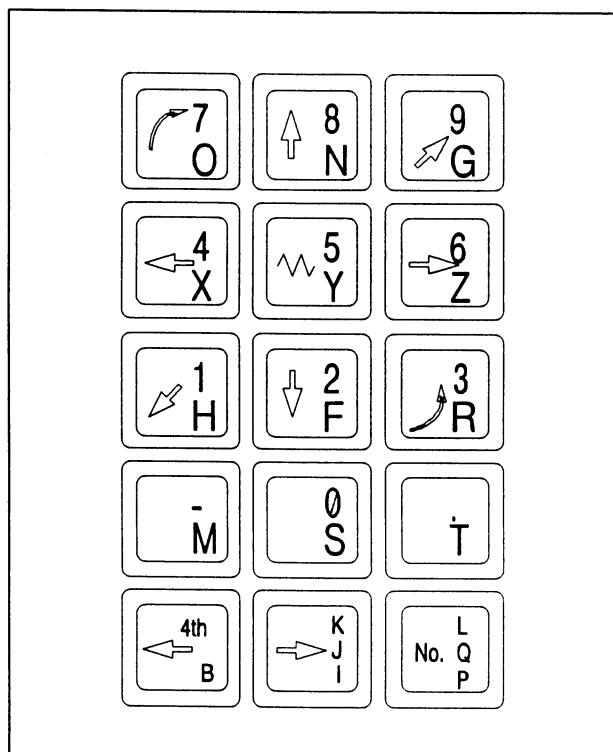
Control Keyboard, Digitizer Overlay



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, End Of Block
 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, va-
 riables
 DGNOS PARAM .. Setting and display of parameter
 and display of diagnostic data
 OPR ALARM..... Alarm and message display
 AUX GRAPH..... Graphic display

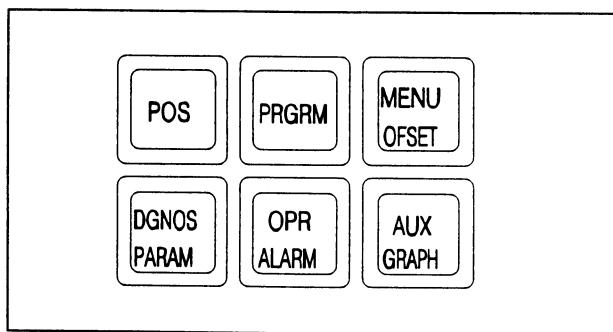


Data input keys

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

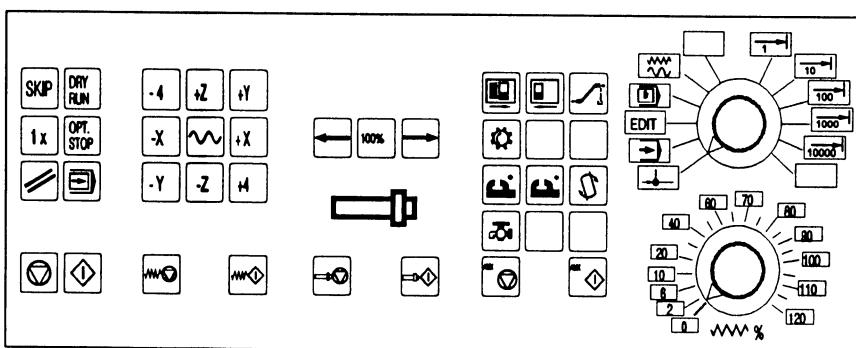
Function Keys

Note for Function Keys

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

Machine Control Keys

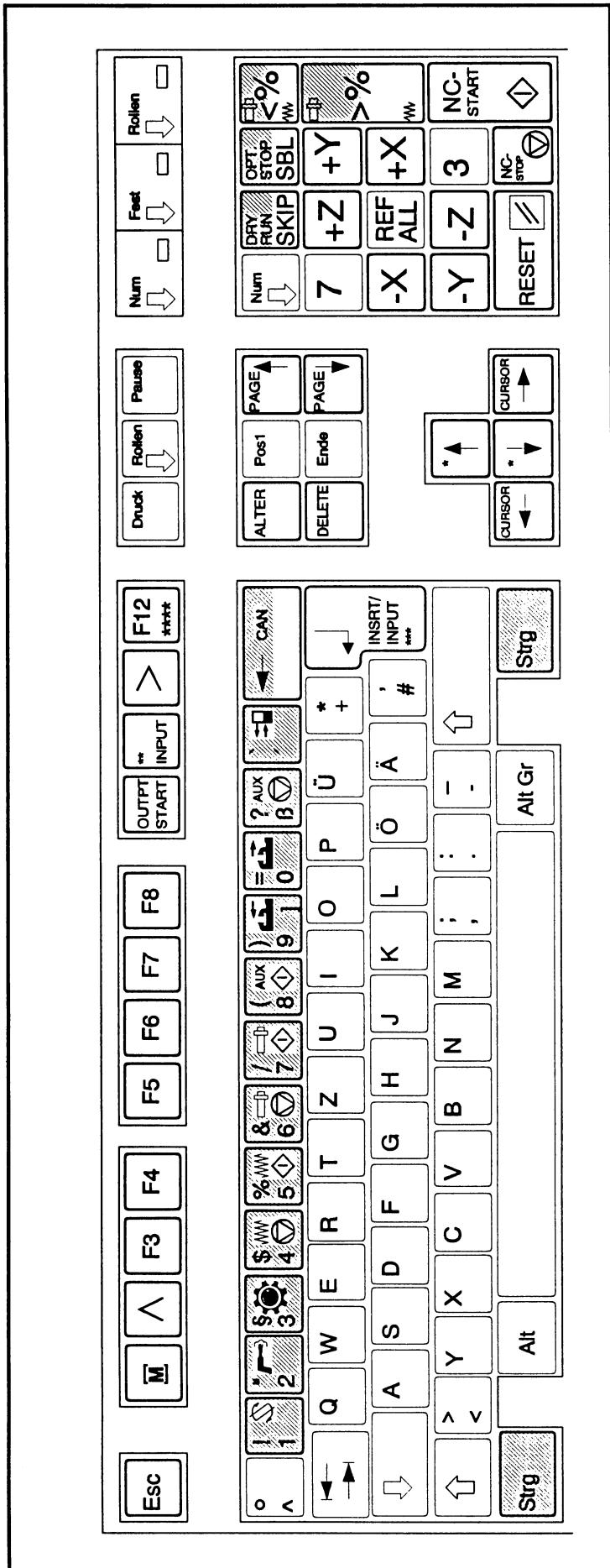
The machine control keys are in the lower block of the control keyboard resp. the digitizer overlay. Depending on the used machine and the used accessories not all functions may be active.



Machine control key block

	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
	feed stop / feed start
	spindle speed decrease/100%/increase
	spindle stop / spindle start; spindle start in the modes JOG and INC1...INC1000: clockwise: press key short, Counterclockwise: press key for at least 1 sec.
	open / close door
	swivel dividing head
	open / close clamping device
	swivel toolholder
	coolant / puff blowing on / off
	AUX OFF / AUX ON (auxilliary drives off / on)
	feed / rapid traverse override switch

PC Keyboard



Bold lined keys are special functions for control and machine, to activate hatched key functions press the Ctrl key at the same time.

Some alarms will be acknowledged with the key ESC.

By pressing the key F1 the modes (AUTO, EDIT, MDI,...) will be displayed in the softkey line.

The meaning of the key combination ctrl 2 depends on the machine:
 EMCO PC TURN 50: Puff blowing ON/OFF
 EMCO PC TURN 120: coolant ON/OFF

The assignment of the accessory functions is described in the chapter "Accessory Functions".

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

- * The keys **↑** and **↓** are active only in the editor, otherwise same functions as **→** and **←**.
- ** only for data input, zero offsets, tool offsets...

**** With F12 the function keys POS, PRGRM, MENUSET, DGNOSPARAM, OPRALARM and AUX GRAPH will be displayed in the softkey line.

B: Basics

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".

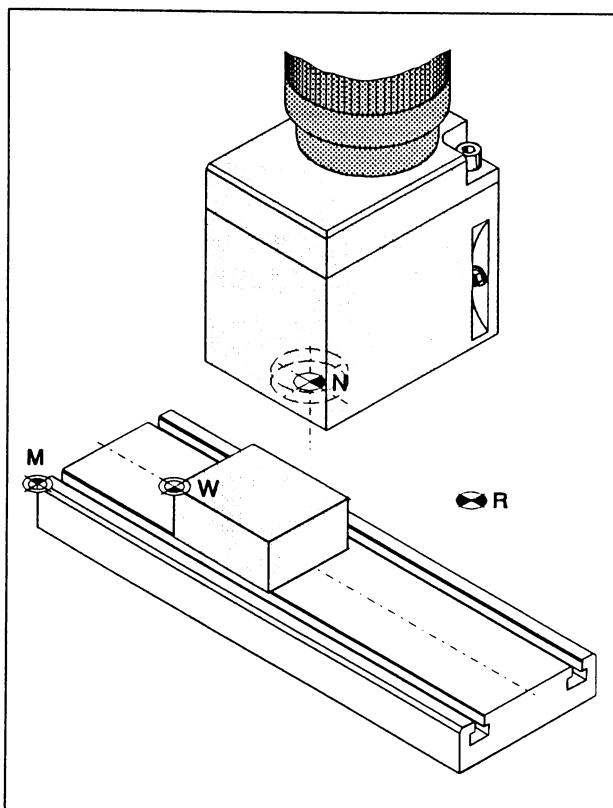
Required after every power failure.

N = Tool mount reference point

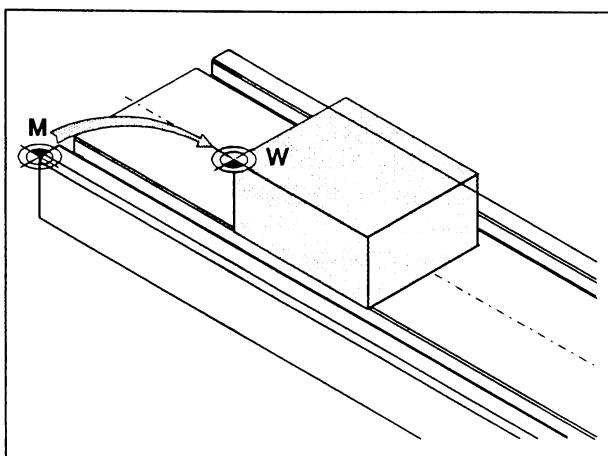
Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

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.



Reference points in the working area



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

Zero Offset

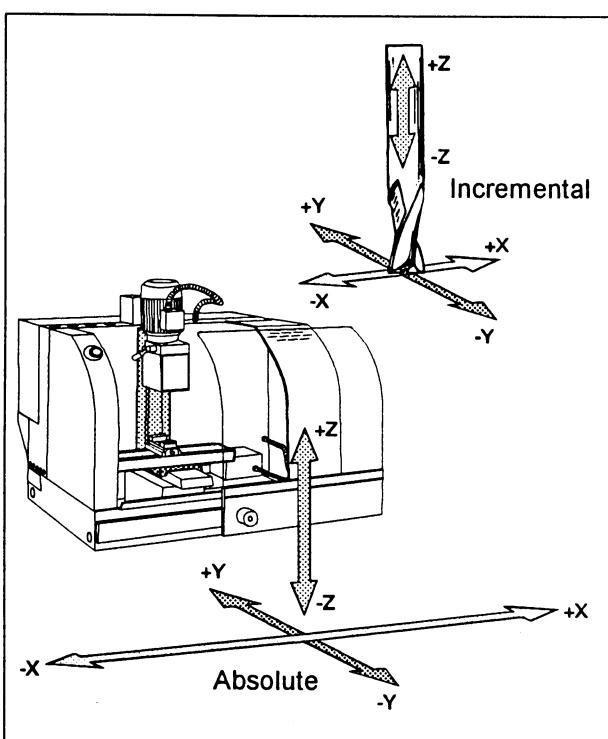
With EMCO milling machines the machine zero "M" lies at the left front edge of the machine table. This position is unsuitable as a starting point for dimensioning. With the so-called zero offset the coordinate system can be moved to a suitable point in the working area of the machine.

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.



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

The Coordinate System

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

F: 100% S: 100%

WORK COORDINATES. O0000 N0000

NO.	DATA	NO.	DATA
00	X 0.000	02	X 0.000
	Y 0.000		Y 0.000
	Z 0.000		Z 0.000

NO.	DATA	NO.	DATA
01	X 0.000	03	X 0.000
	Y 0.000		Y 0.000
	Z 0.000		Z 0.000

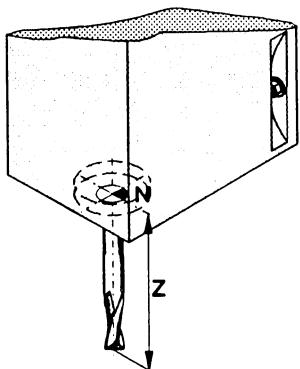
ADRS.

OFFSET		WORK	JOG	
--------	--	------	-----	--

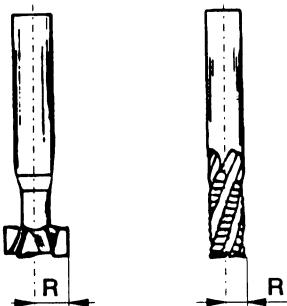
Input pattern for zero offsets

- Press the key
- 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 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 and .
- Enter the desired offset (e.g.: X+30.5) and press the key .
- 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 these 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.

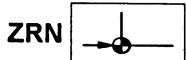
Tool Data Measuring by Scraping

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  and the softkey REL to show the relative position at the screen.
- Press the key  - the Z display flashes
- Reset Z value with  to 0
- Change to MDI mode
- Switch on the spindle (e.g. S1000 M3 NC-Start)
- Change to JOG mode
- Press the key 
- Clamp tool to be measured and 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 in the offset register (OFFSET) with the keys  
- Key in the displayed Z value as parameter in the offset register (OFFSET) and take it over with the  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.

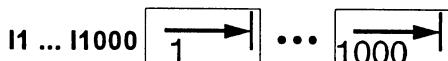


MDI In the MDI mode you can switch on the spindle and swivel the tool holder.

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.



In this operation mode the slides can be traversed for the desired increment (1...10000 in $\mu\text{m}/10^{-4}$ inch) by means of the JOG keys

-X **+X** **-Z** **+Z**.

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

Approach the Reference Point

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

- Change into ZRN (REF) 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**. The settings (SETTING 1) will be displayed.
- 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 paths. 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

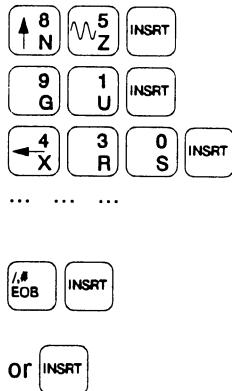
Part programs and subprograms can be entered in the EDIT mode.

Call Up a Program

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

Input of a block

Example:



Block number (not necessary)

1. word

2. word

EOB - End of block (on PC keyboard also )

Note:

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

Search a Word

Enter the address of the word to be searched

(e.g.: X) and press the key .

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 .

Alter a Word

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

Delete a Word

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

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: NO) and press the key .

Delete a Program

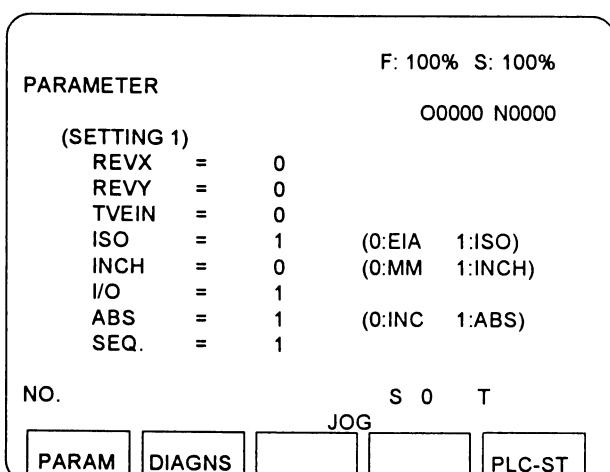
EDIT mode

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

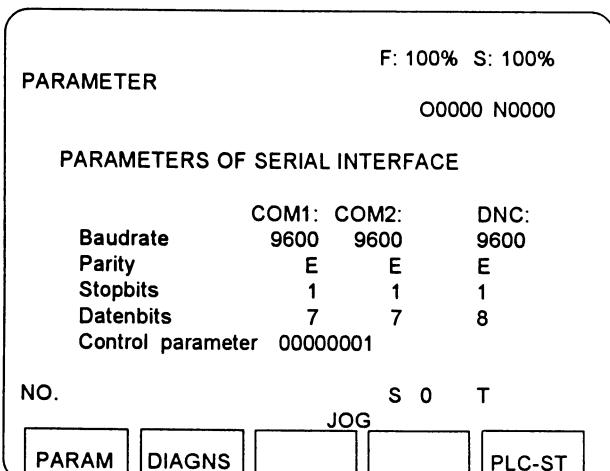
Delete All Programs

EDIT mode

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



Selection of the input/output interface



Adjusting the serial interface

NOTE

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

Data Input - Output

- Press the key **DGNOS PARAM**.
The screen shows (SETTING 3).
- 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

- 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)

Program Output

- EDIT mode
- Enter the receiver in (SETTING 1) below "I/O".
- Press the key .
- Enter the program to be sent (e.g. O22) and press the key .
- When you enter the program number O-9999 all programs will be put out.
- When you press the key  without indicating a program number, the program numbers in the area START - END in SETTING 3 will be put out.

Program Input

- EDIT mode
- Enter the receiver in (SETTING 1) below "I/O".
- Press the key .
- 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 .

Tool Offset Output

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

Tool Offset Input

- EDIT mode
- Enter the receiver in (SETTING 1) below "I/O".
- Press the keys  and .

Print Programs

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

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 .
- Enter the desired part program number (e.g.: O79).
- Press the key .
- 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 . The positions will be shown enlarged at the screen.

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.

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  and  on that block, with which machining should start.
- Change to AUTOMATIC mode.
- Start the program with the key .

Display of the Software Versions

- Press the key .
- Select softkey DGNOS

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

F: 100% S: 100%

ACTUAL POSITION (ABSOLUTE)			
O0001	N0000		
X	0.000		
Y	0.000		
Z	0.000		
RUN TIME	0H 0M	PART COUNT	0
ACT. F	0.00MM/R	CYCLE TIME	0H 0M 0S
AUTO			
ABS	REL	ALL	

Display of part counter and piece time

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 **1** (P) and **CAN** after.

RUN TIME Reset

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

F: 100% S: 100%

PARAMETER			
00000 N0000			
(SETTING 2)			
PWE	= 1 (0: DISABLE 1:ENABLE)		
TAPEF	= 0		
PART TOTAL	= 29		
PART REQUIRED	= 10		
PART COUNT	= 0		
RUN TIME	0H 0M	CYCLE TIME	0H 0M 0S
NR.	-	S	0
JOG			
PARAM	DGNOS		PLC-ST

Presetting the piece counter

Preset of the Part Counter

The part counter can be preset in (SETTING 2). Therefore move the cursor 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).

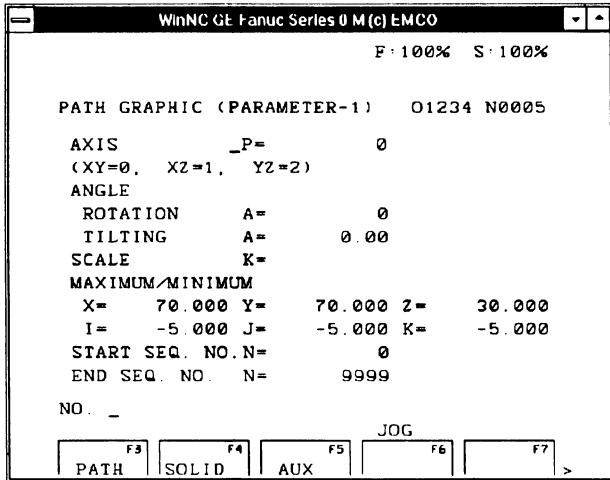
PART REQUIRED:

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.



Input pattern for graphic simulation

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.

Inputs:

AXIS P

Enter the simulation plane here.

0 XY plane

1 XZ plane

2 YZ plane

MAXIMUM/MINIMUM

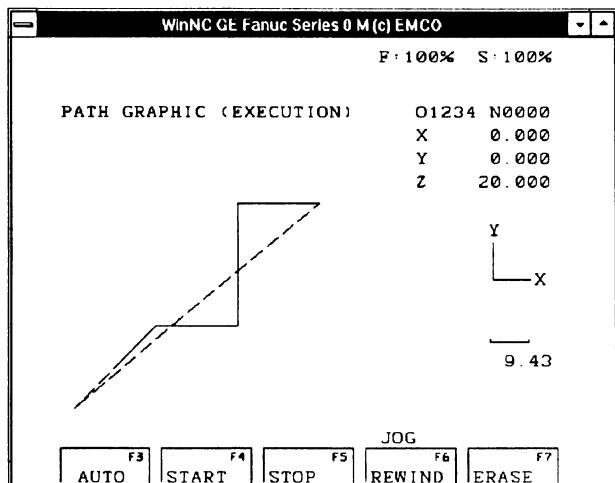
Enter here the right upper (X, Y, Z) and the left lower (I, J, K) edge of the simulation areaein.

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)

Survey of M Commands

M00	Programmed Stop
M01	Programmed Stop, Conditional
M02	Program End
M03	Main Spindle ON Clockwise
M04	Main Spindle ON Counterclockwise
M05 ¹	Main Spindle OFF
M06	Tool Change
M08	Coolant ON
M09 ¹	Coolant OFF
M19	Oriented Spindle Stop
M25	Release Clamping Device
M26	Close Clamping Device
M30	Program End
M71	Puff blowing ON
M72 ¹	Puff blowing OFF
M98	Subprogram Call
M99	Subprogram End

¹ Initial status

² Blockwise effective

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)
; <td>block end</td>	block end

Survey of G Commands

G00¹ Positioning (Rapid Traverse)
 G01 Linear Interpolation
 G02 Circular Interpolation Clockwise
 G03 Circular Interpolation Counterclockwise
 G04² Dwell
 G09² Exact Stop
 G15¹ End Polar Coordinate Interpolation
 G16 Begin Polar Coordinate Interpolation
 G17¹ Plane Selection XY
 G18 Plane Selection ZX
 G19 Plane Selection YZ
 G20 Measuring in Inches
 G21 Measuring in Millimeter
 G28² Approach Reference Point
 G33 Thread Cutting
 G40¹ Cancel Cutter Radius Compensation
 G41 Cutter Radius Compensation left
 G42 Cutter Radius Compensation right
 G43 Tool Length Compensation positive
 G44 Tool Length Compensation negative
 G49¹ Cancel Tool Length Compensation
 G50¹ Cancel Scale Factor
 G51 Scale Factor
 G52² Local Coordinate System
 G53² Machine Coordinate System
 G54¹ 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
 G64¹ Cutting mode
 G73 Chip Break Drilling Cycle
 G74 Left Tapping Cycle
 G76 Fine Drilling Cycle
 G80¹ 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
 G90¹ Absolute Programming
 G91 Incremental Programming
 G92² Coordinate System Setting
 G94¹ Feed per Minute
 G95 Feed per Revolution
 G97¹ Revolutions per Minute
 G98¹ Retraction to Starting Plane (Drilling Cycles)
 G99 Retraction to Withdrawal Plane

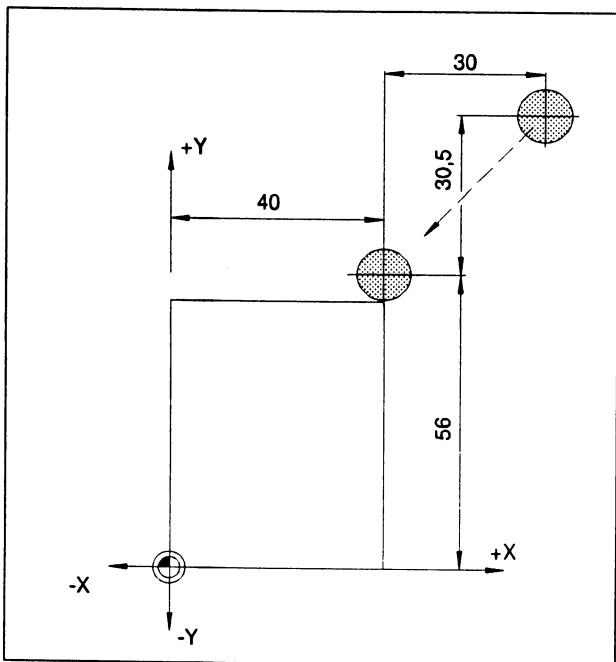
Group Division of G Commands		
Group	Command	Function
0	G04	Dwell
	G09	Exact Stop
	G28	Approach Reference Point
	G52	Local Coordinate System
	G53	Machine Coordinate System
	G92	Coordinate System Setting
1	G00	Positioning (Rapid Traverse)
	G01	Linear Interpolation
	G02	Circular Interpolation Clockwise
	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
10	G87	Back Pocket Drilling Cycle
	G88	Drilling Cycle with Program Stop
11	G89	Reaming Cycle with Dwell
	G98	Retraction to Starting Plane
12	G99	Retraction to Withdrawal Plane
	G50	Cancel Scale Factor
13	G51	Scale Factor
	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
17	G15	End Polar Coordinate Interpolation
	G16	Begin Polar Coordinate Interpolation

¹ Initial status

² Blockwise effective

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

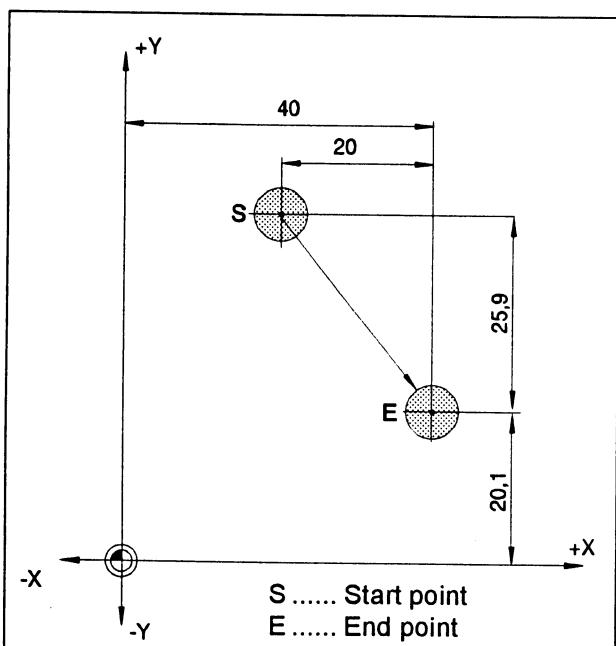
Example

absolute G90

N50 G00 X40 Y56

incremental G91

N50 G00 X-30 Y-30.5



Absolute and incremental measures

G01 Linear Interpolation

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

.....

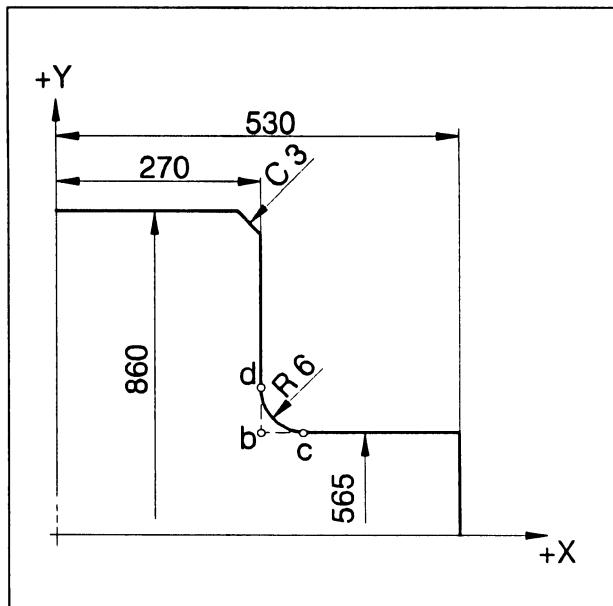
N20 G01 X20 Y-25.9

Chamfers and Radii

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.

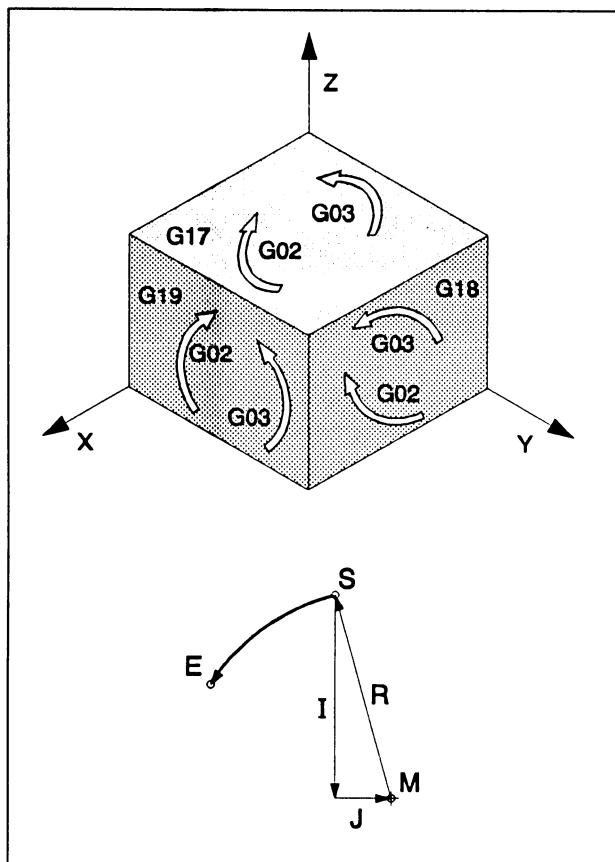
With single block mode the tool starts first at point c and then at point d.

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

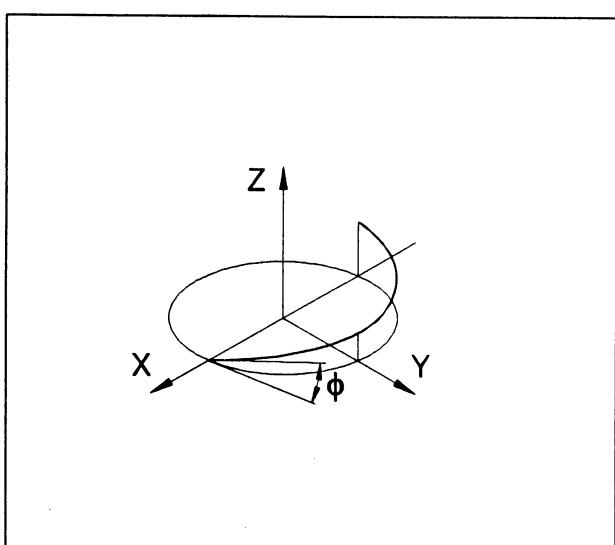
The tool will be traversed along the defined arc with the programmed feed F.

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 curve

Helix Interpolation

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 transitions, 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 - transitions, cleaning drilling ground, exact stop

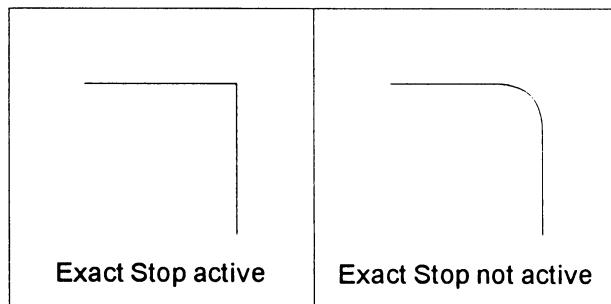
Notes

- 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 transitions will result.

G09 is effective blockwise.

G15 End Polar Coordinate Interpolation

G16 Begin Polar Coordinate Interpolation

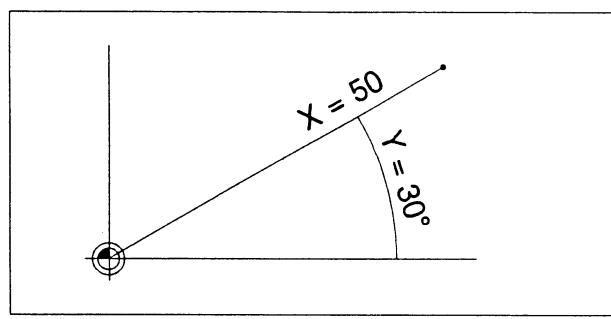
Format

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.



A point determined by polar coordinates

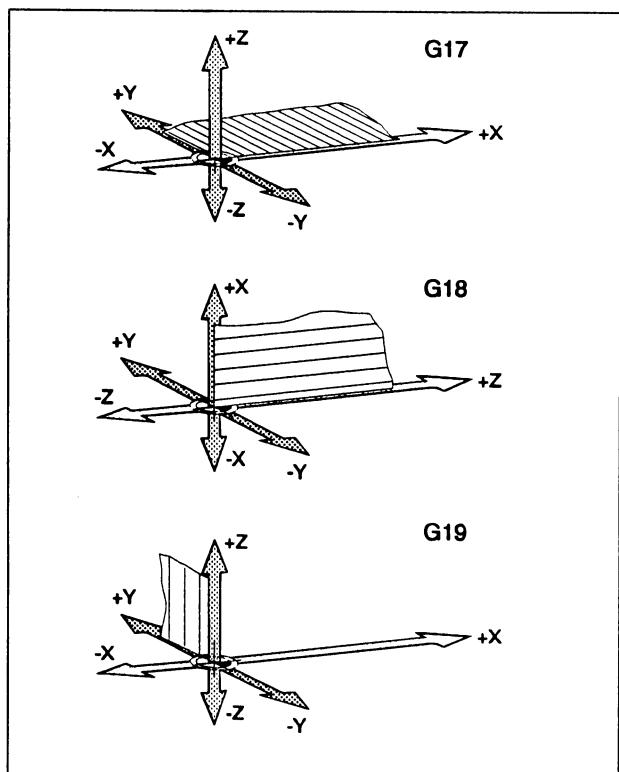
Example

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 paths [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...

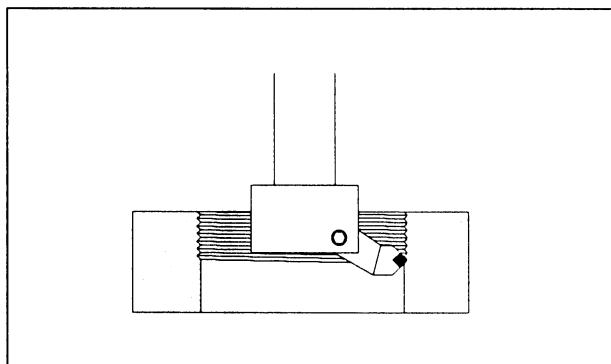
F Thread pitch [mm]

Z Thread depth

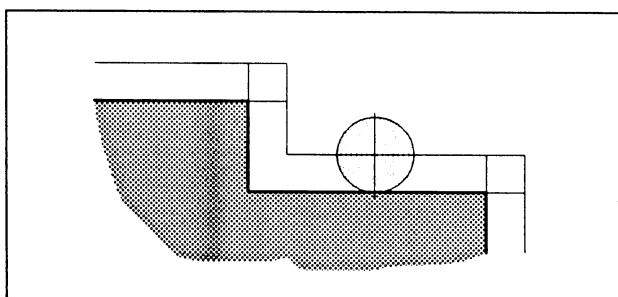
With a fitting tool (boring or facing head) threads can be cut.

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.

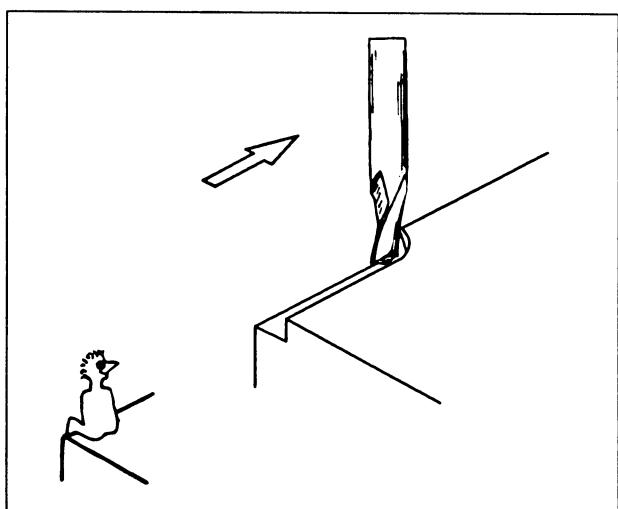
G40 Cancel Cutter Radius Compensation

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 of G41 cutter radius compensation left*

G41 Cutter Radius Compensation left

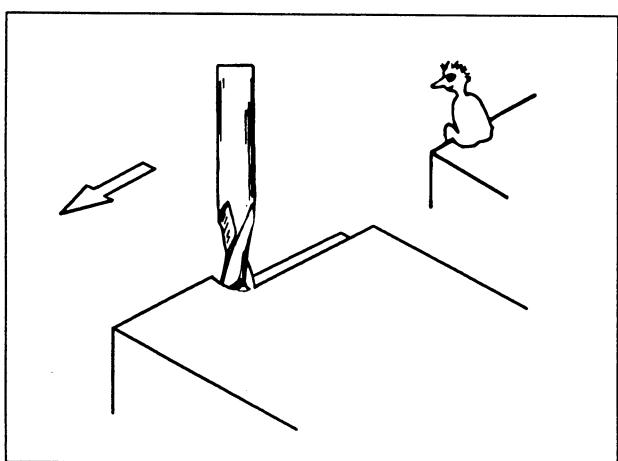
If the tool is (viewed in feed direction) at the **left** side of the contour to be worked, G41 has to be programmed.

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

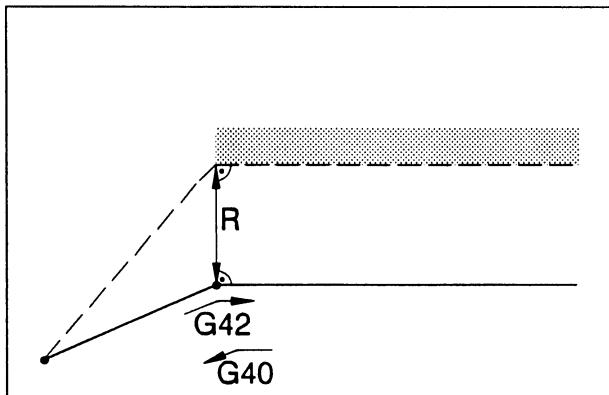
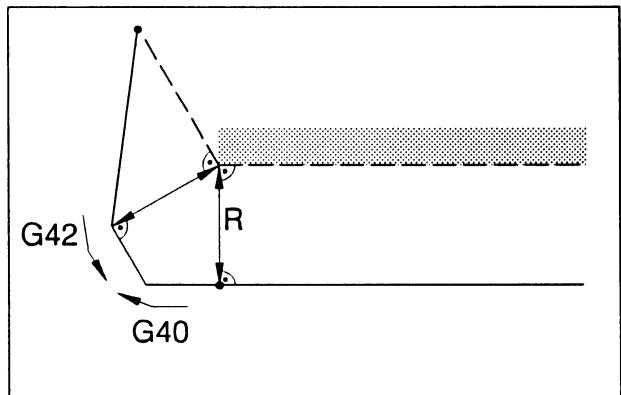
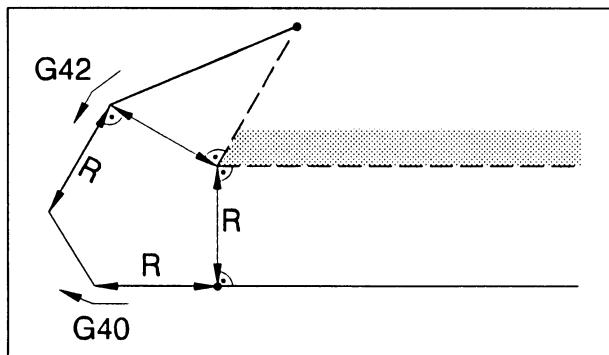
- Direct change between G41 and G42 is not allowed
 - previous cancellation with G40.
- Selection in combination with G00 or G01 necessary
- Programming an H parameter is necessary unconditionally, the H parameter is effective modally.

*Definition of G42 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!

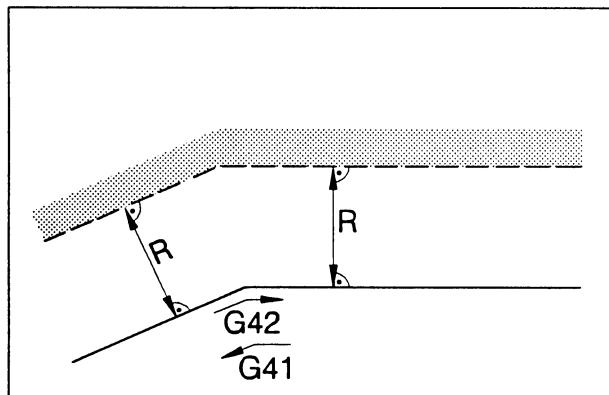
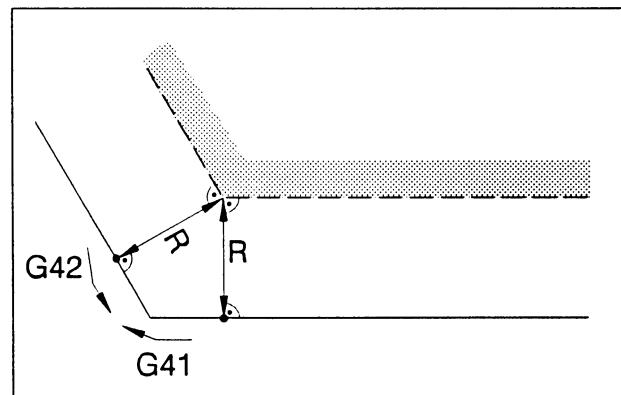
Tool paths with selection / cancellation of the cutter radius compensation*Frontal approach or leaving of an edge point**Approach or leaving an edge point at side behind**Approach or leaving an edge point behind*

— — — 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.

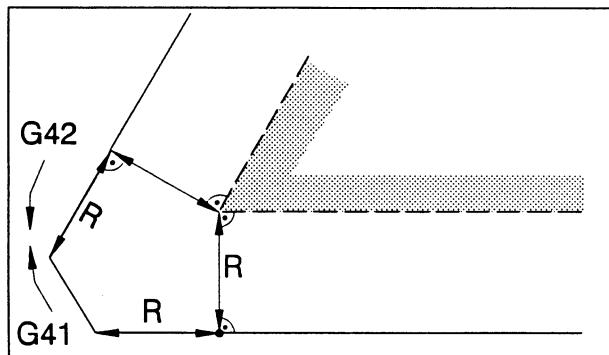
If contour elements are smaller than the tool radius R, contour violations could happen. The software computes three blocks forward to recognize this contour violations and interrupt the program with an alarm.

Tool paths with program run with active cutter radius compensation*Tool path at an inner edge**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.

*Tool path at an outer edge < 90°*

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.

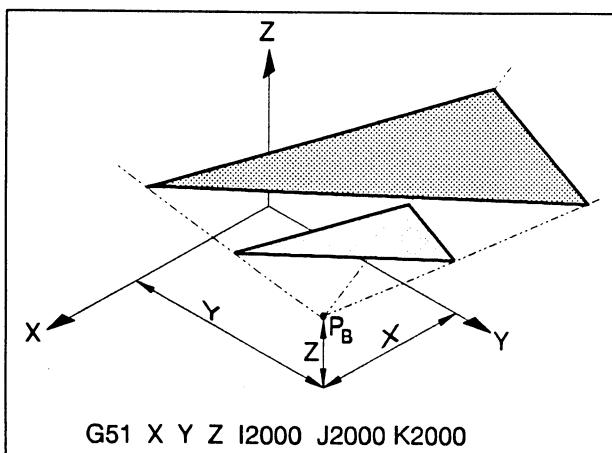
Example:

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:**

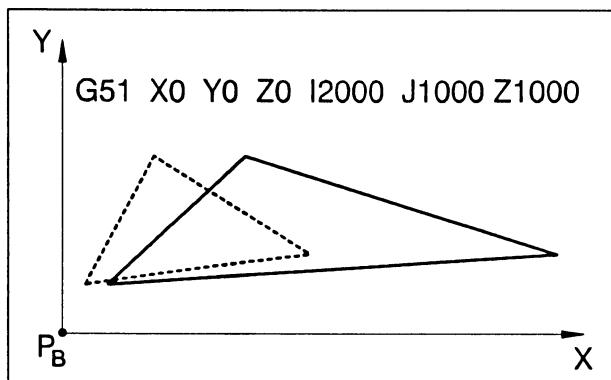
N... G50

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.

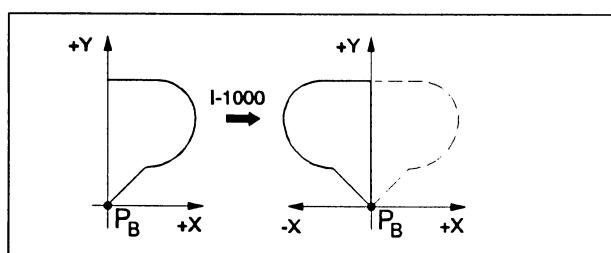
With I, J and K for every axis a scale factor (in 1/1000) can be defined.

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

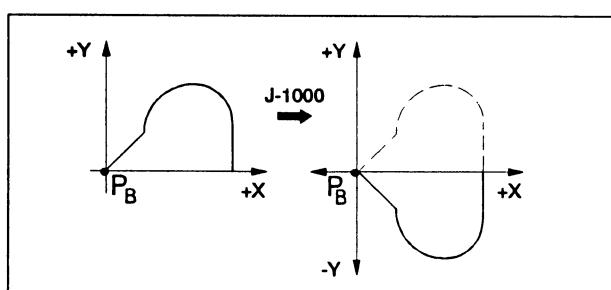
If different scale factors will be defined for the axes, the contour will be distorted.
Circular movements must not be distorted, otherwise alarm.

Mirroring a Contour

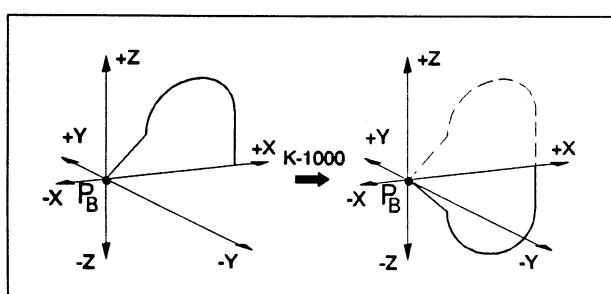
By programming a negative scale a contour will be mirrored around the base point P_B .



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 helded, until another shift will be activated.

G53 Machine Coordinate System

Format:

N... G53

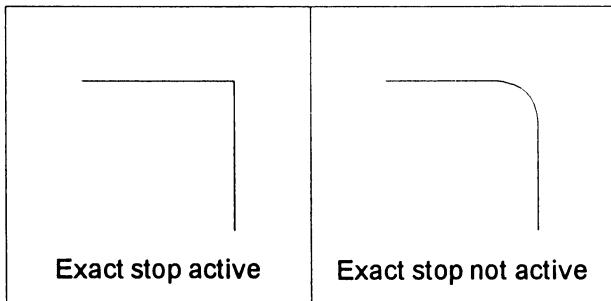
The machine zero point is determined by the machine manufacturer (EMCO milling machines: at the left front machine table corner).

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.

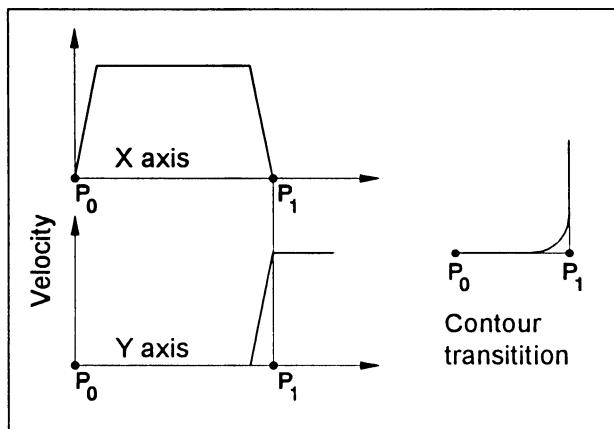


G61 Exact Stop Mode

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 transitions will result.
G61 is active, until it will be deselected with G62 or G64.



Speed reaction of the slides with G62 and G64

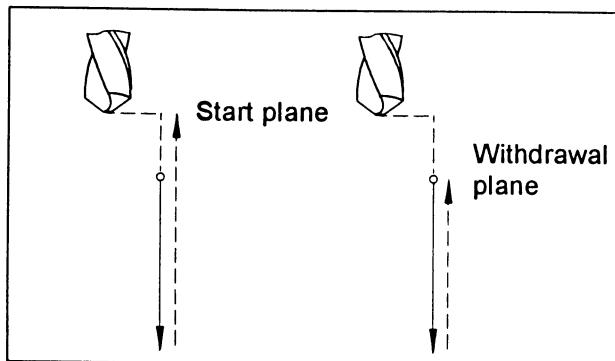
G62 Automatic Corner Override G64 Cutting mode

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 transitions. The contour transition is not exactly sharp-edged (parabola, hyperbola).
The size of the contour transitions is normally within the tolerance of the drawings.

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

If 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 computation of the R parameter is different with incremental and absolute programming:

Absolute programming (G90):

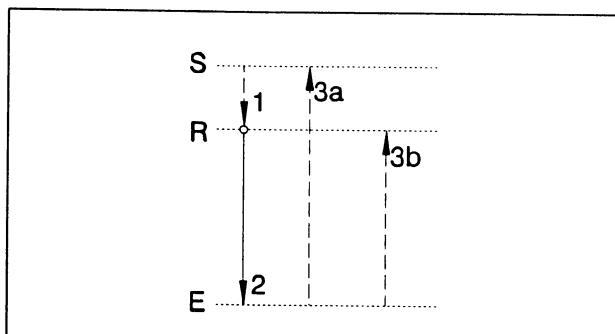
R defines the height of the withdrawal plane over the actual workpiece zero point.

Incremental programming (G90):

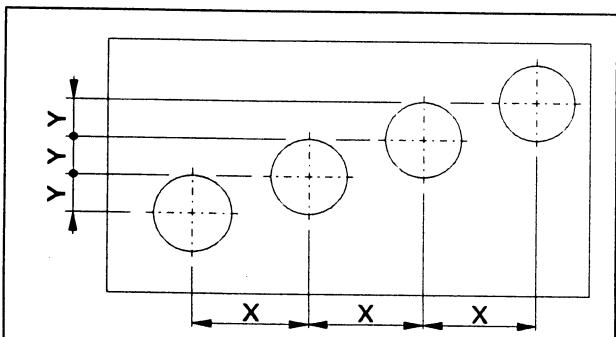
R defines the height of the withdrawal plane related to the last Z position (start position of the drilling cycle). With a negative value for R the withdrawal plane will be below the start position, with a positive value the withdrawal plane will be over the start position

Sequence of movements

- 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 depth (E).
- 3: The withdrawal occurs a: with G98 to the start plane (S) and b: with G99 to the withdrawal plane.



Sequence of movements G98, G99



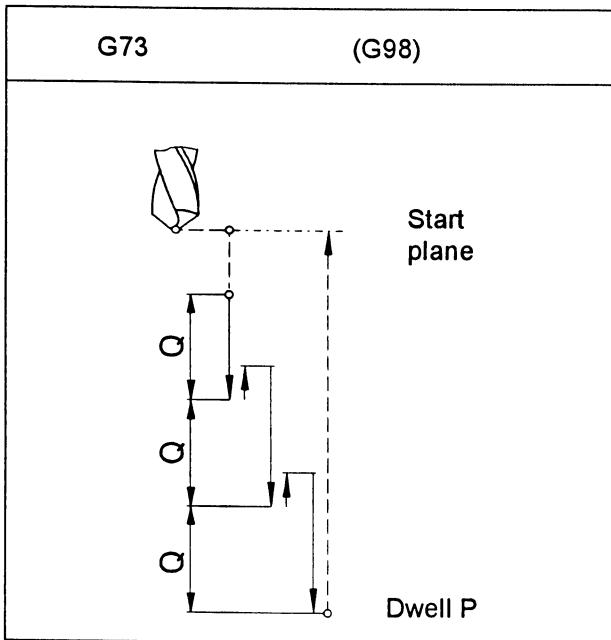
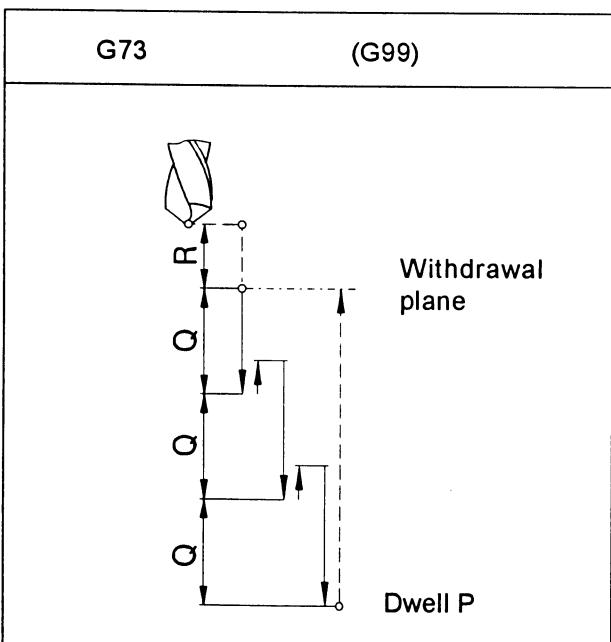
Cycle repetition for a row of holes

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

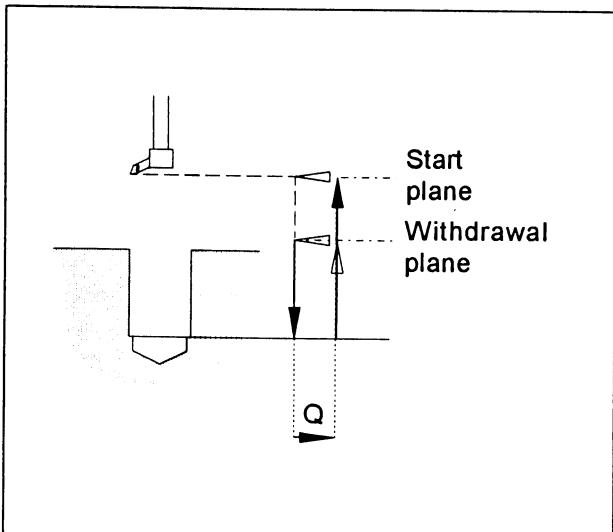
G74 Left Tapping Cycle

Only for PC Mill 100.

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.

Application:

Short drillings, material with good cutting properties

G98(G99)... Retraction to start plane (withdrawal plane)

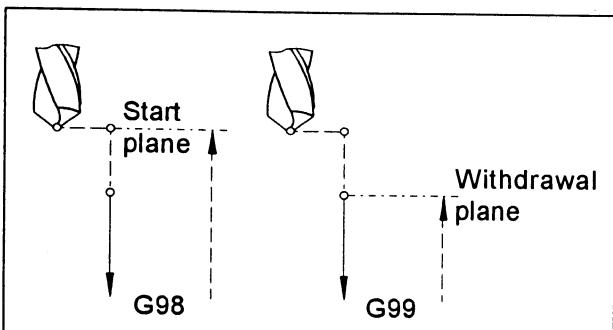
X, Y Hole position

Z Absolute (incremental) drilling depth

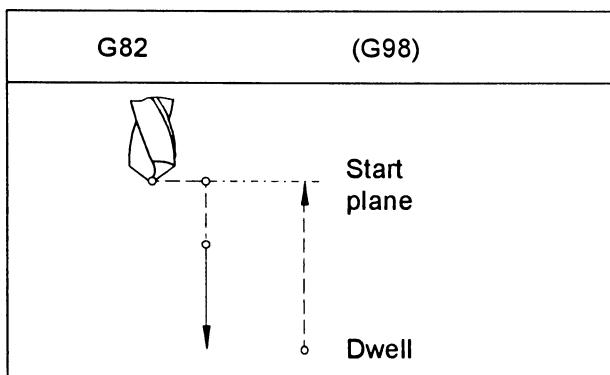
R [mm] Absolute (with G91 incremental) value of the withdrawal plane

F Feed

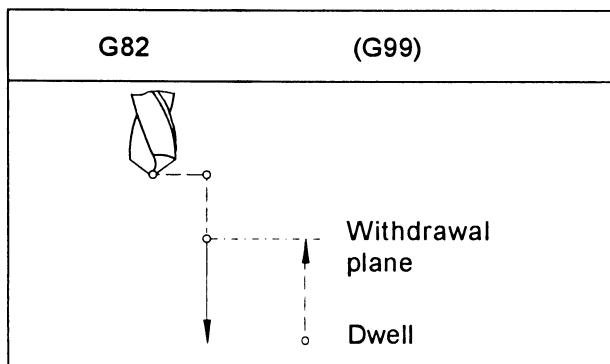
K Number of repetitions



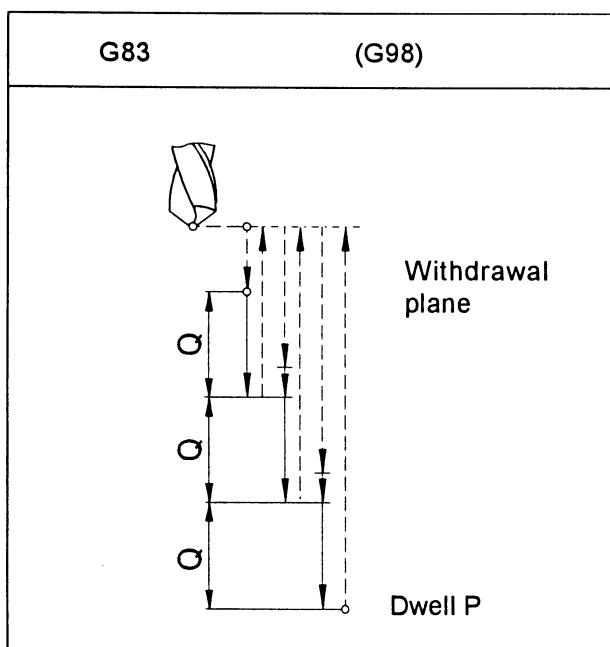
Movements of G81 - drilling cycle



Drilling cycle with dwell and retraction to the start plane



Drilling cycle with dwell and retraction to the withdrawal plane



Movements of G83 with active G98

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

G83 Withdrawal Drilling Cycle

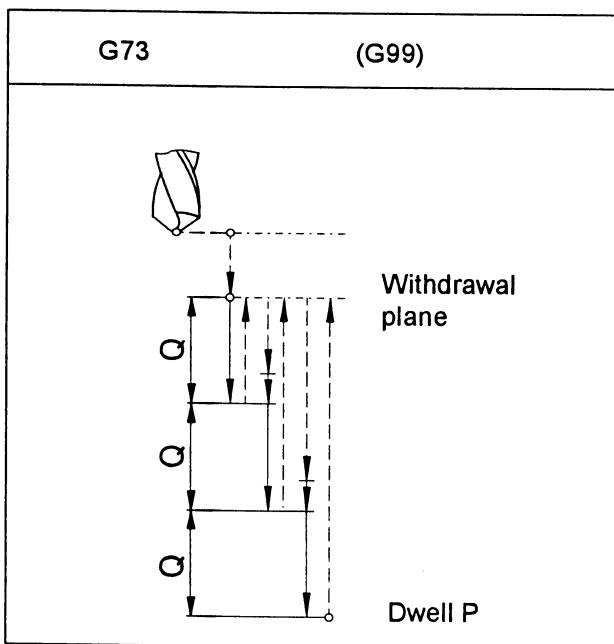
Format

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.

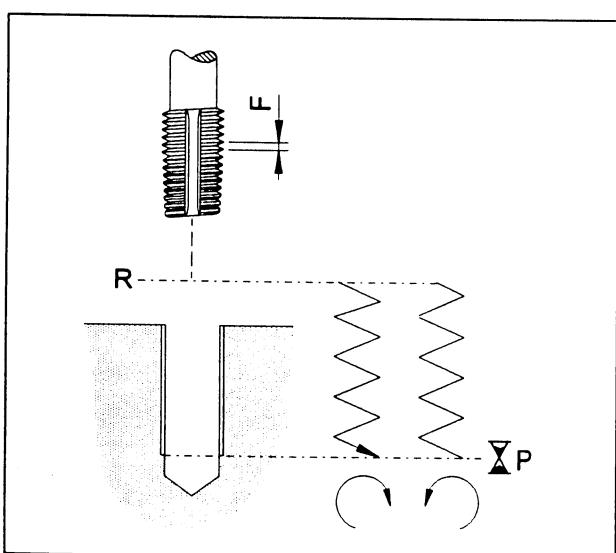
Applications

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
 $P1000 = 1 \text{ sec}$
F Feed rate
Q [mm] Cutting division - infeed per cut
K Number of repetitions



Tapping cycle (with G99)

G84 Tapping Cycle

Only for PC Mill 100.

Format

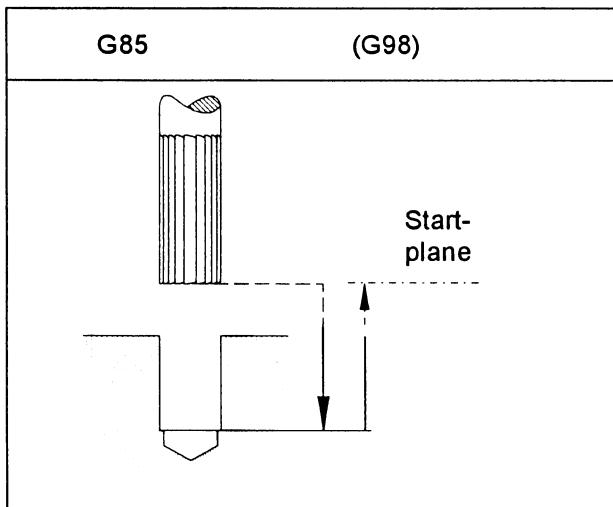
N...G98(G99) G84 X... Y... Z... (R...) F... P... K...

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

*Reaming cycle with withdrawal to the start plane*

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.

G98(G99)... Return to starting plane (withdrawal plane)

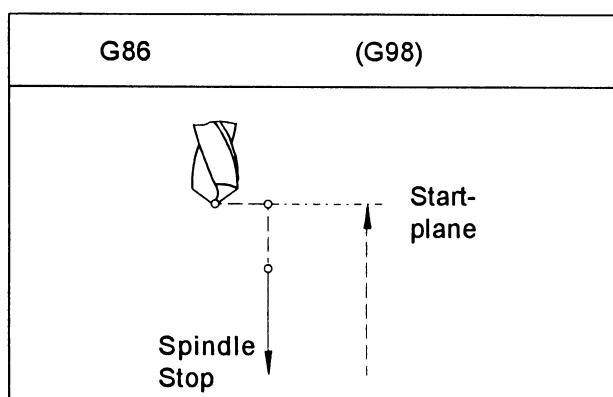
X, Y Hole position

Z Absolute (incremental) drilling depth

R [mm] Absolute (with G91 incremental) value of the withdrawal plane

F Feed rate

K Number of repetitions

*Drilling cycle with spindle stop and withdrawal to the start plane*

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.

G98(G99)... Return to starting plane (withdrawal plane)

X, Y Hole position

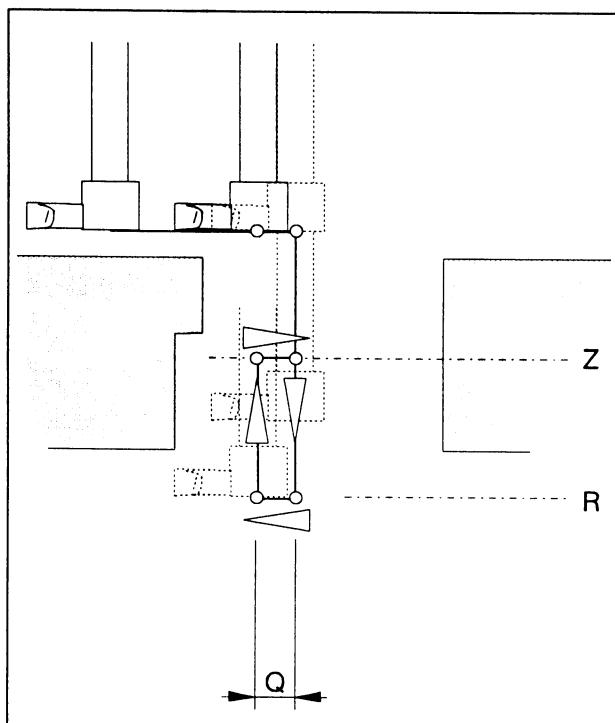
Z Absolute (incremental) drilling depth

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.

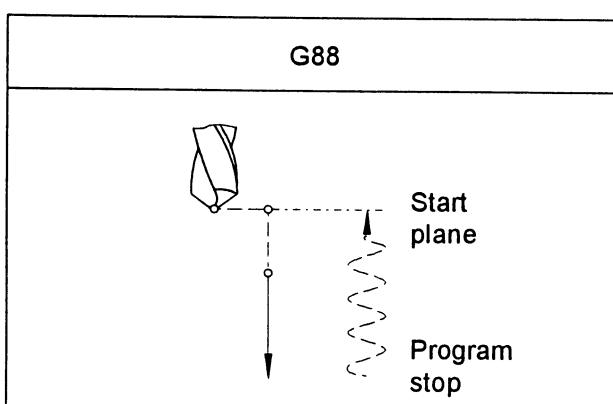
X, Y Hole position

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.

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.

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.

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.

The coolant will be switched on.

M09 Coolant OFF

Only for EMCO PC Mill 100.

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

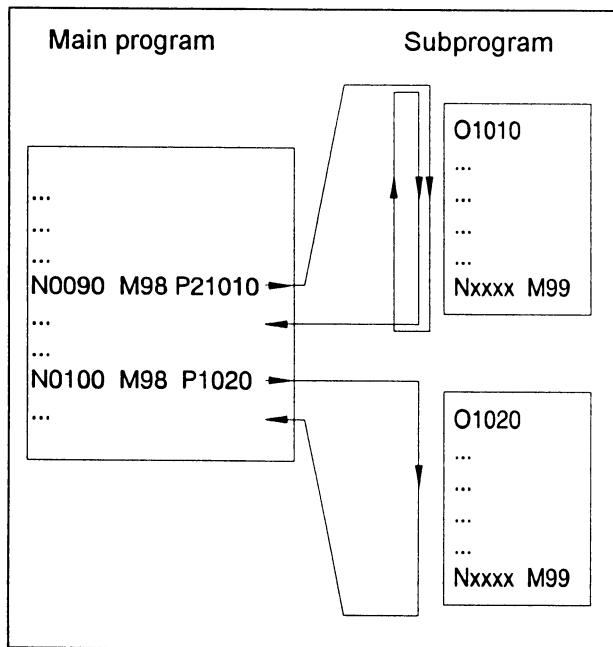
Format

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.

E: Alarms and Messages

Startup Alarms

These alarms can occur only with starting WinNC or WinCTS.

0001 Error creating the file ...

Remedy: Check whether the directories exist, which are entered in the .INI files.
Check whether writing access is possible to these directories.
Check whether the disk is full.

0002 Error opening the file ...

Remedy: Check whether the directories exist, which are entered in the .INI files.
Check whether reading access is possible to these directories (number of simultaneous open files).
Copy the correct file in the corresponding directory.

0003 Error reading the file ...

see 0002.

0004 Error writing the file ...

see 0001

0005 Too less RAM ...

Remedy: Close other WINDOWS applications
Restart WINDOWS

0006 Non compatible software version ...

Remedy: Software update.

0007 Invalid licence version ...

Remedy: Contact EMCO.

0011 Serial interface ... for digitizer is already in use

Cause: The serial interface no. ... is already occupied by another device.

Remedy: Remove the other device and connect the digitizer or define another serial interface for the digitizer.

0012 Serial interface ... for control keyboard is already in use

analogous to 0011

0013 Invalid settings for serial interface ...

Cause: The actual settings of the serial interface are not allowed for WinNC.

Allowed settings:

Baud rate: 110, 300, 600, 1200, 2400, 4800, 9600, 19200

Number of data bits: 7 or 8

Number of stop bits: 1 or 2

Parity: none, even or odd

Remedy: Change the settings of the serial interface in the WINDOWS system control (connections).

0014 Serial interface ... not present

Remedy: Select an existing serial interface.

0015 - 0023 (various alarms)

Remedy: Restart WINDOWS. If these alarm occur repeatedly, contact EMCO.

0024 Invalid entry for control keyboard interface in the profile ...

Cause: The connection of the control keyboard in the file project.ini is invalid.

Remedy: Setting with WinConfig

0025 Invalid entry for digitizer interface in the profile ...

analogous to 0024

0026 Invalid entry for notebook option in the profile ...

Cause: The notebook entry in the file project.ini is invalid.

Remedy: Setting with WinConfig

0027 Error creating the start window

Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

0028 Invalid entry for window representation in the profile ...

Cause: The presentation entry in the file project.ini is invalid.

Remedy: Setting with WinConfig

0029 Error initializing a timer

Remedy: Close all other WINDOWS applications or restart WINDOWS.

0030 Windows 3.1 or higher required

WinNC requires WINDOWS version 3.1 or higher.

0031 - 0036 (various alarms)

see 0002

0037 Memory allocation failure

Remedy: Close all other WINDOWS applications or restart WINDOWS.

0038 Unauthorized software version

Contact EMCO.

0039 Project data non compatible to software version

Possible error after updates, contact EMCO.

0040 Invalid entry for DNC interface in the profile ...

Cause: The DNC entry in the file project.ini is invalid.

Remedy: Setting with WinConfig

0100 Mailslot could not be created

Cause: Insufficient memory in the memory area below 640 kB.

Remedy: Close all other applications, restart WINDOWS. If this is not successfull, remove not necessary device and drivers entries in config.sys or load them in the upper memory area.

0101 For WinCTS Windows for Workgroups 3.11 or higher is required

WinCTS requires WINDOWS for WORKGROUPS version 3.11 or higher.

0102 Error creating the reference table for keybitmaps

Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

0103 Invalid entry for WinCTS status in the profile ...

Cause: The CTS entry in the file project.ini is invalid.

Remedy: Contact EMCO

0104 Error getting the workgroup name

Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

0105 No workgroup found

Remedy: Insert the computer into the workgroup for WinCTS, if necessary set up a workgroup for the WinCTS computers.

0106 Invalid entry for the number of keys to record in the profile ...

Cause: The KeyFifoSize entry in the file winnc.ini is invalid.

Remedy: Correct the number, e.g. 50(see WinConfig).

0107 - 0110 (various alarms)

Remedy: Restart WINDOWS. If this alarm occurs repeatedly, contact EMCO.

Control Alarms

These alarms 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 too 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 too 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 too 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, continuing 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 reproducible
- 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 reproducible
- 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
- 6000 - 7999 Machine alarm s**
see Machine Alarm s
- 8004 ORDxx Failure main-drive unit**
- 8005-8009 ORDxx Internal error AC**
Remedy: Report to EMCO if reproducible
- 8010 ORDxx Syncr. error main drive**
Cause: Main drive synchronisation mark not found
Remedy: Report to EMCO if reproducible
- 8011-8013 ORDxx Internal error AC**
Remedy: Report to EMCO if reproducible
- 8014 ORDxx Axis decel.-time too high**
Remedy: Contact service if reproducible
- 8018 ORDxx Internal error AC**
Remedy: Report to EMCO if reproducible
- 8021 ORDxx Internal error AC**
Remedy: Report to EMCO if reproducible
- 8022 ORDxx Internal error AC**
Remedy: Report to EMCO if reproducible
- 8023 ORDxx Invalid Z value for helix**
Cause: The Z value of the helix must be smaller than the length of arc to be traversed.
Remedy: Program correction
- 9001 unknown parameter !**
Cause: PLC diagnosis, unknown parameter fed in
- 9002 Par.-number not allowed !**
Cause: PLC diagnosis, unknown parameter number fed in

9003 unknown display-format !

PLC diagnosis

9004 DB not existent !

PLC diagnosis

9005 DW not existent !

PLC diagnosis

9006 Invalid COM-port !

PLC diagnosis

9007 Data-transmission active !

Data I/O

9011 No files found !

Data I/O

9014 File already exists

Data I/O

9015 Error while opening file !

Data I/O

9016 Error while reading file !

Data I/O: error with open a file

9017 Error while writing file !

Data I/O: error with write in a file

9018 Invalid RS232 configuration !

Data I/O

9019 Digitizer not initialized !

Cause: A digitizer was declared, but not set up

Remedy: set up digitizer (set up corner points), see "External Input Devices"

9020 No valid input

Cause: Digitizer was activated in an invalid area

9021 Serial interface already occupied

Cause: To the selected interface is already another device connected

9022 No digitizer calibration

Switch on digitizer, check cabling, check interface

9023 Control keyboard not existing

Switch on control keyboard, check cabling, set the interface at the control keyboard (see "External Input Devices"), check plugs, check whether the control keyboard was connected to the correct interface

9024 General RS232 communication error

Remedy: Set RS232 interface, check plugs and cables

9500 No RAM for program

Cause: Too less RAM memory of the PC

Remedy: Close all Windows application, close WinNC, remove resident programs from AUTOEXEC.BAT and CONFIG.SYS, restart the PC

9501 Error with saving program

Cause: Disk full?

9502 To less memory

see alarm 9500

9508 Selected menu not found

Remedy: Report to EMCO if reproducible

9509 Too less memory for picture

Remedy: Report to EMCO if reproducible

9510 Mem.-fail. block-search buffer

Remedy: Report to EMCO if reproducible

9511 Projection error block search

Remedy: Report to EMCO if reproducible

9512 Missing software protection-key !

Remedy: connect dongle

9514 Failure on access to progr.

Remedy: Check file on DOS level

9515 Picture description error**9540 Error in BFM / BFM not found**

Remedy: Report to EMCO if reproducible

Machine Alarms

These alarm s are released by the machine.

The alarm s are different for the PC MILL 50 and the PC MILL 100.

The alarm s 6000 - 6999 normally must be acknowledged with RESET. The alarm s 7000-7999 are messages which will disappear usually when the releasing situation is eliminated.

PC MILL 50

The following alarm s are valid for the PC MILL 50.

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 or too hot, 24 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, fuse defective.
Check fuse.
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 sticks, 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.

6041: TOOL CHANGE TIMEOUT

Tool turret sticks (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 sticks, 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

The following alarm s are valid for the PC MILL 100.

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.

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.

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.

6041: TOOL CHANGE TIMEOUT

Tool drum stuck (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.

6044: 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 toothings 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 it 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 stuck 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

6072: VICE NOT READY

Attempt to start the spindle with an open vice or without clamped workpiece.

Vice stuck 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 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 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

7054: 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

F: Accessory Functions

Activate Accessory Functions

The PC MILL 50 and the PC MILL 100 can be equipped with the following accessories

- Automatic door
- Automatic vice
- Puff blow device
- Robotic interface
- Dividing head
- DNC interface

For using the accessories a PLC must be built-in and activated (authorized dealer or service technician).

Activate the accessories with WinConfig or as on the original control by changing the settings.

Changing the settings:

- Press the key DGNOS PARAM.
- By threefold pressing the key PAGE the page with the four PLC settingbytes SD1 - SD4 will be displayed.
Byte SD2 activates the accessories.

e.g.: SD2 1 1 0 0 1 0 1 1
Bit No. 7 6 5 4 3 2 1 0

Bit 0	automatic door
Bit 1	automatic vice
Bit 2	not used
Bit 3	puff blow device
Bit 4	not used
Bit 5	tool turret (PC MILL 100)
Bit 6	dividing head
Bit 7	robotic interface

Value 0: function not active
Value1: function active

Note

After altering these setting data the machine must be switched off and on.

For accessories the following M codes are in use:

- M25 Close vice (PC MILL 100)
- M26 Open vice (PC MILL 100)
- M27 Swivel dividing head
- M71 Puff blowing ON
- M72 Puff blowing OFF

The accessories and machine functions can be activated with the following keys:

PC keyboard control keyboard, digitizer

Strg	I 1		swivel tool turret (MILL 100)	*
Strg	* 2 2		puff blowing on / off (MILL 50)	
Strg	* 2 2		coolant on / off (MILL 100)	*
Strg	§ 3 3		swivel dividing head	*
Strg	\$ 4		feed stop	*X
Strg	% 5		feed start	*X
Strg	& 6		spindle stop	
Strg	/ 7 {		spindle start	
Strg	(AUX ON	
Strg) 9]		close vice	++
Strg	= 0		open vice	++
Strg	? 8 \		AUX OFF	
Strg	;		door open / close (PC MILL 100: with consent key)	
+ works only with open door				
* works only with closed door				
• on the PC MILL 100 not depending on door status				
x PC MILL 100 only in special operation mode				

Robotic Interface PC MILL 50

The robotic interface for the PC MILL 50 is an accessory. To activate it, a special PLC software (EPROM) has to be installed.

**Caution:**

Inputs and outputs are **NOT** potential free.
(NOT insulated)

Inputs:**Signal level:**

0 V .. 5 V	LOW
15 V .. 24 V	HIGH

Input impedance:

2 kΩ

Signal form:

So long, how a HIGH signal is on input 3.7, "FEED HOLD" will be active

All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal).

Input assignment:

- E 3.0 open door
- E 3.1 close door
- E 3.2 tailstock backward
- E 3.3 tailstock forward
- E 3.4 open chuck / collet
- E 3.5 close chuck / collet
- E 3.7 feed stop
- E 1.7 program start

Outputs:

All outputs are short circuit proof and bearable with 0,2 A.

Signal level:

20 V .. 24 V HIGH

Output assignment:

- A 0.0 program stop (M30, M00, M01)
- A 0.1 chuck / collet open
- A 0.3 chuck / collet closed
- A 0.4 door open
- A 0.5 door closed
- A 0.6 tailstock behind
- A 0.7 tailstock clamped
- A 1.7 alarm output

Robotic Interface PC MILL 100

The robotic interface for the PC MILL 100 is an accessory. To activate it, a special PLC software (EPROM) has to be installed.


Caution:

Inputs and outputs are **NOT** potential free.
(NOT insulated)

Inputs:
Signal level:

0 V .. 5 V	LOW
15 V .. 24 V	HIGH

Input impedance:

2 kΩ

Signal form:

So long, how a HIGH signal is on input 10.6, "FEED HOLD" will be active

All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal).

Input assignment:

- E10.2 AUX ON
- E10.3 Switch NC-mode reference-automatic
- E10.4 Reference axis
- E10.5 NC start
- E10.6 Feed hold
- E11.0 Close vice
- E11.1 Open vice
- E11.4 Close door
- E11.5 Open door

Outputs:

All outputs are short circuit proof and bearable with 0,2 A.

Signal level:

20 V .. 24 V HIGH

Output assignment:

- A10.3 Emergency OFF
- A10.4 Machine ready
- A10.5 NC-mode reference-automatic
- A10.6 Program status
- A10.7 Alarm status
- A11.0 Vice clamped
- A11.1 Vice open
- A11.6 Door closed
- A11.7 Door open

Automatic Vice

The automatic vice works only with open chip guard door. It can be traversed manually by pressing the key or via the robotic interface or the DNC interface, on the PC MILL 100 also with M25/M26 in the program.

Notes for Working with the Automatic Tailstock

- The main spindle can not be switched on when the vice is in an undefined status (neither in back nor in clamped position).
- Moving the tailstock manually is only possible with standing spindle and inactive NC START. On the PC MILL 50 additionally the door must be open.
- The tailstock moves with pressed key until the end position is reached, it can not be positioned by key-tipping.

Door Automatic

Characteristics with activated door automatic:

Open door:

The door can be opened by manual key pressing, via robotic interface or DNC interface.

Additionally the door opens if the following commands are proceeded in the CNC program:

- M00
- M01
- M02
- M30

Close door:

The door can be closed only by manual key pressing, via robotic interface or DNC interface.

PC MILL 50: Door closes after key was pressed.

PC MILL 100: For closing the consent key and the door key must be pressed. The door moves as long as the keys are pressed.

Puff Blowing Device

M71 Puff blowing ON

By M71 in the CNC program the puff blowing device will be switched on.

M72 Puff blowing OFF

By M71 in the CNC program the puff blowing device will be switched off.

On the PC MILL 50 the puff blowing device is switched on and off with the key combination Ctrl + 2.

DNC Interface

The accessory DNC Interface can be used only for WinNC controlling a machine.

Activate the accessory DNC interface with WinConfig.

With the DNC interface the machine can be operated via the PC control together with other machines (flexible machining system).

The DNC interface is set in SETTING DATA - SETTING BITS - DNC.

A master computer coordinates the machines and can load or read the following data and commands via the DNC interface:

- NC start
- NC stop
- NC programs
- zero offsets
- tool data
- RESET
- approach reference point
- peripheric control
- override data, ...

Installation of the DNC Interface

- Switch on your PC.
- Start Windows.
- Insert the installation disk for the DNC interface in drive A.
- Select "File" in the command line of the program manager.
- Select "Run".
- Enter: "a:\setup". Confirm with "OK" (click or ENTER).
- Enter the path in which WinNC is installed.
- Select on which serial interface the DNC interface should be (when you select NONE the DNC will be installed but not activated). Click on "OK".
- Click "OK". The installation is finished.

Activate Tool Turret

The tool turret (PC MILL 100) will be activated like a accessory by Bit 5 in SD2 or with WinConfig. See "Activate Accessory Functions".

WinConfig

General

WinConfig is a configuration software for WinNC and WinCTS.

With WinConfig you can alter the settings of WinNC.

The setting possibilities in the control surfaces (e.g. with setting bytes) are equal to WinConfig, but WinConfig is much more comfortable in operation.

The most important setting possibilities are:

- Language
- Measuring system mm - inch
- Screen display
- Activate accessories
- Interface selection for digitizer and control keyboard

WinConfig also can activate diagnosis functions for service - so you can get fast help.

Some functions of WinConfig are protected by password. This depends on safety.

These functions must be activated only by set-up or service technicians.

Notes for using WinConfig with WinCTS

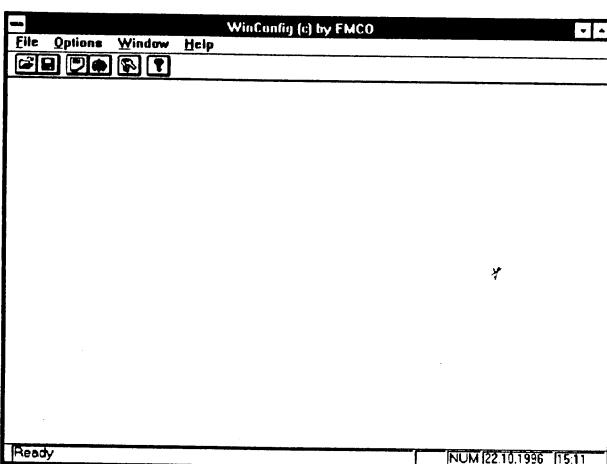
WinConfig in connection with WinCTS is installed at the teachers workplace only. Students have no possibility to alter settings with WinConfig.

The teacher can alter the INI files and the machine data of the students as following:

- In a WINDOWS network (Windows for Workgroups or Windows 95) the installation directories of the students must have read and write access (tip: with password protection, that the students can not connect each other). After that in the WinConfig dialogue window "File - Open" you connect the desired student with the switch button "Network".
- In a network installation (e.g.: Novell) the teacher has direct access to all students (Users) when he is logged in as "Supervisor". In WinConfig you have to select the "Home" directory of the desired student in the Window "File - Open".



Icon for WinConfig



Window for WinConfig

1. Start WinConfig

Double-click on the icon for WinConfig or mark the icon with Ctrl-Tab and the Cursor keys and press Enter.

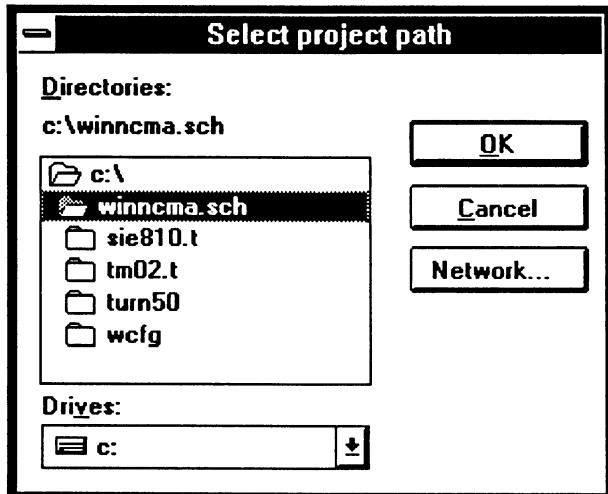
At the screen the window for WinConfig appears.

2. Select Program path of WinNC

Only for WinCTS

Before you can change the settings of WinNC you have to enter where the WinNC software is located.

Select File - Open or click on the symbol .



Selection window for the program path of WinNC

At the screen you can see the selection window for the program path.

Select the program path in which the file WINNC.EXE is located and click on OK.

With NETWORK you can select the program path of a student or of a machine which is defined as student.

WinConfig stores the program path, that means when you start WinConfig at a later time the last used program path is active.

3. Basic Settings for WinConfig

For WinConfig you can define some basic settings. These settings are valid **ONLY** for WinConfig and **NOT** for WinNC.

Select Options in the menu line. You can select Language, Measurement and Password.

Language

You can select English or German.

Measurement

Only in english language version active. You can select whether the data of WinConfig (e.g. position of reference point) are given in mm or inch.

Password

Parameter which touch safety topics are protected by password and can be activated only by set-up or service technicians

4. Change Ini Data of WinNC

Here you can alter data of the software part of WinNC.

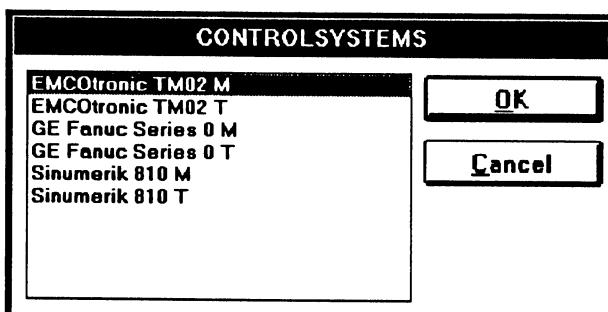
Data of a connected EMCO lathe or milling machine are called Msd data.

As usual with WINDOWS software the Ini data are stored in .ini files.

Select Window - Ini Data or click on the symbol .

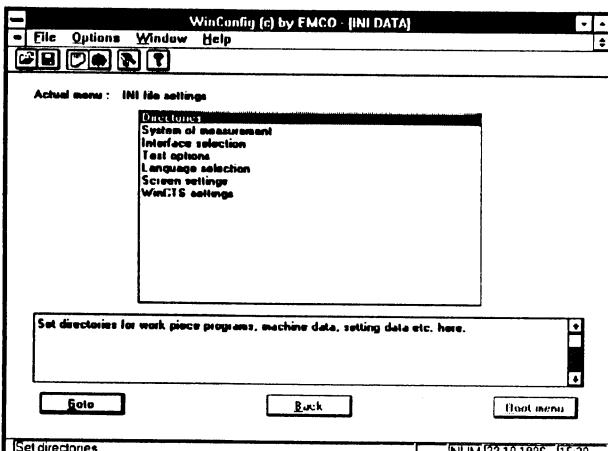
When several control types are installed, the screen shows a selection menu.

Click on the desired control type and on OK.



Selection menu for control type

All following settings are valid for the selected control only.



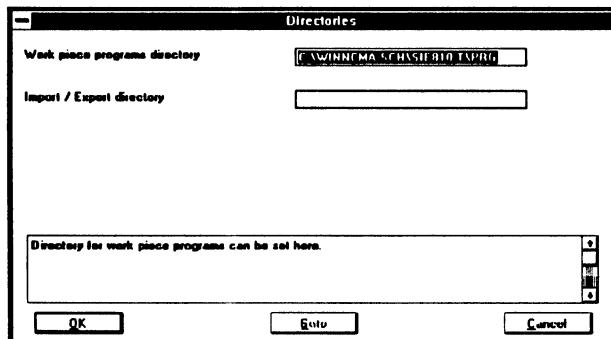
Selection menu for Ini data

The screen shows the selection menu for the Ini data.

4.1 Alter Directories

Select the menu point Directories and press Enter or double-click on Directories.

The screen shows the input window for directories.



Input window for directories

Workpiece Program Directory

Enter the directory in which WinNC opens the workpiece programs.

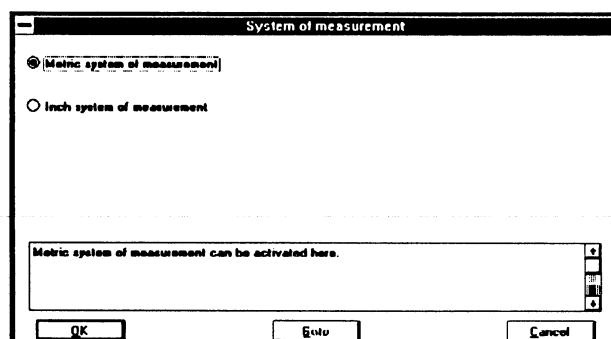
Several users can have their own workpiece directory.

Import / Export Directory

Enter the directory from which data will be imported or to which data will be exported.

You must enter an existing directory, WinConfig does not create directories.

When no directory is entered here WinNC transmits from/to the workpiece directory.

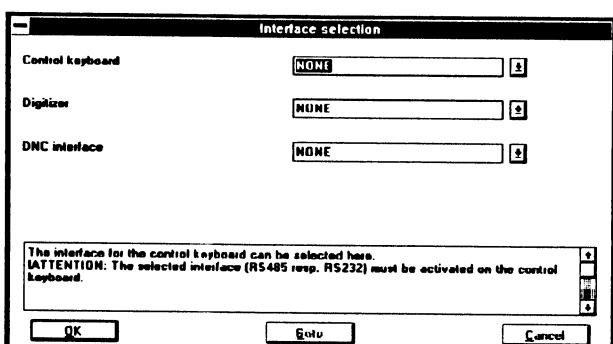


Select measuring system

4.2 System of Measurement

You can select the system of measuring for the control.

Click in the ring for the desired measuring system and on OK.

*Interface selection*

4.3 Interface Selection

Determine which interface you want to use for the single devices.

Control keyboard (accessory)

- | | |
|--------|---|
| NONE | no control keyboard connected |
| RS485 | Control keyboard connected to RS485 interface (recommended for WinNC Machine licence) |
| COM1-4 | Control keyboard connected to serial interface (RS232) 1-4 |

Digitizer (accessory)

- | | |
|--------|---|
| NONE | no digitizer connected |
| COM1-4 | Digitizer connected to serial interface (RS232) 1-4 |

DNC (accessory)

DNC is an interface for communication between several machines and a central control computer in an automatical system (FMS).

The central control computer coordinates the machines and transmitts the necessary data via the DNC interface.

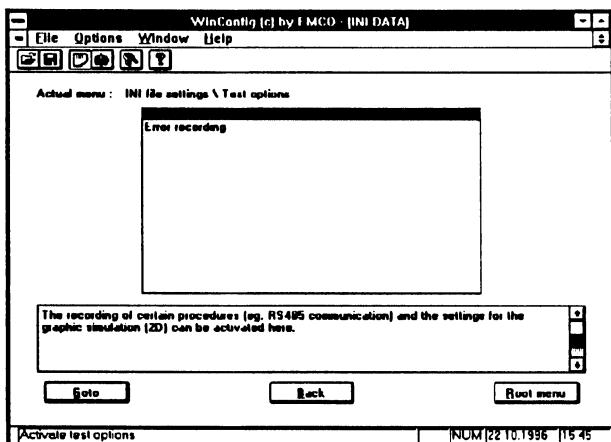
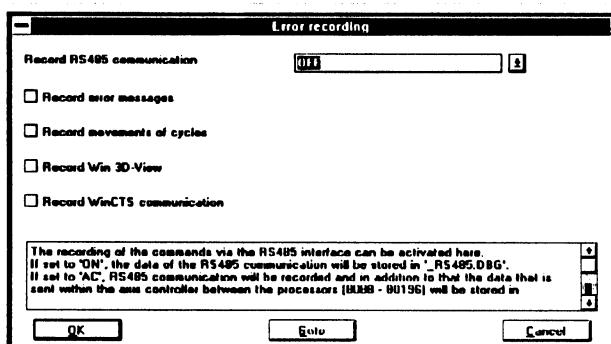
The DNC interface is one of the serial interfaces of the computer.

- | | |
|--------|-----------------------------|
| NONE | no DNC |
| COM1-4 | DNC is serial interface 1-4 |

Note

Consider the explanations in the chapters "External input devices" and "Accessories".



**Submenu Test options****Error recording**

4.4 Test Options - Error Recording

Select the menu point test options.
You enter a submenu.

Without password only the topic Error recording is active.

Select Error recording.

With Error recording you can log the internal command sequences of computer and machine.

This logfile is used for error detection.

Record RS485 Communication

- OFF no recording RS485 communication
- ON RS485 communication will be recorded and stored in the file _RS485TR.DBG.
- AC Additionally to RS485 communication also the internal communication of the axis controller will be recorded and stored in the file _ACTR.DBG.

Record Error Messages

Activate this function (☒) to store the error messages and additional information in the file _ERRLOG.DBG.

Record Movements of Cycles

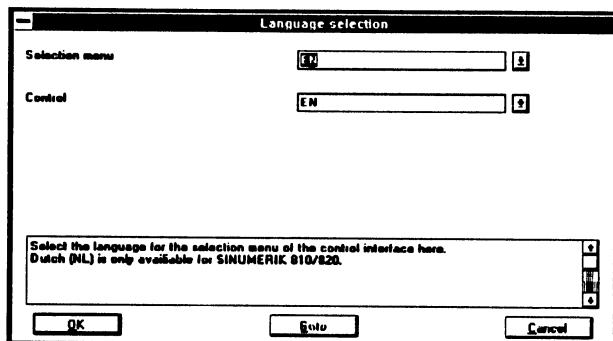
Activate this function (☒) to store the single movement commands in the file _PRGINPT.DBG.

Record Win 3D View

Activate this function (☒) to store the internal commands of the 3D simulation in the file NCSIM.LOG.

Record WinCTS Communication

Activate this function (☒) to store the WinCTS network communication in the file _CTSTR.DBG.



Input window language selection

4.5 Language Selection

Select the menu point language selection.

Selection menu

When several types of controls are installed, after starting WinNC a selection menu is displayed to select the desired type of control.

Here you can determine the language of the selection menu.

Control

Select the desired language for the control.

Available languages:

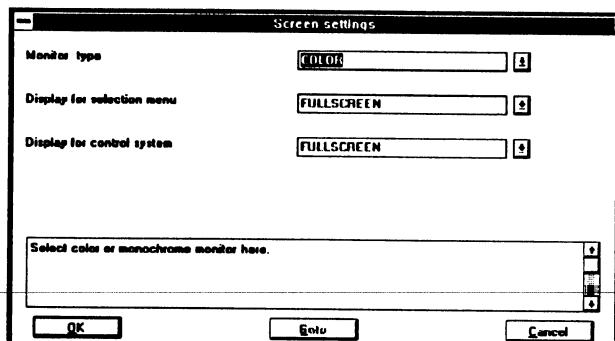
DT German

EN English

FR French

SP Spanish

NL Netherlands (only for SINUMERIK 810/820)



Input window screen settings

4.6 Screen Settings

Monitor type

COLOR Color screen

MONOCHROME Gray screen

Display for selection menu

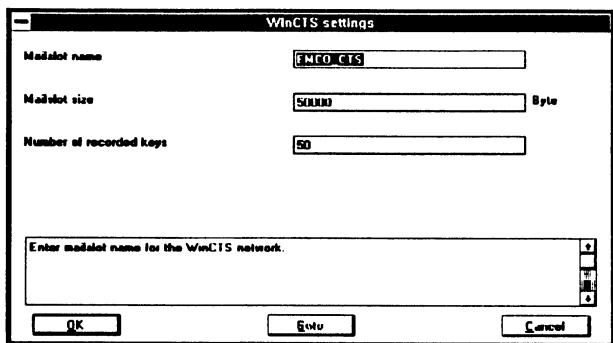
NORMAL The WinNC selection menu uses only a part of the screen.

FULLSCREEN The WinNC selection menu uses the whole screen.

Bildschirmsdarstellung Steuerung

NORMAL The WinNC control window uses only a part of the screen.

FULLSCREEN The WinNC control window uses the whole screen.



Input window WinCTS settings

4.7 WinCTS Settings

Mailslot name

The mailslot is an address for communication in the network.

The complete communication of EMCO WinCTS is done via the mailslot which is determined here. WinCTS works only when all participants have the same mailslot name.

Mailslot size

Here you can enter the size of the mailslot buffer. The mailslot occupies memory in the lower 640 kB RAM.

When other software is disturbed by the memory occupation of the mailslot you can try to reduce the mailslot buffer.

When one computer in the WinCTS network is very slow compared with another computer the mailslot can get an overflow and information will be lost. In this case the mailslot buffer has to be increased.

Number of recorded keys

WinCTS records the operating sequence of the pressed keys and displays it at the screen.

In this way data input can be watched by all.

Here the number of the recorded keys can be determined.

5. Change Msd Data of WinNC

Here you can alter data of the machine part of WinNC.

Data of the WinNC software are called Ini data.

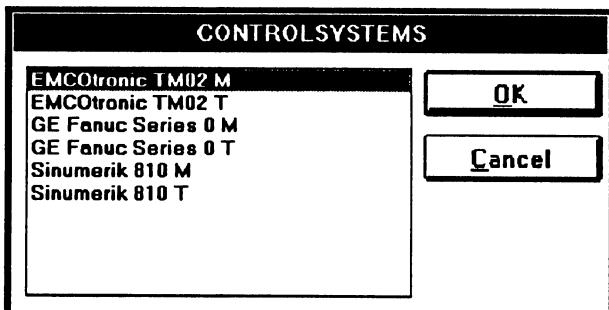
Insert the MSD disk of the machine into drive A or B. The MSD data will be written on the disk. When no disk is inserted you can not store and your alterations will be lost.

Select Window - Msd Data or click on the symbol .

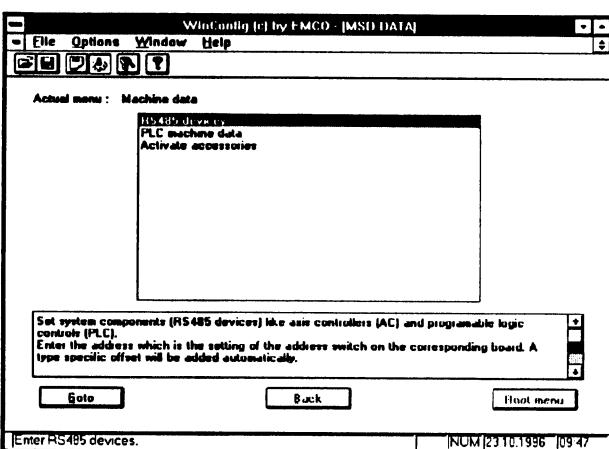
When several control types are installed, the screen shows a selection menu.

Click on the desired control type and on OK.

All following settings are valid for the selected control only.



Selection window for control type



The screen shows the menu for the Msd data.

Menu for Msd data

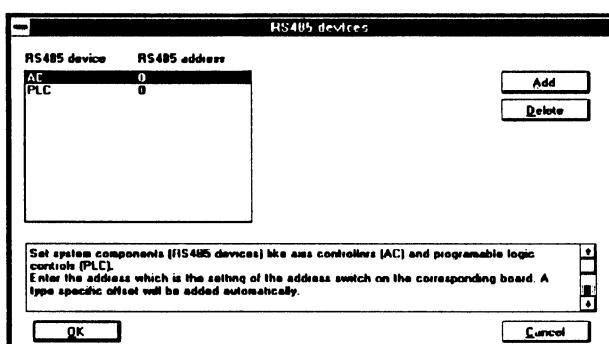
5.1 RS485 Device List



Note

This menu point is needed only for serious alterations on the machine (e.g. mounting a PLC automatisation unit on the milling machine EMCO PC MILL 50).

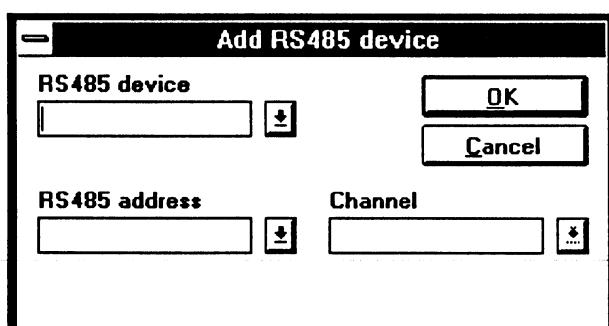
When the device list that is determined here does not fit to the machine configuration, the machine will not work.



Display of the active RS 485 devices

Select the menu point RS485 devices
The actual active RS485 devices are displayed.

You can add or remove RS485 devices.

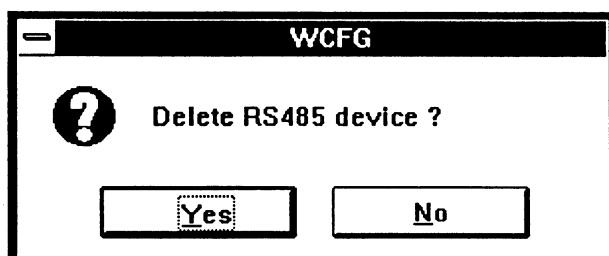


Selection window add RS 485 device

Add RS485 device

Select the switch button Add. The screen shows a selection window.

Determine which device should be added.
As RS485 address you have to enter that address that is set on the device board at the address switch.

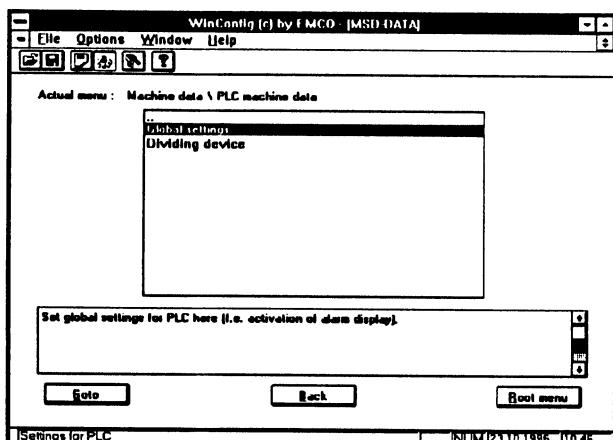


Safety query delete RS485 device

Delete RS485 device

Select the device to be deleted and click on the switch button Delete. The screen shows a safety query.

Confirm with Yes or abort with No.

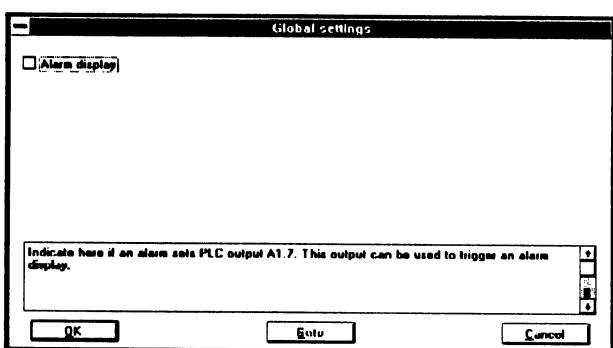
**Setting possibilities PLC machine data**

5.2 PLC Machine Data

Here you can alter the PLC settings.

Select the menu point PCL machine data.

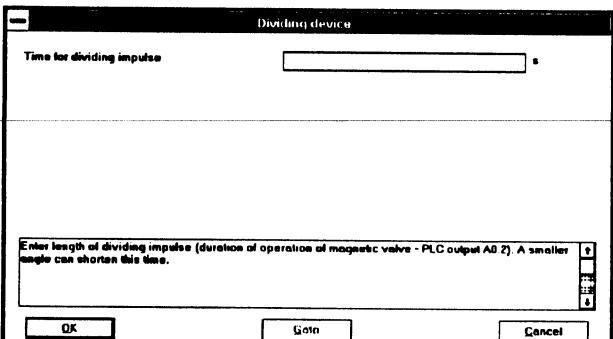
The screen shows the setting possibilities.

**Activate alarm display**

Global settings

You can set whether Output A1.7 is actuated in the case of an alarm.

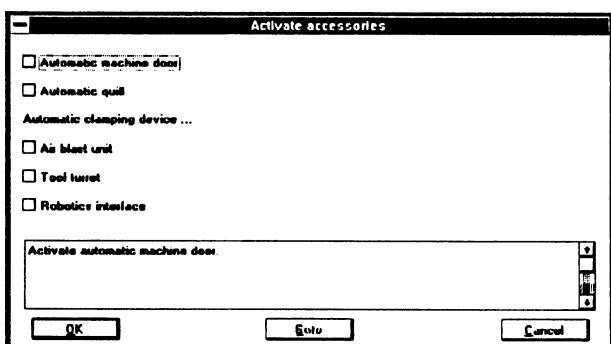
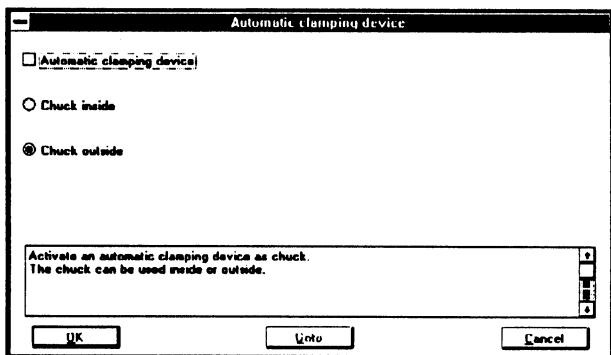
With this output you can e.g. trigger an alarm signal light.

**Set switching time for dividing device**

Dividing device

Setting of the switching time (time duration for which the dividing device is fed with compressed air) for the pneumatic dividing device.

For small dividing angles you can reduce this time and therefore gain a faster program run time.

**Zubehörliste****5.3 Activate Accessories**

When you set-up accessories on your machine these accessories must be activated here.

Activate the existing accessories with and select OK.

When you select an automatic clamping device for a lathe you enter a submenu.

Activate the automatic chuck with .

Chuck inside:

The clamping movement is from outside to inside.

Chuck outside:

The clamping movement is from inside to outside. The jaws are inside the tubular workpiece and are clamping to outside.

Select the desired clamping direction and click on OK.

6. Store Changes

Your alterations must be stored.

Select File - Save or click on the symbol .

When you have changed Msd data, the MSD disk must be inserted in drive A or B. Otherwise storing is not possible and your alterations will be lost.

Software Installation

General

You can install EMCO WinNC for the following CNC control types:

- SINUMERIK 810 T
- SINUMERIK 820 T
- SINUMERIK 810 M
- SINUMERIK 820 M
- GE Fanuc Series 0-TC
- GE Fanuc Series 0-MC
- EMCOTRONIC TM02 T
- EMCOTRONIC TM02 M
- PAL T
- PAL M

If several types are installed, with the start of WinNC a menu is displayed from which you can select the desired control type.

For every CNC control type you can install the following versions:

- Machine licence version:
With this version a PC controlled machine (PC TURN 50, PC TURN 120, PC MILL 50, PC MILL 100) is controlled by WinNC like with a usual CNC control.
- Single licence version:
Programming and operating of the desired CNC control type is simulated at the PC by WinNC.
- Multiple licence version:
WinNC is installed on the server of a PC network. On any number of connected PC's working stations can be installed.
Programming and operating of the desired CNC control type is simulated at this working stations by WinNC.

Notes

- Before you install the software we recommend to make backup copies of all delivered disks (also machine data disk).
If data are deleted accidentally, or if disks become defective due to uncorrect treatment, the original disks are still available.
- To copy the disks use the command "diskcopy" in the DOS operating system or the command "Copy data medium" in the Windows File Manager.
- Mind the minimum configuration of the PC for installing EMCO WinNC:
PC 80386 DX33 IBM compatible
4 MB RAM
8 MB free hard disk memory for all control types
3.5" disk drive 1.44 MB
VGA board
VGA monitor
Windows version 3.1

Software Update

When the installation program of WinNC finds a version of WinNC in that directory, in which installation should happen, the system asks whether you want to:

- proceed a software update
 - install in another directory
 - quit the installation.
-
- Start the old version of WinNC.
 - Output of offset data to drive C (see Software Description EMCO WinNC, Operating - Data Input - Output).
 - Close the old version.
 - Install the new version of WinNC in the same directory as the old version.
 - After the query select "Make an update".
The update occurs without queries.
 - Start the new version.
 - Read in the offset data from drive C (see Software Description EMCO WinNC, Operating - Data Input - Output).
- By this sequence the tool offsets and zero offsets will not be lost while installation.

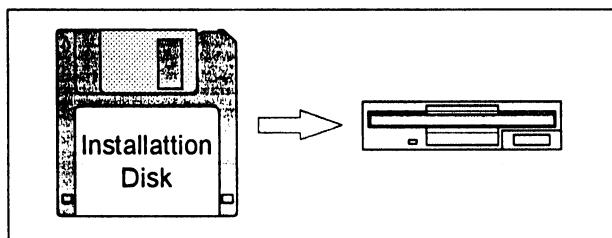
Sequence of Installation

Note:

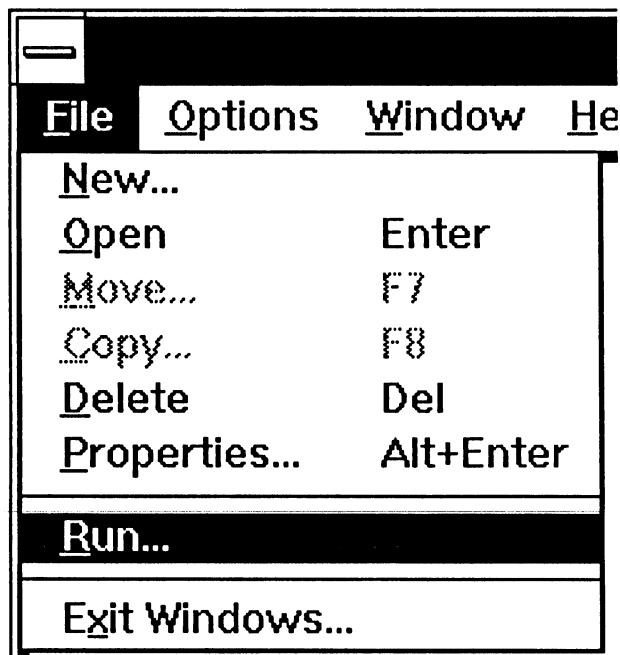
This manual describes the installation of all versions of WinNC.

Work off the following points one by one and skip the points which do not affect your version.

Before a network installation of the multiple licence please read the chapter "Notes for Network Installation".



Insert disk



Program manager - File

All versions:

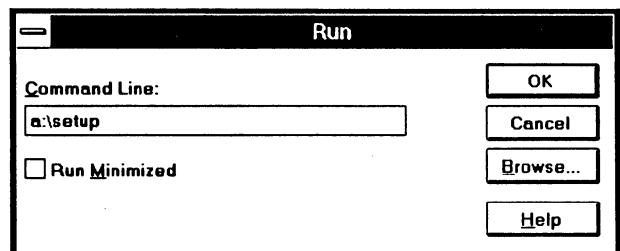
- Switch on your PC.
- Start Windows.
- Insert the installation disk 1 into drive A.

All versions:

- Select "File" in the command line of the program manager.
The screen shows the menu beside.

All versions:

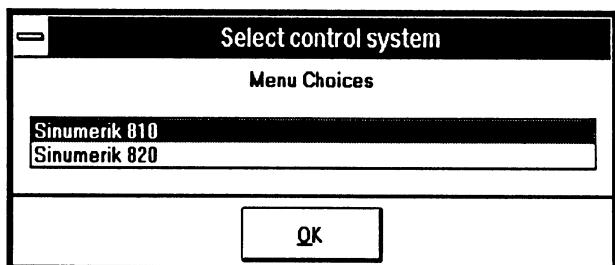
- Select "Run".
The following input window is opened:



Program manager - File - Run

All versions:

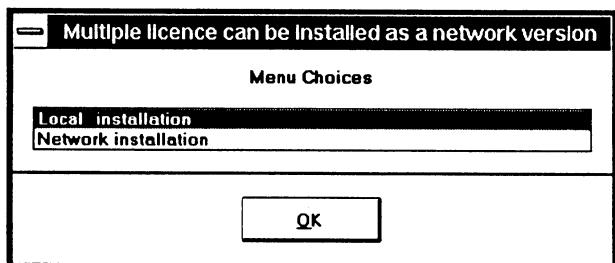
- Enter into the command line: "a:\setup"
- Acknowledge with "OK" (click on or ENTER).
The installation program will be started.



SINUMERIK: selection 810 - 820

Only SINUMERIK:

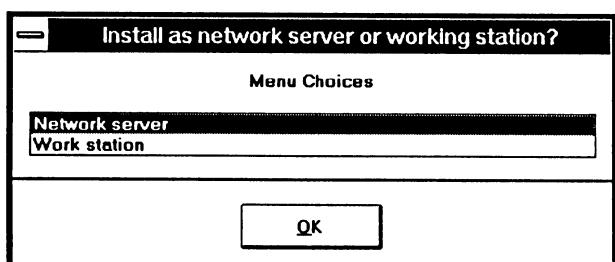
- Select the desired control type and acknowledge with "OK".



Multiple licence: selection net - single station

Only multiple licence version:

- Select whether you want to install a single station (standard installation) or in a network. If you select the standard installation WinNC will be installed like a single licence version.

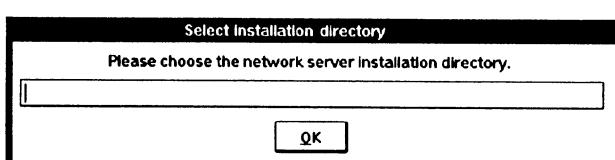


Net installation: select server - working station

Only net installation:

- Select whether you want to install on the network server (master computer of the network) or on a working station.

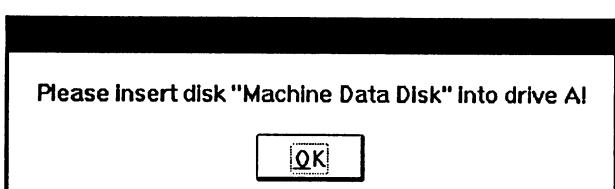
Before you can install on a working station, WinNC must have been installed on the server. With installing the working station only the individual settings will be installed, the core of the software is installed at the server only.



Net working station: enter server directory

Only for net working station:

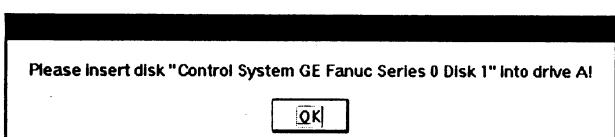
- Enter here the directory in which WinNC was installed at the server.



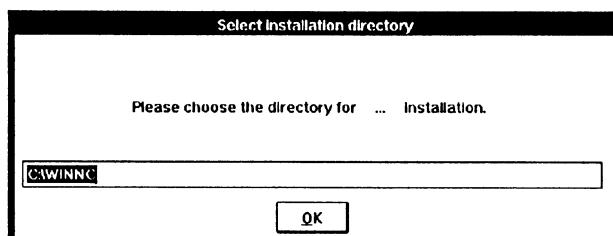
Machine version: insert MSD disk

Only for machine version:

- If you install a machine version now you have to insert the machine data disk.
- Remove the installation disk from drive A, insert the machine data disk and acknowledge with "OK". The machine data will be read in from the MSD disk.
- Insert the installation disk again and acknowledge with "OK".



Machine version: insert installation disk

*Directory for software installation*

All versions except net working station:

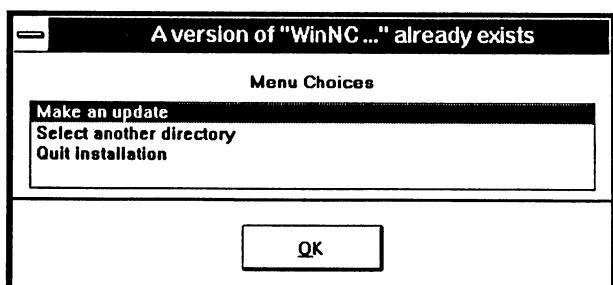
- The installation program needs a directory, in which WinNC should be installed.
As drive enter a hard disk drive or a network drive with at least 3 MB free memory.
- The installation program proposes the directory name "WINNC". You can alter this name.

Note:

If you install several licence types of WinNC (machine version, single licence, multiple licence) or several language versions, you must use different directories for every version.

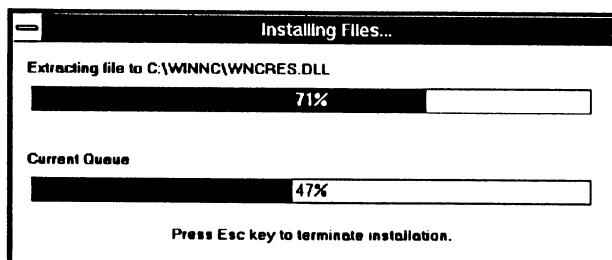


- Take over the proposal or enter an other directory and acknowledge with "OK".

*Query with already installed WinNC*

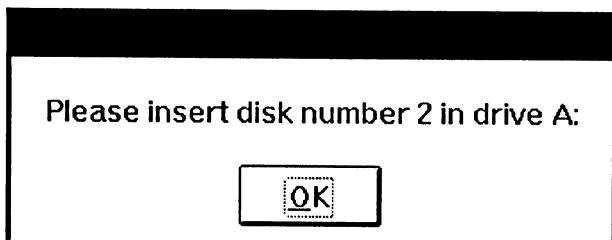
When the installation program of WinNC finds a version of WinNC in that directory, in which WinNC should be installed, you can select one of the following possibilities:

- Make an update:
You can update the software. See "Software Update, Change Settings".
- Select another directory
With this selection you can select another directory to install WinNC.
After that WinNC is installed twice at the computer!
- Quit installation
The installation will be ceased, no alterations will be executed.

*Display of the installation progress*

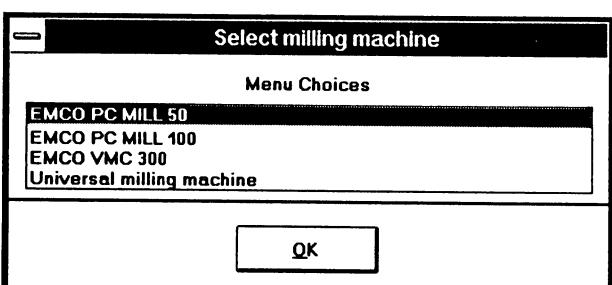
All versions except net working station:

- While the following installation the screen shows how far the installation is advanced.

*Insert second installation disk*

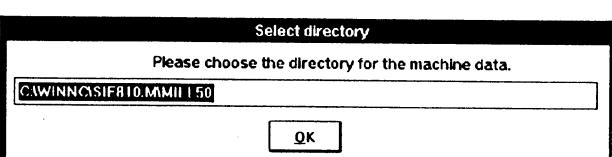
All versions:

- The installation program needs die installation disk 2.
- Insert the installation disk 2 into drive A and acknowledge with "OK".

*Example: selection milling machines*

Only single licence version and net working station:

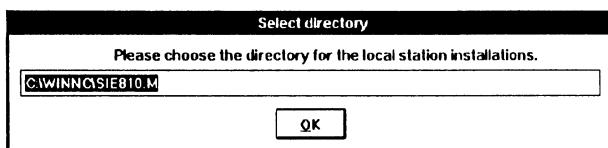
- The installation program proposes different machines, of which you can simulate programming and operating. In the software the limit data of this machine will be set (working area, feeds, speeds, ...). With "Universal... machine" you can simulate an almost unlimited machine.
- Select the desired machine and acknowledge with "OK".

*Directory machine simulation data*

Only single licence version and net working station:

- The installation program needs a directory, in the limit data of the selected machine should be installed.

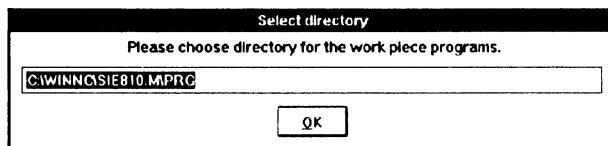
The installation program proposes as directory name the name of the selected machine. You can alter this name.



Net server: directory for local data of the working station

Only net server:

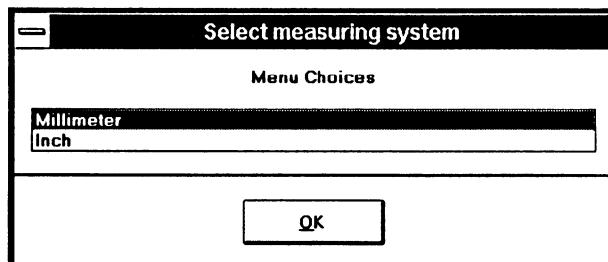
- The installation program needs a directory for the local data of the working station. Here you must enter the directory, on which the drive letter was mapped (see "Notes for Network Installation").



Enter directory for CNC programs

All versions except net server:

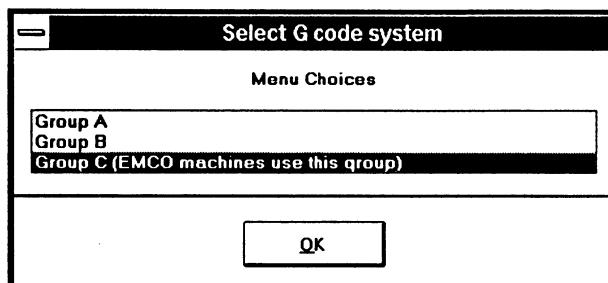
- The installation program needs a directory, in which all CNC programs created by you should be written. The installation program proposes the directory name "PRG". You can alter this name.
- Take over the proposal or enter another directory and acknowledge with "OK".



Select measuring system

All versions except net server:

- Select the desired measuring system and acknowledge with "OK".

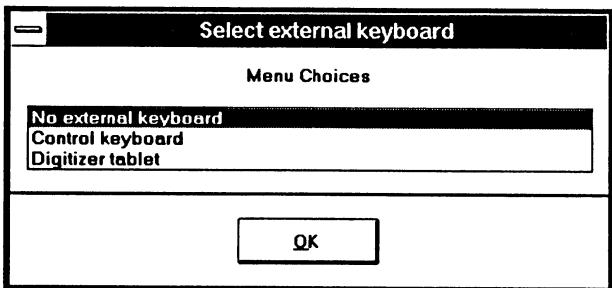


Fanuc 0-TC: enter command group

Only for Fanuc 0-TC except net server:

WinNC for the Fanuc Series 0 control can use all 3 command groups (A, B and C). With the installation of WinNC for Fanuc the command group is asked for.

- Select the desired command group and acknowledge with "OK".

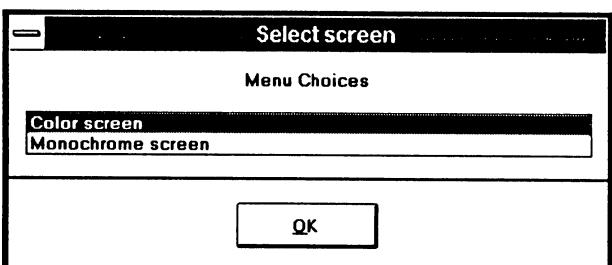
*Enter external input device*

All versions except net server:

- Eventually connected input device.
Select the used input device and acknowledge with "OK".
If you enter an input device the system asks on which interface it is connected (not shown). Select the interface and acknowledge with "OK".

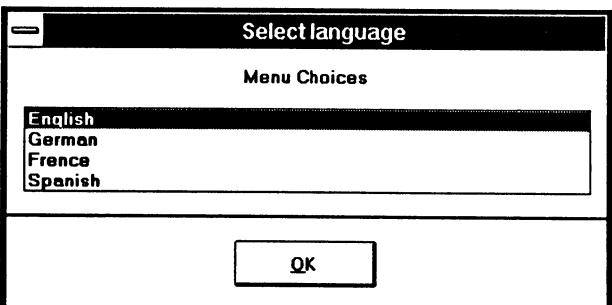
Note:

If you connect an input device at a latertime, you need not to install WinNC again, you can activate this device with an entry in the file PROJECT.INI (see Basic Settings of WinNC).

*Screen selection*

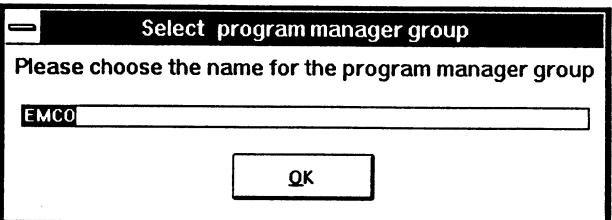
All versions except net server:

- Select the screen used by you and acknowledge with "OK".

*Select language*

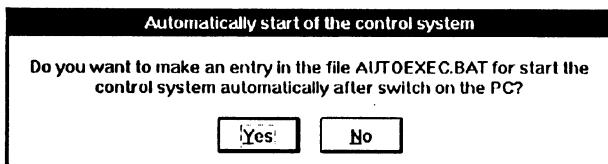
All versions:

- Select the language version for WinNC and acknowledge with "OK".
You also can alter the language in the CNC control pictures or in the basic settings for WinNC afterwards.

*Enter program group name*

All versions except net server:

- The installation program creates the icon for WinNC in a program group (a window for programs in the program manager).
You can enter an existing program group name or create a new program group by entering a new name.
- Enter a program group name and acknowledge with "OK".



Machine version: automatic start YES - NO

Note for WINDOWS 95:

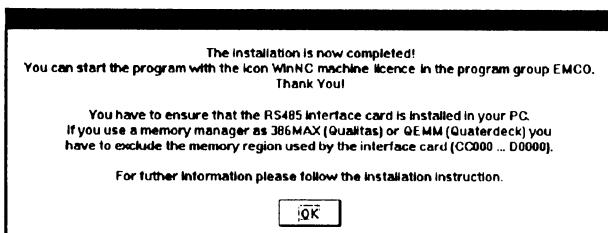
WINDOWS 95 asks whether WinNC should start automatically. The file AUTOEXEC.BAT will not be altered but a link to the AUTOSTART group will be created.

Only machine version:

- The installation program can alter the file AUTOEXEC.BAT for that after switching on the PC WinNC will start automatically.
- Select "YES" if WinNC should start automatically.
- Select "NO" if WinNC should not start automatically

Note:

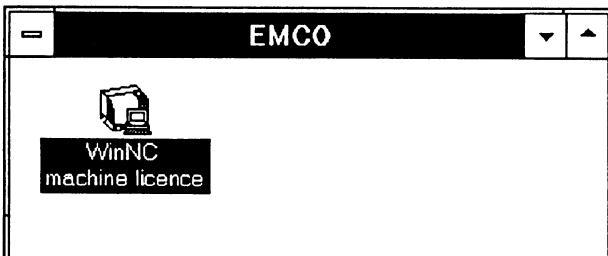
When there is already a call-up of WINDOWS in the file AUTOEXEC.BAT you must remove it before to be able to start WinNC automatically.



Example: End information machine version

All versions:

- The installation program is finished.
- Read the end information and acknowledge with "OK".
- The installation program will be closed.



Example: Icon for WinNC machine version

All versions except net server:

- The program group for WinNC is displayed in the program manager.
- Doubleclick on the icon for WinNC to start WinNC.

Net server:

- If you have installed a net server you have to install the working stations now.

Notes for Network Installation

**Note:**

A network installation is possible with the multiple licence version only.

The network installation must be done by the network administrator (supervisor) only.

WinNC is installed at the server once.
The users have their own (private) directories for workpiece programs and setting data.
The directories for workpiece programs and setting data must be set for every network user separately.

First install WinNC at the server and then at the working stations.

In the installation program "network installation" must be selected and entered whether a server or a working station should be installed.

1. Way

Entering a local directory for the local data of every net working station.

(e.g.: C:\WINNC)

- In the system loginscript (for Novell) a free drive letter must be mapped on the directory.
e.g.: letter H is free:
`map ROOT H:=SYS\USERS\%LOGIN_NAME`
(for Novell nets)
- On that directory in which WinNC is installed at the network server, every user must have execution rights.
- WinNC can be installed at the server now. As directory for local net working station the corresponding user-specific drive letter must be entered
(e.g.: H:\WINNC).
- With the now following installation on the working stations the directory must be entered in which WinNC is installed at the server.

2. Way

This is the most flexible way of network installation. The local data of every net working station also will be stored at the server.

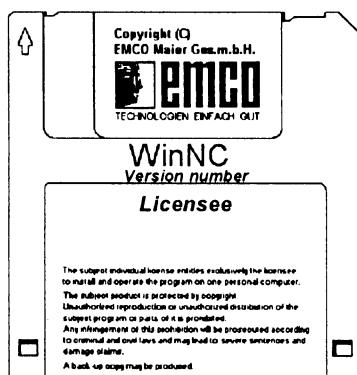
- Create directory for the user:
If not done until now (as usual in the most networks) a directory for every user must be set up, in which only the user has rights.
e.g.: SYS\USERS\USER1
 SYS\USERS\USER2 (Novell)

Starting WinNC

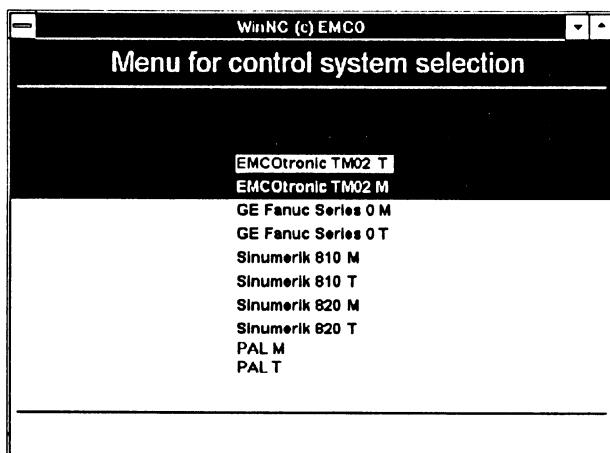
If you have selected "YES" for the last query in the machine version installation (entry in the file AUTOEXEC.BAT), WinNC starts automatically after switching on the PC.

Otherwise act as following:

- Switch on the PC and start Windows.
- The program manager shows the program group for WinNC.
- Start WinNC by doubleclicking on the icon for WinNC.
- The screen shows the start picture. In the start picture the version number of WinNC and the licensee are displayed.



Start picture for WinNC



Selection of the CNC control type

- If you have installed one control type only, it will start immediately.
- If you have installed several control types, the screen shows the selection menu beside.
- Select the desired control type and press ENTER to start it.
- If you use the control keyboard, select the desired control type with the JOG keys and start it with NC-Start .

Closing WinNC

By similar pressing the keys "Alt" and "F4" (PC keyboard) or the keys and (option control keyboard) the control system will be ceased and you are back in the selection menu for the control types.
Press Alt+F4 again to close WinNC.

Mistakes with Installation of the Software

With installation of the software a certain memory area (**memory area CC000 - D0000**) is assigned to the interface card.

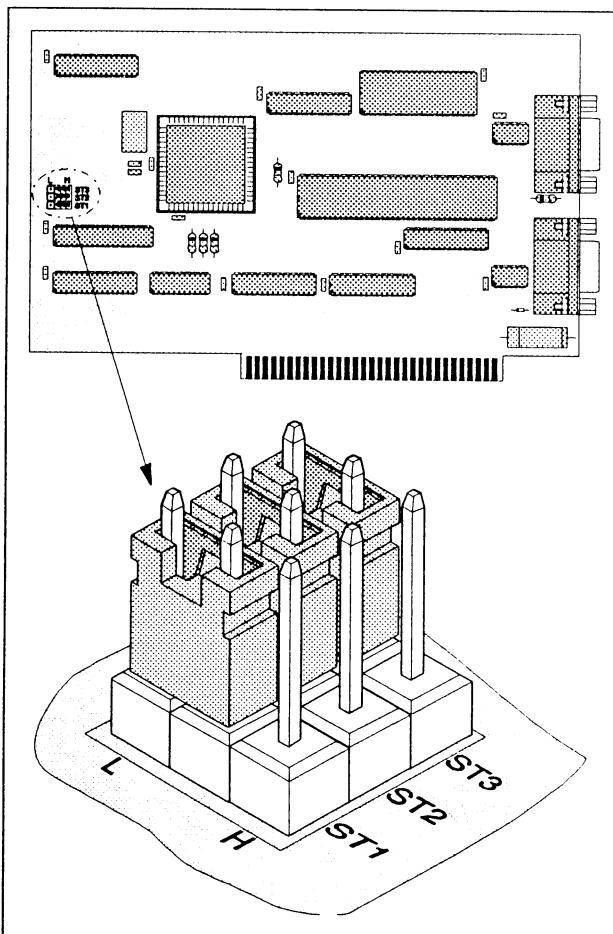
If this area is already occupied, e.g. by another card or an Expanded Memory Manager, the following alarm appears:

"2523 ORDxx INIT error on RS485 PC-board"
"983: RS485 ERROR - INITIALIZING PC BOARD"

After this alarm check the following items:

PC Configuration Mounting of the Interface Card

Make sure that your PC matches to the required minimum configuration (see PC configuration). Check also the correct mounting of the interface card in your PC (see in the machine manual "Installation of the Machine").



Position of the jumpers on the interface card

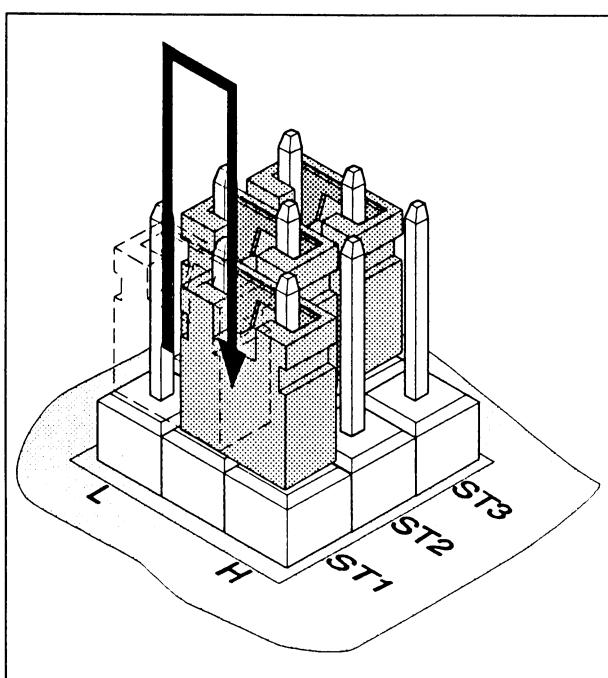
Alter jumper positions

By altering the jumpers the interface card can be switched to another memory area.

The following memory areas are available:

Nr.	Jumper			Hexadecimal Memory Area	
	ST1	ST2	ST3		
1	L	L	L	CC000	- CC7FF
2	H	L	L	CC800	- CCFFF
3	L	H	L	CD000	- CD7FF
4	H	H	L	CD800	- CDFFF
5	L	L	H	CE000	- CE7FF
6	H	L	H	CE800	- CEFFF
7	L	H	H	CF000	- CF7FF
8	H	H	H	CF800	- CFFFF

*) initial position

*Alter the jumper position***Sequence:****Safety regulation:**

Mount and dismount the interface card only while the PC is disconnected to the net.
Pull power cable!

- Dismount the interface card.
- Put ST1 - ST3 into the desired positions (positions 1 - 8 see table on previous page).
- Mount the interface card into the PC.
- Connect the PC to mains and switch it on.
- Try to install the software again.
If the alarm occurs again, try installation again with another jumper setting.

Expanded Memory Manager

If you use an Expanded Memory Manager on your PC, which allows to use the memory area above 640 kB, the memory area from CC000 to D0000 has to be excluded for usage.

Therefore read the manual of your Memory Manager.

**Note:**

When you use the Memory Manager "emm386" (DOS 5.0 or higher) on your PC, the required memory area will be excluded automatically with installation.

Address Conflict with Another PC Card

If further cards are mounted in your PC, and you still cannot install the software, there is possibly a conflict with another PC card which requires the same memory area.

Set the PC card which causes the address conflict to another memory area (see the manual of the respective PC card).

If the change-over is not possible the PC card has to be dismounted.

External Input Devices

By using the EMCO control keyboard (option) or a digitizer tablet (option), EMCO WinNC, WinCTS can be operated in a very comfortable and similar-to-the-original-control way and gets a didactically higher level.

NOTE

When you use an interface expansion card for the digitizer or the EMCO control keyboard (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).

Digitizer Tablet

A digitizer can be connected direct to COM 1 / COM 2, if it supports the format of the "Summa-graphics MM Series".

The digitizer must support directly the Summa-graphics MM format, an emulation is not sufficient. The digitizer will be operated directly via this command interface, no drivers of the manufacturer are necessary.

Recommended digitizer tablets:

- **GRAPHTEC KD 4320**

DIP switch settings:

	1	2	3	4	5	6	7	8
SW1	OFF	ON	OFF	OFF	OFF	OFF	OFF	OFF
SW2	OFF	ON	OFF	ON	OFF	ON	ON	OFF

- **GENIUS HiSketch 1212**

DIP switch settings:

	1	2	3	4
	OFF	OFF	OFF	OFF

- **SummaSketch III**

Setup of the Digitizer Tablet

Before the first use of the digitizer the points of the digitizer overlay have to be defined.

- Fix the digitizer overlay on the digitizer tablet. The frame of the drawing has to be parallel to the edges of the digitizer working area.
- Move the pen or the mouse into the overlay drawing and press pen tip + pen button or left + right mouse button for min. 5 sec.. The beep sound indicates correct input.
- Click (pen tip or left mouse button) first on the left upper and than on the right lower reference point (●). The beep sound indicates correct input.

Now the digitizer is set up.

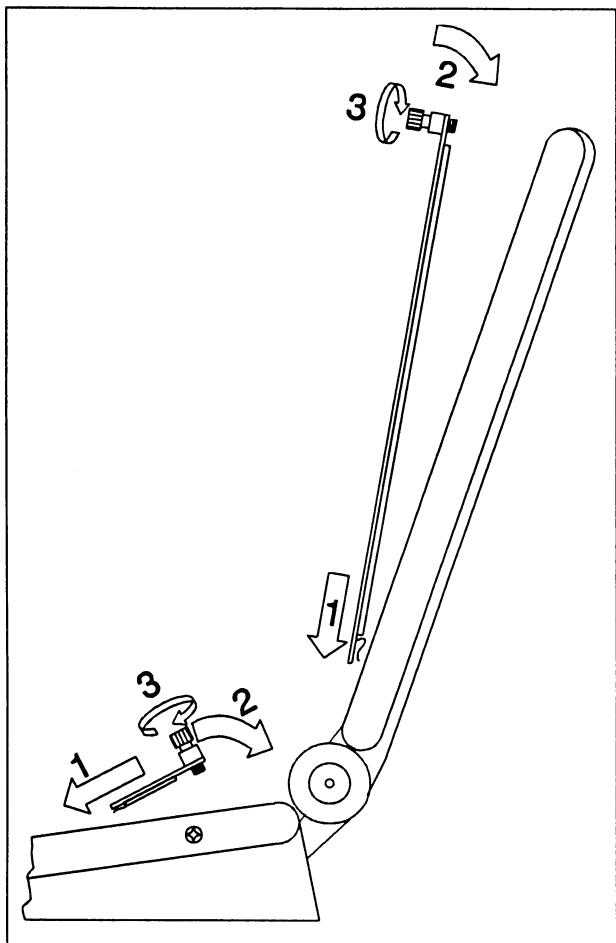
EMCO Control Keyboard

Scope of Supply

The scope of supply for a complete control keyboard consists of three parts:

Basic case
Key module
Power supply

Ref. No.	Description
795 000	Basic case Including 2 cables (RS 232 and RS 485) for connection to the PC.
795 010	Key module SINUMERIK 810 2 key sheets with keys 1 package exchange keys
795 020	Key module SINUMERIK 820 2 key sheets with keys 1 package exchange keys
795 110	Key module FANUC 0 2 key sheets with keys 1 package exchange keys
795 210	Key module EMCOTRONIC TM02 2 key sheets with keys 1 package exchange keys
795 510	Key module PAL 2 key sheets with keys 1 package exchange keys
795 700	Power supply 230 V
795 710	Power supply 115 V

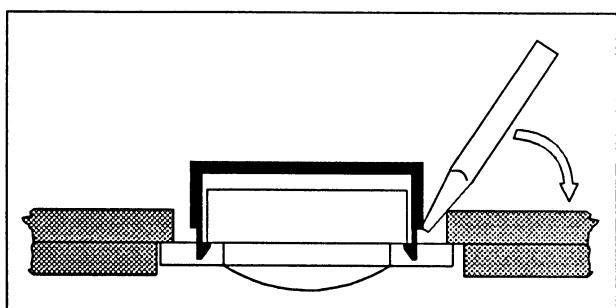


Assembling

- Place the correseponding key sheet with the clips in the basic case (1).
- Pull the key sheet into the basic case, it must be insertet plainly (2).
- Fix the key sheet with the two knurled screws (3).

Note

The key sheets must not be bended, otherwise the switching function can not be warranted.



Exchange of single key caps

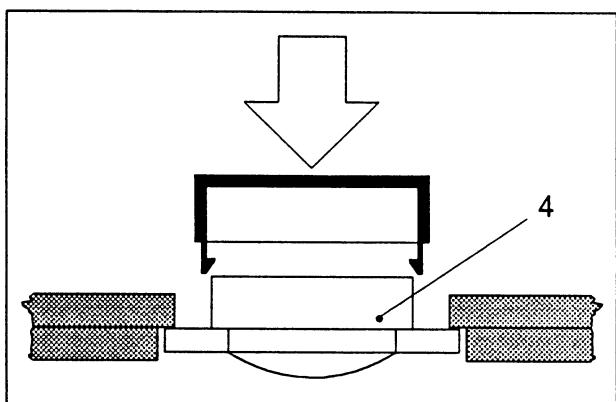
Off works the keyboards are equipped with the keys for turning.

The scope of supply includes a package of exchange key caps to equip the keyboard for milling.

If you want to use the control keyboard for milling, you have to exchange a part of the key caps. Exchange them as shown on the following pages.

Take off

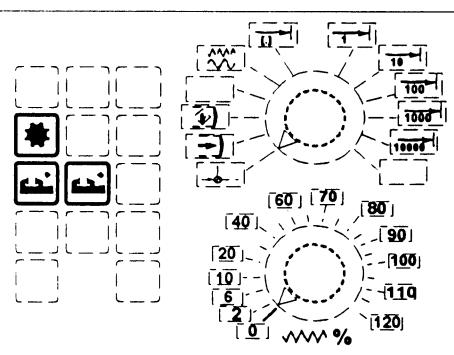
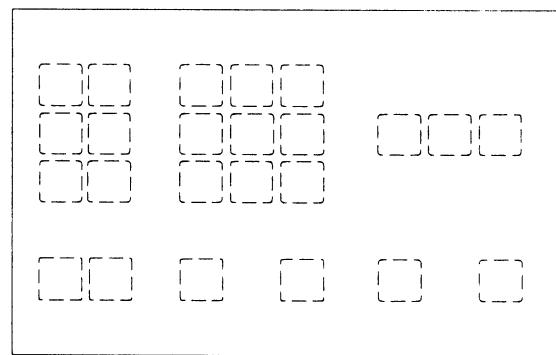
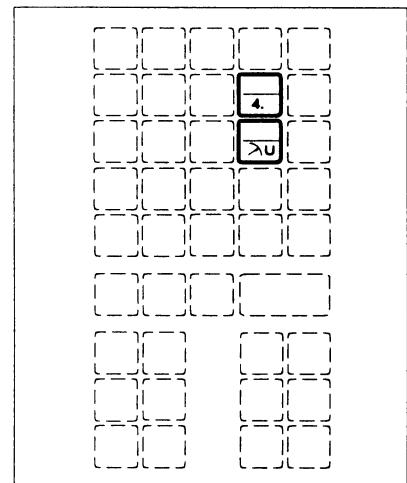
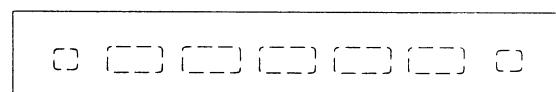
Pull out carefully the key caps to be exchanged with a fine screw driver or a knife.



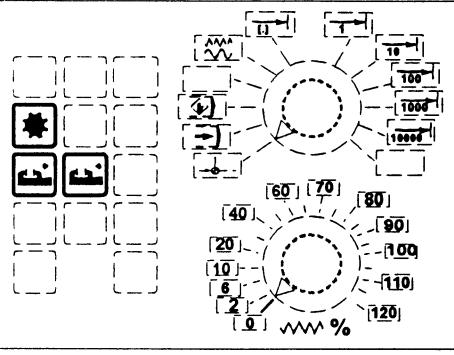
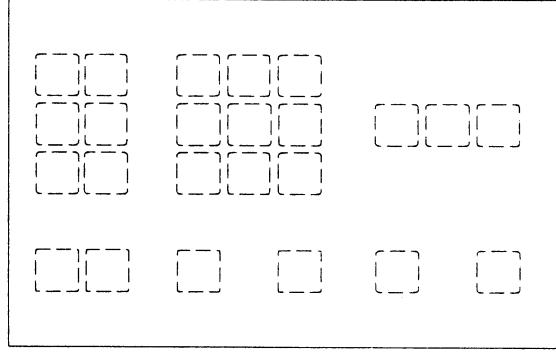
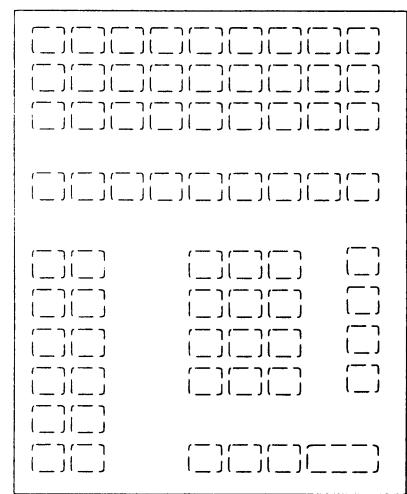
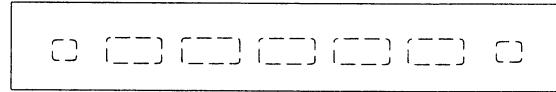
Clip on

Move the key body (4) in the middle of the recess. Push the key cap vertically down onto the key body, until the key cap snaps in tactly.

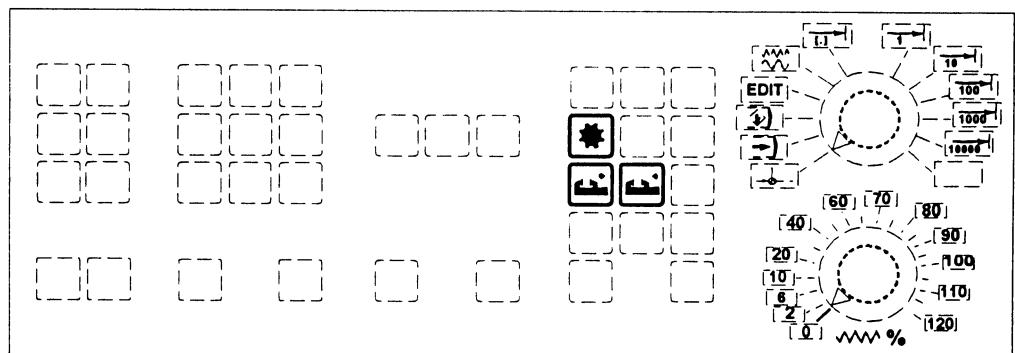
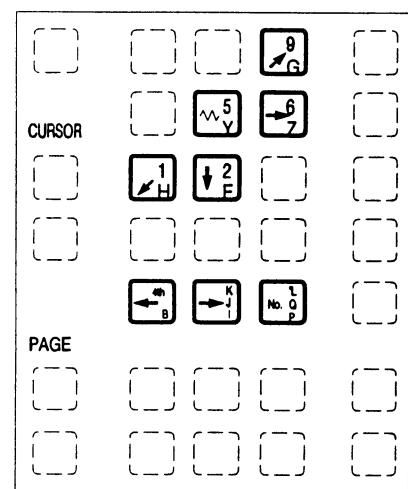
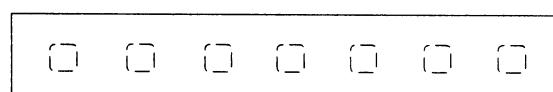
SINUMERIK 810M
Exchange key caps
for milling



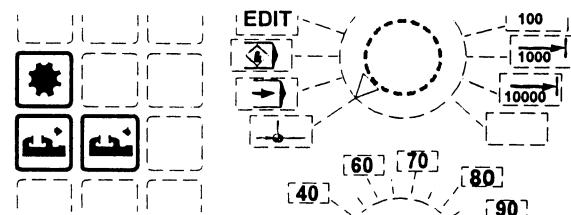
SINUMERIK 820M
and
PAL M
Exchange key caps
for milling



FANUC 0M
Exchange key caps
for milling



EMCOTRONIC M2
Exchange key caps
for milling



Power Supply

The control keyboard is supplied with 9 - 14 V, AC or DC.

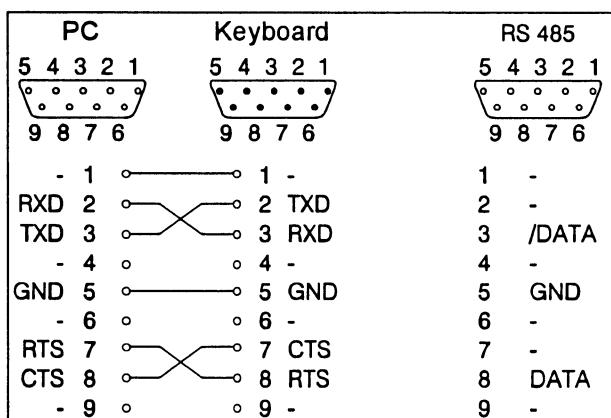
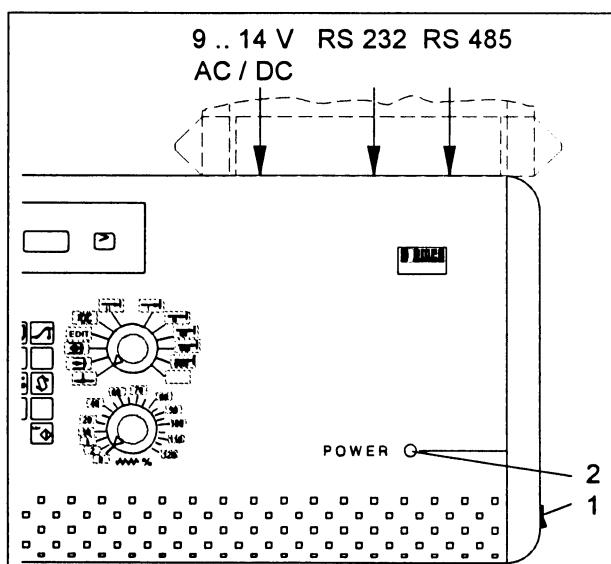
The poles of the power supply can be interchanged, the polarity need not to be considered.

The power supply must be able to deliver at least 250 mA.

The connector is a 5/2.5 mm female cinch jack to be plugged in at the backside of the control keyboard.

Notes for power supply 230 V, Ref. No. 795 700:

- Use the jack adaptor with the green point.
- Set the voltage selector switch to 12 V.

*Pin occupation of the interfaces*

Connection to the PC

The control keyboard can be connected to the PC via RS 485 or RS 232.

Note

If you use the control keyboard in combination with a turning or milling machine one socket at your PC RS 485 board is available. Use this socket to connect it to the RS 485 interface of the control keyboard. If no RS 485 board is built in in your PC use the RS 232 interface (COM1 or COM2) of your PC.

The PC connectors are at the backside of the control keyboard.

The RS 485 connector is outside and is a 9 pole female socket.

The RS 232 connector is inside and is a 9 pole male socket.

Use the corresponding cable to connect the control keyboard to the PC.

Main Switch

The main switch (1) is on the right side of the control keyboard.

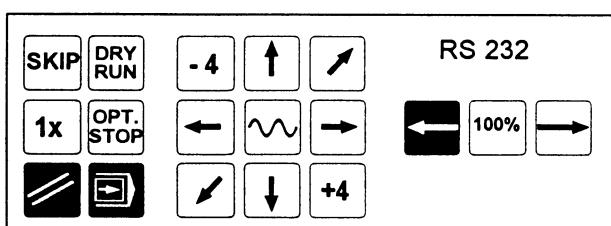
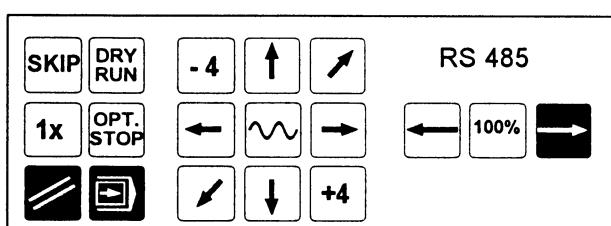
The ON status is displayed by a control lamp (2) on the control keyboard.

Activating the Interface

To activate the selected interface 3 keys on the control keyboard must be pressed at the same time for at least 1 second.

Activating the RS 485 interface with the black displayed keys.

Activating the RS 232 interface with the black displayed keys.



Note:

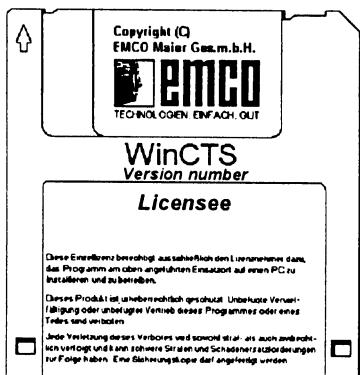
When the control keyboard is connected to the PC via RS232, switch on the PC first and then switch on the control keyboard, otherwise communication problems could occur and the control keyboard would not work.

Start WinNC, WinCTS with the Control Keyboard

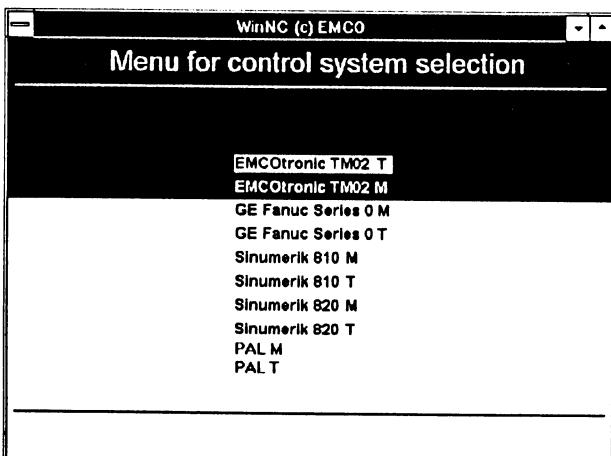
If you have selected "YES" for the last query in the machine version installation (entry in the file AUTOEXEC.BAT), WinNC starts automatically after switching on the PC.

Otherwise act as following:

- Switch on the PC and start Windows.
- The program manager shows the program group for WinNC, WinCTS.
- Start WinNC, WinCTS by doubleclicking on the icon for WinNC, WinCTS.
- The screen shows the start picture. In the start picture the version number of WinNC, WinCTS and the licensee are displayed.

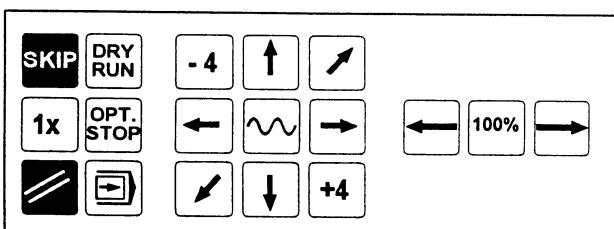


*Start picture for WinNC,
WinCTS*



Selection of the CNC control type

- If you have installed one control type only, it will start immediately.
- If you have installed several control types, the screen shows the selection menu beside.
- Select the desired control type and press ENTER to start it.
- Select the desired control type with the JOG keys and and start it with NC-Start .



Cancelling WinNC, WinCTS with the Control Keyboard

The PC software can be cancelled by pressing the two black displayed keys similarly for at least 1 second.,

This is the same function as Alt+F4 at the PC keyboard.