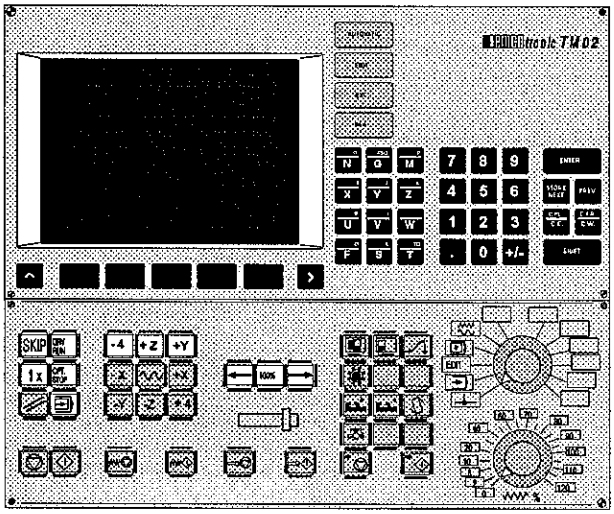
EMCO WinNC EMCOTRONIC M2

Software description/ Software version from 3.34



Software description
EMCO WinNC EMCOTRONIC M2

Ref.No. EN 1800 Edition G1996-10

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

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

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

Internet: www.emco.at

E-Mail: service@emco.co.at

emco
industrial training systems

Preface

The software WinNC EMCOTRONIC M2 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 WinNC for the EMCO PC MILL 50/100 the milling machines EMCO PC MILL 50 and EMCO PC MILL 100 can be controlled directly by the PC keyboard.

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

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

- Instruction manual
- Trainers guide manual
- Overhead slides

Software Description
EMCO WinNC

valid from Version 13

Machine licence version for the machines
EMCO PC TURN 55, EMCO PC TURN 125, EMCO PC MILL 55, EMCO PC MILL 125

Differences to version 3:

- WinNC Version 13 is used to control the machines EMCO PC TURN 55 and EMCO PC TURN 125 as well as EMCO PC MILL 55 and EMCO PC MILL 125.
  The machines EMCO PC TURN 50, EMCO PC TURN 120, EMCO PC MILL 50 and EMCO PC MILL 100 are still controlled WinNC Version 3.
- Operating WinNC Version 13 is the same as operating Version 3.
  See software description EMCO WinNC Version 3.
- WinNC Version 13 is a true 32 bit software and runs under WINDOWS 95.
- The machines of the series 55/125 have a considerable improved dynamic of the drives, additionally computing performance was transferrred from the machine into the PC. For this reason a higher PC minimum configuration is necessary.
- With WinNC Version 13 the EMCO control keyboard can be connected to the RS232 interface only.
- The accessories can be activated with WinConfig only.
- In WinConfig for WinNC Version 13 the used interrupt can be set (see "Problems with software installation").
- Additional alarms.
- Altered installation routine.

Contents

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

PC Minimum Configuration

For the installation of EMCO WinNC Version 13 the PC must have the following minimum configuration:

- PC Pentium 75 IBM-compatible
- 8 MB RAM
- 20 MB free hard disk memory for all control types
- 3,5" disk drive 1,44 MB
- VGA board
- VGA color screen
- Operating sytem Windows95

Software Installation

Note

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.

- Switch on the PC.
- Start Windows95.
- Insert the installations disk into drive A.
- Click on "Start".
- Select "Run".
- Enter "a:\setup" in the command line.
- Confirm with "OK" (Click or ENTER).
- The installation routine will be started.
- The installation is guided by menues. Carry out the single items step by step.

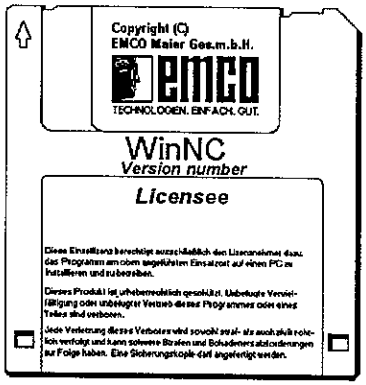
Note:
Version 13 and version 3 must not be installed in the same directory.

Starting WinNC

PC TURN 55, PC MILL 55:
When the last query of the installation (Entry in AUTOSTART) was ticked "YES", WinNC starts automatically after starting the PC.

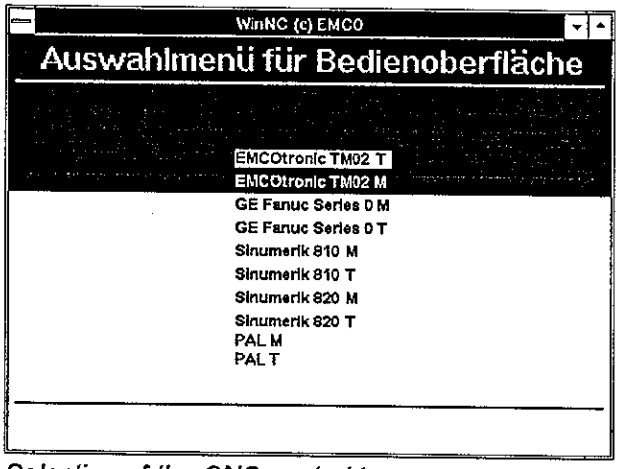
Otherwise act as following:
- Switch on the PC and start Windows95.
- Start EMCO WinNC by double-click on the icon for WinNC.

The screen shows the starting picture. Im Startbild sind die Versionsnummer von WinNC sowie der Lizenznehmer eingetragen.



Starting picture for WinNC

PC TURN 125, PC MILL 125:
WinNC starts automatically.



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 (cursors or mouse) 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 [SKIP] 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.

Problems with Software Installation

PCCOM board

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

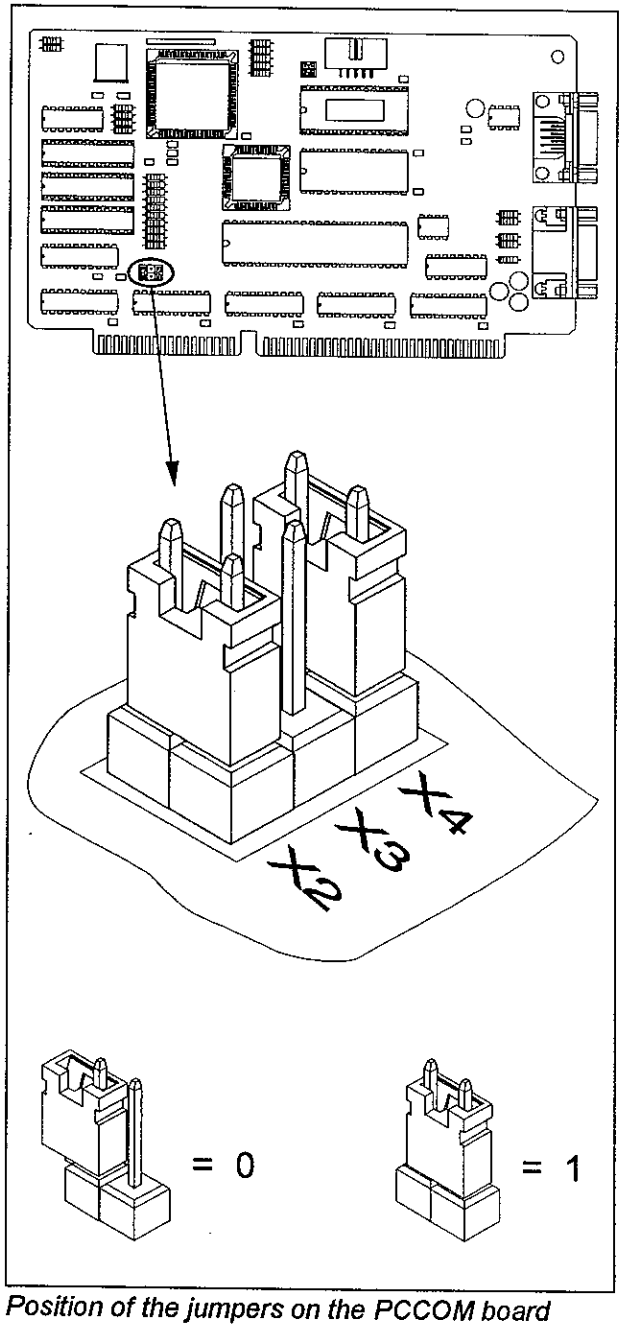
If this area is already occupied, e.g. by another card or an Expanded Memory Manager, the following alarm appears:

8106 No PC-COM board or
8107 PC-COM does not react

When such an alarm appears, act as following:

PC Configuration
Mounting of the Interface Card

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



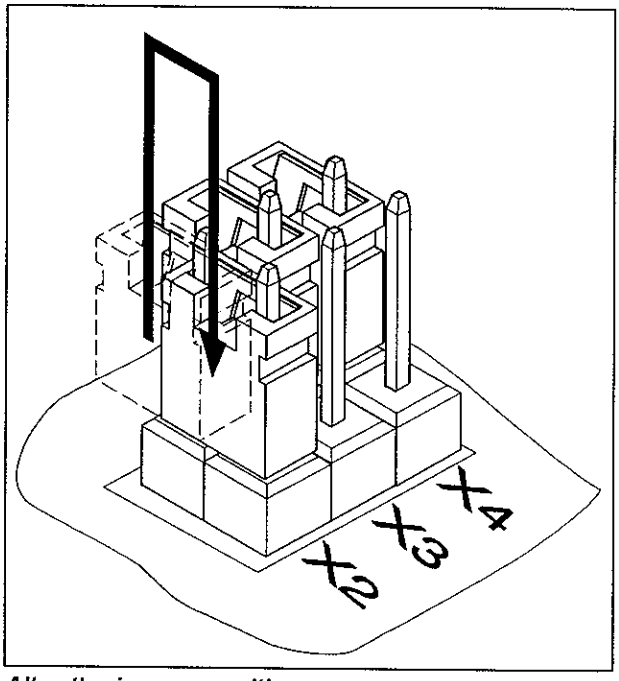
Position of the jumpers on the PCCOM board

Alter jumper positions

By altering the jumpers the interface card can be switched to another memory area.
The following memory areas are available:

	No.	Jumper			Hexadecimal memory area		
		X4	X3	X2			
*	1	0	0	0	CC000	to	CC7FF
	2	0	0	1	CC800	to	CCFFF
	3	0	1	0	CD000	to	CD7FF
	4	0	1	1	CD800	to	CDFFF
	5	1	0	0	CE000	to	CE7FF
	6	1	0	1	CE800	to	CEFFF
	7	1	1	0	CF000	to	CF7FF
	8	1	1	1	CF800	to	CFFFF

*) Initial position



Alter the jumper position

Procedure:

Danger:

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

- Dismount the interface card.
- Put the jumpers into the desired positions (positions 1 - 8 see table on previous page).
- Mount the interface card into the PC.
- Close the PC cover.
- 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, an 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.

Problems with starting WinNC

The PCCOM board is controlled by an so-called interrupt.
While Installation WinNC tries automatically to find a free interrupt.
In exceptional cases problems can occur while starting WinNC e.g.:

- System crash
- Mouse does not work
- Sound board does not work
- Network board does not work etc.

When these problems occure after installing WinNC, you have to select an other interrupt for the PCCOM board.

Procedure

1.Find a free Interrupt:

- Click on the WINDOWS95 Start-Button.
- Click on SETTINGS.
- Click on SYSTEM CONTROL.
- Double-click the icon SYSTEM.
- Click on DEVICE MANAGER.
- The symbol "Computer" is highlighted. Click on the button Attributes.
- The used interrupts are displayed (e.g.: 00 - System timer, 01 - Keyboard etc.).

  The free intrerupts are not displayed.
- For the EMCO PCCOM board you can use the following interrupts:

  05, 07, 10, 11, 12, 03, 04 and 15
- Look in the interrupt list at the PC for one free interrupt of the numbers mentioned above (preferred sequence: 05, 07, 10, 11, 12, 03, 04 and 15) and note the number of this interrupt.

  Note:
  You also may use interrupts which are occupied with printer LPT1 or LPT2 if no other device points on it.
- Select Cancel and close the WINDOWS System control.

2.Enter free interrupt in WinConfig:

- Start WinConfig.
- Select the interrupt number noted before in "General MSD-Data" (selection box).
- Click on the sysmbol "Save".
- Close WinConfig
- Restart WINDOWS.

Aktivating Accessory

From WinNC Version 13 on, activating of accessories is possible in WinConfig only.
The settings in the Setting Bits or in the user monitor are no longer effective.

Additional Alarms

8101 Fatal init error AC
Cause: Internal error
Remedy: Restart software or reinstall when needed, report error to EMCO.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

8106 No PC-COM card found
Cause: PC-COM board can not be accessed (ev. not mounted).
Remedy: Mount board or select other address area with jumpers

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

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

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM Programmpaketquittung fehlt
see 8110.

8116 PC-COM startup error
see 8110.

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

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

8119 PC interrupt no. not valid
Cause: The PC Interrupt number can not be used.
Remedy: Find free interrupt number in Windows95 System control (allowed: 5,7,10, 11, 12, 3, 4 and 15) and enter this number in WinConfig.

8120 PC interrupt no. unmaskable
see 8119

8121 Invalid command to PC-COM
Cause: Internal error or cable defective
Remedy: Check cable (fix plug with screws); restart software or reinstall when needed, report error to EMCO.

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

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

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

8125 Invalid memory for record buffer
Cause: Insufficient RAM, record time too long.
Remedy: Restart software, when needed remove drivers etc. to get free memory, reduce record time.

8126 AC Interpolation overrun
Cause: Ev. insufficient computing performance.
Remedy: Select longer interrupt time with WinConfig. This can cause lower path accuracy.

8127 Insufficient memory
Cause: Insufficient RAM
Remedy: Close other running programs, restart software, when needed remove drivers etc. to get free memory

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

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8131 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The axes combination of linear and circular axes is invalid.
Remedy: Correct program.

8140 Maschine (ACIF) not responding
Cause: Machine not connected or power off.
Remedy: Connect resp. switch on machine.

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

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

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

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

8146 Multiple request for axis
see 8142.

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

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

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

8150 Fatal ACIF error
see 8142.

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

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

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

8155 Invalid FPGA packet acknowledge
see 8142 resp hardware error on ACIF board (call EMCO service).

8156 Sync within 1.5 revol. not found
see 8142 bzw. resp hardware error at Bero (call EMCO service).

8157 Data record done
see 8142.

8158 Bero width too large (referencing)
see 8142 bzw. resp hardware error at Bero (call EMCO service).

8159 Function not implemented
Meaning: This function can not be executed in standard operation.

8160 ORDxx DAC limit axis 4..7
Cause: Axis spins or slide is locked, axis synchronisation was lost
Remedy: Approach reference point.

8164 ORDxx +SW overtravel switch a3..a7
Cause: Axis at the end of traversing area
Remedy: Traverse axis back

8168 ORDxx -SW overtravel switch a3..a7
Cause: Axis at the end of traversing area
Remedy: Traverse axis back

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when needed, report error to EMCO.
Check connection PC-Machine, ev. eliminate source of distortion.

8173 INC while NC program is running

8174 INC not allowed

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software or reinstall when needed, report error to EMCO.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

8182 Gear change not allowed

8183 Gear too high

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message

8191 Invalid jog feed unit

8192 Invalid axis in command

8193 Fatal PLC error
see 8175.

8194 Thread without length

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

8196 Too many axis for thread
Remedy: program max. 2 axes for thread.

8197 Thread not long enough
Cause: Thread too short.
For the transition from one thread to the next the length of the secont thread mut be sufficient to produce a correct thread.
Remedy: Make the second thread longer or replace it by a straight piece (G1).

8198 Internal error (to manny threads)
see 8175.

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

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8202 Internal thread error (IPO)
see 8199.

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

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

8205 PLC Runtime exceeded
Cause: Insufficient computing performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

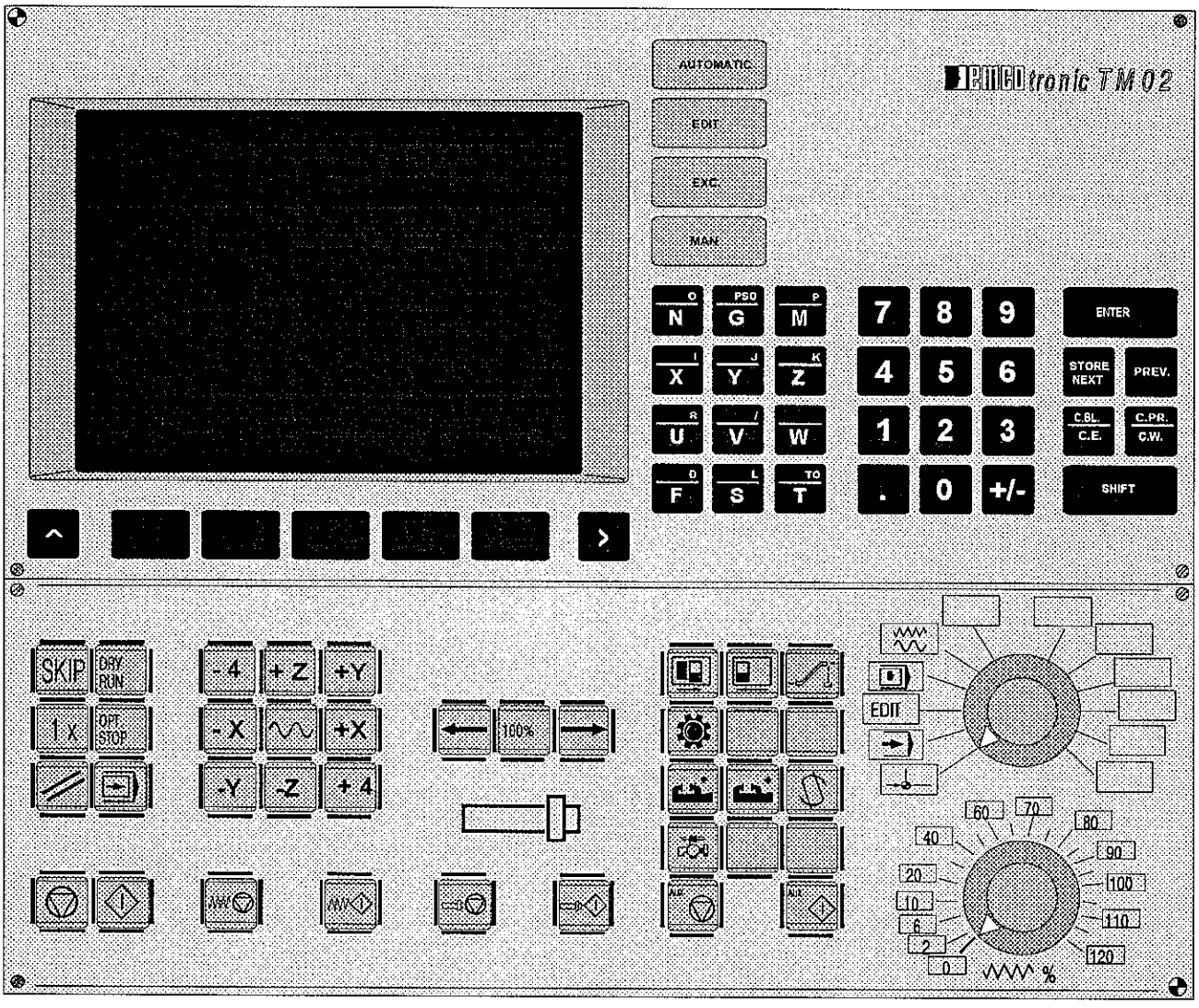
8208 Invalid application message
see 8199.

8211 Feed too high (thread)
Cause: Thread pitch too high / missing, feed for thread reaches 80% of rapid feed
Remedy: Correct program, select lower pitch or slower spindle speed for thread.

Contents

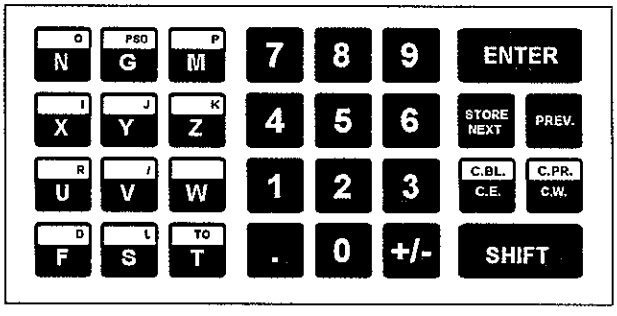
Chapter A: Key Description

Control Keyboard, Digitizer Overlay



Key Functions

MAN	MANUAL mode
EDIT	EDIT mode
EXC.	EXECUTE mode
AUTOMATIC	AUTOMATIC mode
SHIFT	SHIFT key
ENTER	Enter key
STORE NEXT	Store / next line
PREV.	Previous line
C.E.	Clear entry
C.BL.	Clear block
C.W.	Clear word
C.PR.	Clear program



Address and number keys on the control keyboard and on the digitizer overlay

Address and Number Keyboard

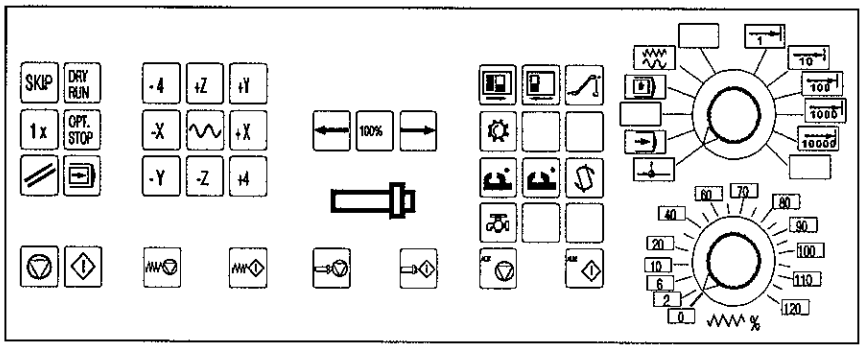
With the SHIFT key at the bottom right the second key function can be selected. Pressing SHIFT again switches back to the first function.

N	function N
SHIFT + N	function O

In this manual the keys are shown without SHIFT, e.g.: N and O.

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.
The mode selection switch is not active on the EMCOTRONIC.



Machine control key block

SKIP (skip blocks will not be excuted)

DRY RUN (test run of programs)

OPT STOP (program stop at M01)

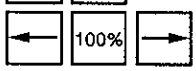

RESET


single block machining


program stop / program start

manual axis movement


feed stop / feed start



spindle speed decrease/100%/increase

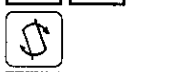

spindle stop / spindle start; spindle start in the mode MANUAL:

clockwise: press key  short, Counterclockwise: press key  for at least 1 sec.


open / close door

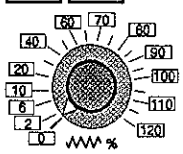

swivel dividing head


open / close clamping device


swivel toolholder

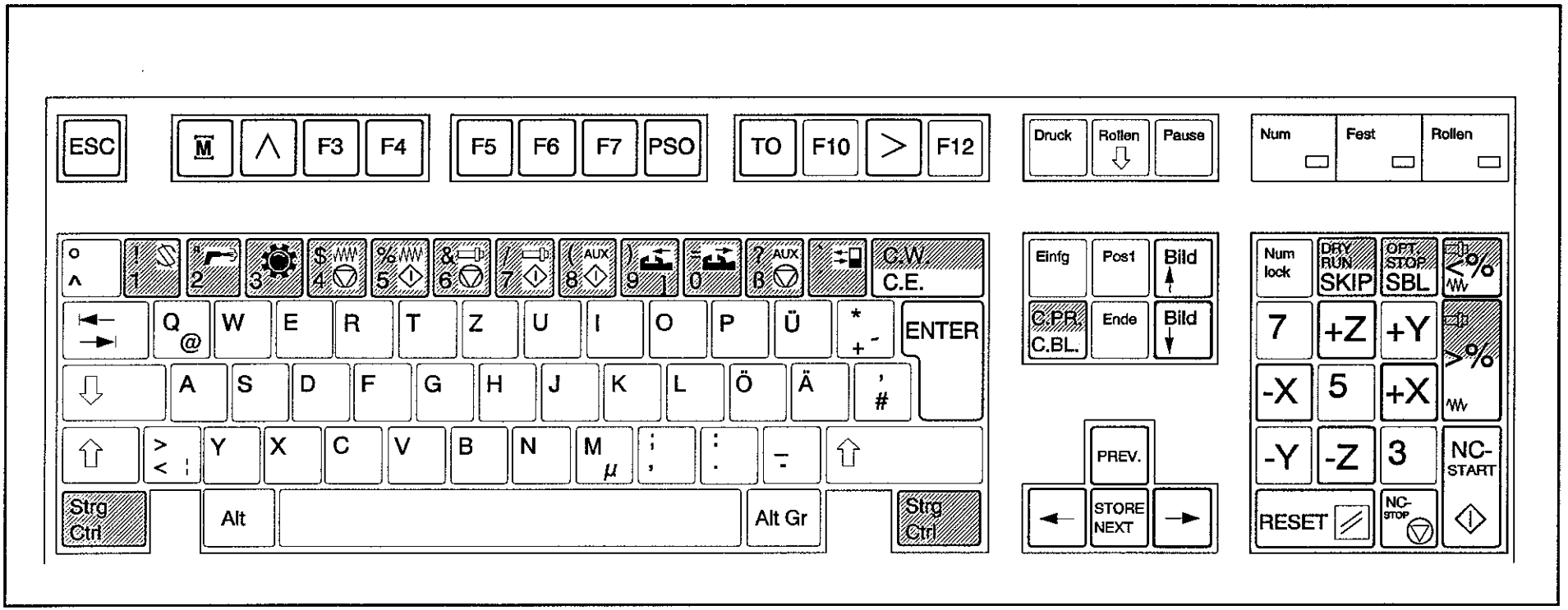

coolant / puff blowing on / off

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.

The ESC key quits some alarms.

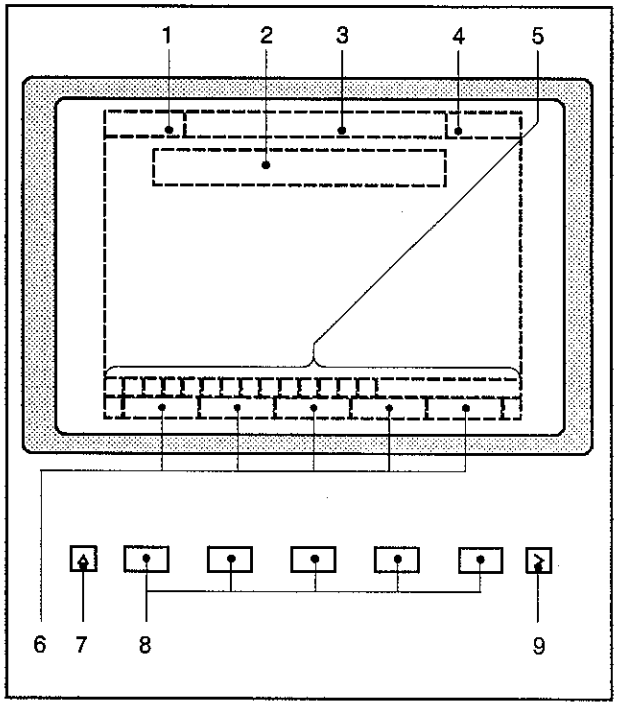
The key F1 shows the modes (AUTOMATIC, EDIT, ...) in the softkeyline.

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

The assignement of the accessory functions is described int the chapter "Accessory Functions".

NUM lock must be inactive for using the machine control functions in the numeric key block.

Screen with Softkeys



Screen with softkeys

At the screen the following areas are determined:

1 Mode display

2 Display of supervising number, messages and alarms

3 Display of control status (SKIP, DRYRUN, INTERFACE, GRAPHIC, ...)

4 Measuring unit display

5 Machine status display

6 Softkey function display

7 Key "jump back to a higher level menu" (key F2 at the PC)

8 Softkeys (keys F3 to F7 at the PC)

9 Taste "further functions in the same menu" (key F11 at the PC)

Softkeys (8) are keys with multiple meanings. The valid meaning will be displayed at the bottom line (6).

Chapter B: Operating Sequences

Survey Modes

MANUAL

With the jog keys -X +X -Y +Y -Z +Z you can traverse the tool manually.
With the corresponding keys you can

- switch on/off the spindle
- swivel the tool holder
- operate the accessories

EXECUTE

You can enter program blocks in the input line.

With the control works off the entered block and deletes the input line for a new entry.
You can also call up a block in a program and work it off.

EDIT EDIT

- Selection and input of programs
- Input and altering of data in the tool offset register (TO)
- Input and altering of data in the position shift register (PSO)
- Access to the user monitor

AUTOMATIC

For working off a part program the control calls up the blocks one by one and proceeds them.
This proceeding includes all correction which are activated by the program.
The blocks proceeded in that way will be worked off one by one.

Survey Submodes

REFERENCE

MANUAL
The reference point will be approached in this submode.

With reaching the reference point the actual position store will be set to the reference point coordinates. By that the control acknowledges the position of the tool in the working area.

With the following situations the reference point must be approached:

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

STATUS

MANUAL, EXECUTE
At the screen the active

- G commands
- M commands
- Speed
- Feed
- Spindle override
- Feed override

will be displayed.

DISKETTE PORT
RS232 PORT
PARALLEL PORT

EDIT
This submodes are used for sending and receiving data.

SINGLE BLOCK
DRY RUN
SKIP

AUTOMATIC
This submodes are for influencing the program run.

Approach Reference Point

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

- Select MANUAL mode.
- Press the softkey REFERENCE.
- With the key  the reference point will be approached in all axes.
- With entering an axis address (e.g. X) and the key  the reference point can be approached in a single axis.

Danger of Collisions

Take care of obstacles in the working area (clamping devices, clamped workpieces etc.).

After reaching the reference point the position of the reference point will be displayed at the screen as actual position. Now the control is synchronized with the machine.

At the screen the software version of WinNC and the eventually connected RS 485 devices will be displayed.

Set Language

Selection of installed languages, the selected language will be activated with restart of the software.

Setting in the user monitor

- EDIT mode
- Access to the user monitor

  M 0 N ENTER
- Select parameter T22

  T 2 2 ENTER
- Enter language (delete old value with C.E.)

  0 for English
  1 for German
  2 for French
  3 for Spanish

Program Input

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

Call Up an Existing or New Program

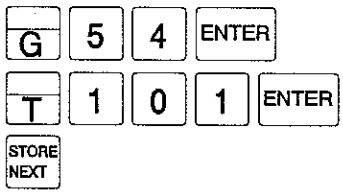
- Select EDIT mode
- Enter program number O...
- Press key ENTER
- If the program already exists the screen shows "O.... FOUND"
- If the program does not exist the screen shows "O.... NEW"

  Press key ENTER to confirm "NEW", "NEW" disappears at the screen
- Press the key STORE NEXT to enter the program
- Now you can work with the program

Block Input

Example:
The screen shows "N0000 NEW".

Press key ENTER to take over N0000 or enter a new block number and press ENTER.



Enter 1. word

Enter 2. word

With STORE NEXT the entered words will be written into the program, ath the same time the next block number will be supposed (jumps by ten).

The screen shows "N0010 NEW".

Press key ENTER to take over N0010.

etc.

Jump Forward/Backward Blockwise

Jump forward with the key STORE NEXT, backward with PREV.

Select Block

Enter the block number N.... and press ENTER. The selected block will be displayed.

Delete Block

Select the desired block (e.g.: N110 ENTER) and press the key C.BL.

Insert Block

Between the jumps 10 by 10 further blocks can be inserted, e.g.:

N101 ENTER (enter block number)

At the screen "N0101 NEW" will be displayed.

ENTER (confirm block number)

STORE NEXT (Block N0101 will be stored in the program between N100 and N110.)

Alter Block Number

Select the block to be altered, e.g.:

N0101 ENTER

Press the key C.E. as often, until all digits of the block number are deleted.

Enter the new block number and press ENTER.

Jump Forward/Backward Wordwise

Jump forward with the key ENTER, jump to the first word in the block with SHIFT ENTER.

Select Word

A block must be selected. Enter the address of the word (e.g. X) and press ENTER. The cursor jumps on the desired word.

Delete Word

Select the desired word (e.g.: X ENTER) and press the key C.W.. With STORE NEXT the new block contents will be taken over into the program.

Insert Word

Set the cursor on that word, that should be before the inserted word and enter the new word (address and value) and press ENTER. With STORE NEXT the new block contents will be taken over into the program.

Alter Word

Select the word to be altered (e.g.: X ENTER). Enter the new value and press ENTER and STORE NEXT.

Delete Entry

With this function you can alter entries before they were confirmed with ENTER or STORE NEXT.

Press the key C.E. to delete a number digit by digit.

Wrong entered addresses you can alter by entering the correct address.

Mark Skip Blocks

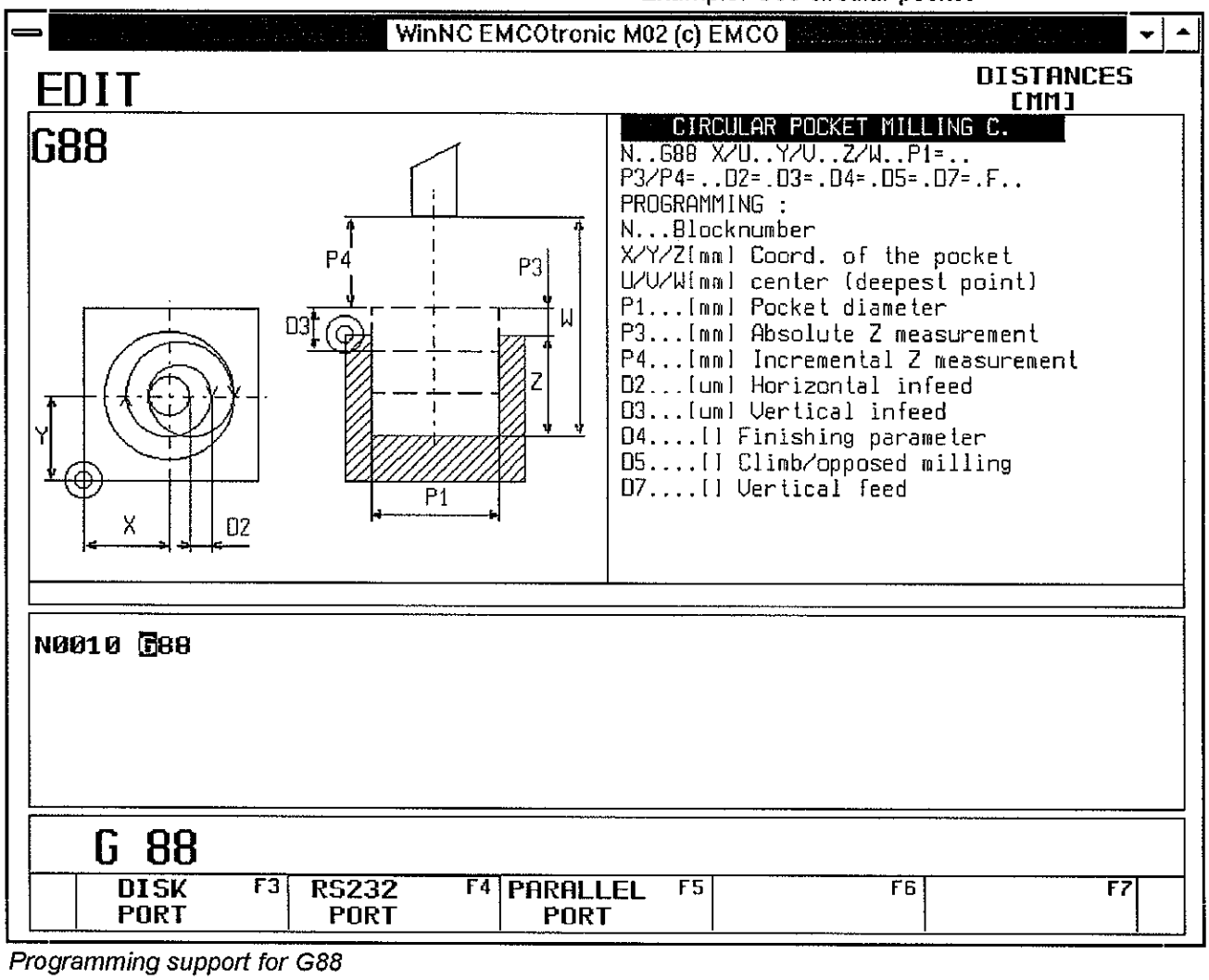
Skip blocks are marked with a "/". These blocks will be omitted when the function SKIP is active.

Move the cursor with STORE NEXT or PREV. to the skip block. The cursor must be on the block number to insert or remove the skip sign. Press the key / to mark the block as skip block or the key C.W. to remove the "/" skip sign.

Program Input with Programming Support

You can enter complicate commands into a program with a graphical help.
If you enter a G code and a support picture is available for this G code, this picture will be shown at the screen.
In this picture addresses and parameter of the G code are shown.

Example: G88 circular pocket



Programming support for G88

Program Administration

EDIT mode

List Programs
Enter:

L ENTER

All existing programs will be listed.

Rename Program
Example:

- The program O20 should be renamed to O25.
- Select the program.

  O 2 0 ENTER

- Press the key C.E. as often, until all digits (also leading zeros) are deleted.
- Enter the new program number (with leading zeros).

  O 0 0 2 5 ENTER.

Delete Program
Example:

- The program O25 should be deleted.
- Select the program.

  O 2 5 ENTER

- Press the key C.PR.

Delete all Programs

- No program must be selected.
- Enter:

  P R O G K I L L

- Press the key C.PR.

Data Input-Output

Data input and output occurs in the EDIT mode. Data input and output will be activated with the softkeys DISKETTE PORT, RS 232 PORT and PARALLEL PORT.

Data Input and Output to a PC Drive

Softkey DISKETTE PORT.
The EMCOTRONIC TM02 has a built-in cassette or diskette drive, which will be approached with this softkey.
WinNC uses the drives of the PC.

- The Parameter O_{06} in the user monitor determines the drive.
  O_{06} = 0 drive A:
  O_{06} = 1 drive B:
  O_{06} = 2 drive C: or import / export directory, see WinConfig, 4.1 "Alter Directories".
- Press the softkey DISKETTE PORT.
  The active drive will be shown, e.g.:
  "INTERFACE A SELECTED"
- The softkey line shows the softkeys INPUT, OUTPUT, INPUT ALL, OUTPUT ALL.
- When you use disks with EMCOTRONIC format, you have to set parameter O_{07}:
  O_{07} = 0 DOS disk
  O_{07} = 1 EMCOTRONIC disk

  Notes:
  - Formatting the EMCOTRONIC disk is not possible with WinNC. At the PC you can only delete all programs, formatting the EMCOTRONIC disk is possible only at the machine.
  - With WINDOWS 95 all functions for the EMCOTRONIC disk are locked, you can work with DOS disks only

Store Program:

- Press the softkey DISKETTE PORT.
- select the desired program, e.g.:

  O 2 0 ENTER
- Press the softkey OUTPUT.
  The program will be stored on the respective drive.

Load Program:

- Press the softkey DISKETTE PORT.
- Select the desired program, e.g.:

  O 2 0 ENTER
- Press the softkey INPUT.
  The program will be loaded from the respective drive.
- If the program to be loaded already exists in the program directory of WinNC, overwriting depends on the user monitor parameter O_{03}.
  O_{03} Bit 0 = 0
  The screen shows "ALREADY EXISTS". If you delete this message with C.PR. the program will be overwritten, cancel with /.
  O_{03} Bit 0 = 1
  The existing program will be overwritten without message.

Store / Load PSO and TO Data:

- Storing and loading PSO and TO data occurs in the same way as with programs. With programs the address O will be selected, with PSO and TO data the address TO.
- PSO and TO data are stored as a package with a number

  e.g.: TO 2 0 ENTER
  Select any number for this package. It makes sense to take the same number for this package as for a program belonging to it.
- All 5 PSO data will be stored or loaded.

Listing of the stored TO/PSO data:

T L ENTER

Store All Programs:

- Press the softkey DISKETTE PORT.
- Press the softkey OUTPUT ALL.
  All programs in the WinNC program directory will be stored to the respective drive.

Load All Programs:

- No program must be selected.
- Press the softkey DISKETTE PORT.
- Press the softkey INPUT ALL.
  All programs from the respective drive will be loaded in the WinNC program directory.
- If one of the programs to be loaded already exists, overwriting depends on the user monitor parameter O_{00} ab.
  $O_{00} = 0$
  The screen shows "ALREADY EXISTS". If you delete this message with C.PR., the program will be overwritten, cancel with /.
  $O_{00} = 1$
  The existing program will be overwritten without mesage..

Note:
A program can not bve loaded directly from an EMCOTRONIC disk. Use the EMCOTRONIC DISK UTILITY (delivered with with the EMCOTRONIC TM02 control), to convert the disk in a PC readable form.

Data Input and Output on the Serial Interface

Softkey RS232 PORT.

- The parameter O_{05} in the user monitor determines the interface.
  $O_{05} = 0$ COM1
  $O_{05} = 1$ COM2
- Press the softkey RS232 PORT.
  The active interface is displayed, e.g.:
  "INTERFACE 1 SELECTED"
- The softkey line shows the softkeys INPUT and OUTPUT.
- For a transmission via the serial interface the interfaces of the sender and the receiver must have the same settings. Setting the interface occurs in the user monitor with the parameters D_{00} and O_{01}.

Store Program:

- Press the softkey RS 232 PORT.
- Select the desired program, e.g.:
  O 2 0 ENTER
- Press the softkey OUTPUT.
  The program will be transmitted to the respective interface.

Load Program:

- Press the softkey RS 232 PORT.
- Select the desired program, e.g.:
  O 2 0 ENTER
- Press the softkey INPUT.
  The program will be loaded from the respective interface.
- An alarm occurs, if the interface receives another program.

Store / Load PSO and TO Data:

- Storing and loading PSO and TO data occurs in the same way as with programs. With programs the address O will be selected, with PSO and TO data the address TO.
- PSO and TO data are stored as a package with a number
  e.g.: TO 2 0 ENTER
  Select any number for this package. It makes sense to take the same number for this package as for a program belonging to it.
- All 5 PSO data will be stored or loaded.
- With loading or storing PSO and TO data no overwrite warning occurs.

Print Data

Softkey PARALLEL PORT.

Print Program:

- Select the desired program, e.g.:
  O 2 0 ENTER
- Press the softkey PARALLEL PORT.
  The program will be printed.

Print PSO and TO Data:

- Select PSO and TO data with the key TO.
- Press the softkey PARALLEL PORT.
  The data will be printed.

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 AUTOMATIC mode.
- Enter the desired part program number (e.g.: O79:

  O 7 9 ENTER).
- If the program exists the screen shows "O79 FOUND". The program can be started.
- Press the key  .
- The program starts, if no alarm is active.
- In the machine status display (second line from bottom) the field "CYCLE START" is displayed inverse.

Program Influence

Running programs can be influenced by actuating the softkeys SINGLE, SKIP and DRYRUN.

SINGLE
If the softkey SINGLE is active, The program will be stopped after every block.

Continue the program with the key  .

SKIP
Blocks which are marked with a slash before the block number (/N ...), will not be worked off when the softkey SKIP is active.

DRYRUN
Activate this function for a test run without workpiece. All movements with programmed feed (G01, G02, G03, G33, ...) traverse with test run feed instead of the programmed feed, the spindle stands still, the coolant is off.

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.

- Press the key RESET ().
- Select AUTOMATIC mode.
- Enter the desired part program number (e.g.: O79:

  O 7 9 ENTER).
- Enter the number of the desired block (e.g.: N100:

  N 1 0 0 ENTER).
- Instead of entering the block number you can step to the desired block with the keys STORE NEXT and PREV.
- Press the key  .
- The input line shows "SIMULATING".
- Previous G, M, S and T codes in the program will be activated, the tool traverses to the end point of the previous block. The program runs from the selected block, if no alarm is active.

Program Interruption

1. Way

Press RESET ().

- the slides will be stopped immediately
- the main spindle will be switched off
- the coolant will be switched off
- all offsets and G41/G42 will be deselected
- the program will be reset to N0000.

Note:
Because of the immediate stop e.g. threads in work will be destroyed.

2. Way

Press the key  .
The drives will be stopped in accordance to the programmed path.

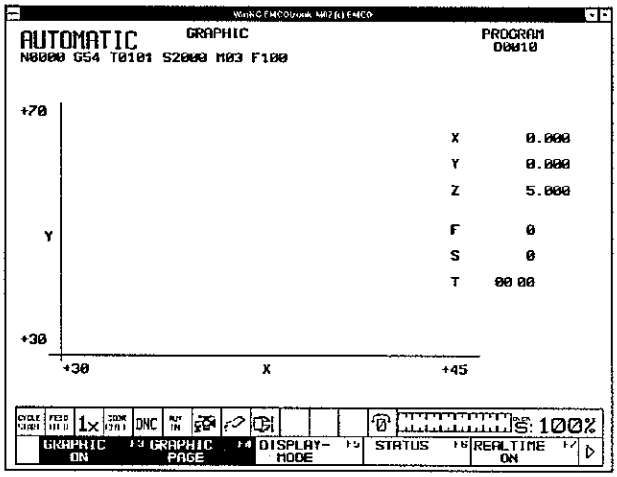
3. Program Stop with M00, M01
If M00 is reached in the program run, the program will be stopped. M01 stops the program only when OPT. STOP is active.

Continue the program with  .

Graphic Simulation

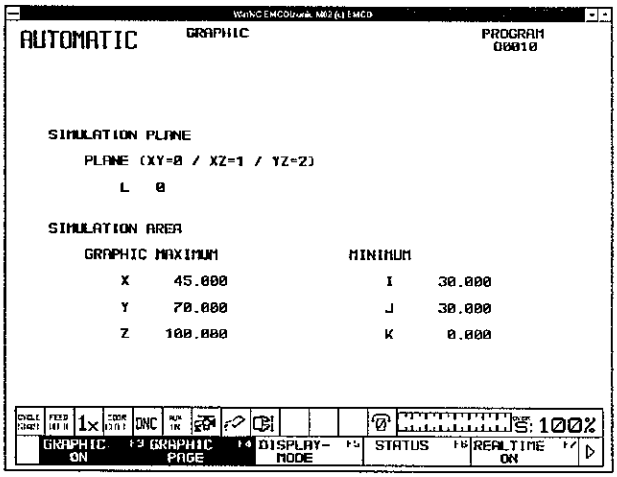
A program run can also be simulated graphically at the screen.
Therefore you must switch to the graphic page and enter a simulation area.

- Select a workpiece program to be simulated, e.g.: O20.
- Press the softkeys GRAPHIC ON and GRAPHIC PAGE.
  The screen shows the graphic page.



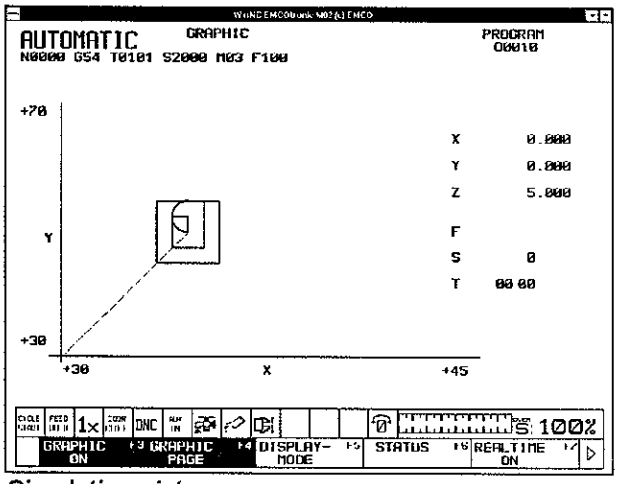
Graphic page

- Press the keys PAGE ↓ or PAGE ↑ at the PC or on the control keyboard / digitizer the keys SHIFT STORE NEXT or SHIFT PREV.
  The screen shows the borders for the simulation area.
  With the addresses X, Y, Z, I, J and K you can enter the borders of the simulation window.
- Enter the simulation plane with the address L (XY plane: L1, XZ plane: L2, YZ plane: L3)

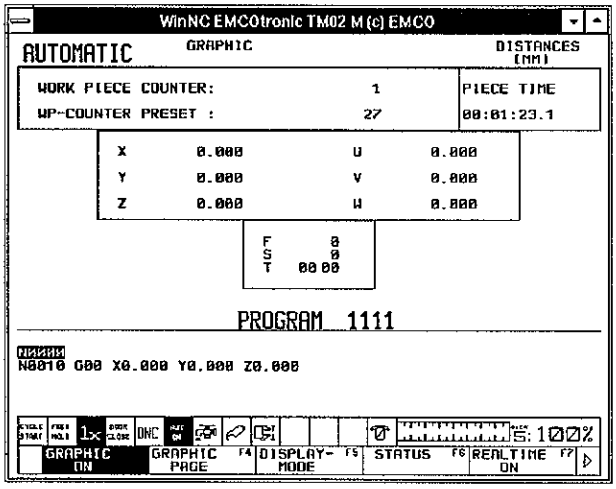


Borders for simulation

- Press the key ◇ to start the simulation run.
  Rapid movements are shown dashed, working movements as full lines.



Simulation picture



Anzeige Stückzähler, Stückzeit

Work Piece Counter and Work Piece Time

The work piece counter and the work piece time will be shown over the position display in AUTOMATIC mode.

The work piece counter displays, how often a program was running. Every M30 increases the conter value for 1.

PIECE TIME shows the whole running time of all program runs.

Work Piece Counter Reset

With parameter D_{03} in the user monitor the counter can be set to a value (e.g. reset to 0 with D_{03}=0).

Work Piece Number Preset

With parameter D_{02} the number of work pieces can be set in the user monitor.

Machines with PLC:
(PC MILL 100 or PC MILL 50 with automatisation)

When the preset number is reached, the program will be stopped and message 7043 PIECE COUNT REACHED will be displayed.

Then the program can be started only after resetting the work piece counter or entering a higher preset number.

To disactivate the preset enter 0 as preset number.

Reset of the Work Piece Time

Reset the workpiece time to 0 with the key C.PR. in the AUTOMATIC mode..

C: Programming Basics

Program Structure

Program structure of the EMCOTRONIC TM 02 conform to DIN 66025 and ISO 1056.

A CNC program continues all commands and informations, which are necessary to machine a workpiece.
The EMCOTRONIC TM 02 differences between main and subprograms.

Every program consists of:

1. Program start
   The program start is the program number. The program number has the address **O** (character **O**).
2. Program contents
   NC blocks
3. Program end
   M30 for a main program
   M17 for a subprogram

Program Numbers

You can use the numbers O0000 - O6999 for main programs and the numbers O0000 - O0255 for subprograms.

Program Blocks, NC Blocks

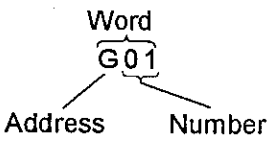
O0015

N0000
N0010
N0020

Address: N
Possible block numbers N 0000 to N 9999.
A block consists of block numbers and words. The words are the contents of a block. It makes sense to select the block numbers in jumps of ten. By that way blocks can be inserted afterwards without renumbering the rest of the program.
With input of a program this jump of ten for the block number is automatically supposed.

Words

N0010 G01 X40. F120



A block consists of one or more words.

A word consists of a character (address) and a number. Every address has a certain meaning, on which the corresponding number depends. The addresses and their meaning is the main contents of this programming manual.

Addresses

O..... Program number

N..... Block number

G ..... Path commands
- G00 Rapid traverse
- G01 Linear interpolation
- G02 Circular interpolation clockwise
- G03 Circular interpolation counter-clockwise
- G04 Dwell
- G17 Plane selection XY
- G18 Plane selection ZX
- G19 Plane selection YZ
- G25 Subprogram call
- G27 Jump instruction
- G40 Cancel tool path compensation
- G41 Tool path compensation left
- G42 Tool path compensation right
- G50 Cancel scale factor
- G51 Scale factor
- G53 Cancellation of zero offset 1 and 2
- G54 Zero offset 1
- G55 Zero offset 2
- G56 Cancellation of zero offset 3, 4, 5
- G57 Zero offset 3
- G58 Zero offset 4
- G59 Zero offset 5
- G70 Measuring in inch
- G71 Measuring in mm
- G72 Definition circular boring pattern
- G73 Execute circular boring pattern
- G74 Definition rectangular boring pattern
- G75 Execute rectangular boring pattern
- G81 Drilling cycle
- G82 Drilling Cycle with dwell
- G83 Withdrawal drilling cycle
- G84 Tapping cycle
- G86 Chip break drilling cycle
- G87 Rectangular pocket milling cycle
- G88 Circular pocket milling cycle
- G89 Slot milling cycle
- G92 Set zero offset 5
- G94 Feed in mm/min (1/100 inch/min)
- G95 Feed in µm/rev (1/10000 inch/rev)
- G98 Retraction to start plane
- G99 Retraction to withdrawal plane

X, Y, Z ..... Absolute coordinates

U, V, W.... Incrementalcoordinates

I, J, K ..... Interpolation parameters

P_0 .. P_7 ..... Auxiliary parameters

D_0 .. D_7 ..... Auxiliary parameters

F ..... Feed, Thread pitch

S ..... Spindle speed

T ..... Tool call-up, selection of tool length compensation

L ..... Subroutine numbers / repetitions, jump target

M ..... Miscellaneous functions
- M00 Programmed stop
- M01 Programmed stop conditional
- M03 Milling spindle ON clockwise
- M04 Milling spindle ON counterclockwise
- M05 Milling spindle OFF
- M08 Coolant ON
- M09 Coolant OFF
- M17 Subprogram end
- M27 Swivel dividing head
- M30 Program end
- M38 Precise stop ON
- M39 Precise stop OFF
- M71 Puff blowing ON
- M72 Puff blowing OFF
- M90 Cancel mirror function
- M91 Mirror X axis
- M92 Mirror Y axis
- M93 Mirror X and Y axis

Syntax Regulations

Block length

The maximum block length depends on the programmed words and is three or four lines. When exceeding the maximum block length ALARM 650 will occur.
For the clearness of a program we recommend a logical structure.

Sequence of words

No regulation for the sequence of words. For the clearness of a program you should use the following sequence:

- Every block starts with the block number.
- After the block number follows the G command.
- Words for the coordinates X(U), Y(V), Z(W).
- For G02, G03 program the interpolation parameter I, J and K after X(U), Y(V), Z(W).
- For cycles program the parameter after X(U), Y(V), Z(W).
- The F word (feed, thread pitch).
- The S word (spindle speed, cutting speed).
- The T word (tool address).
- The M word (additional functions).

Several G and M Functions of the Same Group

Where two or more G and M functions are in one block (no sense), the last programmed function is effective. Group divisions see "Group divisions and initial stati of the G commands" and "Group divisions and initial stati of the G commands".

Same Words in a Block

The last programmed word is valid.

Decimal Point Programming

X, Y, Z, U, V, W, P, I, J, K values must be programmed with a decimal point. Without decimal point this values would be computed as µm (G71) or as 1/10000 inch (G70).
Leading and following zeros need not to be programmed.

Skip Blocks

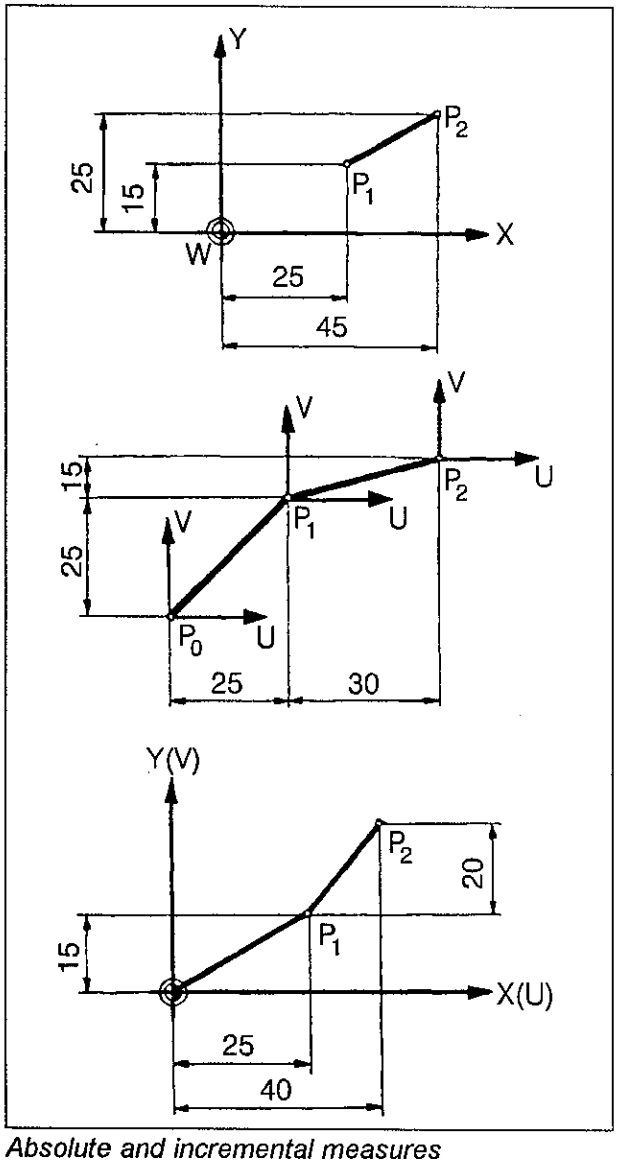
For some machining conditions (variant machining) it is useful that blocks can be skipped.
Skip blocks are marked with a slash. The slash is written after the block number.

Move the cursor with STORE NEXT or PREV. to the skip block. The cursor must be on the block number to insert or remove the skip sign. Press the key / to mark the block as skip block or the key C.W. to remove the "/" skip sign.

N0090 G00 X20. Y20. Z30.
N0100/ M00 ..... skip block

Program Sequence

SKIP key pressed: Skip blocks will not be executed
SKIP key not pressed: Skip blocks will be executed



Absolute and incremental measures

Absolute and Incremental Value Programming

Absolute Value Programming

The description runs under the addresses **X, Y, Z.** The X, Y and Z values always relate to the actual origin of the coordinate system (siehe position shift offsets).

Example
P_1 X = 25, Y = 15
P_2 X = 45, Y = 25

Incremental Value Programming

The description runs under the addresses **U, V, W.** The U, V and W values always relate to the start position of each block.

Example
P_1 U = 25, V = 25
P_2 U = 30, V = 10

Mixed Programming

The programming can also be mixed.

Example
P_1 X = 25, Y = 15
P_2 X = 40, V = 20

Format Description of the Addresses

Specific addresses are assigned to most G commands.

Example

G00 X±....,... Y±....,... Z±....,...
or
G01 X±....,... Y±....,... Z±....,... F....

For a short and easy to understand description of pertaining addresses (format description) the data are encoded.

N $\underbrace{\cdots}_{4}$ → N 4

1. Instead of giving the possible inputs, the number of decades is given.
   e.g. instead of N from 0 to 9999 or N.... , N 4 is written.

X $\underbrace{\cdots}_{4}\underbrace{\cdots}_{3}$ → X43

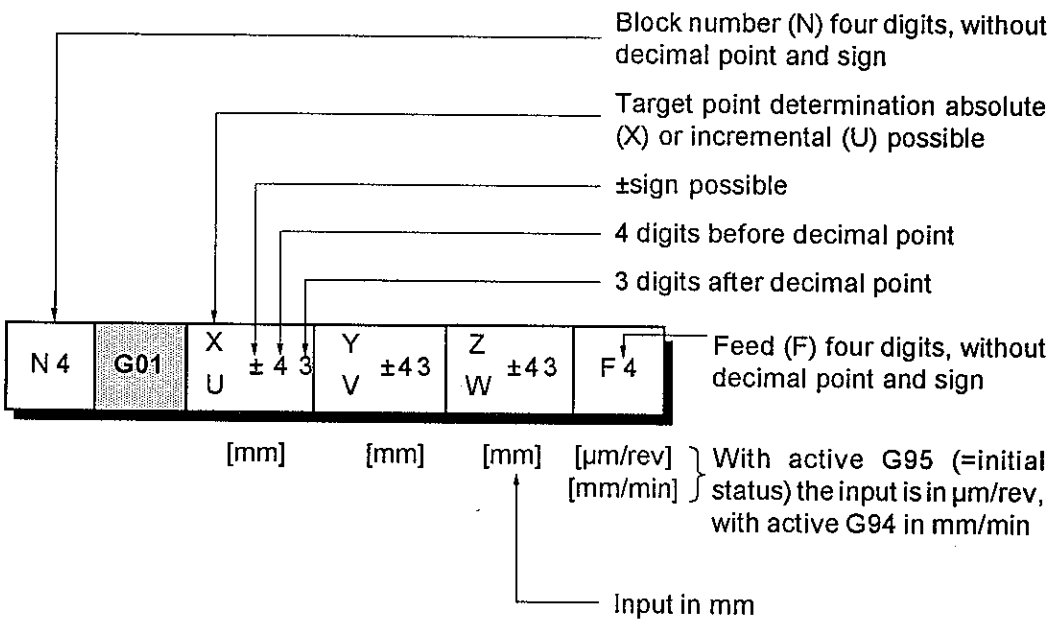
2. The specification of the possible decades before or after a decimal point is coded with two numbers.
   The first number: decades before the decimal point
   The second number: decades after the decimal point

X ±43

3. If values can be positive or negative, a ± sign is written between address and number.

Remark
Below the format box the input dimensions are written.

Example
The linear moving G01 is descibed as following:



Self-holding Functions, Words

The majority of the G and M commands and other words are self-holding. That is they remain active until they will be overwritten or deselected.
This is a simplification and reduction for programming.

G and M Commands

The G and M commands are divided into groups.
The self-holding G and M commands are active, until they will be overwritten by another command of the same group. (see at the end of this chapter)
Some G and M functions can directly be deselected.
e.g. G41 will be deselected by G40

Example
Takeover of G00 in block N0110.
In block N0120 G00 is deselected by G01. G01 is active.

N0100	G00	X50.000	Y+10.000	
N0110		X36.000	Y+2.000	
N0120	G01	X40.000	Y-10.000	F ....

Example
M03 will be activated in block N0050. In the blocks N0050 to N0120 M03 is effective. M03 will be deselected in block N0120 by M04. M04 is active from block N0120.

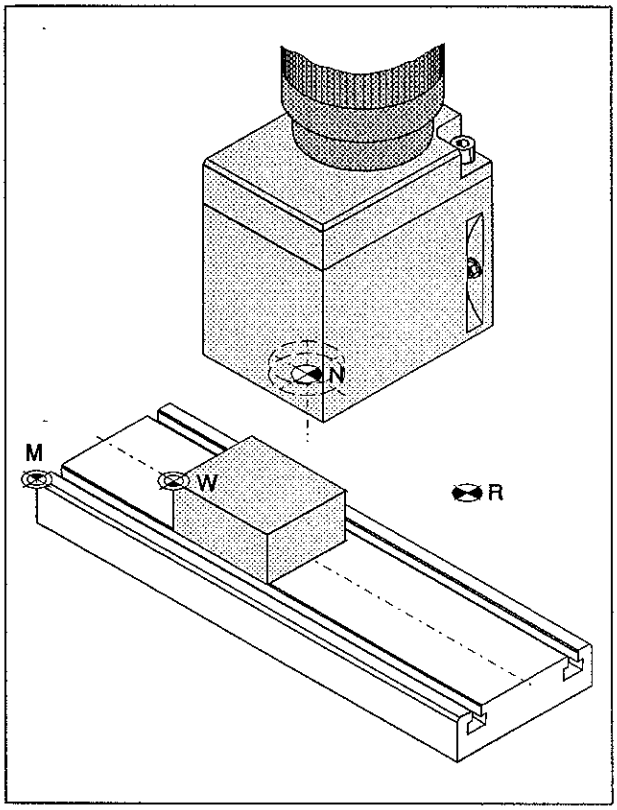
N0050				M03
N0060				
.				
.				
.				
N0120				M04

Takeover of Words and Word Contents

X(U), Y(V), Z(W), F, S, T word contents will be taken over in the following blocks. The contents will be overwritten by programming another value.

Example
Words which were taken over from a previous block are shown as grey box.

N0040	G01	X40.000	Y10.000	F120	S1500	T0303
N0050		X35.000				
N0060			Y18.000			
N0070		X48.000	Y20.000			



Reference points in the working area

Reference Points of the EMCO Milling Machine

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.

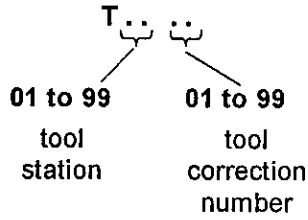
N = Tool mount reference point
Starting point for the measurement of the tools. "N" lies at a suitable point on the tool holder system and is established by the machine manufacturer.

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.

Tool Programming

Tools are programmed under the T address with a 4 digit number.

Encoding

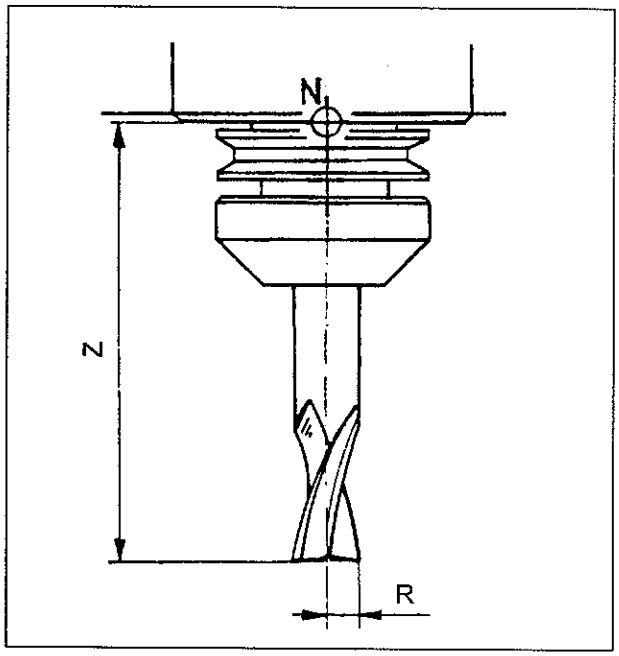


1. Tool Station

The first two digits determine the tool station in the tool holder. With calling up the T command the tool holder swivels this station in working position. If the programmed station is higher than the number of stations at the machine, an alarm occurs.

2. Tool Correction Number

Tool correction values are stored in the tool offset register (TO) with a tool correction number.



Tool correction values

The Tool Correction Values

The Tool Data

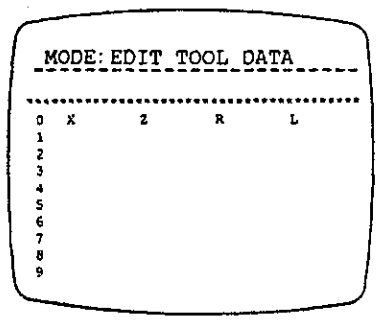
Imagine the coordinate system in the point N.
From point N on (tool mount reference point) the tool length will be measured (-Z value). These measure is written in the tool offset register.

The Cutter Radius R

The tool radius R (cutting radius) must be entered into the register.

When working with G40, G41, G42 and some cycles R is needed.

Tool Offset Register



Tool offset register at the screen

The tool correction values will be entered into the tool offset register in the EDIT mode or measured in the MANUAL mode.

Tool length: Below Z address
Cutter radius: Below R

Entering the tool offset values in the EDIT mode:

Press the key [TO] to call up the tool offset register.

Key in the tool correction number to be altered and press ENTER. The cursor jumps on the first value. Key in the desired value and press ENTER. The cursor jumps on the next value etc.

Incremental altering of the tool data:
The value which is marked with the cursor can be altered with the JOG keys +X, -X, +Y, -Y, +Z, -Z.

+X, -X increase/decrease for 0.001 mm (0.0001")
+Y, -Y increase/decrease for 0.01 mm (0.001")
+Z, -Z increase/decrease for 0.1 mm (0.01")

Compution
When a tool is called up with tool correction number in the program, the control aquisites the tool data Z and R which are stored in the tool offset register.
e.g. Call up of T . . 20 - the control calculates the values which are stored under correction number 20..

Call up the T Address

Example:

```
N0090   M00
N0100   G00     X...   Y...   Z...   T0202
N0110   M3      S...
```

Example:

```
N0100   T0202
N0110   G94     F130   M3     S...
N0120   G54
N0130   G00     X...   Y...   Z...
```

No tool correction

T.. 00

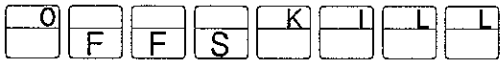
When T.. 00 is programmed, the measuring system relates on the tool mount reference point N.
The tool holder swivels in the called up tool station, no tool correction will be calculated, an eventually previous programmed tool correction will be deselected.

Programming Notes

Specification of the correction numbers:
Correction number and tool number need not to be identical, e.g. T0501. For clearness it makes sense, to take identical tool numbers and correction numbers.
E.g.: T0303, T0313, T0323 etc.

Delete all Tool and Offset Data

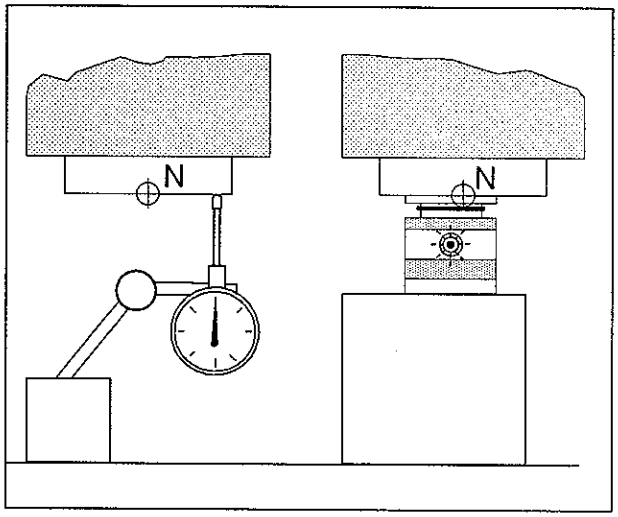
- Enter:



- Press the key 

.
- All tool data (TO) and offset data (PSO) will be deleted.

Tool Data Measuring with a Metering Clockwork or a Measuring Cell

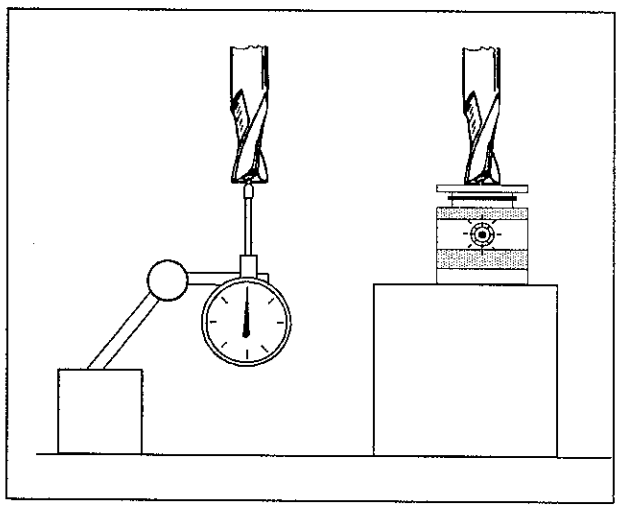


Traverse with the tool mount reference point onto the metering clockwork or measuring cell

Sequence of operation

- Set up the metering clockwork or the measuring cell so in the working area, that the measuring point can be reached with the tool mount reference point and with all tools to be measured.
- Change into MANUAL mode [MAN]
- Traverse with the tool mount reference point onto the metering clockwork and set it to zero or onto the measuring cell until the lamp lights up.
- Store the position of the tool mount reference point as reference position.
  Input:
  [TO] [0] [0] [ENTER]

Note:
At the EMCO PC MILL 100 the tool mount reference point is on the face centre of the reference tool. Clamp this tool for the sequence described above.



Traverse with the tool tip onto the metering clockwork or measuring cell

- Clamp (PC MILL 50) or swivel in (PC MILL 100) the tool to be measured and traverse with it onto the metering clockwork until 0 is displayed or onto the measuring cell until the lamp lights up.
- Store the offset of the tool in the desired line of the tool offset register, e.g.:
  [TO] [0] [1] [ENTER]
- Measure next tool etc.

Note

The tool radius R must be entered into the tool data register subsequently.

Zero Offsets

The origin of the coordinate system can be shiftet in a position selected by you. Through a call command shift values that have been previously set in the position shift register are activated.

Position Shift Register

The dimensions for zero point offsets are entered in the position shift registers 1-5 with correct sign.

Call command Position shift register 1 - 5

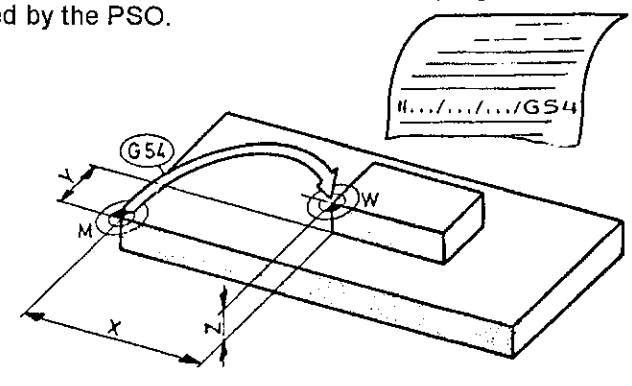
	Position Shift			
		X	Y	Z
G54 →	1	...	...	...
G55 →	2	...	...	...
G57 →	3	...	...	...
G58 →	4	...	...	...
G59 →	5	...	...	...

On the following pages the values of the position shift registers 1 - 5 are also called PSO (Position Shift Offset).

Call Commands

When a call command is programmed in the CNC program, the coordinate system is shifted by the PSO.

Example
N.... G54 ....



Input possibilities

1. Manual input in the position shift register 1-5:

   Entering the position shift values in the EDIT mode:

   Press the key PSO to call up the position shift register.

   Key in the offset number to be altered and press ENTER. The cursor jumps on the first value. Key in the desired value and press ENTER. The cursor jumps on the next value etc.

   Incremental altering of the offset data:
   The value which is marked with the cursor can be altered with the JOG keys +X, -X, +Y, -Y, +Z, -Z.
   +X, -X increase/decrease for 0.001 mm (0.0001")
   +Y, -Y increase/decrease for 0.01 mm (0.001")
   +Z, -Z increase/decrease for 0.1 mm (0.01")

2. Reading in position shift register from disk or serial interface.

3. Special event:
   G92 - Setting of PSO 5 in the CNC program
   G59 - Activation of PSO 5 in the CNC program

Call / Cancellation of the Position Shift Offset

Group division of the commands:

Group 3	G53	Cancellation of G54, G55
	G54=PSO 1 G55=PSO 2	Call of position shift offset (PSO) 1, 2.
Group 5	G56	Cancellation of G57, G58, G59
	G57=PSO 3 G58=PSO 4 G59=PSO 5	Call of position shift offset (PSO) 3, 4, 5.

Group division and shift / cancellation of a shift

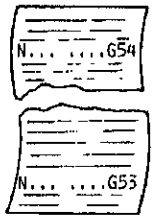
Several commands of the same group in a program:
Always the last programmed command is valid.
The previous command is cancelled by the next (see examples).

Two commands from different groups:
Commands from different groups are added vectorially. (They do not cancel each other! See examples)

Shift deselection:
G53 deselects G54 and G55
G56 deselects G57, G58 and G59

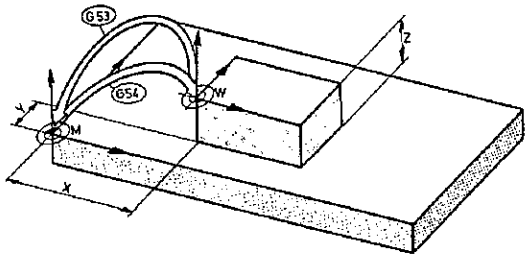
Examples G53 - G59

Example 1



G54: The coordinate zero will be shifted to workpiece zero point W.
The X, Y, Z values are in position shift register 1.

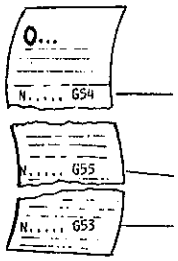
G53: G53 cancels the shift G54.
The coordinate zero will be set back to machine zero point M.



Example 2

If in a program two or more zero offsets of the same group are called, the later one cancels the previous.

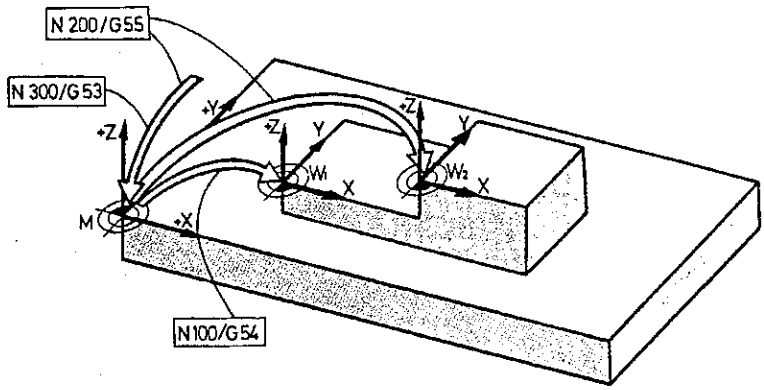
The values for W1 and W2 are entered in the position shift register



activating W1

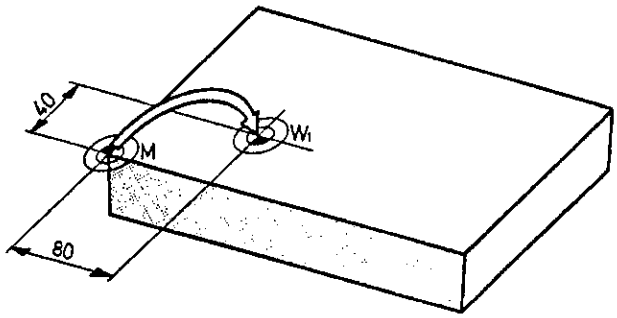
cancelling W1 and activating W2

cancelling W2, the coordinate zero is in point M



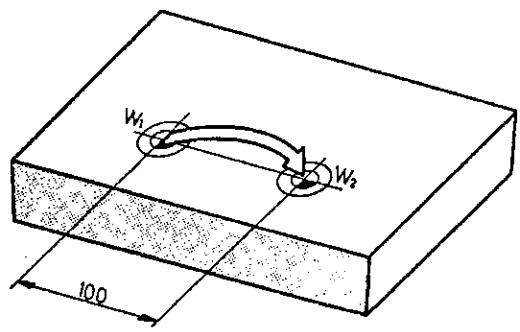
Example 3

G commands from different groups do not cancel, they will be added.



Calling G54 (group 3):
Zero shift from M to W1

Values for PSO 1
X=80 Y=40 Z=0



Calling G57 (group 5):
Zero shift from W1 to W2

Values for PSO 3
X=100 Y=0 Z=0

Total shift = G54 + G57 values

G54:	X= 80	Y= 40	Z= 0
G54:	X=100	Y= 0	Z= 0
Total:	X=180	Y= 40	Z= 0

Special Event G92, G59

G92 Set Zero Offset 5
G59 Zero Offset 5

Regulations

Programming the shift values
The shift values are written under G92 in the part program.

Example
N.... G92 X-14.2 Y13. Z14.

Activation of the position shift
With programrun the shift offset will be taken over in the position shift register 5 (PSO 5).
With G59 the zero offset will be executed.

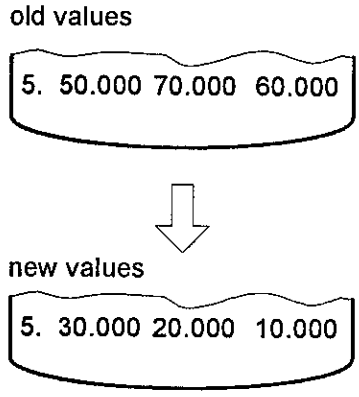
Syntax

- G59 can not be programmed in the same block with G92, but must be programmed in the following blocks.
- When G59 is active with proceeding the G92 block, alarm 700 occurs.
- When G59 is programmed together with G54 or G55 both zero offsets will be added.

Cancellation

Cancellation occurs with G56.

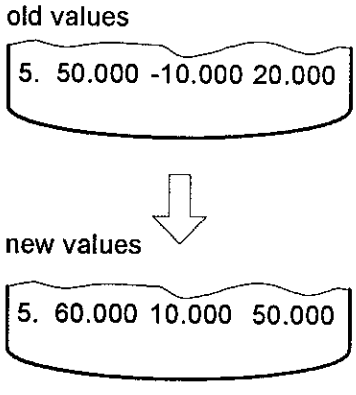
Types of Dimension Input G92



1. Absolute values:

If the shift values for G92 are described with X, Y, Z the old values in PSO 5 will be deleted and the new values for G92 are active.

N0100 G92 X30.000 Y20.000 Z10.000

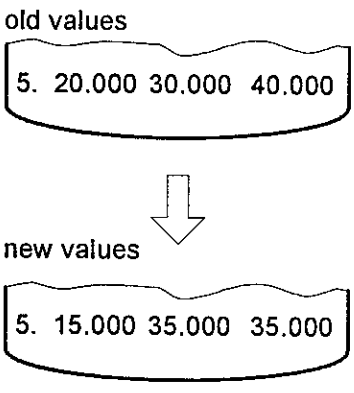


2. Incremental values:

If the shift values for G92 are described with U, V, W, the U, V, W values will be added to the values in PSO 5.

N0100 G92 U10.000 V20.000 W30.000

Note
If shift values will be programmed incrementally they will be added to PSO 5 with each program run.



3. Mixed values:

With mixed input for G92 (absolute with X, Y, Z and incremental with U, V, W)

- the absolute G92 values will be taken over into the register,
- the incremental G92 values will be added to PSO 5.

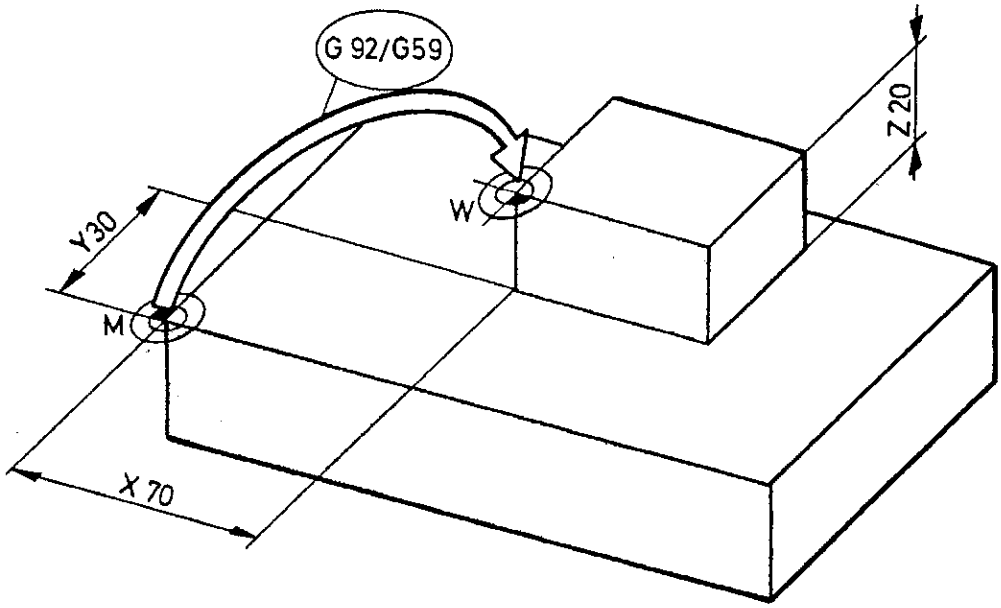
N.... G92 X15.000 V5.000 W-5.000

Examples G92

Example 1

Shift from machine zero M point to workpiece zero point W

Input of a zero offset with G92.
Activate with G59.
Cancel with G56.



Program:

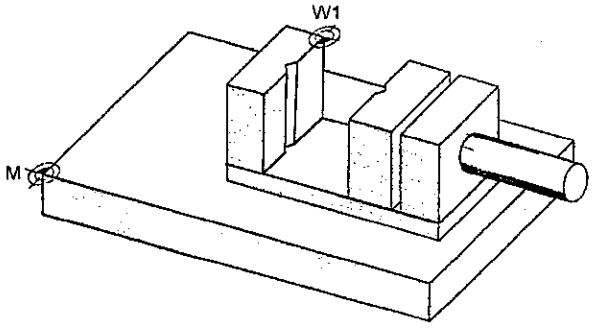
```
N100 G92  X70.      Y30.      Z20.
...
...
...
N150 G00  ...       ...       ...       G59
```

The zero shift occurs in block N150.

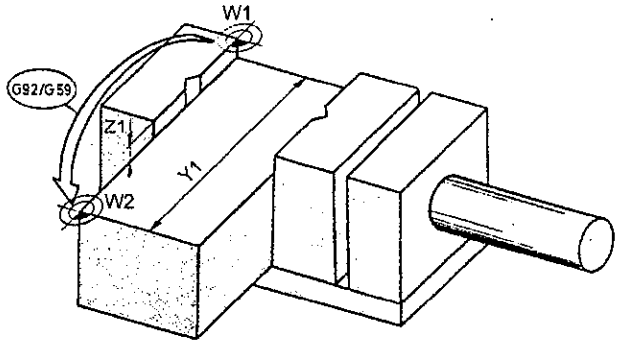
Example 2

Shift from machine zero M point to workpiece zero point W1 with group 3 command and shift from W1 to W2 with G92, G59

This is a very practicable way when using fix mounted clamping devices. One edge (or stop) of the clamping device is W1 and can be programmed with an offset register.
The shift from W1 to W2 depends only on the workpiece dimensions and can be programmed with G92 and called with G59.

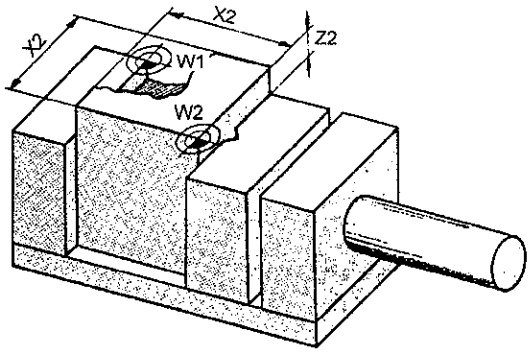


The coordinate zero will be shifted from M to W1 with a group 3 command (e.g.: G54).



Example for shifting:

```
N.... G54
N.... G92  X0.000     Y1          Z1
N.... G59
```



Example for shifting:

```
N.... G54
N.... G92  X2         Y2          Z2
N.... G59
```

Group Division and Initial Stati of the G Commands

Group 0	 * 	G00: Rapid traverse G01: Linear interpolation G02: Circular interpolation clockwise G03: Circular interpolation counterclockwise G04: Dwell G72: Definition circular boring pattern G74: Definition rectangular boring pattern G81: Drilling cycle G82: Drilling cycle with dwell G83: Withdrawal drilling cycle G84: Tapping cycle G86: Chip break drilling cycle G87: Rectangular pocket milling cycle G87: Circular pocket milling cycle G87: Slot milling cycle
Group 2	**	G94: Feed in mm/min (1/100 inch/min) G95: Feed in µm/rev (1/10.000 inch/rev)
Group 3	**	G53: Cancellation of zero offset 1 and 2 G54: Zero offset 1 G55: Zero offset 2
Group 4	*	G92: Set zero offset 5
Group 5	** 	G56: Cancellation of zero offset 3, 4, 5 G57: Zero offset 3 G58: Zero offset 4 G59: Zero offset 5
Group 6	* *	G25: Subprogram call G27: Jump instruction
Group 7	▭ ▭	G70: Measuring in inch G71: Measuring in mm
Group 8	** 	G40: Cancel cutter radius compensation G41: Cutter radius compensation left G42: Cutter radius compensation right
Group 9	** 	G17: Plane selection XY G18: Plane selection ZX G19: Plane selection YZ
Group 11	** 	G98: Withdrawal to start plane G99: Withdrawal to withdrawal plane
Group 12	* *	G73: Execute circular boring pattern G75: Execute rectangular boring pattern
Group 15	** 	G50: Cancel scale factor G51: Scale factor

* effective blockwise
** initial status
▭ initial status setable with WinConfig

Group Division and Initial Stati of the M Commands

Group 0	 **	M03: Milling spindle ON clockwise M04: Milling spindle ON counterclockwise M05: Milling spindle OFF
Group 1	 **	M38: Precise stop ON M39: Precise stop OFF
Group 2	* * * *	M00: Programmed stop M01: Programmed stop conditional M17: Subprogram end M30: Program end
Group 3	 **	M08: Coolant ON M09: Coolant OFF
Group 4	*	M27: Swivel dividing head
Group 7	 **	M71: Puff blowing ON M72: Puff blowing OFF
Group 10	**	M90: Cancel mirror function M91: Mirror X axis M92: Mirror Y axis M93: Mirror Z axis

* effective blockwise
** initial status

Addresses and their Input Dimensions

Address	metric	inch
Pathaddresses X, Y, Z, U, V, W	±[mm]	±[inch]
Circle interpolation parameter I, J, K	±[mm]	±[inch]
F - thread pitch (G84)	[µm]	[1/10000 inch]
F - feed per minute (G94)	[mm/min]	[1/100 inch/min]
F - feed per revolution (G95)	[µm/rev]	[1/10000 inch/rev]
S - direct speed programming (G97)	[rev/min]	[rev/min]

D Parameter in the Program

Parameter		Default Option
D_0	G72: number of boring pattern elements	
	G74: number of horizontal boring pattern elements	
D_1	G74: number of vertical boring pattern elements	
D_2	G72: start angle [°x10]	0°
	G88: horizontal infeed	1.7 x tool radius
	G89: slot angle related to X axis [°x10]	0°
D_3	G72: total angle [°x10]	3600
	G83: infeed per cut [µm, 1/10000 inch]	no cut division
	G86: infeed per cut [µm, 1/10000 inch]	no cut division
	G87: infeed per cut [µm, 1/10000 inch]	no cut division
	G88: vertical infeed	work off in 1 cut
	G89: vertical infeed	work off in 1 cut
D_4	G04: dwell [1/10 s]	no dwell
	G82: dwell [1/10 s]	no dwell
	G86: dwell [1/10 s]	no dwell
	G88: finishing parameter	1
	G89: finishing parameter	1
D_5	G83: percentual decrease	0
	G86: percentual decrease	0
	G87: climb milling / opposed milling	3
	G88: climb milling / opposed milling	3
	G89: climb milling / opposed milling	3
D_7	G72: parameter take-over	0 (no take-over)
	G74: parameter take-over	0
	G87: vertical infeed parameter	1

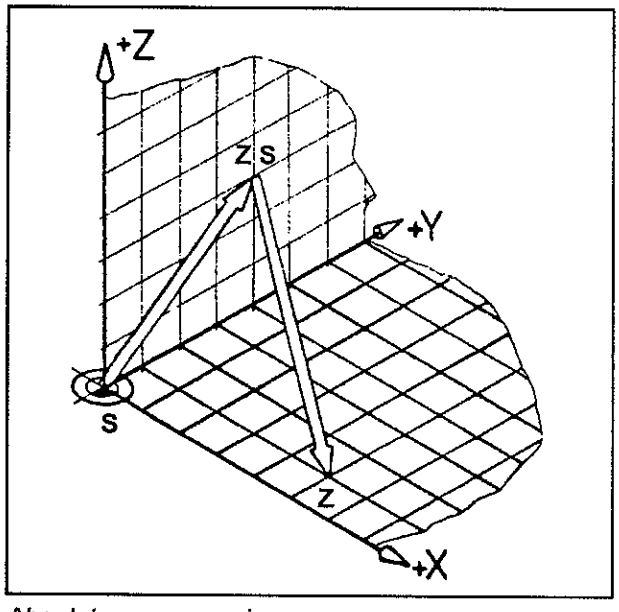
Default option means standard interpretation for not given parameters. The parameters signed with "Def" can but need not to be programmed. If they will not be programmed the presetted value is active (see table).

P Parameter in the Program

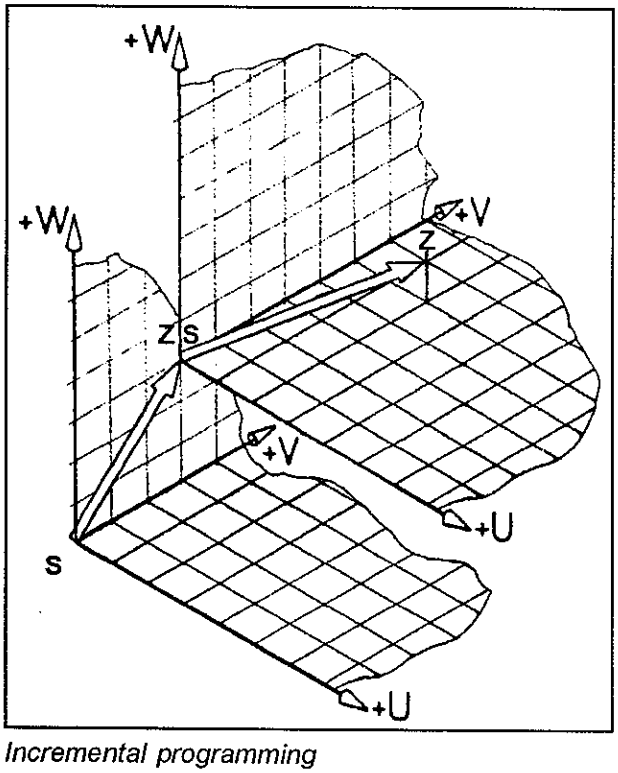
Parameter		Default Option
P_0	G72: circle diameter [mm, inch]	
	G74: horizontal distance [mm, inch]	
	G87: pocket length in X [mm, inch]	
	G89: length of the slot [mm, inch]	
P_1	G74: vertical distance [mm, inch]	
	G87: pocket length in Y [mm, inch]	
	G88: pocket diameter [mm, inch]	
	G89: width of the slot [mm, inch]	
P_3	G81 - G89: definition of the retraction plane absolute from the workpiece zero	
P_4	G81 - G89: definition of the retraction plane incremental from the start plane	
P_7	G51: scale factor	

D: G Commands

N 4	G00	X U ±43	Z W ±43	Z W ±43
		[mm] [inch]	[mm] [inch]	[mm] [inch]



Absolute programming



Incremental programming

G00 Rapid Traverse

G00 (rapid traverse) is a pure traversing movement - not a working movement!
The speed of rapid traverse is specified at works for the particular machine type.

Programming

N ..... block number
G00 ..... rapid traverse
X, Y, Z } Absolute, incremental coordinates
U, V, W } of the target point Z
S ..... start point
Z ..... target point

Notes

The sequence of X (U), Y(V), Z (W) is immaterial. you also can program mixed (absolute and incremental) within a block.

E.g.: G00 X44. W-9.

Programming absolute

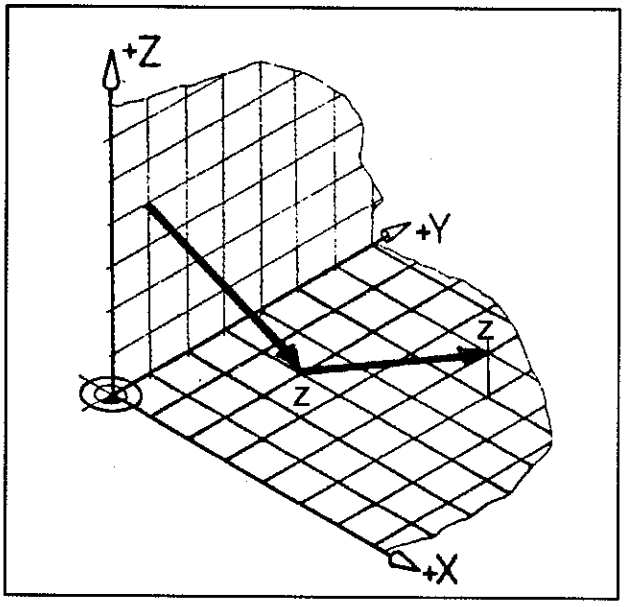
N090 .....
N100 G00 X0 Y4. Z3.
N110 G00 X5. Y1. Z0

Programming incremental

N100 .....
N110 G00 U-6. W-9.
N120 .....

G01 Linear Interpolation

N 4	G01	X ±43 U	Y ±43 V	Z ±43 W	F 4
		[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm/min] [1/100 inch / min] [µm/rev] [1/10000 inch / rev]



Absolute programming

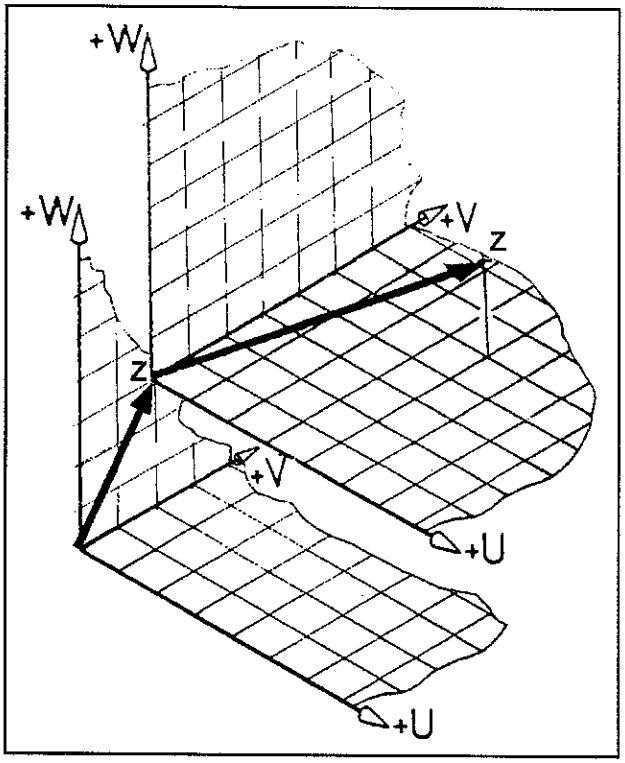
G01 is a linear working movement. The feed must be programmed. It can be entered in mm/min (G94) or in µm/rev (G95).
The feed (F) is self-holding.

Programming

N ..... block number
G01..... function linear interpolation
X, Y, Z } Absolute, incremental coordinates
U, V, W } of the target point Z
F ..... feed

Programming absolute

N090	.....				
N100	G01	X2.	Y3.	Z0	F...
N110	G01	X5.	Y5.	Z1.	F...
N120	.....				



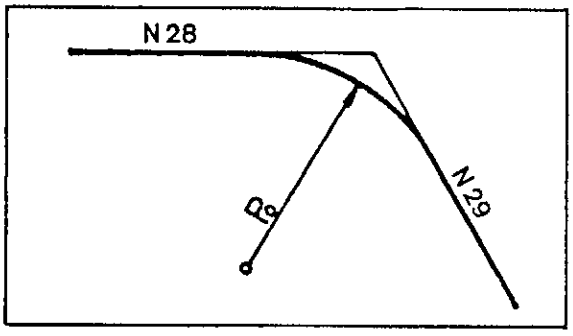
Incremental programming

Programming incremental

N090	.....				
N100	G01	U0.	V2.	W3.	F...
N110	G00	U4.	V5.	W2.	F...
N120	.....				

Inserting Chamfers and Radii

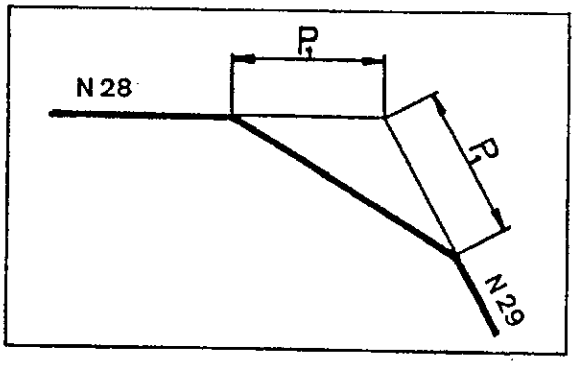
N 4	G01	X U ±43	Y V ±43	Z W ±43	P_0 P_1 ±43	F 4
		[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm/min] [1/100 inch / min] [µm/rev] [1/10000 inch / rev]



N28 G01 X.... Z.... F.... P_0
N29 G01 X.... Z.... F....

Radius

- Between two straight movements (e.g. block N28 and N29) a radius can be inserted.
- The radius is defined by parameter P_0.
- P_0 will be added to the first (N28) of the two blocks.



N28 G01 X.... Z.... F.... P_1
N29 G01 X.... Z.... F....

Chamfer

- Between two straight movements (e.g. block N28 and N29) a chamfer can be inserted.
- The chamfer is defined by parameter P_1.
- The chamfer is put in the edge symmetrically, that means the length P_1 is equal on both containing straights.
- P_1 will be added to the first (N28) of the two blocks.

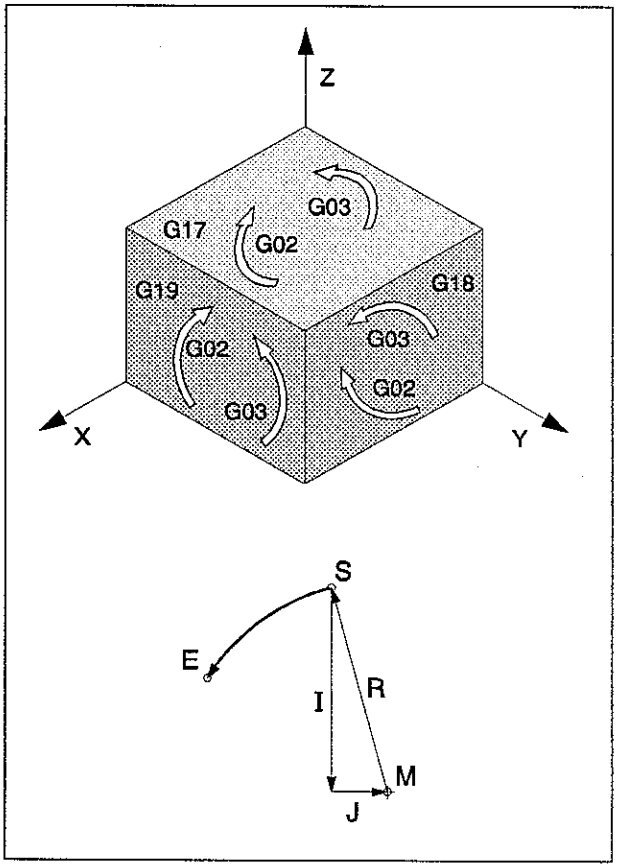
Conditions

- The length P_1 of an inserted chamfer must not be longer than the shorter one of the two straights, otherwise no intersection point would result.
- For calculating the chamfer resp. the radius the block in which the chamfer/radius is programmed and the following block is needed.

  In these blocks no PSO alterations, no tool change and no scale factor alteration must happen.

Circular Interpolation
G02 Clockwise
G03 Counterclockwise

N 4	G02 G03	X U ±43	Y V ±43	Z W ±43	I±43	J±43	K ±43	F 4
		[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm/min] [1/100 inch / min] [µm/rev] [1/10000 inch / rev]



Rotational directions of G02 and G03

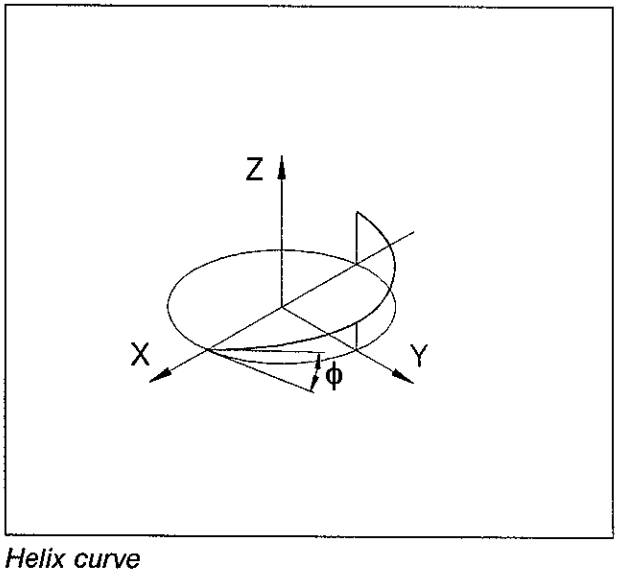
Programming

N..... block number
G02..... circular interpolation clockwise
G03..... circular interpolation counterclockwise
X, Y, Z } Absolute, incremental
U, V, W } coordinates of the target point
I, J, K ..... circle centre coordinates (incremental
..... from the circle start point)
F ..... feed

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.
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

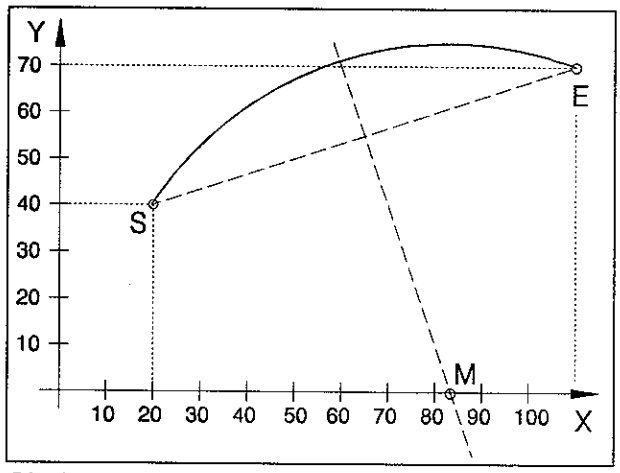
- A helix interpolation is possible with G17 only.
- The gradient angle ϕ must be less than 45°.
- If the spatial tangents differ more than 2° with block transitions, an exact stop will be proceeded in every case before/after the helix.

Notes for programming an arc

The centre point is on the symmetric axis between the start and end point of the arc.

By programming one centre coordinate an arc is completely determined.

The centre coordinate must be programmed with which the more exact intersection would result. This is that coordinate, in in which direction the distance from start to end point is shorter.



Start, centre and end point of the arc

Example:
Start point S: X=20 Y=40
End point E: X=110 Y=70
Centre point M: X=83,3333 Y=0
G01 X20. Y40.
G02 X110. Y70. (I83.3333) (J0)
In this case J must be programmed, because the distance from start to end point in Y (70-40=30) is shorter than in X (110-20=90).

If both centre coordinates will be programmed, the control uses that one with the more exact intersection. The second centre coordinate will be supervised whether it is within the tolerance. If not, an alarm will occur.

G04 Dwell

[1/10 s]

With G04 and the parameter D_4 a dwell will be programmed.

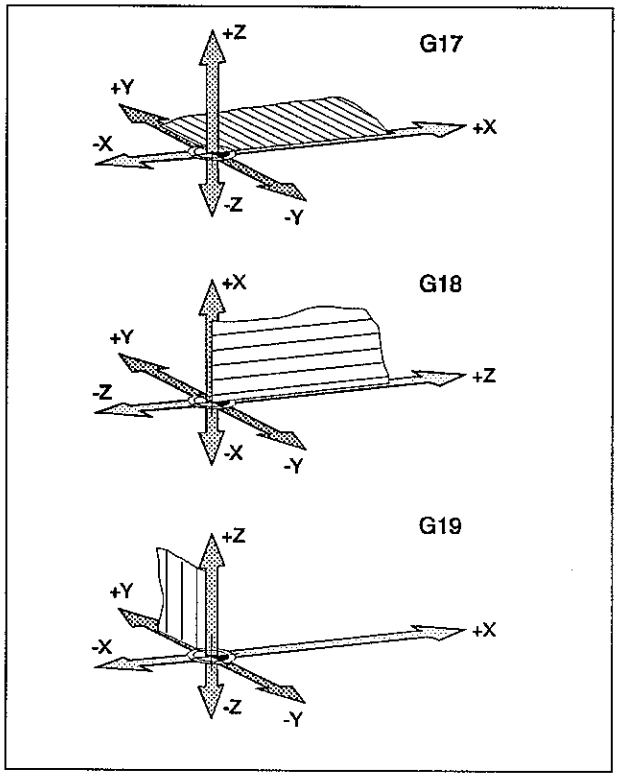
Input range
1 - 10 000 (0.1 s - 1000 s)

Note
G04 is effective blockwise and is active last in the block, irrespective of whether the dwell is .

Example

N0100 G04 D_4=20 M03
N0110 G00 X40. Z-10.

Block 100:
The main spindle is switched on (rotation clockwise = M03). Before block N110 the programmed dwell of 2 seconds will be executed.



Definition of the main planes

G17-G19 Plane Selection

N 4	G17 G18 G19

With G17 to G19 the plane will be defined, in which the circular interpolation and 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

G25 Subprogram Call

N 4	G25	L 5

Subprogram numbers: O 0000 - O 0255

Max. nesting depth: 10

A subprogram will be called from a main program or a subprogram. The subprogram has the same structure like a main program.

Structure Subprogram

- Program number
  Possible program numbers O 0000 - O 0255
- Program blocks
- M17: Program end (jump-back command)

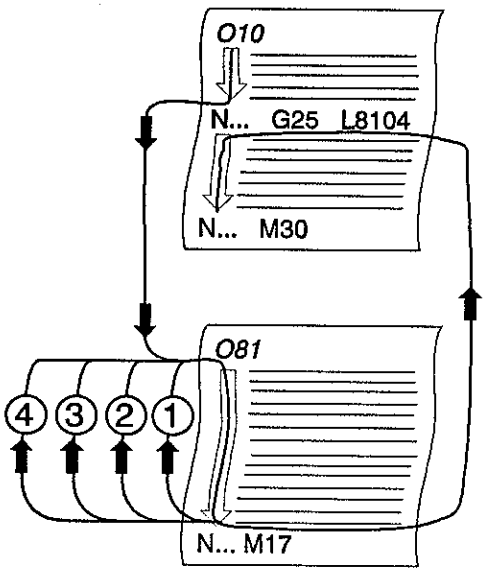
```
O 81
N
N
N
N
N
N
M17
```

Subprogram Call

G25 Subprogram call

L.. address for subprogram number and number of subprogram runs.

```
L ... ..
subprogram          Number of
number 0-255        runs 1-99
```



Example

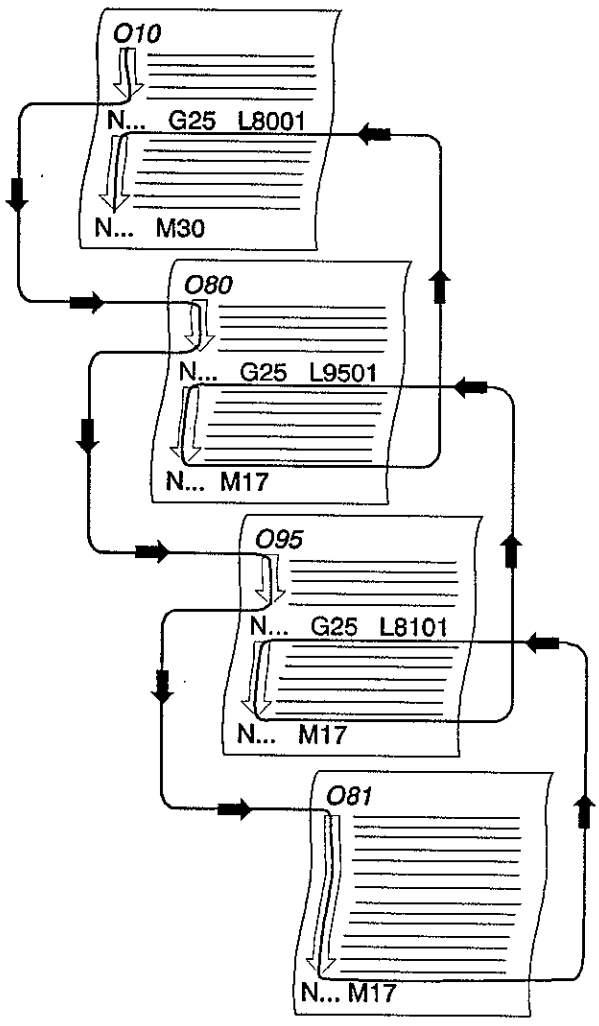
subprogram with 4 runs

main program O 10

N.... G25 L8104
N....

subprogram O 81

M17



Example

Nesting of subprograms:
From subprograms other subprograms can be called (nesting).
The EMCOTRONIC TM 02 allows a ten fold nesting.

Main program O10

Subprogram 80
nesting depth 1

Subprogram 95
nesting depth 2

Subprogram 81
nesting depth 3

Program Numbers for Subprograms

For a better clearness main programs and subprograms should be distinguished by the number.

Possible main program numbers:
O 0000 - O 6999
Possible subprogram numbers:
O 0000 - O 0255

For the main program the numbers O 0000 - O 6999 can be used. (in this sense do not use the numbers O 0000 - O 0255 for main programs if you also use subprograms.)
For subprograms only the numbers O 0000 - O 0255 can be used, otherwise alarm 630.

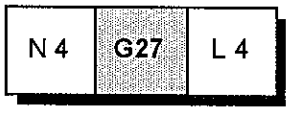
Remark

With program transmission to the original control for the address L only four digit numbers are allowed.

Example
the call **G25 L11013** will result in **G25 L1101** while transmission.

Remedy
Insert missing digit manually or use only two digit program numbers for subprograms.

G27 Jump Instruction



G27 causes a jump in the program run. With the parameter L a block number will be determined as jump target.

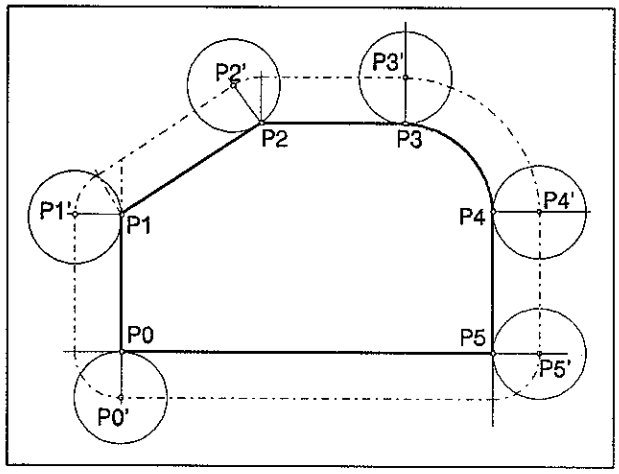
Example
N100 G27 L250

The program jumps from block N100 to block N250.

N 4	G40, G41, G42

G40 - G42 Tool Path Compensation

G40 Cancel Tool Path Compensation
G41 Tool Path Compensation Left
G42 Tool Path Compensation Right



Contour points and base points for the tool path

Purpose of the Tool Path Compensation

The contour of the workpiece is dimensioned in the technical drawing (P0, P1, P2, P3, P4, P5).

To produce this contour the tool must traverse an eqidistant path to the contour.

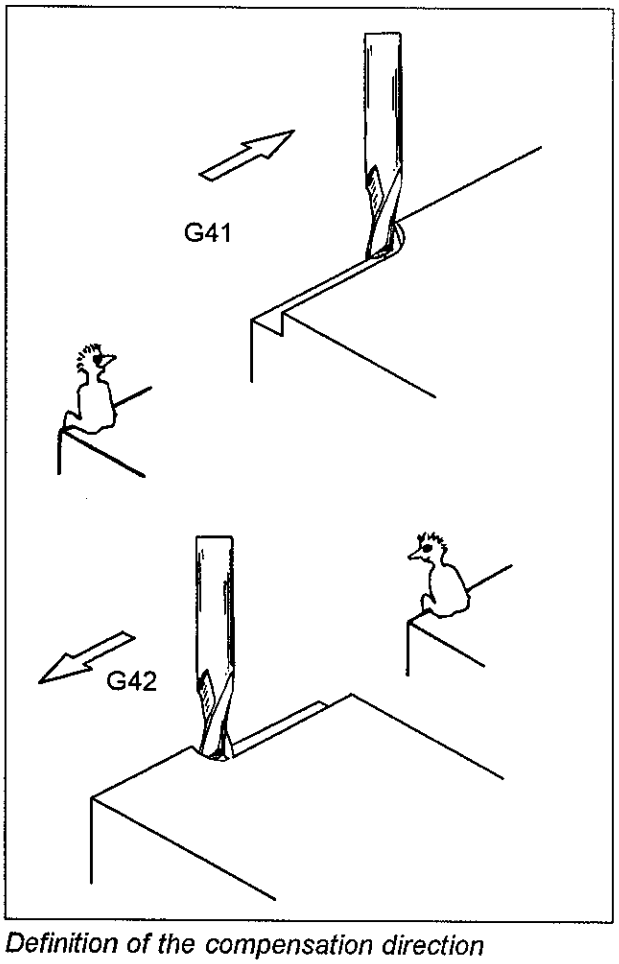
The base points for this equidestant path must be calculated (P0', P1', P2', P3', P4', P5').

With programmed tool path compensation the control calculates this base points automatically.

Required Details

The control compensates the effect of the tool radius. It computes traversing pathes which produce the actually programmed contour.
Therefore the control needs the necessary informations.
These are:

- tool radius R
  Input in mm (inch) with decimal point under the R address in the tool offset register
- tool path compensation left or right



Definition of the compensation direction

Definition G41, G42

G41 Tool path compensation left:

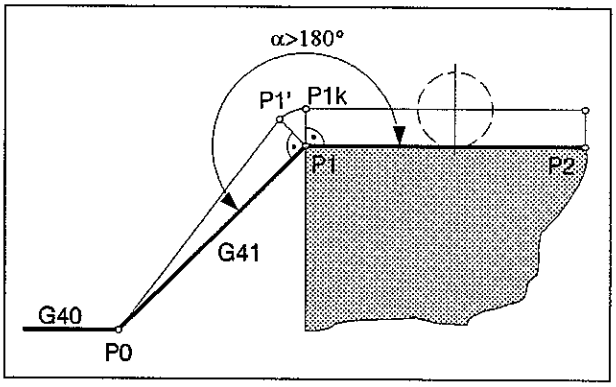
The tool is located to the left of the workpiece when viewed in the direction of the relative tool movement.
Rule:
"Sit down on the workpiece and gaze in feed direction, if the tool is at the left side - G41."

G42 Tool path compensation right:

The tool is located to the right of the workpiece when viewed in the direction of the relative tool movement.
Rule:
"Sit down on the workpiece and gaze in feed direction, if the tool is at the right side - G42."

G40, G41, G42 are self-holding functions, initial status is G40.

Tool pathes with Calling the Tool Path Compensation



Approach angle > 180°

Angle between programmed path P_0P_1 and path P_1P_2 is larger than 180°.

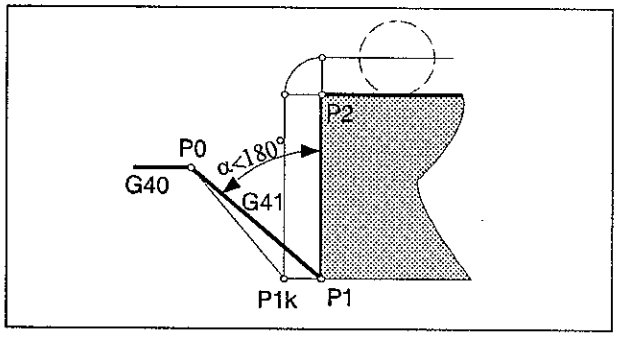
Block N100
The tool traverses with the centre point on point P_0.

Block N110
The tool traverses with the centre point on P_1', than on an arc to P_{1K}.
The arc radius is equal to the radius of the tool.
P_1P_1' is a normal line to P_0P_1 in the point P_1.
P_1P_{1K} is a normal line to P_1P_2.

The tool circumference touches P_1.

Programming:

N....	G40			
N100	G00	X_{P0}	Y_{P0}	
N110	G01	X_{P1}	Y_{P1}	G41
N120	G01	X_{P2}	Y_{P2}	



Approach angle < 180°

The angle between the programmed path P_0P_1 and path P_1P_2 is less than 180°.

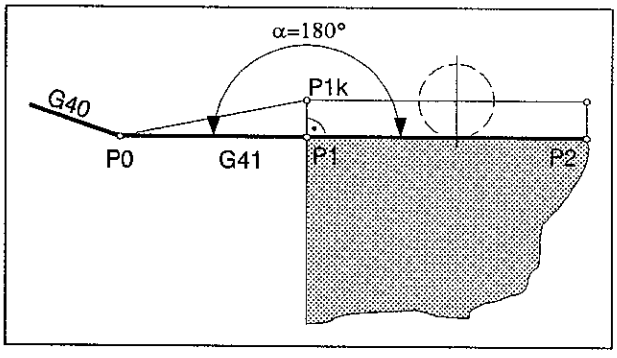
Block N100
The tool traverses with the centre point on point P_0.

Block N110
The tool traverses with the centre point on P_{1K}.
P_1P_{1K} is a normal line to P_1P_2.

The tool circumference touches P_1.

Programming:

N....	G40			
N100	G00	X_{P0}	Y_{P0}	
N110	G01	X_{P1}	Y_{P1}	G41
N120	G01	X_{P2}	Y_{P2}	



Approach angle = 180°

Angle between programmed path P_0P_1 and path P_1P_2 is 180°.

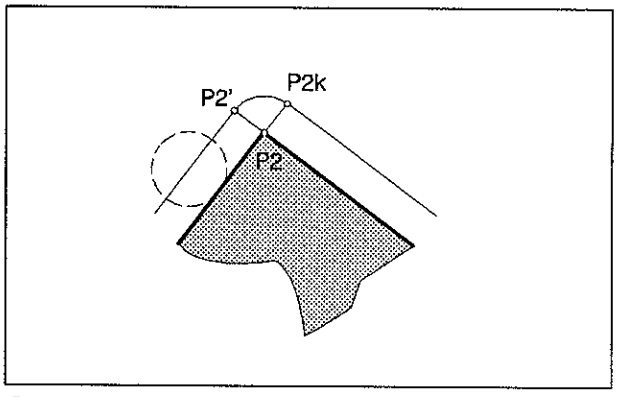
Block N100
The tool traverses with the centre point on point P_0.

Block N110
The tool circumference touches point P_1.

Programming:

N....	G40			
N100	G00	X_{P0}	Y_{P0}	
N110	X_{P1}	Y_{P1}	G42	
N120	G01	X_{P2}	Y_{P2}	

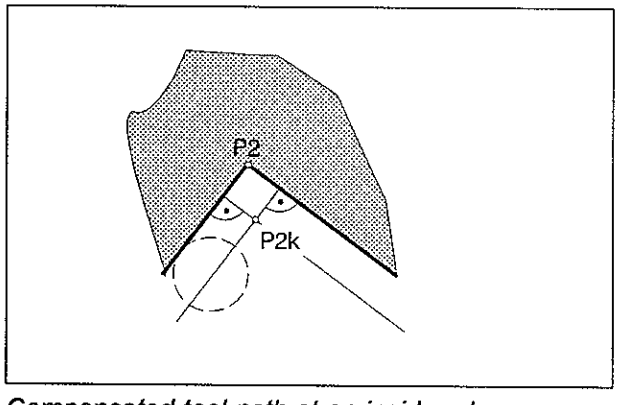
Tool Pathes with Active Tool Path Compensation



Compensated tool path at an outside edge

Outside edge

The tool centre moves to end point of the equidistant path (P_2') and rolls around the edge to the start point of the next traverse movement (P_{2k}).

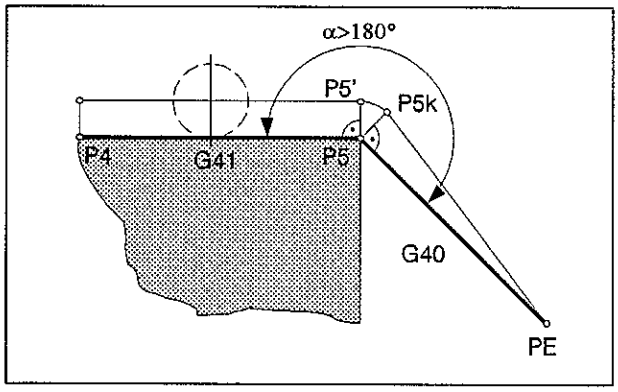


Compensated tool path at an inside edge

Inside edge

The tool centre moves to intersection point of the equidistant pathes (P_{2k}).

Tool Pathes with Cancellation of the Tool Path Compensation

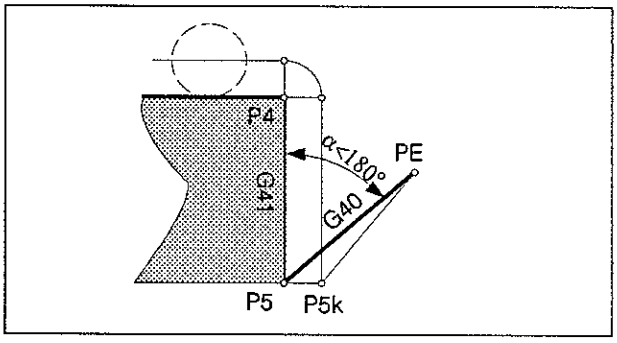


Cancellation angle > 180°

Block N140
The centre point traverses on point P_5' to point P_{5K}. P_{5K} is on the normal to the path P_5P_E.

Block N150
The tool traverses with the theoretical cutter point on point P_E.

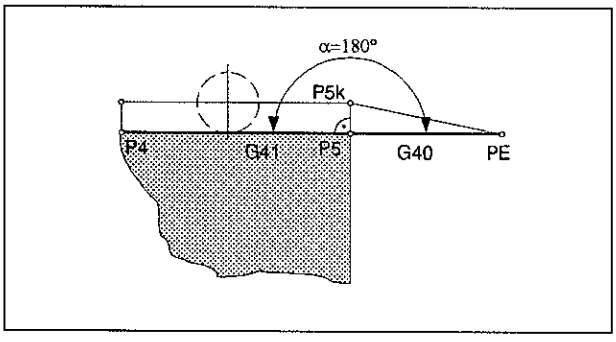
Programming:
N.... G41
N140 G01 X_{P5} Y_{P5}
N150 G00 X_{PE} Y_{PE} G40



Cancellation angle < 180°

Block N150
The tool traverses with the centre point on point P_E.

Programming:
N.... G41
N140 G01 X_{P5} Y_{P5}
N150 G00 X_{PE} Y_{PE} G40



Cancellation angle = 180°

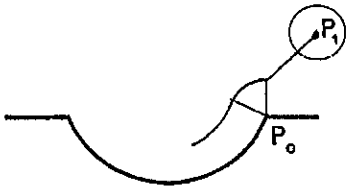
Block N150
The tool traverses with the centre point on point P_E.

Programming:
N.... G41
N140 G01 X_{P5} Y_{P5}
N150 G00 X_{PE} Y_{PE} G40

Syntax Specifications

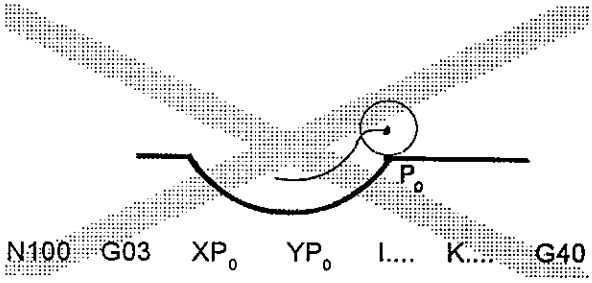
G40, G41, G42 must be programmed only in conjunction with G00, G01.

Right



N100 G03 XP_0 YP_0 I.... K....
N110 G01 XP_1 YP_1 G40

Wrong



N100 G03 XP_0 YP_0 I.... K.... G40

The first G00, G01 block must be programmed at the latest 5 blocks after G40, G41, G42.

Right

N100 G41 G00 X.... Y....

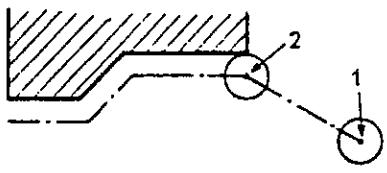
N100 G41
N110 M03
N120 F120
N130 G00 X.... Y....

Wrong

N100 G41
N110 M03
N120 F120
N130 S2000
N140 M08
N150 G04 $D_4$10
N160 G01 X.. Y..

Alarm 500
...more than 5 blocks

In the G00, G01 block a change of the X or Y or XY value must be programmed.



With G41/G42 a "traverse command to the start point" should be programmed.
That means the tool position (1) while the selection of G41/G42 must be different to the start point (2) of the tool path compensation.

If this traverse command is not programmed, alarm 520 can occur.

N.... G00 X_1 Y_1 } both traverse com-
N.... G41/G42 G00 X_2 Y_2 } mands are necessary

With selected tool path compensation at least two blocks with a change of the X or Y or XY value must be programmed.

Right

N100 G41 G00 X_{P0} Y_{P0}
N110 G01 X_{P1} Y_{P1}
N120 X_{P2} Y_{P2}
N130 X_{P3} Y_{P3}
N140 G40 G00 X_{P4} Y_{P4}

Wrong

N100 G41 G01 X_{P0} Y_{P0}
N110 G40 G00 X_{P1} Y_{P1}

Alarm 510!

A direct change from G41 to G42 and vice versa is not allowed, the tool path compensation must be cancelled before a change.

Geometry Specification

The tool radius must not be too large in relation to the smallest contour element.

N 4	G50

G50 Cancel Scale Factor

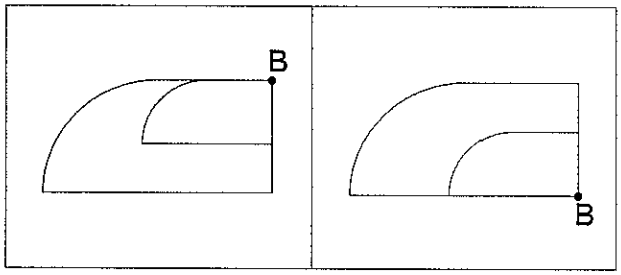
N 4	G51	X U ±43	Y V ±43	Z W ±43	P_7 43
		[mm] [inch]	[mm] [inch]	[mm] [inch]	

G51 Scale Factor

A tool path can be enlarged or reduced related to a reference point B.

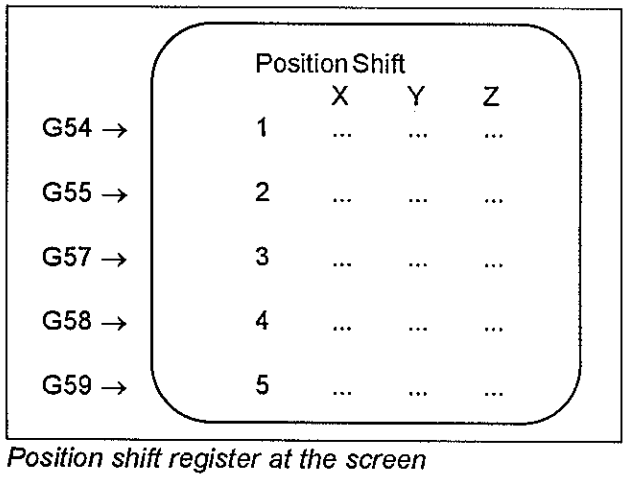
Required details

- Tool path:
  The tool path to be enlarged or reduced is written in the program between G51 and G50. It can be a open or closed contour.
- Reference point B:
  The reference point is described absolute with X, Y, Z or incremental with U, V, W.
  It can be at any position.
- Scale factor:
  With P_7 the factor for enlarging or reducing is determined. Input range from >0 to 9999.999.
  e.g. M1:2 $P_7 = 0.5$ reducing
  M7.38:1 $P_7 = 7.38$ enlarging



Scaling related to the reference point B

G53-G59 Zero Offset with Position Shift Register



Position shift register at the screen

The shift values are written with X, Y and Z in the position shift register.

With G54, G55, G57, G58, G59 you can call the shift values of the position shift register.

G53 cancels G54 and G55.
G56 cancels G57, G58 and G59.

Details see chapter Programming Basics, zero offsets.

G70 Measuring in Inch

If G70 is programmed at the program start all dimensions will be computed as inches.

Notes for G70/G71

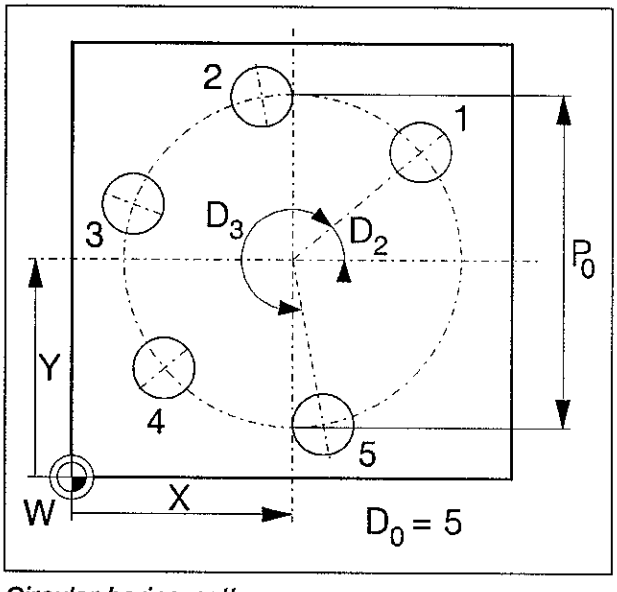
- In the user monitor (MON) the initial status G70 or G71 can be determined with the parameter O_{11} Bit 0.
- G70 and G71 are selfholding commands from the same group.
- Setting at works:
  USA G70
  Other countries G71

G71 Measuring in mm

If G71 is programmed at the program start all dimensions will be computed metrical.

G72 Definition Circular Boring Pattern

N4	G72	X U ±43	Y V ±43	P_0 ±43	D_0 5	D_2 4	D_3 4	D_7 1	F 4
		[mm] [inch]	[mm] [inch]	[mm] [inch]		[°x10]	[°x10]		[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]



Circular boring pattern

Programming

N ..... block number
G72 ..... definition circular boring pattern
X (U), Y (V) } Absolute resp. incremental coordinates of the pattern centre point
P_0 ..... circular boring pattern diameter
D_0 ..... number of borings
D_2 ..... start angle (Def.) 0 - 3600 (= 0° - 360°) related to the +X direction
D_3 ..... total angle (Def.) 0 - 3600 (= 0° - 360°)
D_7 ..... parameter take-over (Def.), 0 or 1
If D_7 = 1 is programmed, parameters from a preceding pattern definition will be taken over for the actual pattern definition. X, Y U, V values will not be taken over. If D_7 = 1 is not programmed, all other parameters must be programmed. For programming D_7 = 1 a preceding pattern definition must exist.
F ..... feed

G73 Execute Circular Boring Pattern

N4	G73	G...	Z W ±43	.....
			[mm] [inch]	

Programming

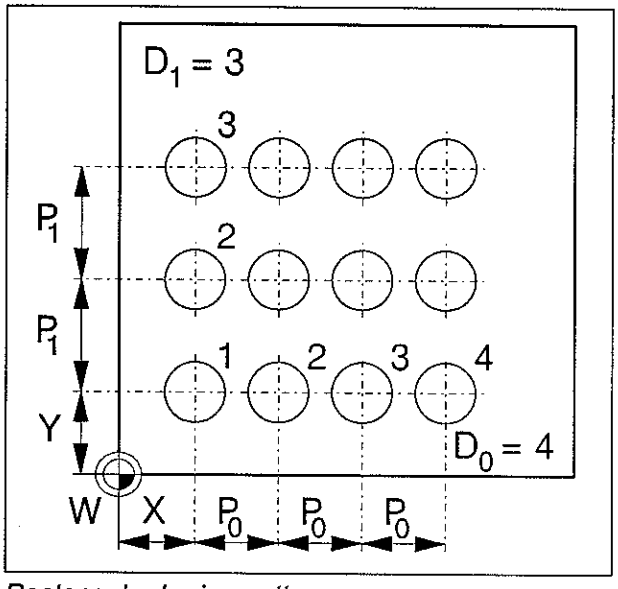
N ..... block number
G73 ..... execute circular boring pattern
G... ..... cycle for single boring
Z, W ..... boring depth
.... parameter of the boring cycle (only in Z)

The execution command follows the definition command of the circular boring pattern.
Machining starts with boring no. 1 and will be continued counterclockwise.
After machining the last boring the tool stops over the last boring.

Note:
The boring pattern elements must be programmed in Z(W) direction only (including the cycle parameter).

G74 Definition Rectangular Boring Pattern

N4	G74	X U ±43	Y V ±43	P_0 ±43	P_1 ±43	D_0 5	D_1 5	D_7 1	F 4
		[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]				[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]



Rectangular boring pattern

Programming

N ..... block number
G74 ..... definition rectangular boring pattern
X (U), Y (V) } Absolute resp. incremental centre coordinates of the first pattern element
P_0 ..... distance of the borings in X
P_1 ..... distance of the borings in Y
D_0 ..... number of borings in X
D_1 ..... number of borings in Y
D_7 ..... parameter take-over (Def.), 0 or 1
If D_7 = 1 is programmed, parameters from a preceding pattern definition will be taken over for the actual pattern definition. X, Y U, V values will not be taken over. If D_7 = 1 is not programmed, all other parameters must be programmed. For programming D_7 = 1 a preceding pattern definition must exist.
F ..... feed

G75 Execute Rectangular Boring Pattern

N4	G75	G...	Z W ±43	.....
			[mm] [inch]	

Programming

N ..... block number
G75 ..... execute rectangular boring pattern
G.... ..... cycle for single boring
Z, W ..... boring depth
.... parameter of the boring cycle (only in Z)

The execution command follows the definition command of the circular boring pattern.
Machining starts with the boring defined by X(U) and Y(V).
After machining the last boring the tool stops over the last boring.
Note:
The boring pattern elements must be programmed in Z(W) direction only (including the cycle parameter).

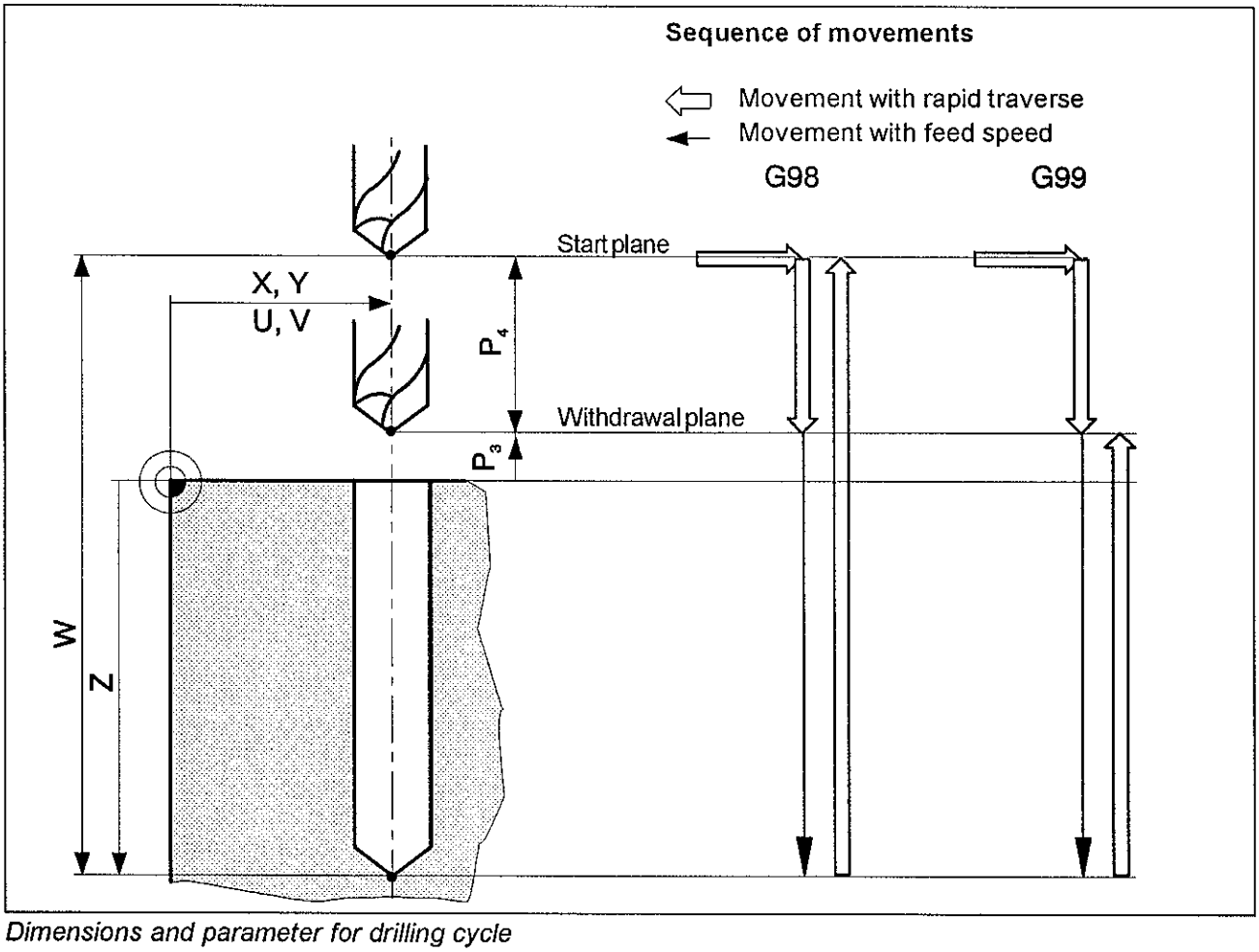
G81 Drilling Cycle

N 4	G81	G98 G99	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Programming

N..... block number
G81..... drilling cycle
G98..... retraction to start plane
G99..... retraction to withdrawal plane
X (U) ... traverse movement before drilling
Y (V) ... traverse movement before drilling
Z, W.... absolute (incremental) drilling depth
P_3..... start of machining feed movement in Z
P_4..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle.
- Last Z position of the tool

F ..... feed



Dimensions and parameter for drilling cycle

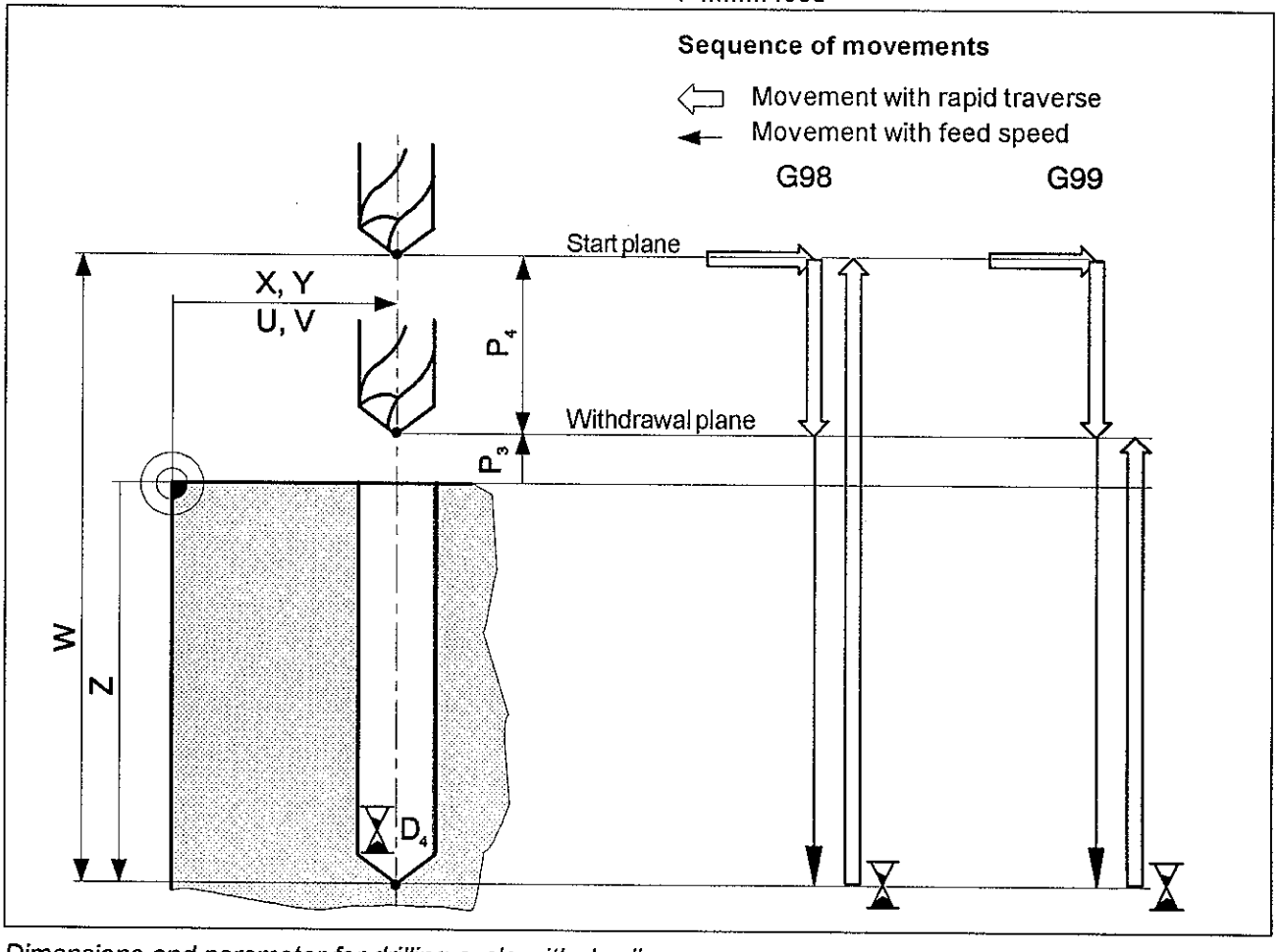
G82 Drilling Cycle with Dwell

N 4	G82	G98 G99	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	D_4 5	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[1/10 s]	[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Programming

N..... block number
G82..... drilling cycle with dwell
G98..... retraction to start plane
G99..... retraction to withdrawal plane
X (U) ... traverse movement before drilling
Y (V) ... traverse movement before drilling
Z, W.... absolute (incremental) drilling depth
P_3..... start of machining feed movement in Z
P_4..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle
- Last Z position of the tool

D_4 ..... dwell at the drilling ground
F ..... feed



Dimensions and parameter for drilling cycle with dwell

G83 Withdrawal Drilling Cycle

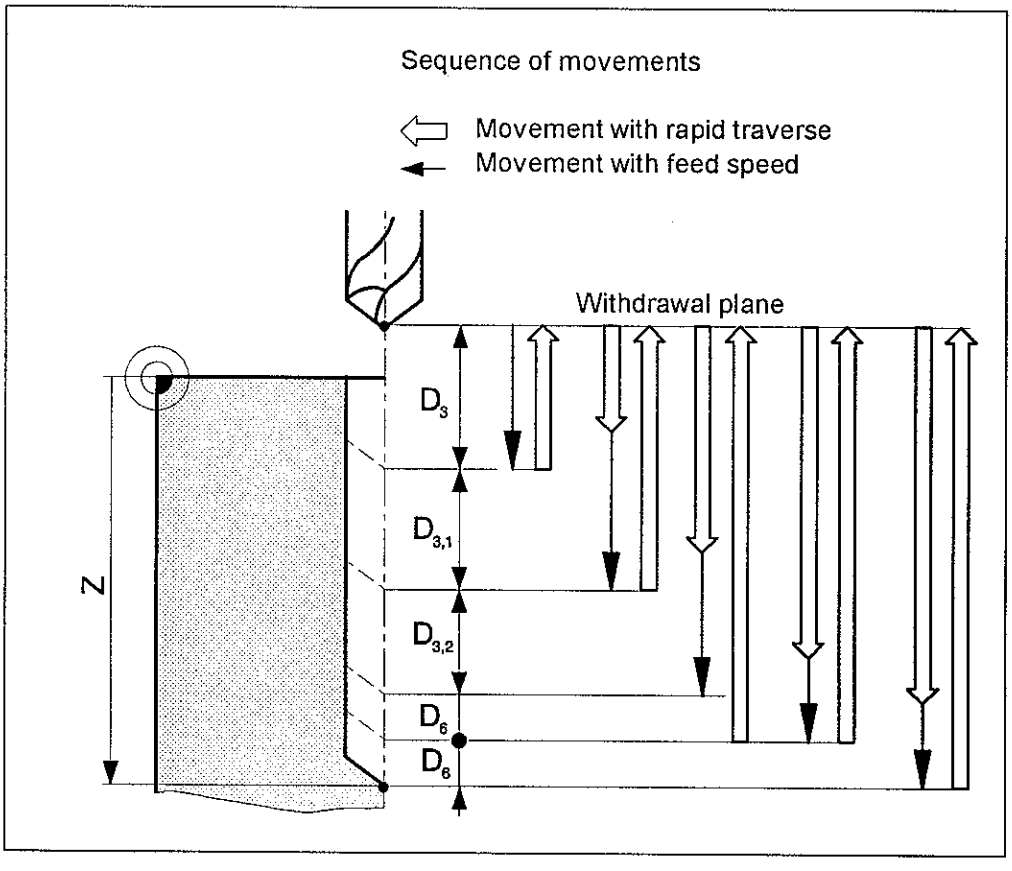
N 4	G83	G98 G99	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	D_3 5	D_5 5	D_6 5	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[µm] [1/10000 inch]	[%]	[µm] [1/10000 inch]	[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Settings:
The minimum infeed of 100 µm is activ if D_6 was not programmed.

Note:
Programming D_5 reduces the infeed by the entered percentage. The control calculates the infeed reduction as following:
$D_{3xn}=D_{3xn-1}xD5/100$

Programming
N..... block number
G83..... withdrawal drilling cycle
G98 ..... retraction to start plane
G99 ..... retraction to withdrawal plane
X, U .... traverse movement before drilling
Y, V .... traverse movement before drilling
Z, W .... absolute (incremental) drilling depth
P_3..... start of machining feed movement in Z
P_4..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle
- Last Z position of the tool

D_3 ..... drilling depth of the 1. cut (Def.)
D_5..... percentage of cutting depth reduction (Def.)
D_6 ..... minimum drilling depth (Def.)
F ..... feed



Dimensions and parameter for withdrawal drilling

G84 Tapping Cycle

N 4	G84	G98 G99	X U ±43	Y V ±43	Z W ±43	P_3 P_4 ±43	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[µm/rev] [1/10000 inch/rev]

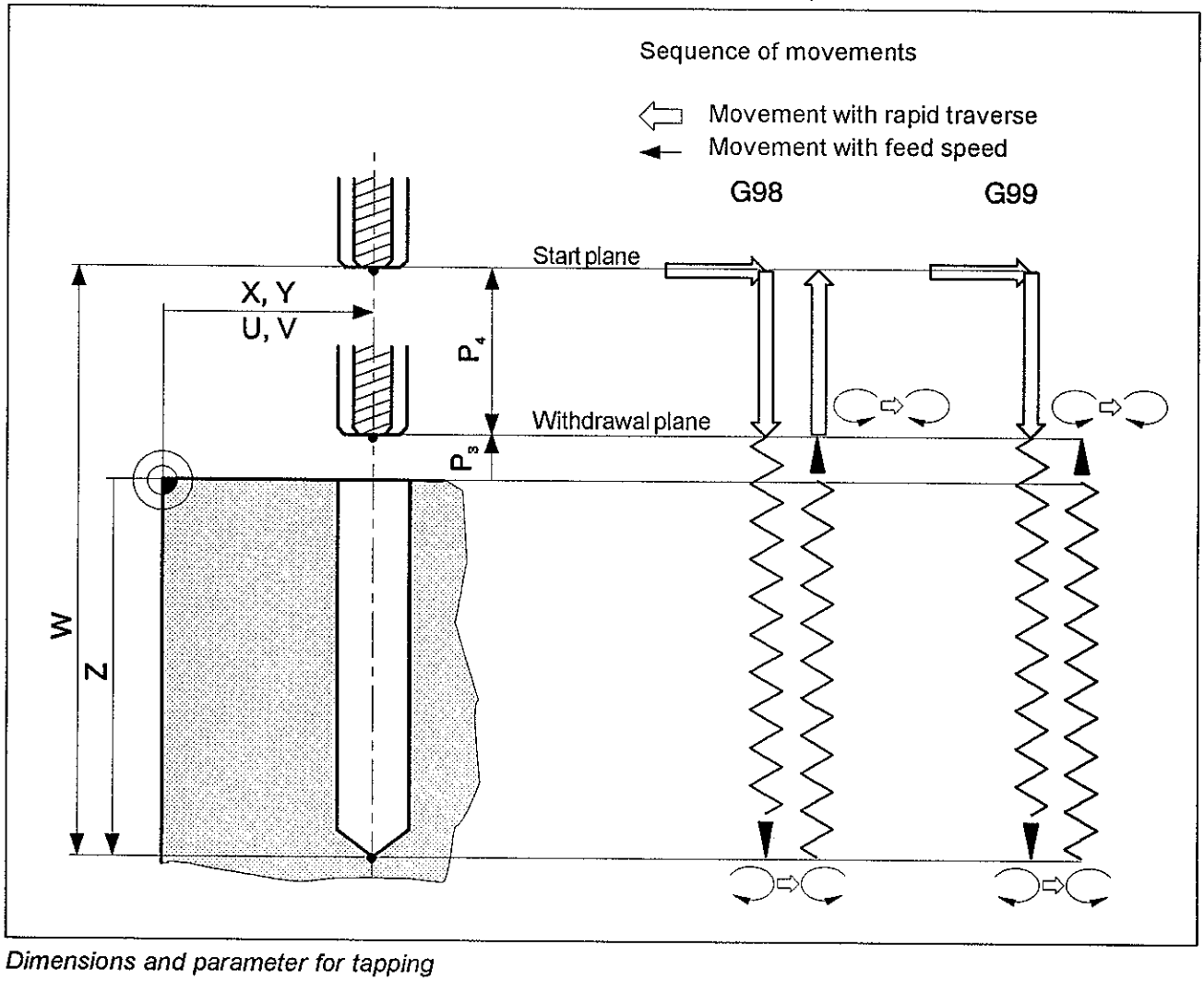
Programming like G81.
For a right thread M03 has to be programmed before, for a left thread M04 has to be programmed before.

Sequence of machining:
The tool traverses down to the programmed end depth.
The turning direction will be changed.
The tool traverses back to the start or withdrawal plane.
The turning direction changes to the origin direction.

For this cycle a tap holder with length compensation must be used, otherwise the thread could be damaged or the tool could break.

Programming
N..... block number
G84..... tapping cycle
G98..... retraction to start plane
G99..... retraction to withdrawal plane
X (U) ... traverse movement before tapping
Y (V) ... traverse movement before tapping
Z, W.... absolute (incremental) tapping depth
P_3..... start of tapping feed movement in Z
P_4..... start of tapping feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle.
- Last Z position of the tool

F ..... thread pitch



Dimensions and parameter for tapping

G86 Chip Break Drilling Cycle

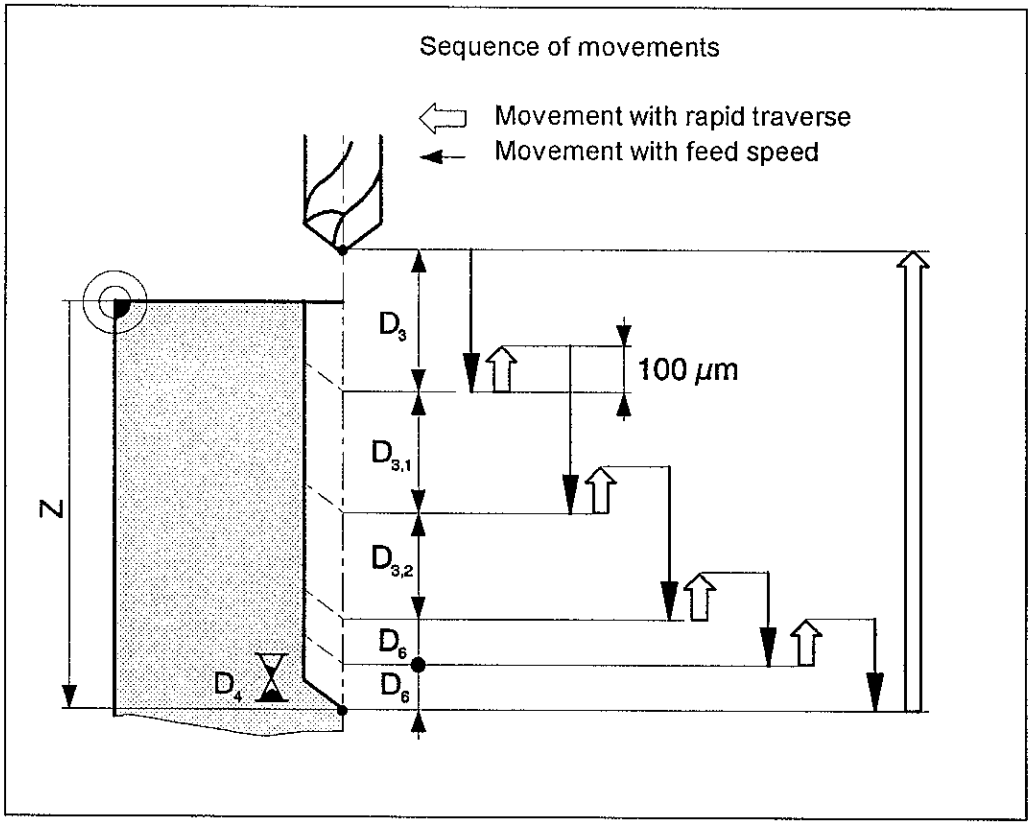
N 4	G86	G98 G99	X ±43 U ±43	Y ±43 V ±43	Z ±43 W ±43	P_3 ±43 P_4 ±43	D_3 5	D_5 5	D_6 5	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[µm] [1/10000 inch]	[%]	[µm] [1/10000 inch]	[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Settings:
The minimum infeed of 100 µm is activ if D_6 was not programmed.
The tool retracts after every infeed for 100 µm.

Note:
Programming D_5 reduces the infeed by the entered percentage. The control calculates the infeed reduction as following:
$D_{3xn}=D_{3xn-1}xD5/100$

Programming
N..... block number
G86..... chip break drilling cycle
G98 ..... retraction to start plane
G99 ..... retraction to withdrawal plane
X, U .... traverse movement before drilling
Y, V .... traverse movement before drilling
Z, W ... absolute, incremental drilling depth
P_3..... start of machining feed movement in Z
P_4..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle
- Last Z position of the tool

D_3 ..... drilling depth of the 1. cut (Def.)
D_4 ..... dwell at the drilling ground (Def.)
D_5..... percentage of cutting depth reduction (Def.)
D_6 ..... minimum drilling depth (Def.)
F ..... feed



Dimensions and parameter for chip break drilling

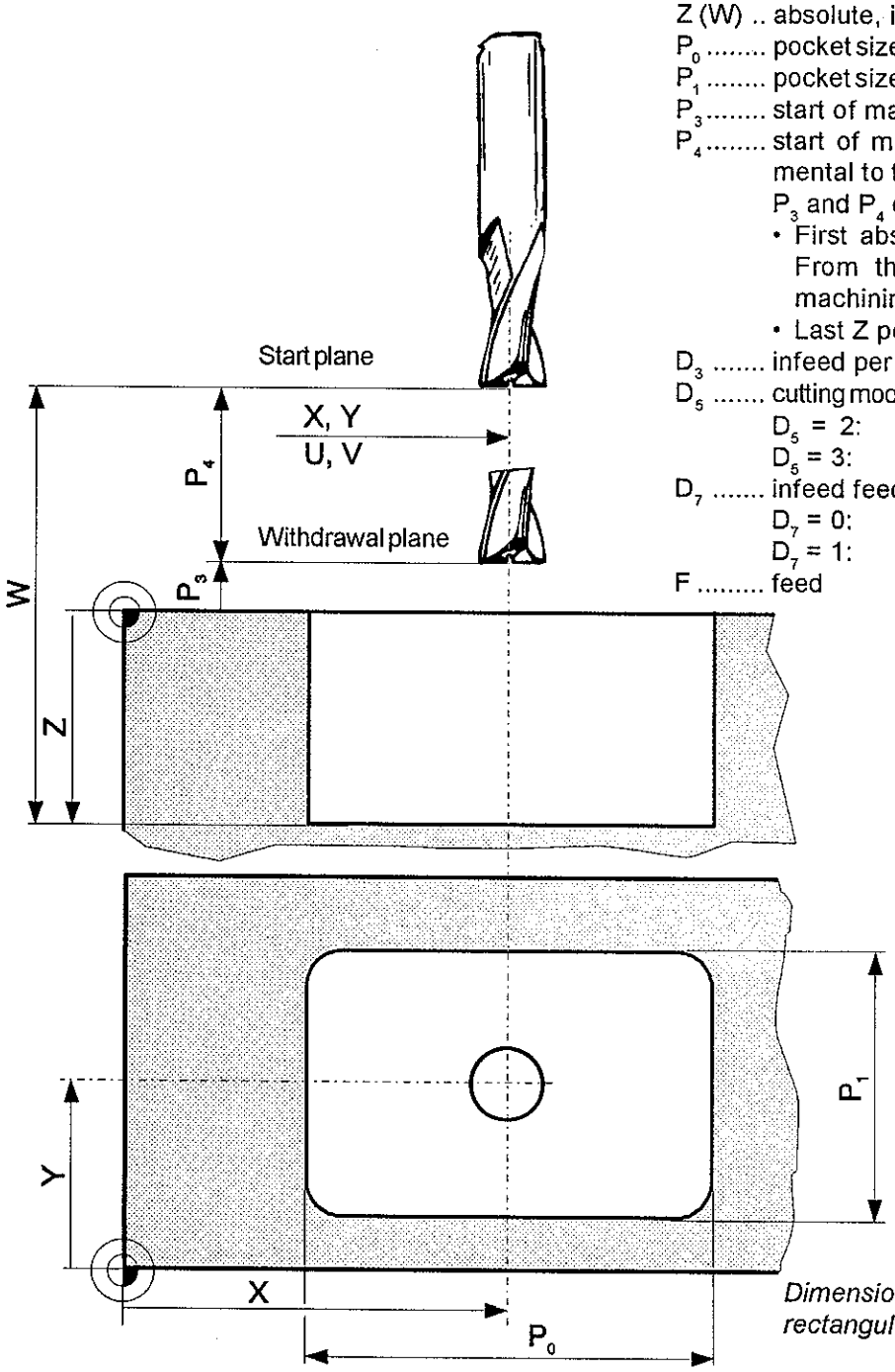
G87 Rectangular Pocket Milling Cycle

N 4	G87	G98 G99	X U ±43	Y V ±43	Z W ±43	P_0 43	P_1 43	P_3 P_4 ±43	D_3 5	D_5 1	D_7 1	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[µm] [1/10000 inch]			[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Programming

N..... block number
G87..... rectangular pocket milling cycle
G98..... retraction to start plane
G99..... retraction to withdrawal plane
X (U) ... pocket centre in X
Y (V) ... pocket centre in Y
Z (W) .. absolute, incremental depth of the pocket
P_0 ..... pocket size in X
P_1 ..... pocket size in Y
P_3..... start of machining feed movement in Z
P_4..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle
- Last Z position of the tool

D_3 ..... infeed per cut
D_5 ..... cutting mode
D_5 = 2: climb milling
D_5 = 3: opposed milling (Def.)
D_7 ..... infeed feed in Z
D_7 = 0: infeed with programmed feed F
D_7 = 1: infeed with F/2 (Def.)
F ..... feed



Dimensions and parameter for rectangular pocket milling

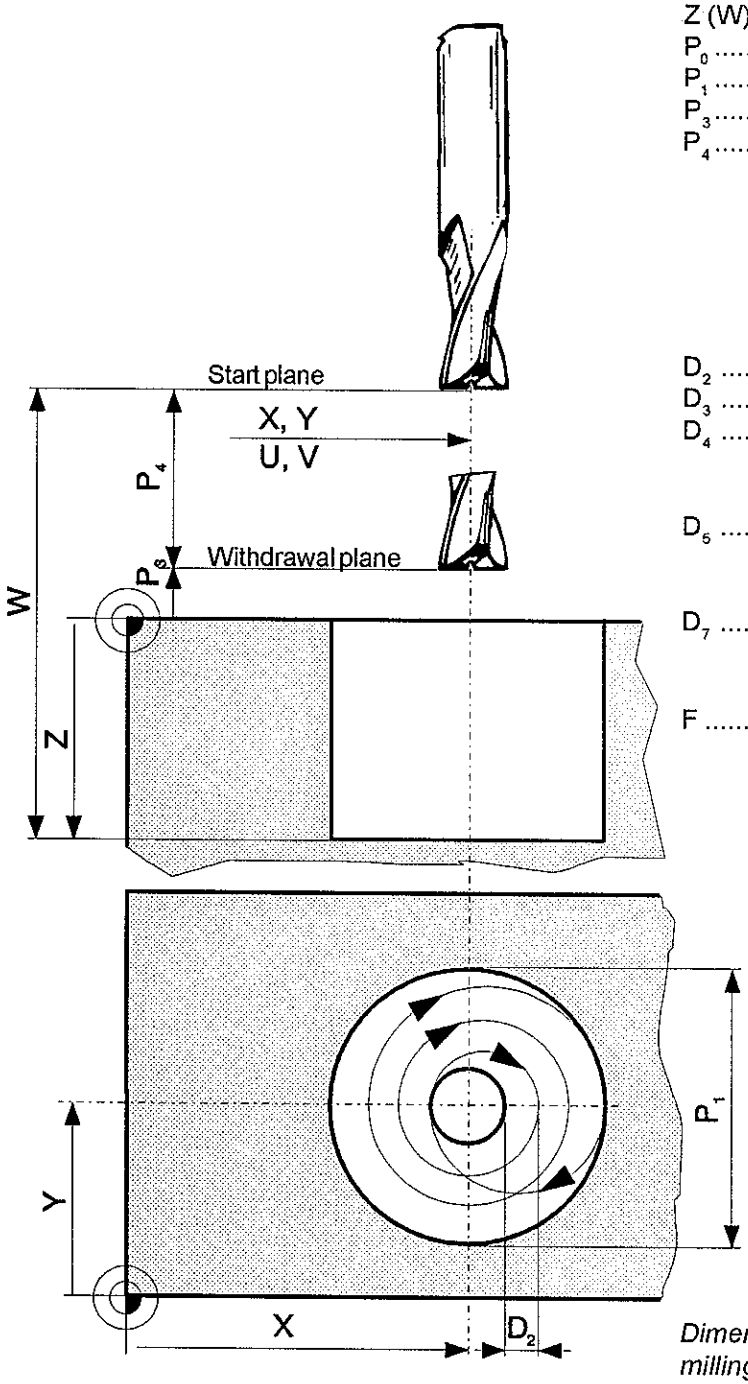
G88 Circular Pocket Milling Cycle

N 4	G88	G98 G99	X U ±43	Y V ±43	Z W ±43	P_1 43	P_3 P_4 ±43	D_2 5	D_3 5	D_4 1	D_5 1	D_7 1	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[µm] [1/10000 inch]					[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Programming

N..... block number
G88..... circular pocket milling cycle
G98..... retraction to start plane
G99..... retraction to withdrawal plane
X (U) ... pocket centre in X
Y (V) ... pocket centre in Y
Z (W) .. absolute, incremental depth of the pocket
P_0 ..... pocket size in X
P_1 ..... pocket size in Y
P_3 ..... start of machining feed movement in Z
P_4 ..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle
- Last Z position of the tool

D_2 ..... horicontal infeed, D2 < tool diameter
D_3 ..... infeed per cut
D_4 ..... feed at the pocket circumference
D_4 = 0: infeed with programmed feed F
D_4 = 1: infeed with F/2 (Def.)
D_5 ..... cutting mode
D_5 = 2: climb milling
D_5 = 6: opposed milling (Def.)
D_7 ..... infeed feed in Z
D_7 = 0: infeed with programmed feed F
D_7 = 1: infeed with F/2 (Def.)
F ..... feed



Dimensions and parameter for circular pocket milling

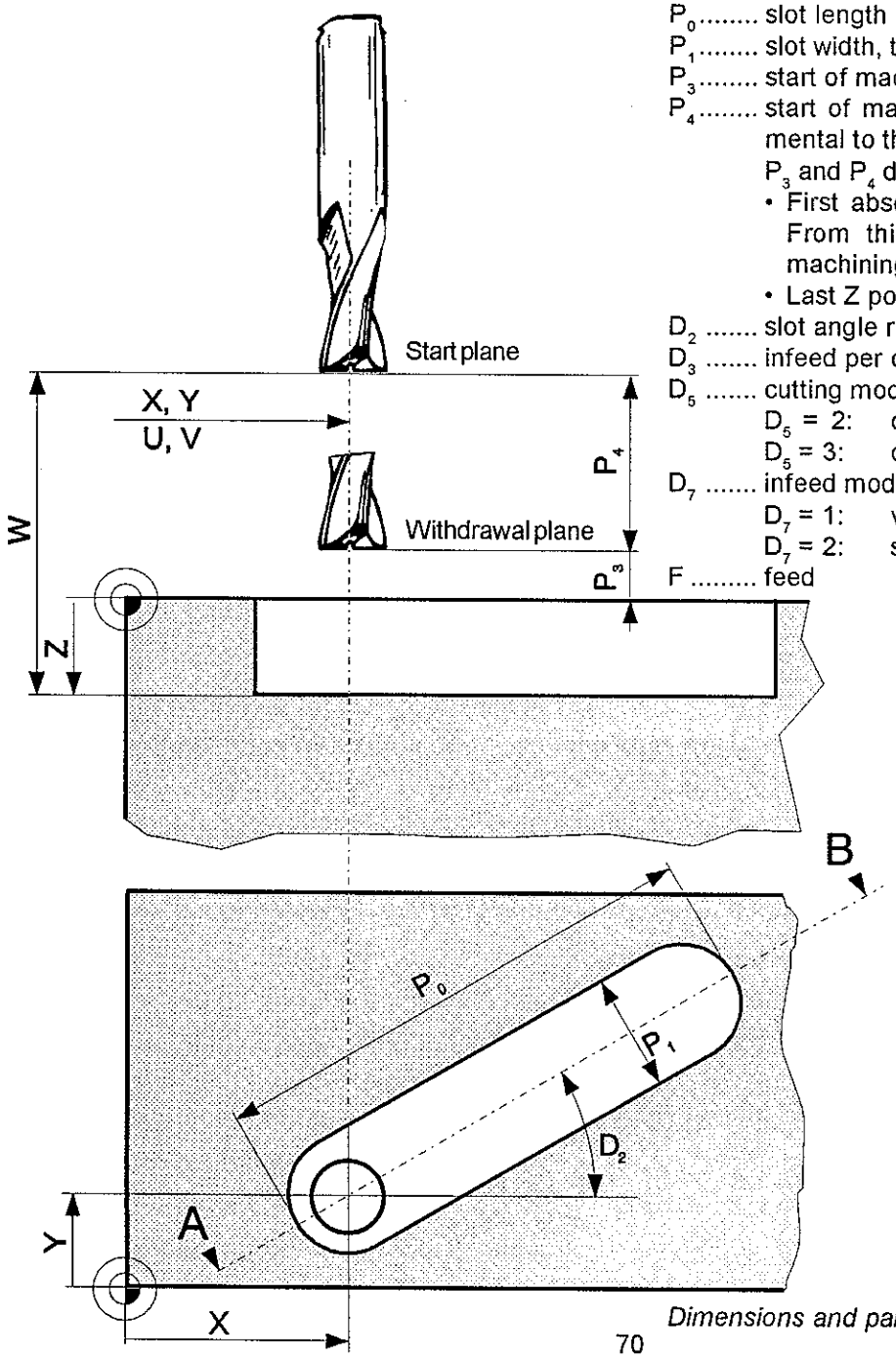
G89 Slot Milling Cycle

N 4	G89	G98 G99	X U ±43	Y V ±43	Z W ±43	P_0 43	P_1 43	P_3 P_4 ±43	D_2 4	D_3 5	D_5 1	D_7 1	F 4
			[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[mm] [inch]	[1/10 °]	[µm] 1/10000 inch]			[mm/min] [1/100 inch/min] [µm/rev] [1/10000 inch/rev]

Programming

N..... block number
G89..... slot pocket milling cycle
G98..... retraction to start plane
G99..... retraction to withdrawal plane
X (U) ... slot start in X
Y (V) ... slot start in Y
Z (W) .. absolute, incremental depth of the slot
P_0..... slot length
P_1..... slot width, tool diameter ≤ P_1 ≤ 3 x tool dia.
P_3..... start of machining feed movement in Z
P_4..... start of machining feed movement incremental to the start plane
P_3 and P_4 define the following position:
- First absolute G00 position of the tool. From this position occurs the further machining of the cycle
- Last Z position of the tool

D_2 ..... slot angle related to +X direction
D_3 ..... infeed per cut
D_5 ..... cutting mode
D_5 = 2: climb milling
D_5 = 3: opposed milling (Def.)
D_7 ..... infeed mode in Z
D_7 = 1: vertical infeed (Def.)
D_7 = 2: slant infeed
F ..... feed



Dimensions and parameter for slot milling

G92 Set Zero Offset 5

N 4	G92	X / U ±43	Y / V ±43	Z / W ±43
		[mm] [inch]	[mm] [inch]	[mm] [inch]

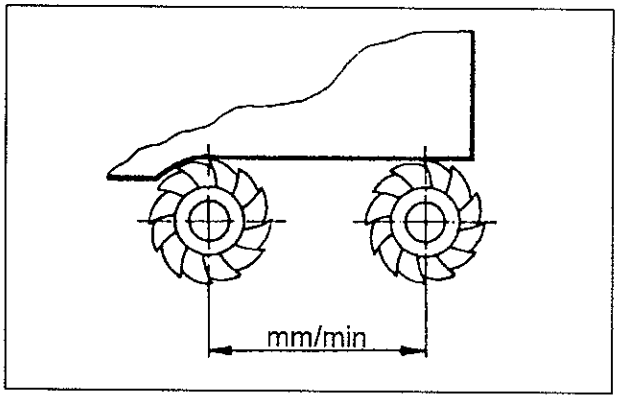
With G92 shift values are written in PSO 5. The shift values are determined with X, Y and Z. When running a G92 block the new X, Y and Z values overwrite the old values in PSO 5.
If the values for the G92 block are given with U, V and W, the U, V and W are added to resp. subtracted from the old values.

Activation of the zero offset
With G59 the zero offset will be executed.

Note
G59 must not be programmed in the same block as G92.

Details see chapter zero offsets.

N 4	G94



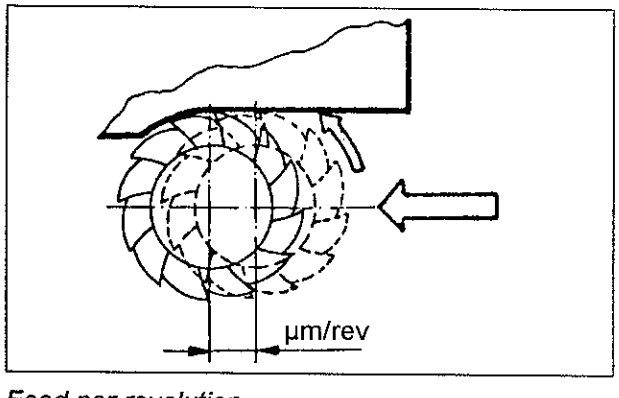
Feed per minute

G94 Feed in mm/min (1/100 inch/min)

G94 is the initial status of the control. When G94 is programmed, the entered feed values are interpreted as mm/min (1/100 inch/min).

Possible input range see technical data of the machine!

N 4	G95



Feed per revolution

G95 Feed in µm/rev (1/10000 inch/rev)

If G95 is programmed, all feed values will be interpreted as µm/rev (1/10000 inch/rev) automatically.

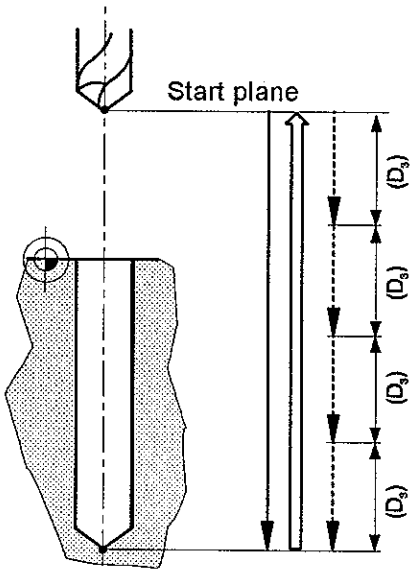
G98 Retraction to Start Plane

G99 Retraction to Withdrawal Plane

For cycles a withdrwal plane can be programmed with the parameters P_3 and P_4.

When a withdrawal plane is programmed for a cycle, the D parameters will be active below this withdrawal plane.

G98, no P_3 or P_4 programmed

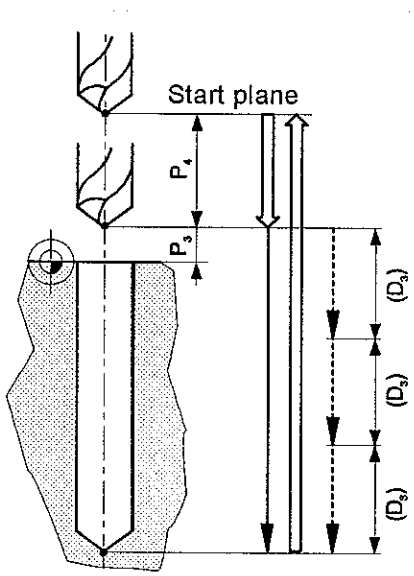


Infeed from start plane down to end depth with G01.

The D parameter (e.g. D_3) are active from the start plane.

Retraction to the start plane.

G98, P_3 or P_4 programmed

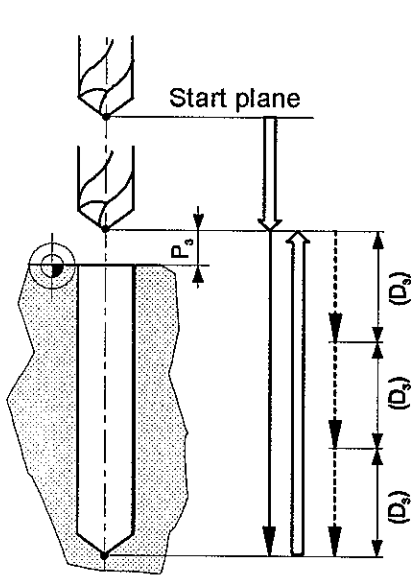


Infeed from start plane down to the withdrawal plane with G00, from the withdrawal plane down to end depth with G01.

The D parameter (e.g. D_3) are active from the withdrawal plane.

Retraction to the start plane.

G99, P_3 programmed



Infeed from start plane down to the withdrawal plane with G00, from the withdrawal plane down to end depth with G01.

The D parameter (e.g. D_3) are active from the withdrawal plane.

Retraction to the withdrawal plane.

With G99 only P_3 can be programmed.

E: M Commands

Programming

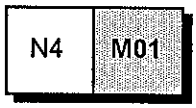
M commands are switching or addidional functions (miscalleneous). The M commands can stand alone in a program block or together with other commands. Commands of the same group cancel each other, that means the M command programmed last cancels the previously programmed M command of the same group. Group division see chapter C "Programming Basics".

Remark
The following pages describe the M commands, which can be called by the software. Whether these M commands are executable depends on the type of the machine and the used accessories.

M00 Programmed Stop

The slides will be stopped, main spindle and coolant will be switched off.
Application: Measuring and tests while the production process etc.

By pressing the key "Cycle Start" the program continues. The main spindle will be switched on with all previously active values automatically after continuing the program.



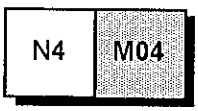
M01 Programmed Stop Conditional

If the key OPT. STOP (Ctrl + x in the numeric key block) was pressed, M01 works like M00, otherwise M00 is not effective.
This command is not available at the original EMCOTRONIC control.

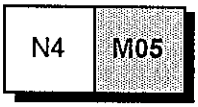
By pressing the key "Cycle Start" the program continues. The main spindle will be switched on with all previously active values automatically after continuing the program.

M03 Milling Spindle ON Clockwise

(viewed in direction working area)



M04 Milling Spindle ON Counterclockwise



M05 Milling Spindle OFF

M30 at the program end activates M05 automatically.

M08 Coolant ON

only PC MILL 100

M09 Coolant OFF

only PC MILL 100
M30 at the program end activates M09.

M17 Subprogram End

The subprogram will be closed with M17. M17 causes a jump-back to the next higher level of the part program.
Details see G25.

M27 Swivel Dividing Head

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

M30 Program End

Effect
Block end / program end, jump back to the program start

M30 causes additionally
- coolant off
- main spindle off
- G40

N4 M38

M38 Precise Stop ON

Effective at: block start

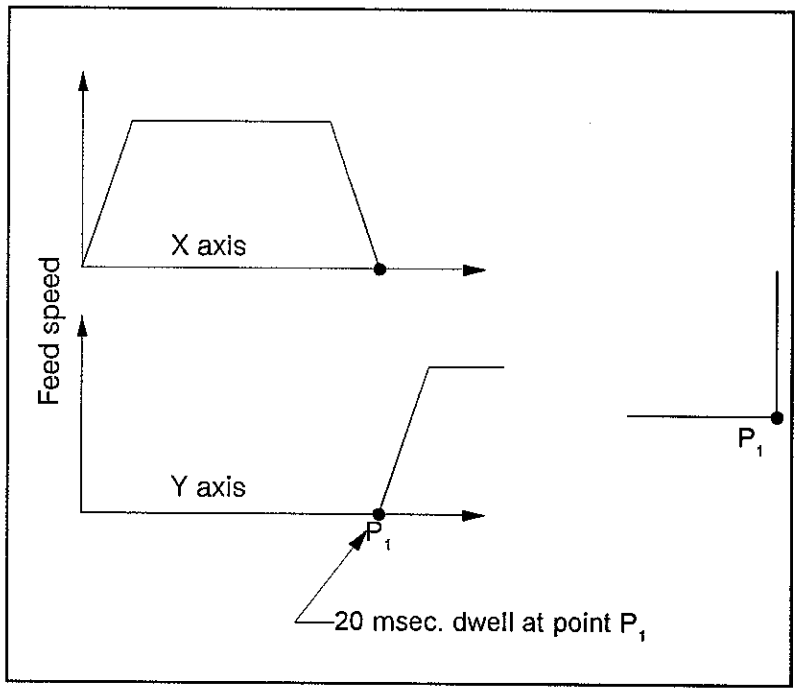
If you want to produce sharp edges, you have to program M38. First the axis movements will be stopped completely at the programmed target point and the the next block will be ran.

Altering the feed speed causes altering the cutting conditions. Complete stopping the slides costs time.

Remark

- Measure the time difference for a workpiece program with and without precise stop.
- The control knows the contents of the next traverse command.

Contour transitition with M38



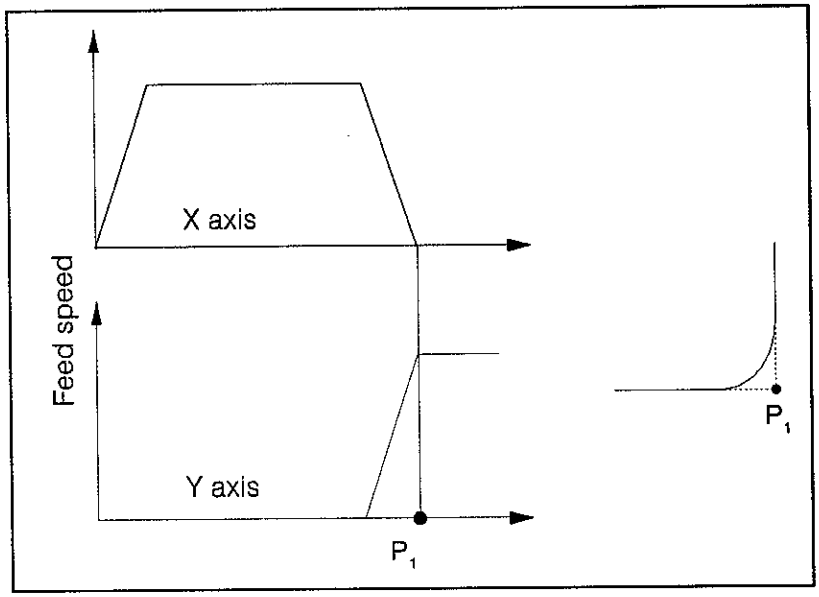
Speed Characteristics with M38

M39 Precise Stop OFF

The EMCOTRONIC TM 02 is designed that before reaching the target in X direction the Y axis will 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 tolerances of the drawing.

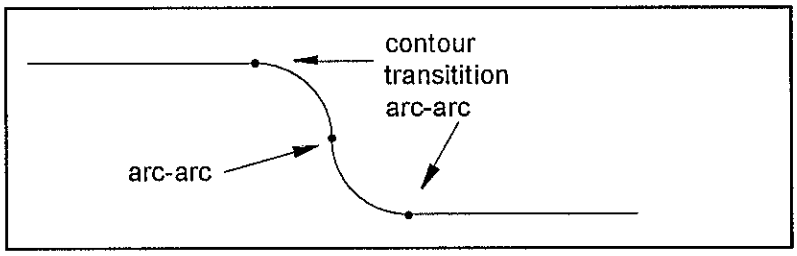
The faster the feed, the larger the transition curves. (see drawing: feed $F_1<F_2<F_3$).

Contour transition with M39



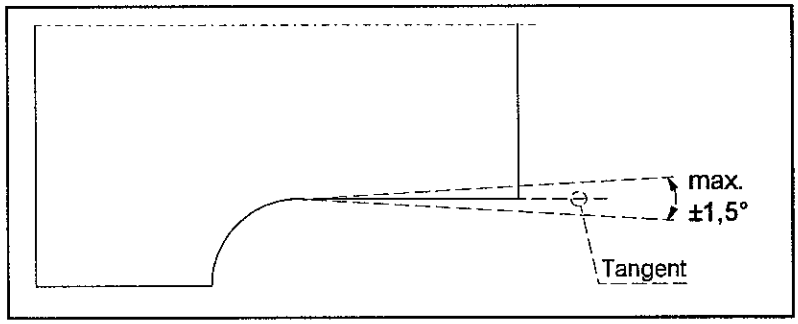
Speed characteristics with M39

Block Transitions to Arcs without Stop

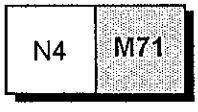


Conditions:

- M39 "precise stop off" must be active.
- The contour transitition must be tangential. A maximum declination of ±1,5° is allowed.

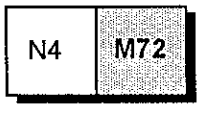


- The feed override switch must not be actuated before the block transitition.
- The movement must not be traversed in rapid traverse.
- When a contour element is too short or the feed too fast, the control has not enough time to calculate the following transitition and a precise stop will be executed.



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.

N4 M90

M90 Cancel Mirror Function

N4 M91

M91 Mirror X Axis

N4 M92

M92 Mirror Y Axis

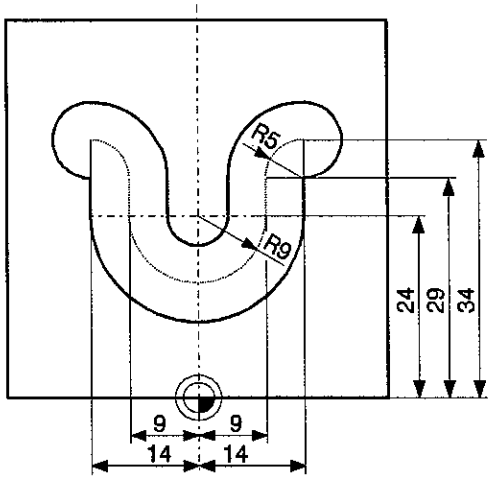
N4 M93

M93 Mirror X and Y Axis

Conditions for mirroring

- In NC blocks which contain a mirror command M90 - M93 nor a circular interpolation command (G02, G03) nor a tool path compensation call command (G41, G42) may be programmed.
- Calling or cancelling a mirror function is allowed with G40 active.
- Mirroring of G02, G03, G41 and G42:
  Depending on the kind of mirror function it is possible that the turning direction of a circular interpolation or the kind of tool path compensation will be altered.
  When an arc is at the beginning of the subroutine, the start point of the arc must be programmed within the subroutine - WinNC does not take over the start position.
- The definition commands for boring patterns (G72, G74) must be programmed with M90.
  The execution command can be mirrored, but for circular boring patterns the parameter D_3 must be D_3=3600 (full circle).

Example



Tool diameter: 10 mm, milling depth: 5 mm

Programming (absolute):

```
N....
N....  M90
N....  G01   Z-5.    F...
N....  G25   L7001
N....  G00   Z5.
N....  M92
N....  G25   L7001
N....  M90
N....
```

Subroutine O 0070:

```
N....  G00   X14.   Y34.
N....  G03   X9.    Y29.   J-5.   F...
N....  G01   X9.    Y24.
N....  G02   X0     Y15.   I-9.
N....  M17
```

Chapter F: User Monitor

In the user monitor (MON) machine and control stati can be altered by the user. The status will be determined by entering parameter.

Group Division of the Parameters in the User Monitor

D	Common Monitor Data
L	Peripherical Data
O	Common Setting Data
R	Reference Positions
T	Language

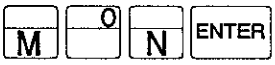
Survey of the Parameters in the User Monitor

D	L	O	R	T
00 02 03 08	00 01 02 03	00 01 05 06 07 08 11	00 01 02	22

Data Input

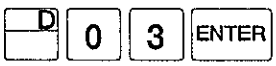
1. Calling the User Monitor

The user monitor (MON) is called in the EDIT mode. An eventually active workpiece program must be cancelled before (RESET).



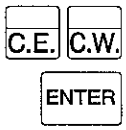
Enter the characters M, O, N , ENTER, the control reports in the user monitor.

2. Calling a Parameters



Example
By input of D_{03} this parameter is displayed at the screen.

3. Input and Storing a Parameters



- Correction of the displayed value with CLEAR ENTRY or CLEAR WORD key and input of the desired value.
- ENTER, take over into memory

4. Exit from the User Monitor

Cease input by pressing any mode key or RESET. With RESET EDIT stays active.

D Parameters - Common Monitor Data

D_{00} Entry of the Baud Rate for the Serial Interface

D_{00} determines the speed (=Baudrate) of the data transmission via the V24 interface.
Input range: 150 - 9600 Baud.

The baud rate depends on the connected peripheral device (see manual for peripheral device).

D_{02}/D_{03} Workpiece Counter and Workpiece Number Presetting

1. Workpiece Number Display

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

2. Workpiece Number Presetting

Entering the number of automatic runs D_{02} (specified number)
Enter the number with parameter D_{02}.

Example
16 automatic runs.
Input: D_{02}=16
The program stops after 16 runs.

D_{08} Baud Rate for DNC

Input as same as for the serial interface

L Parameters - Peripheral Data

L_{00} - L_{03} PLC Setting Bits 1 - 4

L_{00} PLC setting bit 1
L_{01} PLC setting bit 2
L_{02} PLC setting bit 3
L_{03} PLC setting bit 4

O Parameters - Common Setting Data

O_{00} Overwrite Warning while Data Input

Determine with parameter O_{00} wheter while data input a overwrite warning should occur.
O_{00} = 0 warning
O_{00} = 1 overwrite without warning

O_{01} Data Format for the Serial Interface

	Bit 0	Bit 1	Bit 2	Bit 3	Bit 4	Bit 5	Bit 6	Bit 7
	not used	end of workpiece program	character length		Parity check	Parity odd/even	number of stop bits	
Bit=0 (low)					no Parity check (disable)	odd		
value		0	0	0	0	0	0	0
Bit=1 (high)		transmission end with ETX			Parity check (enable)	even		
value		2	4	8	16	32	64	128

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

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

O_{01} Bit 0

not used

O_{01} Bit 1

Bit 1=High
Data transmission will be stopped when the ETX sign was received.

O_{01} Bit 2/Bit 3

Bit 2 and bit 3 are combinated. They determine the character length. Usual are character lengthes of 7 or 8 bits.

O_{01} Bit 6 / Bit 7

Determination of the number of stop bits. The number depends on the connected peripheral device (see manual of the periperal device).

O_{01} Bit 4 / Bit 5

Bit 5 determines wheter an odd ore an even parity should be checked. This check can be enabled or disabled in bit 4.

Example parameter O_{01}

		Wert
Bit 0		0
Bit 1	no ETX	0
Bit 2	Character	0
Bit 3	7 bits	8
Bit 4	Parity check	16
Bit 5	Even Parity	32
Bit 6	1 stop bit	64
Bit 7		0
	Input value O_{01}	120

O_{05} Number of the Serial Interface

O_{05} determines on which serial Interface data will be sent and from which serial interface data will be read.
O_{05} = 0 COM1
O_{05} = 1 COM2

O_{06} Output Drive

O_{08} determines on which PC drive data will be sent and from which PC drive data will be read.
O_{06} = 0 drive A:
O_{06} = 1 drive B:
O_{06} = 2 drive C: (hard disk)

O_{07} Disk Format

Parameter O_{07} sets the output for EMCOTRONIC or DOS disks.
O_{07} = 0 DOS disk
O_{07} = 1 EMCOTRONIC disk

O_{08} Settings for the DNC Interface

Setting the DNC interface is the same as setting the serial interface (like O_{01})

O_{11} Measuring Unit

O_{11} determines the initial status of the measuring unit.
O_{11} = 0 metric
O_{11} = 1 inch

R Parameters - Reference Positions

R_{00} Reference Point in X

R_{01} Reference Point in Y

For lathes: 0.000

R_{02} Reference Point in Z

T Parameters - Language

T_{22} Setting the Language

With T_{22} you can select one of the installed languages for the software texts.
0 for English
1 for German
2 for French
3 for Spanish

G: Alarms

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.

ALARMS 000 - 190: AXIS CONTROLLER

ALARM 001: X-AXIS: SOFTWARE LIMIT SWITCH OVERTRAVELLED
EXECUTE/AUTOMATIC mode: The programmed path is supervised by software limit switches which will release ALARM 001, 002, 003. (evtl. wrong data in PSO, wrong tool data or arc overtravels the admitted working area although the start and target point are in the admitted area)
MANUAL: The software limit switches are valid after approaching the reference point and release the alarm and an axis stop when they were overtravelled.
Remedy: Traverse back in MANUAL mode.

ALARM 002: Y-AXIS: SOFTWARE LIMIT SWITCH OVERTRAVELLED
see ALARM 001

ALARM 003: Z-AXIS: SOFTWARE LIMIT SWITCH OVERTRAVELLED
see ALARM 001

ALARM 020 MAIN DRIVE NOT READY

This ALARM will be released due to an error of the main drive, when:

- after switching on the control no ready signal of the main drive is given
- trying to switch on the main drive without ready signal
- an error occurs while the main drive is running
- a fault of the mains of machine or main drive occurs

Acknowledge this ALARM by switching off and on after repairing the fault at the main drive.

ALARM 100: AC SYNTAX ERROR

A command to the axis controller (AC) does not have the correct format. In normal operation this error should not occur. Switch off and on the control and the machine after this alarm. If this alarm occurs repeatedly, contact EMCO.

ALARM 101: X-AXIS: PROXIMITY DETECTOR ERROR

The inductive approximation switch for standstill supervising of the X axis is defective.

ALARM 102: Y-AXIS: PROXIMITY DETECTOR ERROR

See ALARM 101

ALARM 103: Z-AXIS: PROXIMITY DETECTOR ERROR

See ALARM 101

ALARM 110: AC OUTPUT BUFFER OVERFLOW

Status messages of the axis controller (AC) are not proceeded fast enough. In normal operation this error should not occur. Switch off and on the control and the machine after this alarm. If this alarm occurs repeatedly, contact EMCO.

ALARM 130: VALUE OUT OF RANGE OR INVALID INPUT DATA

This error occurs, if setting data were sent to the axis controller which can not be proceeded.
Remedy: New installation. If this alarm occurs repeatedly, contact EMCO.

ALARM 140: MAIN DRIVE SYNCHRONISATION ERROR

The axis controller gets incorrect signals to execute a traverse command with rotational feed.
Causes:

- The speed encoder of the main drive does not work.
- Hardware defect on the axis controller.
- Speed breakdown because of overload of the main drive.
- Synchronisation signal missing.

ALARM 150: AXIS OUT OF SYNCHRONISATION. REFERENCE POSITION LOST

This alarm occurs with step motor driven drives.
If the axis can not be traversed after quitting the alarm and restarting of the auxiliary drives, the exact error cause is displayed ba the LED's at the step motor board.
Possible causes:

- Overtemperature of the step motor board
- Overcurrent because of a defective step motor
- Over or undervoltage because of bad connections

If the slide can be traversed without switching off/on the control the following error causes are possible:

- Overload (e.g. with collisions)
- Wrong machine data
  Remedy: Reinstall MSD disk
- The slide stucks mechanically (lubrication)
- The distance of the inductive proximity switch defective is too large
- The inductive proximity switch defective is defective

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

Cause: Overload of the feed motor.
Remedy: Approach the reference position, alter the cutting values in the CNC program.

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

See ALARM 151

ALARM 153: Z-AXIS OUT OF SYNCHRONISATION. REFERENCE POSITION LOST

See ALARM 151

ALARM 160: BAD PARAMETER FOR G02 OR G03

If this alarm occurs repeatedly, contact EMCO.

ALARM 161: INVALID Z VALUE FOR HELIX

With a helix command a wrong parameter or a parameter with an invalid value was programmed. The Z path of the helix interpolation must be smaller than the length of the arc.

ALARM 170: TRIED TO START WITH FEED = 0

This alarm occurs with attempts of traverse movements which are impossible because of the following reasons:

- feed per minute: F=0 active (no F programmed)
- feed per revolution:
  a) F=0 active (no F programmed)
  b) M5 is active
  c) S=0 active (no speed programmed)
- threads: no pitch (F) programmed

Remark: The zero position of the feed override switch does not trigger this alarm if the movement was programmed correctly.

ALARM 171: G95 FEED LIMIT EXCEEDED

If this alarm occurs repeatedly, contact EMCO.

ALARM 180: WRONG CENTRE COORDINATE SPECIFIED

If this alarm occurs repeatedly, contact EMCO.

ALARM 190: RADIUS TOO LARGE

If this alarm occurs repeatedly, contact EMCO.

ALARMS 200 - 281: MACHINING CYCLES

ALARM 200: INVALID VALUE OF D OR P PARAMETER

An invalid D or P parameter was entered.:

- G04: the maximum value for D_4 (10000, = 1000sec dwell time) was exceeded.
- G85: D_3 was not programmed or with the value 0, for D_5 an invalid value was programmed (only 0°,40°,55°,60° and 80° admittable).
- D_6 is larger than the distance between start and target point in infeed direction.
- G86/87/88: the maximum value for D_4 (10000, = 1000 sec dwell time) was exceeded.

ALARM 201: P1 MUST BE POSITIVE

With the pocket milling cycle a negative diameter is not allowed.

ALARM 202: NO ANGLEPARAMETER D_2 ALLOWED

ALARM 203: NO MIRRORING ALLOWED

For some cycles mirroring is not allowed.

ALARM 204: G72/G74: M90 MUST BE ACTIVE

For G72 and G74 mirroring is not allowed.

ALARM 204: G72/G74: M90 MUST BE ACTIVE

ALARM 208: P_1 LARGER OR EQUAL 3 X CUTTER DIAMETER

Enlarge P_1 or use a smaller cutter diameter.

ALARM 209: D_2 LARGER THAN CUTTER DIAMETER

Program a smaller D_2 or use a larger cutter diameter.

ALARM 210: INVALID TAPER PARAMETER (P_0, P_2)

- G84/85 with cutting division: The sign of a taper parameter in the infeed axis must correspond to the infeed direction.
- G84/85 without cutting division: the amount of the taper parameter in the infeed direction is larger than the whole infeed, while the sign of this taper parameter is opposite to the infeed direction.
- G84/85: the amount of the taper parameter in the non infeed axis is larger than the distance start - target point belonging to, while the sign of the taper parameter reduces the distance start - target point.

ALARM 211: INVALID P_0 OR P_1

ALARM 220: INVALID REMAINDER
G84: A contour offset programmed with D_0/D_2 is larger than the whole infeed of the corresponding axis.

ALARM 221: INVALID D_0 OR D_1

ALARM 222: INVALID D_2

ALARM 223: INVALID D_3

ALARM 224: INVALID D_4

ALARM 225: INVALID D_5

ALARM 225: INVALID D_6

ALARM 227: INVALID D_7

ALARM 230: INVALID CYCLE TARGET
A wrong target point was entered for a cycle, e.g.:
- G84: Start and target point coordinates may be similar in one axis only if in this axis a valid taper parameter is programmed. In this case in the other axis no taper parameter may be programmed.
- G85/86: Start and target point coordinates must be different in every axis.
- G87/88: The drilling path must be unequal 0.

ALARM 240: NO OR INVALID STEP DEPTH
For a cycle a wrong cutting division was entered e.g.:
- G84: D_3=0 was programmed.
  D_0/D_2 was programmed but no cutting division.
- G85: D_3 is larger than D_6 or larger than the distance start point - target point.
- G86: D_3 is larger than the distance start point - target point.

ALARM 250: D OR P PARAMETER FOR GIVEN CYCLE MISSING
E.g.:
- G85: D_3 is not programmed
- G86: D_3 is not programmed

ALARM 260: DRILL NOT CENTERED
G87/88: At the start of a drilling cycle the drill must be on centre position (Z = 0).

ALARM 270: NO OR INVALID WITHDRAWAL PLANE

ALARM 280: CYCLE MUST START WITH G40 ACTIVE

ALARM 281: MIRROR START OR END ONLY WITH G40 ACTIVE

ALARM 283: CHANGE OF PLANE ONLY WITH G40 ACTIVE

ALARM 284: THREAD ONLY WITH G40 ACTIVE

ALARMS 290 - 340: PROGRAM RUN (SUBPROGRAMS, G27)

ALARM 300: MORE THAN 10 SUBROUTINES NESTED
Nesting of more than 10 subroutines. If this alarm occurs repeatedly after CNC program correction, contact EMCO.

ALARM 310: SUBROUTINE NOT IN MEMORY
- A subprogram called with G25 was not found in the program directory.
- The called subprogram contains no block.

Remedy: Correct CNC program or subroutine, create subprogram, check the path entries in the .INI files.

ALARM 311: CNC PROGRAM NOT IN MERMORY
A program which was started in AUTOMATIC does not exist yet.
Remedy: Create CNC program, check the path entries in the .INI files.

ALARM 330: M17 WITHOUT G25 OR M30 IN A SUBROUTINE
- Main program end without M30.
- M30 in a subprogram called by G25.

ALARMS 350 - 440: WORKPIECE PROGRAM INTERPRETER

ALARM 350: INVALID CUTTER RADIUS
G41/42: The radius of the active tool is 0. No tool radius compensation is active.
Cycles: Milling tool radius 0 not allowed.

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

With active tool radius compensation no new tool correction can be called.

ALARM 376: SCALE OUT OF RANGE

A too large or negative scale factor or scale factor = 0 was programmed.

ALARM 380: INVALID L WORD FOR G25/G27

A wrong jump target was programmed.

ALARM 382: MISSING POSITION PARAMETER FOR CHAMFER / RADIUS

- The block after a programmed chamfer or radius must contain position parameter (absolute or incremental).
- Too many blocks without new position were programmed.
- Decoding single block (with PC TURN 50 also M00) was programmed.
- With the programmed NC addresses no valid contour draft can be created.

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

The offset register must not altered in the block with chamfer / radius, otherwise chamfer / radius could not be calculated.

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

In the block with chamfer / radius no tool change is allowed, no tool change must be proceeded, otherwise chamfer / radius could not be calculated. The tool radius compensation G40-G42 also must not be altered.

ALARM 389: PROGRAMMED CHAMFER / RADIUS TOO GREAT

The programmed chamfer / radius is too large. The inserted chamfer / radius must not be larger than the shorter of the two straights between which the chamfer / radius should be inserted.

ALARM 410: INVALID G-CODE

- Too many G codes were programmed in a block.
- A G code of the same group was already programmed in the block.
- An invalid G code was programmed.
- A combination of these G codes within a block is not allowed.

ALARM 416: BAD PARAMETER FOR G02 OR G03

With a circular traverse command a wrong parameter or a parameter with a wrong value was programmed. In the following vases this alarm will occur:

- No centre point coordinate entered.
- Centre point coordinate out of the numeric range of the machine (the second not programmed coordinate).
- A centre point coordinate does not fit to the circle (tolerance range).

ALARM 418: WRONG CENTRE COORDINATE SPECIFIED

The centre point coordinates must be in the active main plane.

ALARM 419: RADIUS TOO LARGE

The radius of the circular movement has a too large/ small value.

ALARM 420: INVALID M-CODE

- Too many M codes were programmed in a block.
- An invalid M code was programmed.
- An M code is not allowed.

If this alarm occurs repeatedly after program correction, contact EMCO.

ALARM 421: NO M03/M04 PROGRAMMED

A spindle direction and speed must be programmed.

ALARM 430: INVALID T WORD FOUND

- Too many tools were programmed in a CNC block.
- An invalid T word was programmed.

ALARM 431: INVALID TOOL TYPE

No or an invalid tool type (L - cutter position) was entered in the TO register.

ALARM 432: NO TOOL OFFSET ACTIVE

For some cycles a tool and a tool correction number must be selected.

ALARM 433: CUTTER RADIUS = 0

Enter the correct cutter radius in the TO register.

ALARM 434: CUTTER RADIUS TOO LARGE
Use a smaller tool.

ALARM 435: TOOL TO WIDE
Use a smaller tool.

ALARM 440: TARGET LIMITS EXCEEDED
EXECUTE/AUTOMATIC: The programmed target points will be supervised by software limit switches which release ALARM 440. (evtl. wrong data in PSO or wrong tool data).

- The target was programmed out of the working area.
- An arc overtravels the working area.

ALARM 441: HELIX Z LENGTH TOO LARGE
The pitch of a helix must be lower than 45°.

ALARMS 450 - 490: GENERAL OPERATING ERRORS

ALARM 460: REFERENCE POSITION NOT ACTIVE
Only after approaching the reference point in all axes the coordinate system of the machine becomes active, only after that absolute positions can be displayed and approached.
Approach the reference point.

ALARM 480: NO OR INVALID PARAMETER FOR G GROUP 0

- A circle centre parameter was programmed although no G02 or G03 was programmed.
- In a cycle of the G code group 0 an invalid D or P parameter was programmed
- G04: Parameter D_4 (dwell) not programmed.
- G84/85/86: The target point must be entered in both axes.
- G87/88: The target point must only be programmed in Z.

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

ALARMS 500 - 580: CUTTER RADIUS COMPENSATION

ALARM 500: TOO MANY BLOCKS WITHOUT SLIDE OPERATION
With G42/G42 more than 5 blocks without XY value alteration were programmed (with PC TURN 50 also M00).

ALARM 520: ERROR AT COMPENSATION START OR END

- The first traverse command after start or end of the tool radius compensation must be G00 or G01.
- With start or end of the tool radius compensation the XZ value must be altered related to the following or previous values. Altering X or Z alone is also alowed.
- The length of the start or end traverse path must be at least the tool radius.

ALARM 540: BAD CIRCLE PARAMETER
Arcs must be programmed in the active plane.

ALARM 570: RADIUS TOO LARGE FOR GIVEN CONTOUR
Possible causes:

- Programming an arc with a radius smaller than the tool radius.
- Programming small contour elements related to the tool radius, if a contour violation would occur in the last proceeded block.
- Programming an internal edge between two arcs, with special geometrical circumstances (mainly if the tool radius is very larger than the smallest programmed radius).

Remark: Contour violations which are later than in the following block resp. blocks which will be worked off later can not be recognized.

ALARMS 600 - 710: EDITOR

ALARM 600: INCORRECT EDITING SEQUENCE

Illogical or contradictory inputs e.g.:

- CNC block structure error.
- Too many M codes in a block.
- Too many S values in a block.
- Too many F values in a block.
- An axis, radius was programmed twice.
- A dwell time was already programmed in the block.
- An arc was already programmed in the block.
- An arc with more than two axes was programmed.
- A thread pitch was already programmed in the block.
- Too many axes were programmed in a block with a thread.
- Too many axes were programmed in a block with mirroring.
- Scale factor was already programmed in the block.
- Wrong input sequence.

ALARM 610: INVALID PARAMETER ENTERED

- An invalid address (a wrong character) was entered in the program.
- Data input/output: unknown data type.
- The programmed address is not allowed.

ALARM 620: INPUT VALUE OUT OF NUMERIC RANGE

- The block number is not allowed (too high).
- The programmed speed is too high/low.
- The programmed feed is too fast.
- The programmed dwell is too long.
- The programmed thread pitch is too large.
- Data input/output: incorrect data.

ALARM 640: BLOCK NUMBER ALRTEADY EXISTS

Attempt to renumber a block to a number which already exists in the active program.

ALARM 650: BLOCK MEMORY OVERFLOW

Attempt to enter a too long block.

ALARM 651: ERROR WHILE SAVING PROGRAM

- write error on disk, hard disk
- wrong drive specified
- disk, hard disk full

ALARM 690: INVALID INDEX ENTERED

- EDIT and EXECUTE mode: Attempt to enter a P or D parameters with index > 7.
- Tool or PSO data: Attempt to enter a tool index > 99 or PSO index > 5.

ALARM 700: NO CHANGE OF ACTIVE TOOL DATA / PSO

- EDIT: Attempt to alter the active tool data or the active zero offset data. Alteration is possible only after deselection of the tool or the PSO. Deselect just by pressing the RESET key or in EXECUTE mode by working off a block with the deselection functions (other tool or T0 or other PSO or G53/56).
- AUTOMATIC/EXECUTE: Attempt to alter the PSO register 5 with G92, although G59 is active.

ALARM 710: PROGRAM NUMBER ALREADY EXISTS

Attempt to renumber a program to a number which already exists in the program directory.

ALARM 742: TOOL TOO LARGE

ALARM 746: INVALID SIMULATION AREA

Wrong input for the graphic simulation area.

ALARMS 800 - 870: DATA INTERFACE (DISK, RS232)

ALARM 800: CASSETTE / DISK DRIVE NOT READY

- No cassette / disk inserted
- Cassette / disk full
- Hardware error of the device

ALARM 850: PROGRAM NOT FOUND

- Attempt to read in a not stored program.
- Attempt to write out a not existing program.

ALARM 860: INTERFACE OPERATING ERROR

Wrong operating sequence while reading in data from the serial interface.

No number was entered for input/output of the tool offsets.

ALARM 880: INTERFACE ERROR

Check, whether the settings of the serial interface (baud rate, parity bit, stop bit, ...) are similar at sender and receiver.

- character overflow
- parity error, parity setting error
- data frame error, different settings of sender and receiver
- setting of the serial interface not allowed
- general communication error

ALARM 881: COM-PORT INVALID / ALREADY USED

The serial interface which you want to use for data input / output does not exist (e.g. no COM 2 established in the PC) or is already used by another device (e.g. Digitizer).

ALARM 882: CONTROL KEYBOARD / DIGITIZER NOT AVAILABLE
Check, whether control keyboard or digitizer are switched on, connected to the correct interface, cables, plugs, ...

ALARM 883: NO DIGITIZER INITIAL DATA FOUND
Before working the digitizer must be initialized, see chapter External Input Devices.

ALARM 885: FILE ACCESS ERROR
Drive not ready, file does not exist, hard disk full, ...

ALARMS 970 - 998: FATAL ERROR! CONTACT EMCO

ALARMS 970 - 998
These Alarms with the numbers 970 to 998 should never occur (except alarms 981 - 983)!
Restart WinNC with no other windows applications running.
With alarm 974 check whether the hard disk is full.
With alarm 979 check the block structure and the cycle call with G code.
If these alarms occur repeatedly contact the next EMCO representation.

ALARM 981: RS485 ERROR
An RS485 device (control keyboard, machine) did not report or is defective. Switch on machine or control keyboard, select interface on control keyboard, check cables and plugs.

ALARM 982: RS485 ERROR - XX MISSING
XX: AC axis controller
SPS PLC
MT control keyboard
see 981.

ALARM 983: RS485 ERROR - INITIALIZING PC BOARD
Act as described in the chapter Installation - "PC configuration - mounting the interface board".

ALARM 999: NEW SETTING DATA GENERATED
This alarm can occur after software updates. The PSO and TO data could have been altered. Check these data and reload if necessary.

Machine Alarms

These alarms are released by the machine.
The alarms are different for the PC MILL 50 and the PC MILL 100.
The alarms 6000 - 6999 normally must be acknowledged with RESET. The alarms 7000 - 7999 are messages which will disappear usually when the alarm releasing situation is eliminated.

PC MILL 50
The following alarms are valid for the PC MILL 50.

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

ALARM 6001: CYCLE TIME EXCEEDS LIMIT
Contact EMCO Service.

ALARM 6002: NO PLC PROGRAM LOADED
Contact EMCO Service.

ALARM 6003: DB NOT EXISTENT
Contact EMCO Service.

ALARM 6004: RAM ERROR ON PLC BOARD
Contact EMCO Service.

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

ALARM 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.

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

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

ALARM 6013: MAIN DRIVE NOT READY
Main drive power supply defective, cable defective, fuse defective.
Check fuse.
Contact EMCO service.

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

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

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

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

ALARM 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.

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

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

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

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

ALARM 6042: TOOL CHANGE TIMEOUT
see alarm 6041.

ALARM 6043: TOOL CHANGE TIMEOUT
see alarm 6041.

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

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

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

ALARM 6049: INTERLOCKING TIME EXCEEDED
see alarm 6048

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

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

ALARM 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.

ALARM 7017: GO FOR REFERENCE POINT
Approach the reference point.

ALARM 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.

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

ALARM 7050: NO PART CLAMPED
After switching on or after an alarm 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.

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

PC MILL 100
The following alarms 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 stucks (collision?), main drive not ready, fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

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 toothing while the machine was switched off.
A running CNC program will be stopped.
Contact EMCO service.

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

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

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

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

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

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

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

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

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

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

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

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

7021: INITIALIZE TOOL TURRET
The tool turret operating was interrupted.
No traversing operation is possible.
Press 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

H: Accessory Functions

Activate Accessory Functions

The PC MILL 50 and 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:
- Enter the user monitor.
- The parameter L01 activates the accessories.

e.g.: L01 1 0 0 0 1 1 1 1 (→L01=143)
Bit Nr. 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 1 — swivel tool turret (MILL 100) ..... *

Strg 2 — puff blowing on / off (MILL 50)

Strg 2 — coolant on / off (MILL 100) ..... *

Strg 3 — swivel dividing head ..... •

Strg 4 — feed stop ..... •x

Strg 5 — feed start ..... *x

Strg 6 — spindle stop

Strg 7 — spindle start ..... *

Spindle start in the mode MANUAL:
clockwise: press short
ccw: press at least 1 sec.

Strg 8 — AUX ON

Strg 9 — close vice ..... +•

Strg 0 — open vice ..... +•

Strg ß — 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 assignement:

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 assignement:

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 assignement:

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 assignement:

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.

Notes for Working with the Automatic Tailstock

- The main spindle can not be switched on and NC START ( ) will not be accepted when the vice is in an undefined status (neither in back nor in clamped position).
- Moving the tailstock manually is only possible with open door, standing spindle and inactive NC START.
- 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
- 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 servie - 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.

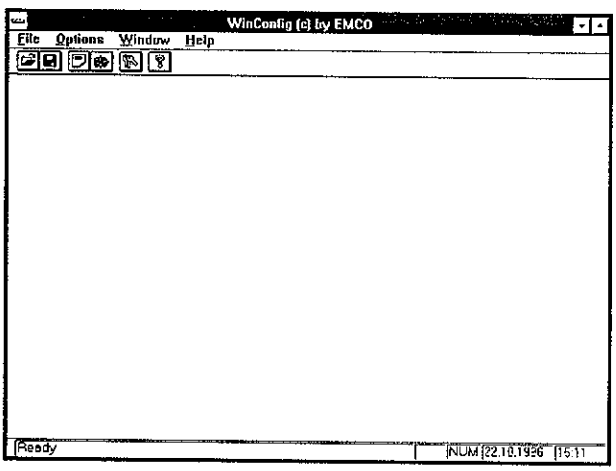
Notes for using WinConfig with WinCTS

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

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

- In a WINDOWS network (Windows for Work-groups 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.

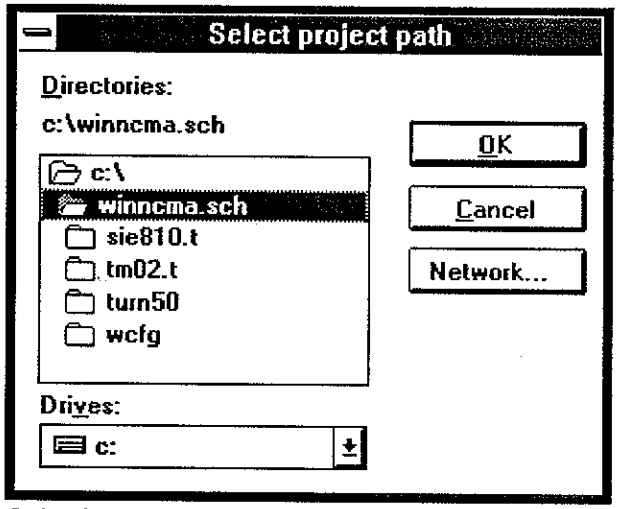
At the screen the window for WinConfig appears.

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.

Merasurement

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.

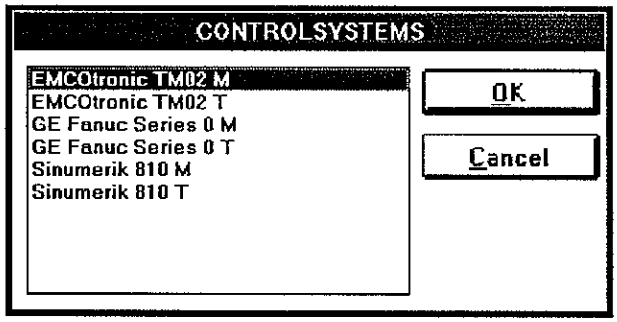
As usual with WINDOWS software the Ini data are stored in .ini files.

Select Window - Ini Data or click on the symbol.

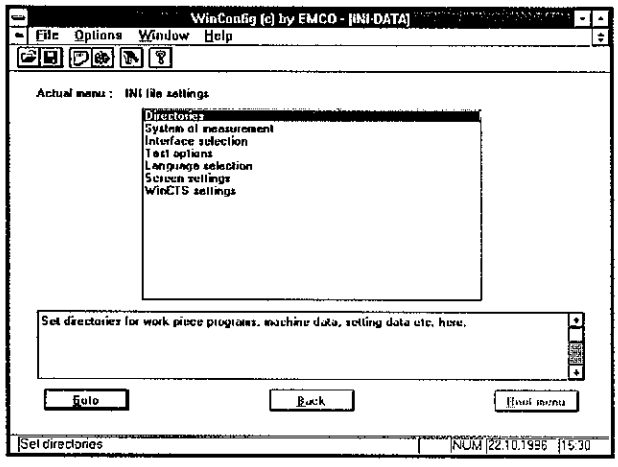
When several control types are installed, the screen shows a selection menu.

Click on the desired control type and on OK.

All following settings are valid for the selected control only.



Selection menu for control type



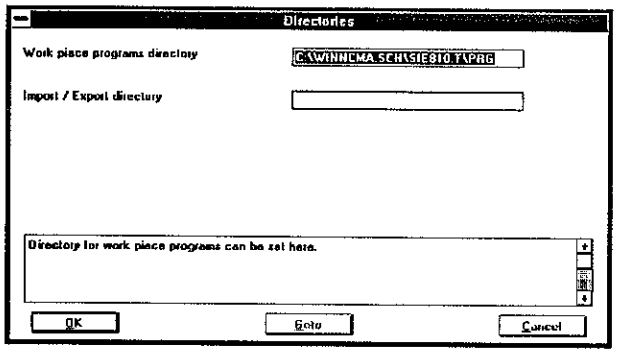
Selection menu for Ini data

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.



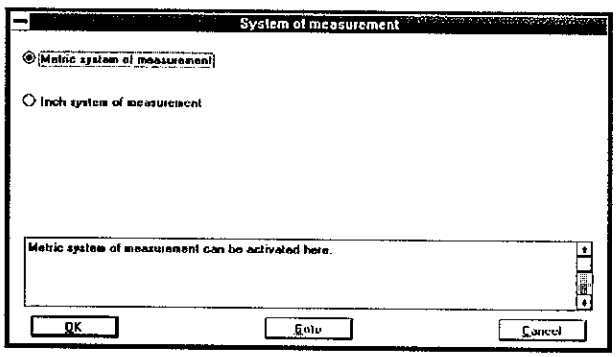
Input window for directories

Workpiece Program Directory

Enter the directory in which WinNC opens the workpiece programs.
Several users can have their own workpiece directory.

Import / Export Directory

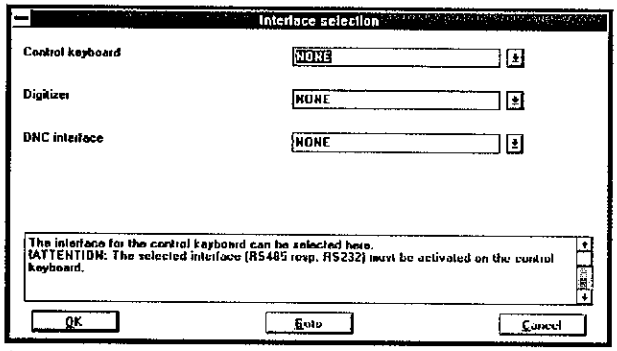
Enter the directory from which data will be imported or to which data will be exported.
You must enter an existing directory, WinConfig does not create directories.
When no directory is entered here WinNC transmits from/to the workpiece directory.



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

Note

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

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

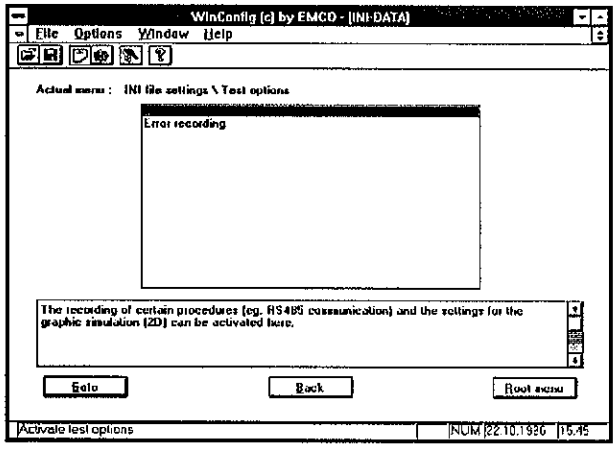
Digitizer (accessory)

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

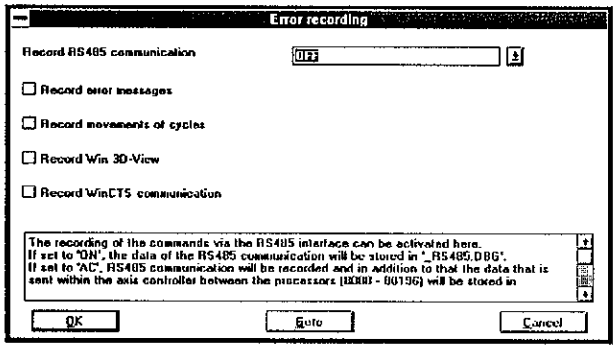
DNC (accessory)

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



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 communicationwill 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

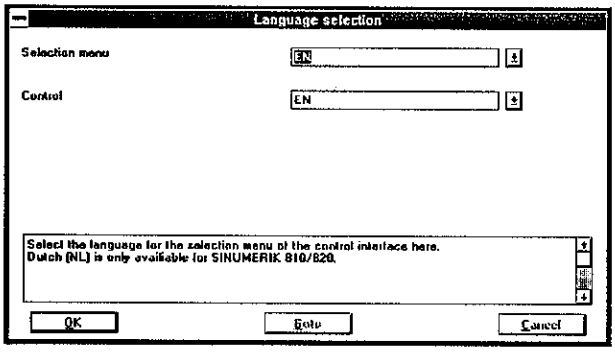
Activate this function (☒) to store the single movement commands in the file _PRGINPT.DBG.

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

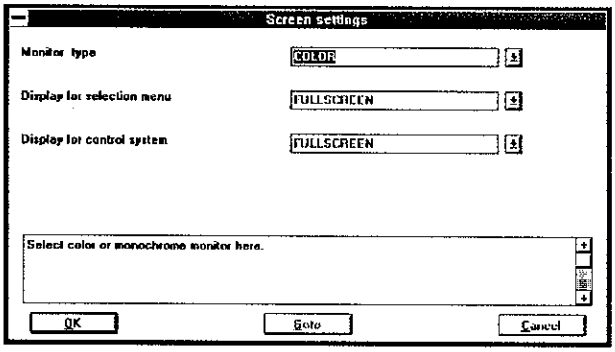
When several types of controls are installed, after starting WinNC a selection menu is displayed to select the desired type of control.
Here you can determine the language of the selection menu.

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

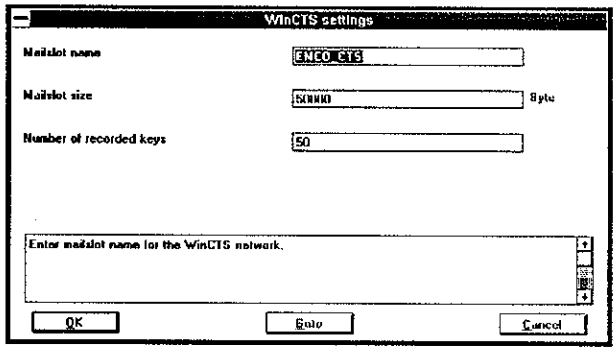
COLOR	Color screen
MONOCHROME	Gray screen

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.

Bildschirmdarstellung 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 mailsot 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 reduze 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.

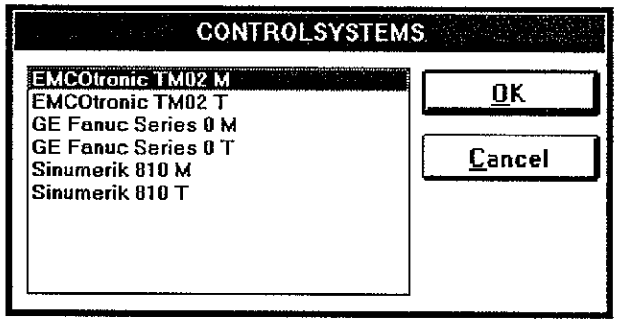
Insert the MSD disk of the machine into drive A or B. The MSD data will be written on the disk. When no disk is inserted you can not store and your alterations will be lost.

Select Window - Msd Data or click on the symbol.

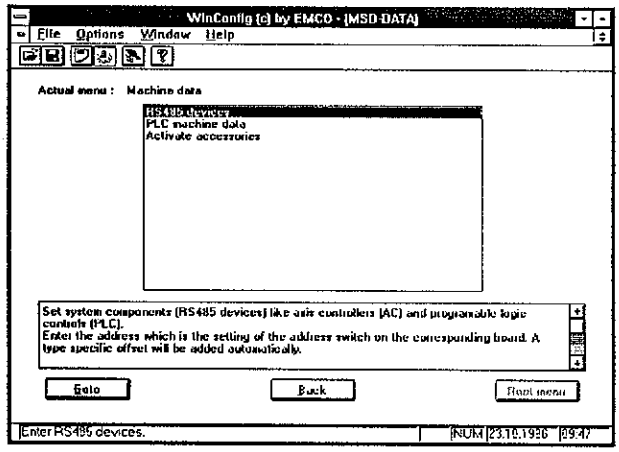
When several control types are installed, the screen shows a selection menu.

Click on the desired control type and on OK.

All following settings are valid for the selected control only.



Selection window for control type



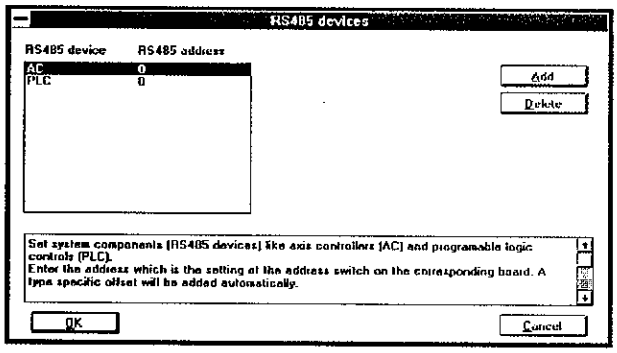
Menu for Msd data

The screen shows the menu for the Msd data.

5.1 RS485 Device List

Note

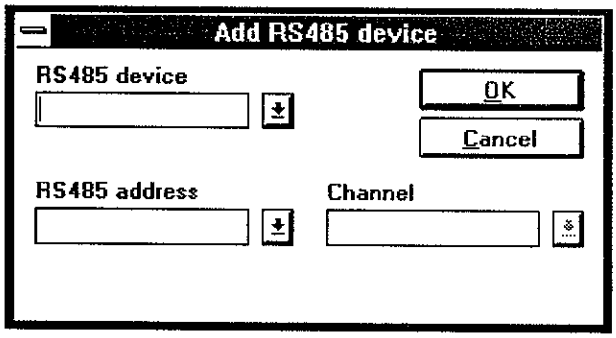
This menu point is needed only for serious alterations on the machine (e.g. mounting a PLC automatisation unit on the milling machine EMCO PC MILL 50).
When the device list that is determined here does not fit to the machine configuration, the machine will not work.



Display of the active RS 485 devices

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

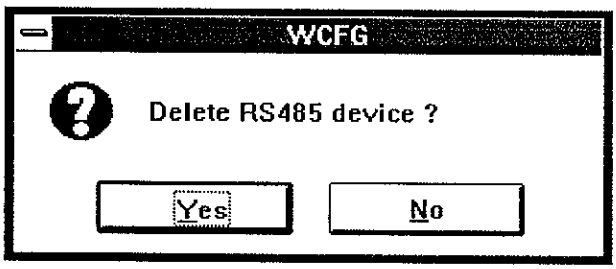
You can add or remove RS485 devices.



Selection window add RS 485 device

Add RS485 device

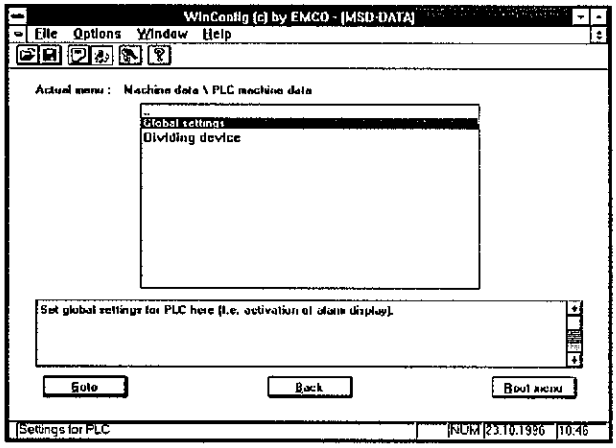
Select the switch button Add. The screen shows a selection window.
Determine which device should be added.
As RS485 address you have to enter that address that is set on the device board at the address switch.



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.



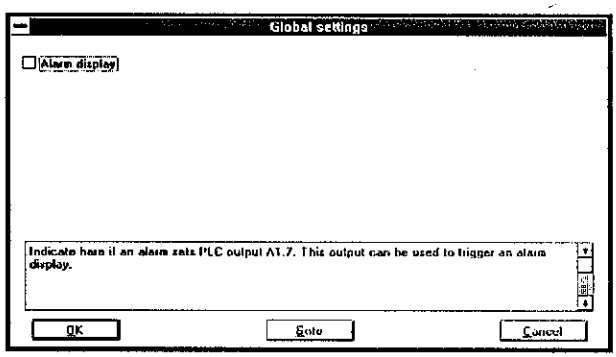
Setting possibilities PLC machine data

5.2 PLC Machine Data

Here you can alter the PLC settings.

Select the menu point PCL machine data.

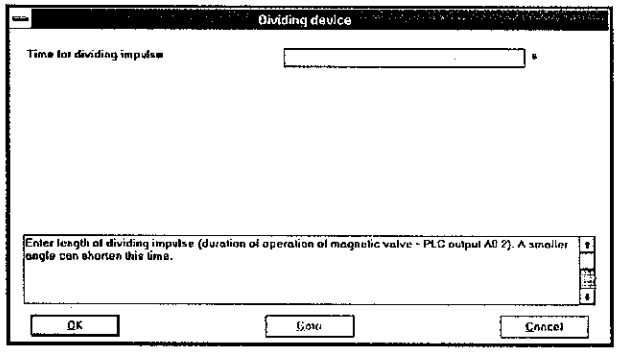
The screen shows the setting possibilities.



Activate alarm display

Global settings

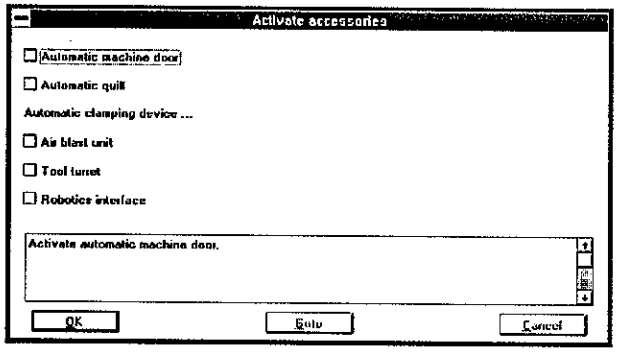
You can set whether Output A1.7 is actuated in the case of an alarm.
With this output you can e.g. trigger an alarm signal light.



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.

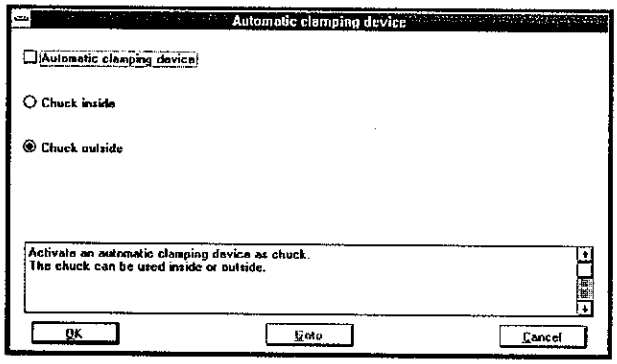


Zubehörliste

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 systen 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

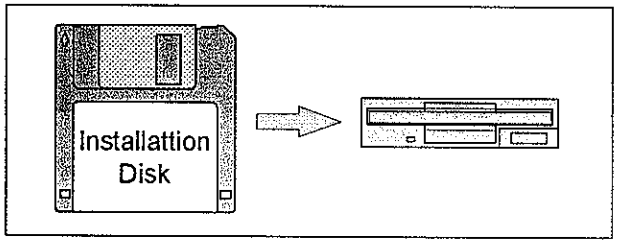
When the installation program of WinNC finds a version of WinNC in that directory, in which installation should happen, the system asks whether you want to:

- 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

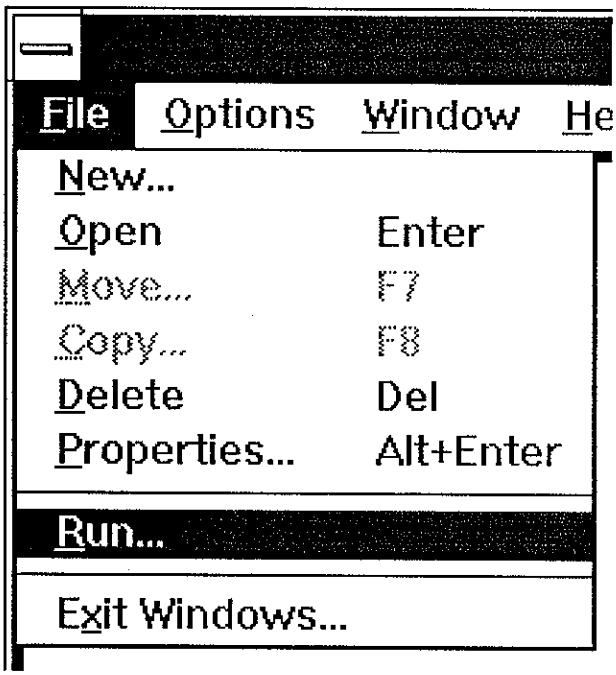
Note:
This manual describes the installation of all versions of WinNC.
Work off the following points one by one and skip the points which do not affect your version.
Before a network installation of the multiple licence pleas read the chapter "Notes for Network Installation".



Insert disk

All versions:

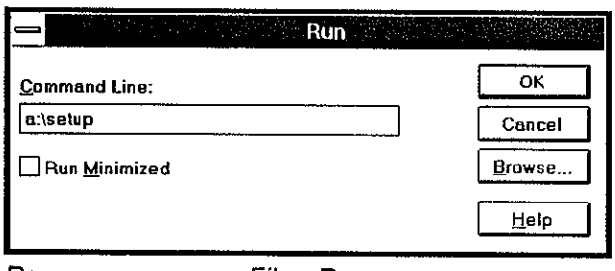
- Switch on your PC.
- Start Windows.
- Insert the installation disk 1 into drive A.



Program manager - File

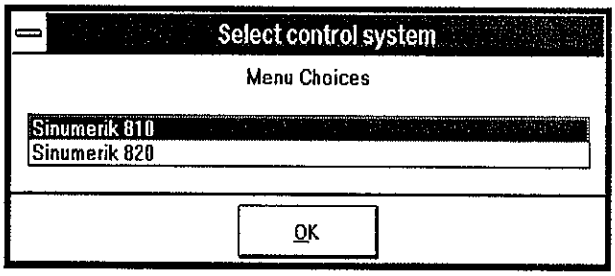
All versions:
- Select "File" in the command line of the program manager.
  The screen shows the menu beside.

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



Program manager - File - Run

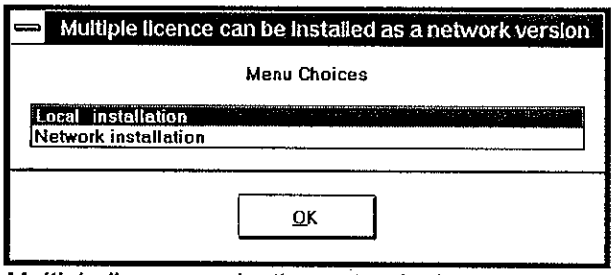
All versions:
- Enter into the command line: "a:setup"
- Acknowledge with "OK" (click on or ENTER).
  The installation program will be started.



SINUMERIK: selection 810 - 820

Only SINUMERIK:

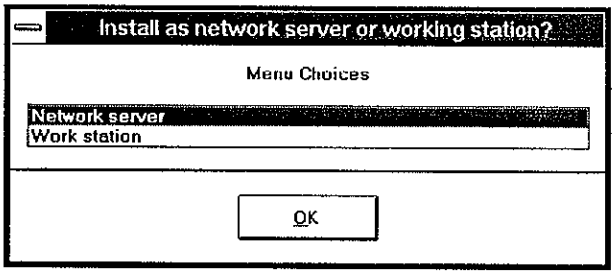
- Select the desired control type and acknowledge with "OK".



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

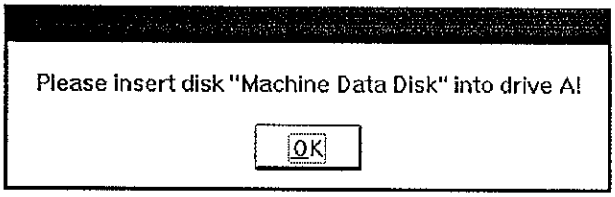
Only net installation:

- 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:

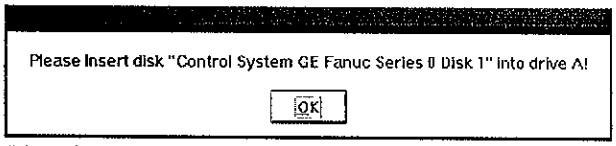
- Enter here the directory in which WinNC was installed at the server.



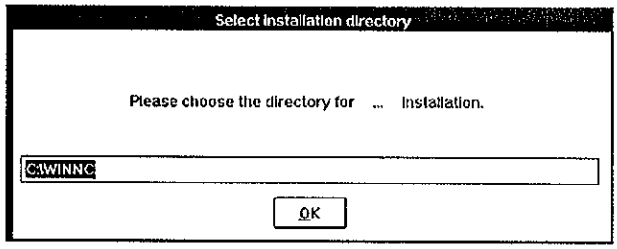
Machine version: insert MSD disk

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



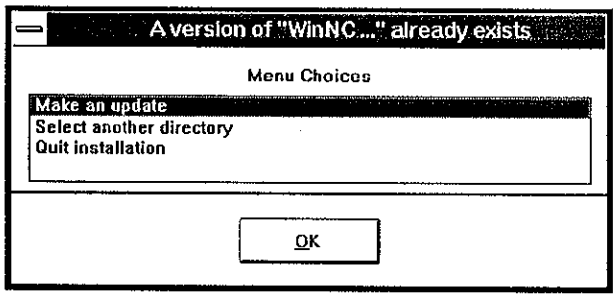
Directory for software installation

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.

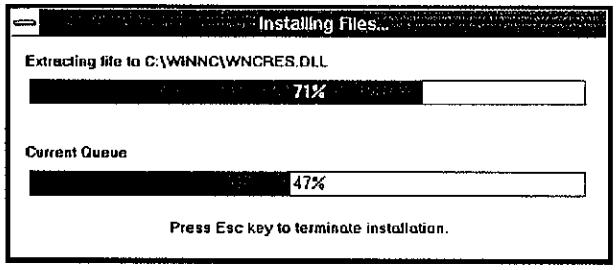
- Take over the proposal or enter an other directory and acknowledge with "OK".



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:

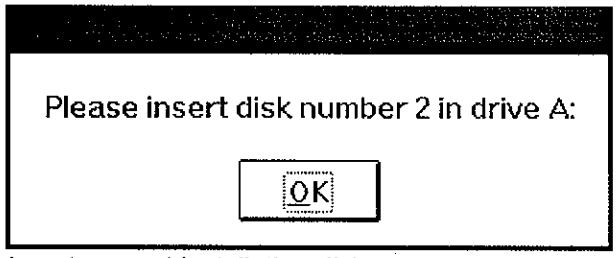
- Make an update:
  You can update the software. See "Software Update, Change Settings".
- Select another directory
  With this selection you can select another directory to install WinNC.
  After that WinNC is installed twice at the computer!
- Quit installation
  The installation will be ceased, no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



Display of the installation progress

All versions except net working station:

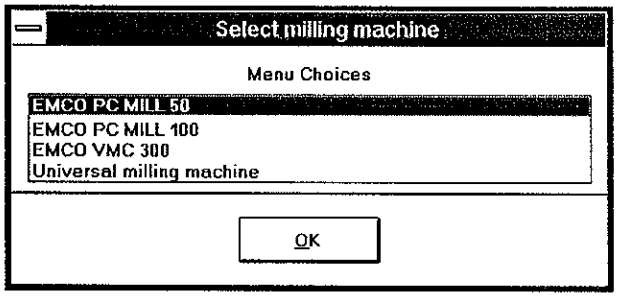
- While the following installation the screen shows how far the installation is advanced.



Insert second installation disk

All versions:

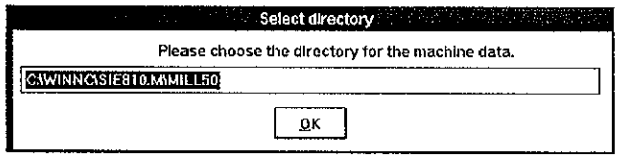
- The installation program needs die installation disk 2.
- Insert the installation disk 2 into drive A and acknowledge with "OK".



Example: selection milling machines

Only single licence version and net working station:

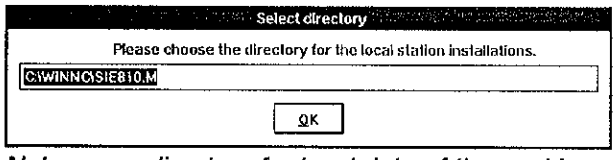
- The installation program proposes different machines, of which you can simulate programming and operating. In the software the limit data of this machine will be set (working area, feeds, speeds, ...).
  With "Universal ... machine" you can simulate an almost unlimited machine.
- Select the desired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



Directory machine simulation data

Only single licence version and net working station:

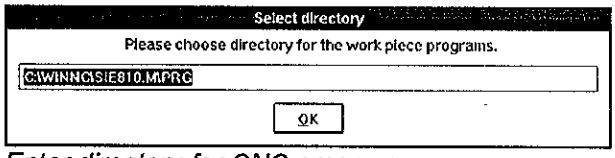
- The installation program needs a directory, in the limit data of the selected machine should be installed.
  The installation program proposes as directory name the name of the selected machine. You can alter this name.



Net server: directory for local data of the working station

Only net server:

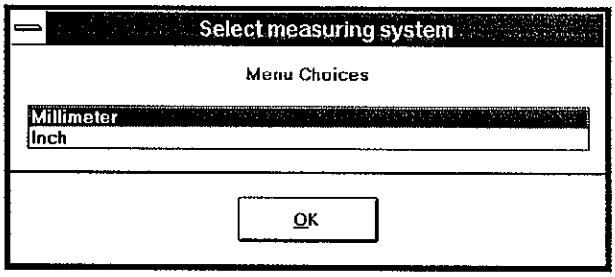
- The installation program needs a directory for the local data of the working station.
  Here you must enter the directory, on which the drive letter was mapped (see "Notes for Network Installation).



Enter directory for CNC programs

All versions except net server:

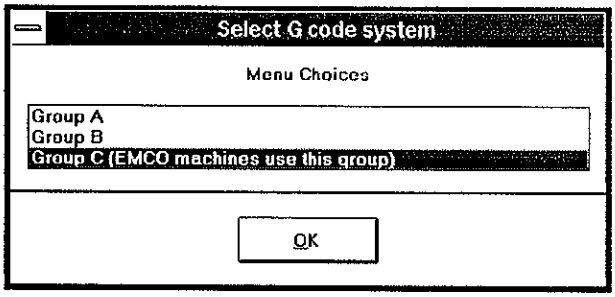
- The installation program needs a directory, in which all CNC programs created by you should be written.
  The installation program proposes the directory name "PRG". You can alter this name.
- Take over the proposal or enter an other directory and acknowledge with "OK".



Select measuring system

All versions except net server:

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

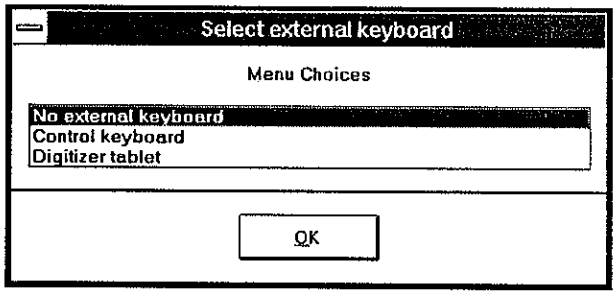


Fanuc 0-TC: enter command group

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".

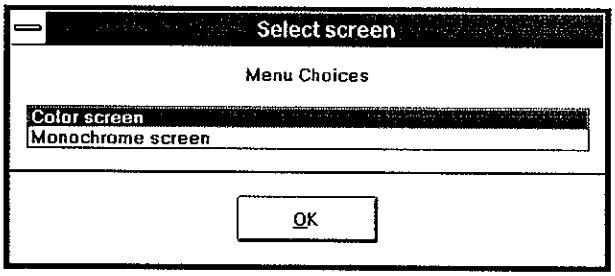


Enter external input device

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".

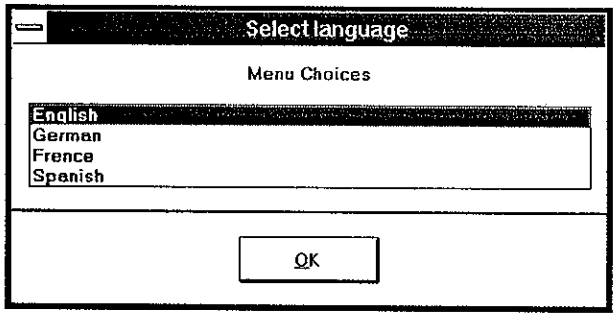
  Note:
  If you connect an input device at a later time, you need not to install WinNC again, you can activate this device with an entry in the file PROJECT.INI (see Basic Settings of WinNC).



Screen selection

All versions except net server:

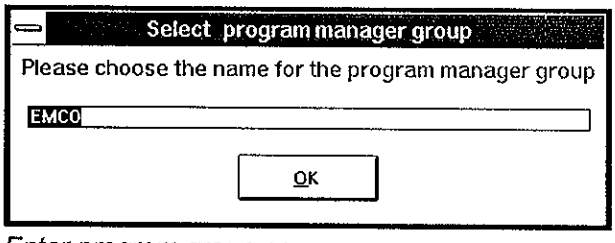
- Select the screen used by you and acknowledge with "OK".



Select language

All versions:

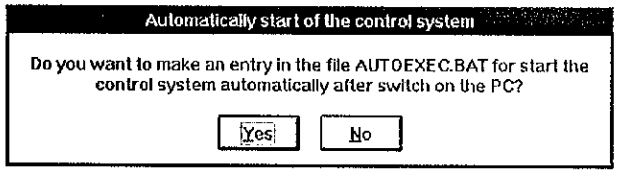
- Select the language version for WinNC and acknowledge with"OK".
  You also can alter the language in the CNC control pictures or in the basic settings for WinNC afterwards.



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".

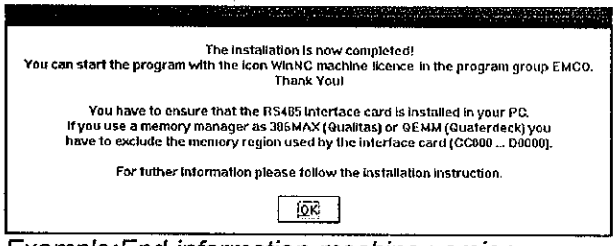


Machine version: automatic start YES - NO

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.

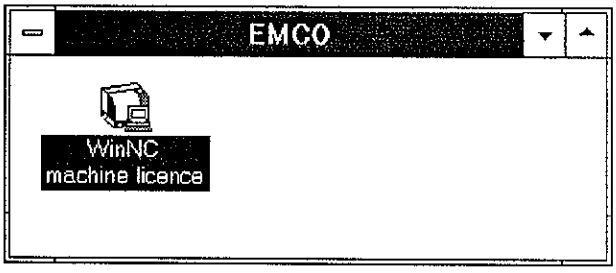
Only machine version:
- The installation program can alter the file AUTOEXEC.BAT for that after switching on the PC WinNC will start automatically.
- Select "YES" if WinNC should start automatically.
- Select "NO" if WinNC should not start automatically

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)

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)

- 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).

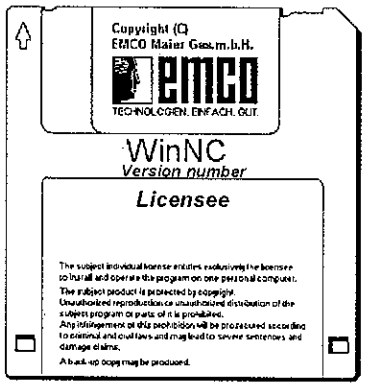
- With the now following installation on the working stations the directory must be entered in which WinNC is installed at the server.

Starting WinNC

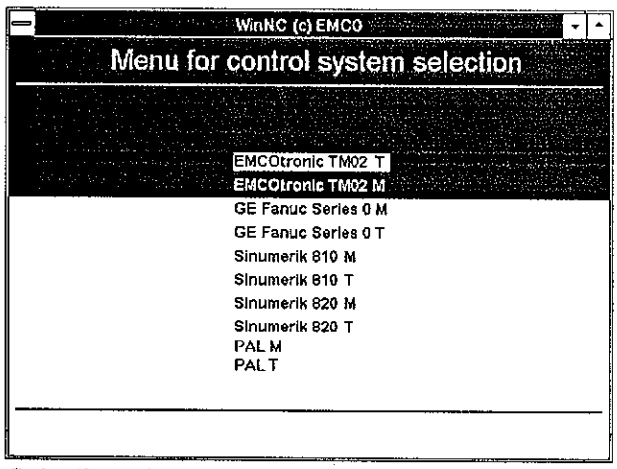
If you have selected "YES" for the last query in the machine version installation (entry in the file AUTOEXEC.BAT), WinNC starts automatically after switching on the PC.

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 SKIP 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.

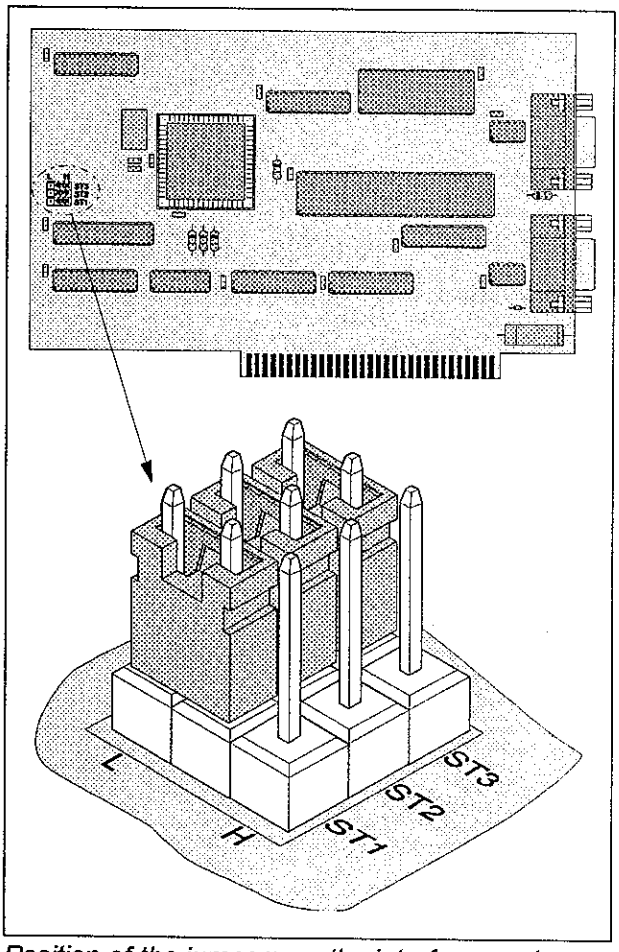
If this area is already occupied, e.g. by another card or an Expanded Memory Manager, the following alarm appears:

"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").



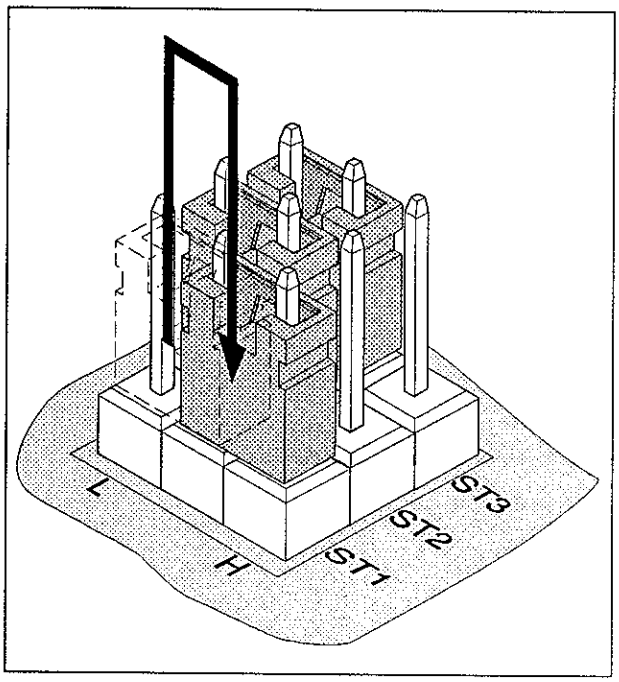
Position of the jumpers on the interface card

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, an 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 didactivcally 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 "Summagraphics MM Series".
The digitizer must support directly the Summagraphics MM format, an emulation is not sufficient. The digitizer will be operated directly via this command interface, no drivers of the manufactorer 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.

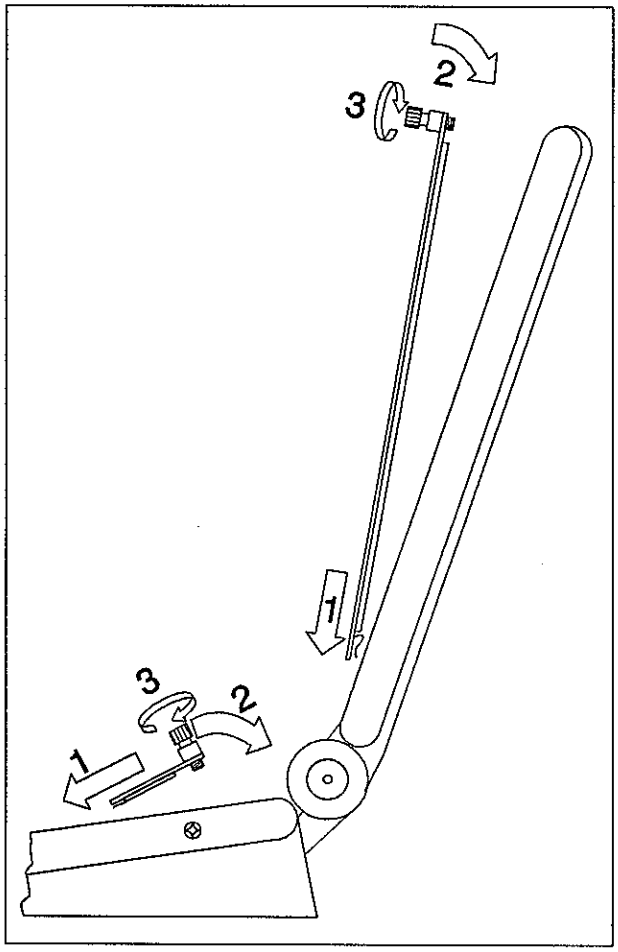
EMCO Control Keyboard

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



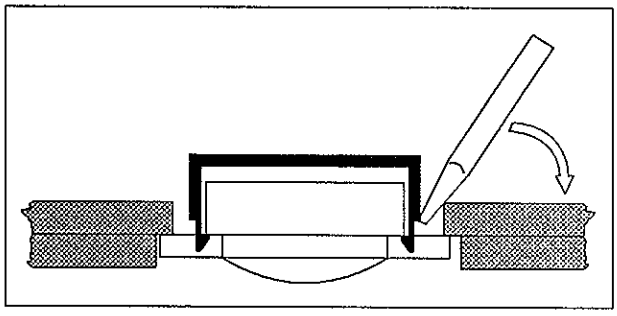
Assembling

- Place the corresponding 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).

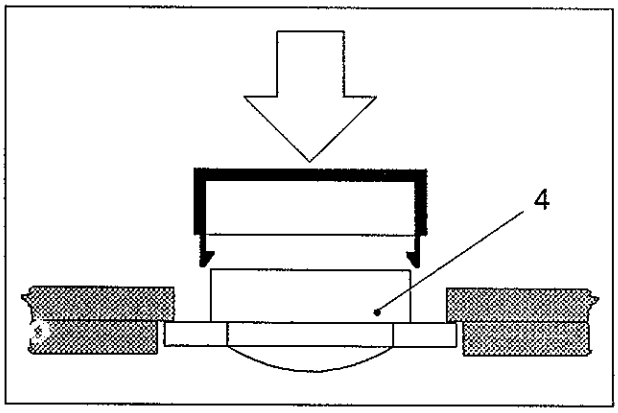
Note
The key sheets must not be bended, otherwise the switching function can not be warranted.

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.

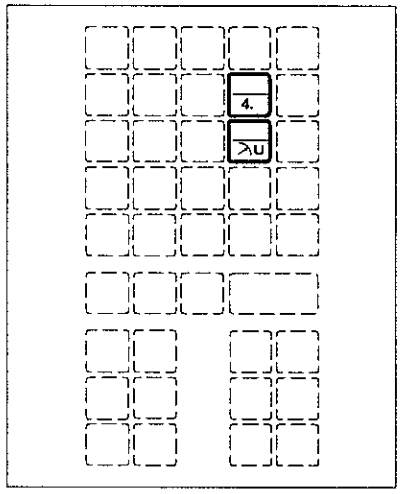


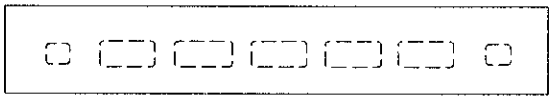
Take off
Pull out carefully the key caps to be exchanged with a fine screw driver or a knife.

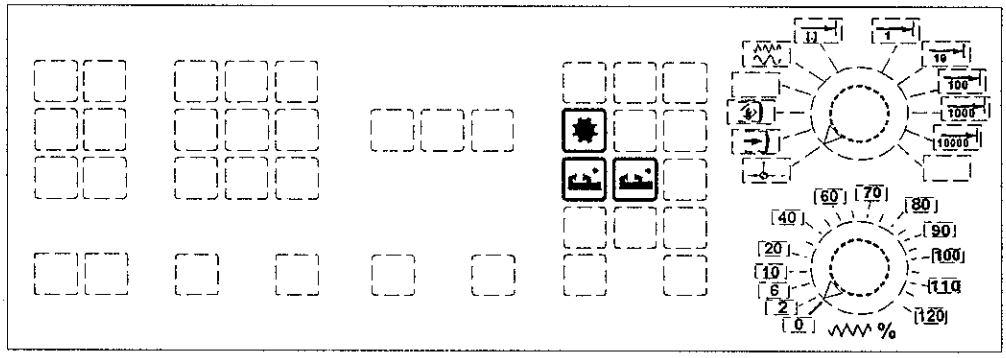


Clip on
Move the key body (4) in the middle of the recess. Push the key cap vertically down onto the key body, until the key cap snaps in tactily.

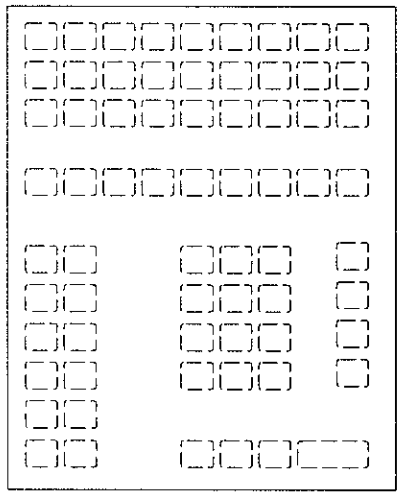
SINUMERIK 810M
Exchange key caps
for milling

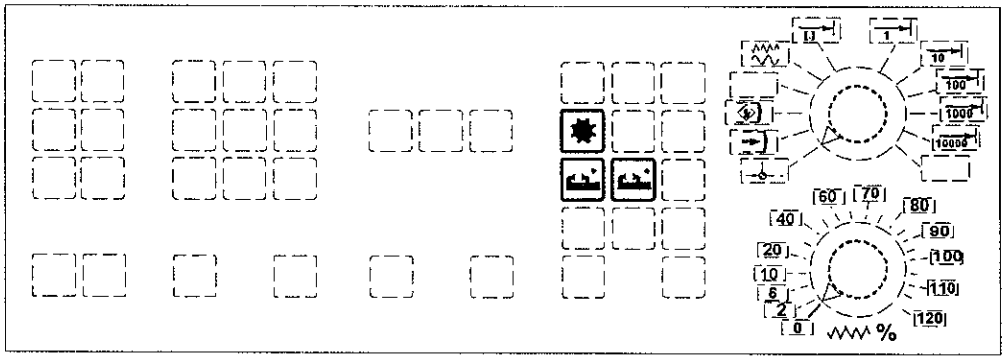




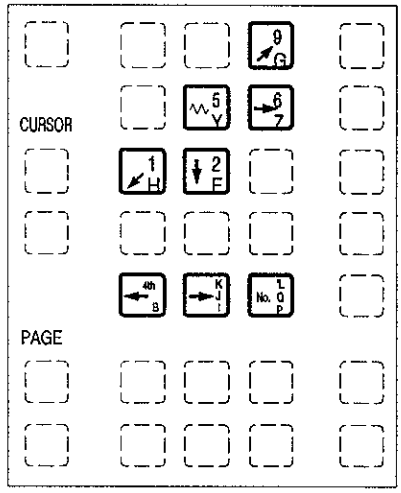


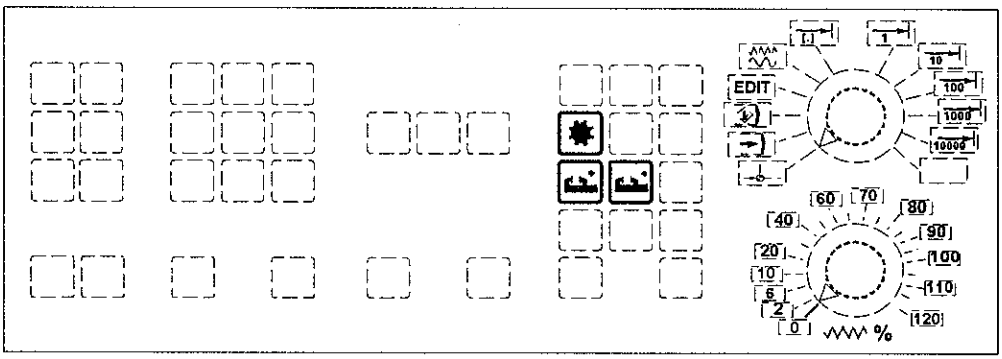
SINUMERIK 820M
and
PAL M
Exchange key caps
for milling



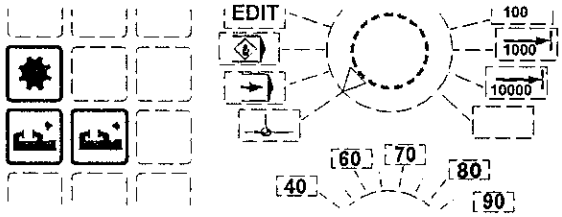


FANUC 0M
Exchange key caps
for milling





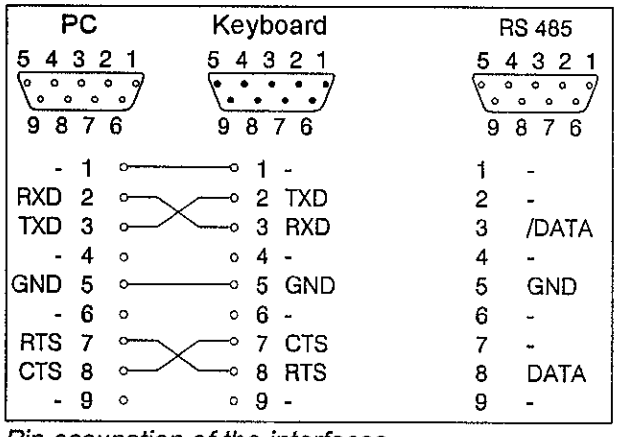
EMCOTRONIC M2
Exchange key caps
for milling



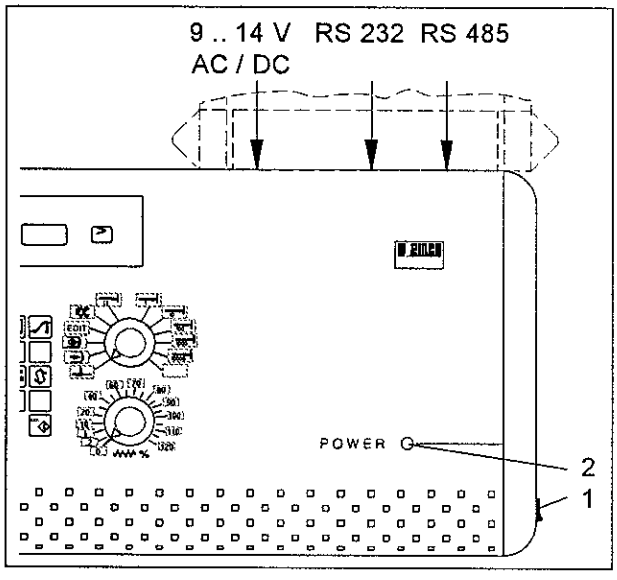
Power Supply

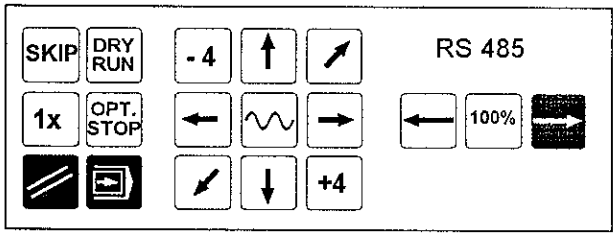
The control keyboard is supplied with 9 - 14 V, AC or DC.
The poles of the power supply can be interchanged, the polarity need not to be considered.
The power supply must be able to deliver at least 250 mA.
The connector is a 5/2.5 mm female cinch jack to be plugged in at the backside of the control keyboard.

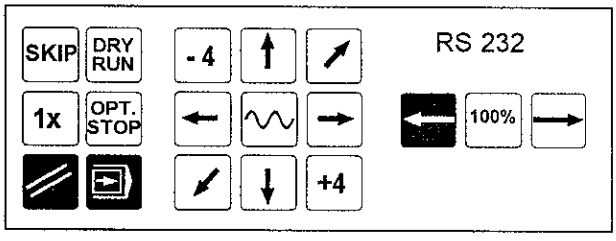
Notes for power supply 230 V, Ref. No. 795 700:
- Use the jack adaptor with the green point.
- Set the voltage selector switch to 12 V.



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.

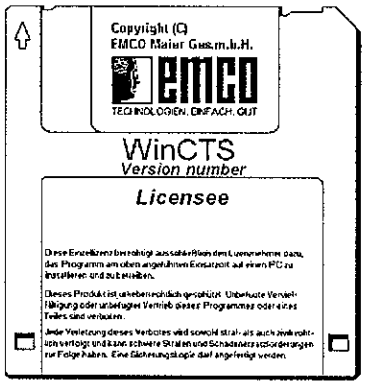
Note:
When the control keyboard is connected to the PC via RS232, switch on the PC first and then switch on the control keyboard, otherwise communication problems could occur and the control keyboard would not work.

Start WinNC, WinCTS with the Control Keyboard

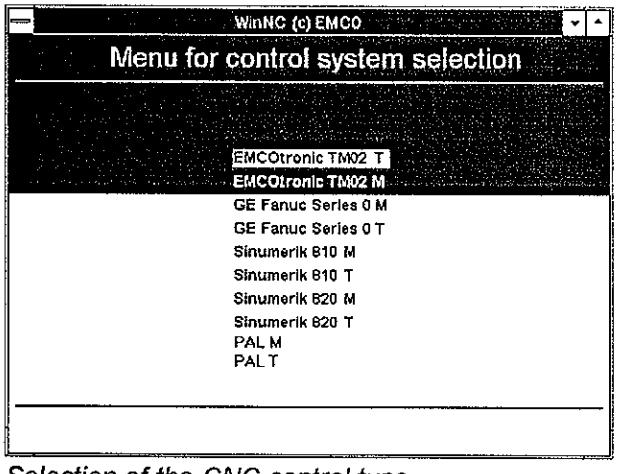
If you have selected "YES" for the last query in the machine version installation (entry in the file AUTOEXEC.BAT), WinNC starts automatically after switching on the PC.

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 piccture the version number of WinNC, WinCTS and the licensee are displayed.

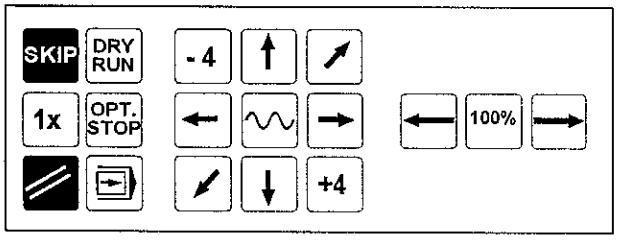


Start picture for WinNC, WinCTS



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 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 similarely for at least 1 second,.
This is the same function as Alt+F4 at the PC keyboard.

Software Description- Update
EMCO WinNC

Because of the adjustment of EMCO WinNC- controls from 16 Bit to 32 Bit versions the following capters become invalid and have to be replaced by the enclosure.

- **Accessory functions**
- **WinConfig**
- **External input devices**
- **Software installation**

Contents

EMCO

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

W: Accessory Functions

Activate Accessory Functions

The milling machines can be equipped with following accessories:

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

The accessory will be activated with WinConfig.

Note
After changing these settings the machine must be switched off and on.

The following M commands are used for the accessories:

M10 Lock dividing head
M11 Unlock dividing head
M25 Close vice
M26 Open vice
M27 Swivel dividing head
M71 Puff blowing on
M72 Puff blowing off

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

PC keyboard **Control keyboard, Digitizer**

Swivel tool turret ..... *

Puff blowing on / off

Coolant on / off ..... *

Swivel dividing head ..... •

Feed stop ..... •x

Feed Start ..... *x

Spindle stop

Spindle start ..... *

Spindle start manual:
Clockwise: press short
CCW: press min. 1 sec.

AUX ON

Close vice ..... +•

Open vice ..... +•

AUX OFF

Open / close door: consent key

\+ works with open door only
* works with closed door only
• not depending on door status
x PC MILL 100/105/125/155 only in special operation mode

Robotic interface PC MILL 55

The robotic interface for the PC Mill 55 is an accessory. To activate it, a special PLC software has to be installed. The robotic interface controls the inputs and outputs of the PLC directly.

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

As long as an HIGH signal is at input 5.7 "FEED HOLD" is active.
All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal)

Input assignement

E 5.1 robotic / Open door
E 5.0 robotic / Close door
E 5.2 robotic / Open vice
E 5.3 robotic / Close vice
E 5.7 robotic / Feed stop
E 5.6 robotic / Program START

Outputs

All outputs are short circuit proof and bearable with 0.2 A.

Signal level

20 V .. 24 V HIGH

Output assignement

A 5.0 robotic / Program STOP
(M30, M00, M01, M02)
A 5.3 robotic / Door open
A 5.4 robotic / Door closed
A 5.5 robotic / Vice back position
A 5.6 robotic / Vice clamped
A 5.7 robotic / Alarm output

Robotic interface PC MILL 105

The robotic interface for the PC Mill 105 is an accessory. To activate it, a special PLC software has to be installed. The robotic interface controls the inputs and outputs of the PLC directly.

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

As long as an HIGH signal is at input 6.6 "FEED HOLD" is active.
All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal)

Input assignement

*E 6.0 robotik / close door 2nd channel input
E 6.2 robotic / AUX ON
E 6.3 robotic / Switch NC-mode reference - automatic
E 6.4 robotic / Approach reference point
E 6.5 robotic / NC sart
E 6.6 robotic / Feed stop

E 7.0 robotic / Close vice
E 7.1 robotic / Open vice
E 7.5 robotic / Open door

Outputs

All outputs are short circuit proof and bearable with 0.2 A.

Signal level

20 V .. 24 V HIGH

Output assignement

A 0.3 robotic / alarm active
*A 6.0 robotic / close door 2nd channel output
A 6.3 robotic / EMERGENCY OFF
A 6.4 robotic / Machine ready
A 6.5 robotic / NC-mode reference
A 6.6 robotic / M0, M1, M2 or M30 is activ

A 7.0 robotic / Vice clamped
A 7.1 robotic / Vice open
A 7.6 robotic / Door closed
A 7.7 robotic / Door open

* With the help of two potential-free safety-contacts, the door will be closed.
One contact bridges the consent-key, the second connects the SPS-output 6.0 with the SPS-input 6.0. The safety-relais has to be switched until the door is closed, then it has to be switched off.

Robotic interface PC MILL 125

The robotic interface for the PC Mill 125 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

As long as an HIGH signal is at input 6.6 "FEED HOLD" is active.
All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal)

Input assignement

E 6.2 robotic / AUX ON
E 6.3 robotic / Switch NC-mode reference - automatic
E 6.4 robotic / Approach reference point
E 6.5 robotic / NC sart
E 6.6 robotic / Feed stop
E 7.0 robotic / Close vice
E 7.1 robotic / Open vice
E 7.4 robotic / Close door
E 7.5 robotic / Open door

Outputs

All outputs are short circuit proof and bearable with 0.2 A.

Signal level

20 V .. 24 V HIGH

Output assignement

A 6.3 robotic / EMERGENCY OFF
A 6.4 robotic / Machine ready
A 6.5 robotic / NC-mode reference
A 6.6 robotic / M00, M01, M02 or M30 aktiv
A 4.7 robotic / Alarm status
A 7.0 robotic / Vice clamped
A 7.1 robotic / Vice open
A 7.6 robotic / Door closed
A 7.7 robotic / Door open

Robotic interface PC MILL 155

The robotic interface for the PC Mill 155 is an accessory. To activate it, a special PLC software has to be installed. The robotic interface controls the inputs and outputs of the PLC directly.

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

As long as an HIGH signal is at input 6.6 "FEED HOLD" is active.
All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal)

Input assignement

*E 6.0 robotik / close door 2nd channel input
E 6.2 robotic / AUX ON
E 6.3 robotic / Switch NC-mode reference - automatic
E 6.4 robotic / Approach reference point
E 6.5 robotic / NC sart
E 6.6 robotic / Feed stop
E 7.0 robotic / Close vice
E 7.1 robotic / Open vice
E 7.5 robotic / Open door

Outputs

All outputs are short circuit proof and bearable with 0.2 A.

Signal level

20 V .. 24 V HIGH

Output assignement

A 0.3 robotic / alarm active
*A 6.0 robotic / close door 2nd channel output
A 6.3 robotic / EMERGENCY OFF
A 6.4 robotic / Machine ready
A 6.5 robotic / NC-mode reference
A 6.6 robotic / M00, M01, M02 or M30 aktiv
A 7.0 robotic / Vice clamped
A 7.1 robotic / Vice open
A 7.6 robotic / Door closed
A 7.7 robotic / Door open

* With the help of two potential-free safety-contacts, the door will be closed.
One contact bridges the consent-key, the second connects the SPS-output 6.0 with the SPS-input 6.0. The safety-relais has to be switched until the door is closed, then it has to be switched off.

Robotic interface Concept MILL 105

The robotic interface for the Concept Mill 105 is an accessory. To activate it, a special PLC software has to be installed. The robotic interface controls the inputs and outputs of the PLC directly.

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

As long as an HIGH signal is at input 7.4 "FEED HOLD" is active.
All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal)

Input assignement

*E 6.6 robotik / close door 2nd channel input
E 7.0 robotic / AUX ON
E 7.1 robotic / Switch NC-mode reference - automatic
E 7.2 robotic / Approach reference point
E 7.3 robotic / NC sart
E 7.4 robotic / Feed stop

E 7.6 robotic / Close vice
E 7.7 robotic / Open vice
E 8.3 robotic / Open door

Outputs

All outputs are short circuit proof and bearable with 0.2 A.

Signal level

20 V .. 24 V HIGH

Output assignement

A 3.3 robotic / alarm active
*A 4.0 robotic / close door 2nd channel output
A 4.3 robotic / EMERGENCY OFF
A 4.4 robotic / Machine ready
A 4.5 robotic / NC-mode reference
A 4.6 robotic / M0, M1, M2 or M30 is activ

A 5.0 robotic / Vice clamped
A 5.1 robotic / Vice open
A 5.6 robotic / Door closed
A 5.7 robotic / Door open

* With the help of two potential-free safety-contacts, the door will be closed.
One contact bridges the consent-key, the second connects the SPS-output 4.0 with the SPS-input 6.6. The safety-relais has to be switched until the door is closed, then it has to be switched off.

Robotic interface Concept MILL 155

The robotic interface for the Concept Mill 155 is an accessory. To activate it, a special PLC software has to be installed. The robotic interface controls the inputs and outputs of the PLC directly.

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

As long as an HIGH signal is at input 6.6 "FEED HOLD" is active.
All other inputs need a HIGH impulse with minimum 1 second duration, to switch the accessories (no steady signal)

Input assignement

*E 5.3 robotik / close door 2nd channel input
E 5.5 robotic / AUX ON
E 5.6 robotic / Switch NC-mode reference - automatic
E 5.7 robotic / Approach reference point
E 6.0 robotic / NC sart
E 6.1 robotic / Feed stop
E 6.2 robotic / Close vice
E 6.3 robotic / Open vice
E 6.5 robotic / Open door

Outputs

All outputs are short circuit proof and bearable with 0.2 A.

Signal level

20 V .. 24 V HIGH

Output assignement

A 3.3 robotic / alarm active
*A 5.3 robotic / close door 2nd channel output
A 4.0 robotic / EMERGENCY OFF
A 4.1 robotic / Machine ready
A 4.2 robotic / NC-mode reference
A 4.3 robotic / M00, M01, M02 or M30 aktiv
A 4.4 robotic / Vice clamped
A 4.5 robotic / Vice open
A 4.6 robotic / Door closed
A 4.7 robotic / Door open

* With the help of two potential-free safety-contacts, the door will be closed.
One contact bridges the consent-key, the second connects the SPS-output 5.3 with the SPS-input 5.3. The safety-relais has to be switched until the door is closed, then it has to be switched off.

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. At the PC Mill 100/105/125/155 also at the program with M25/M26

Notes for Working with the Automatic Tailstock

- The main spindle can not be switched on an when the vice is in an undefined status (neither in back nor in clamped position).
- Moving the tailstock manually is only possible with open door, standing spindle and inactive NC START. At the PC MILL 50/55 the doors 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/ 55: The door closes on pressed key
PC MILL 100/105/125/155: Press the door key and the consent key simultanously to close the door. The door moves only 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 50/55 the puff blowing device is switched on and off with the key combination Ctrl + 2.

Dividing head

M10 Lock dividing head

M11 Unlock dividing head

The dividing head will be activated like a accessory with WinConfig.
See WinConfig.

Activate Tool Turret

The tool turret (PC MILL 100/125/155) will be activated like a accessory with WinConfig. See "WinConfig".

DNC Interface

The accessory DNC interface can only be installed for a machine version of WinNC.

The accessory DNC interface is activated with WinConfig by indicating TCP/IP or a serial interface for the DNC.

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

The setting of the serial interface parameters DNC is carried out as during the data transmission via the serial interface in the operating area SERVICES via the softkeys V24 USER and SETTING, with the serial interface of DNC to be selected.
The DNC-Format Reduced ASCII requires 7 data bits for the data transmission.
The DNC-Format Full Binary requires 8 data bits for the data transmission.
With *WinNC SINUMERIK 810 D / 840 D* the correct number of data bits must be selected. Other WinNC controls automatically select the correct data bits.

If the interface DNC is operated with TCP/IP, incoming connections on port 5557 are waited for.
The description of the DNC protocol is on the installation disc and/or on the installation CD.

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 from CDROM

- Switch on your PC.
- Start Windows.
- Insert the installation CDROM for the DNC interface into the CDROM floppy drive.
- The installation program starts automatically.
- Enter the path in which WinNC is to be found.
- Select the serial interface or TCP/IP (if you select NONE, the DNC is installed but not activated). Click on „OK".
- Click on „OK". The installation is terminated.

Installation of the DNC interface from disks

- Switch on your PC.
- Start Windows.
- Insert the installation disk for the DNC interface into floppy drive A.
- Select "File" in the command line of the program manager.
- Select „Exceute".
- Enter „a:\setup" into the command line. Confirm with „OK" (click or ENTER).
- Enter the path in which WinNC is to be found.
- Select the serial interface or TCP/IP (if you select NONE, the DNC is installed but not activated). Click on „OK".
- Click on „OK". The installation is terminated.

X: 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 servie - 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.

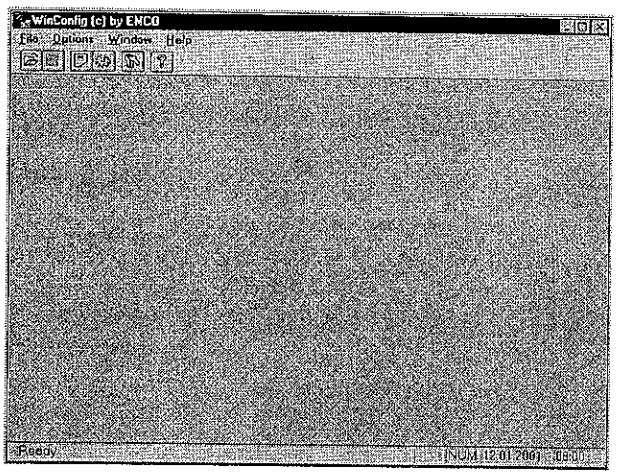
Notes for using WinConfig with WinCTS

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

Start WinConfig

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

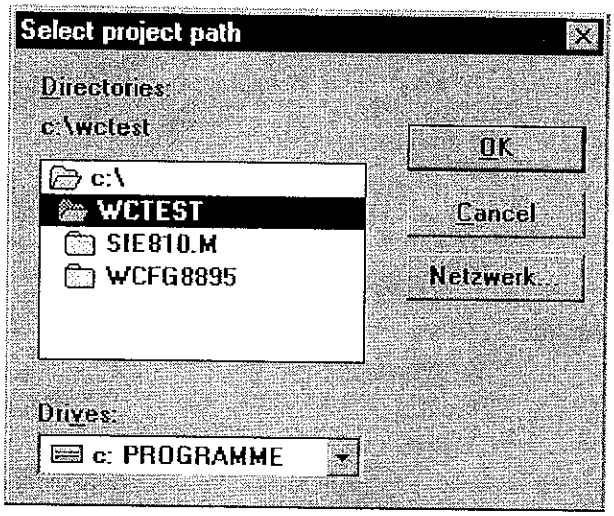
At the screen the window for WinConfig appears.

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 EXE- files (WINNC_88.EXE, WINNC_95.EXE, WINNC32.EXE, NC96:EXE, Sie840D.EXE) are 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.

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.

Merasurement

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

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.

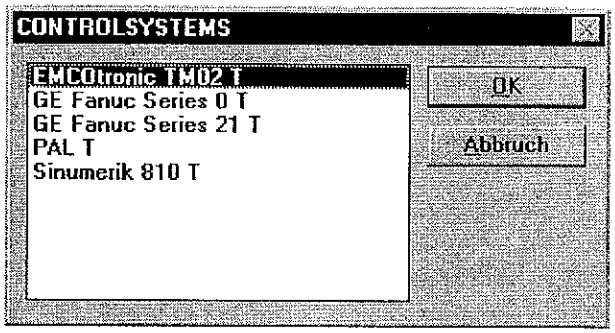
As usual with WINDOWS software the Ini data are stored in .ini files.

Select Window - Ini Data or click on the symbol  .

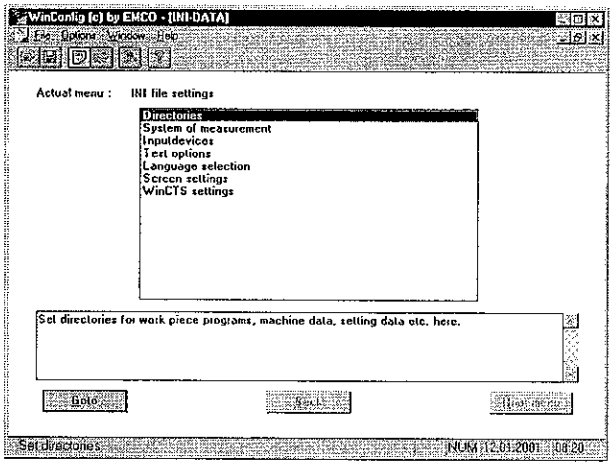
When several control types are installed, the screen shows a selection menu.

Click on the desired control type and on OK.

All following settings are valid for the selected control only.



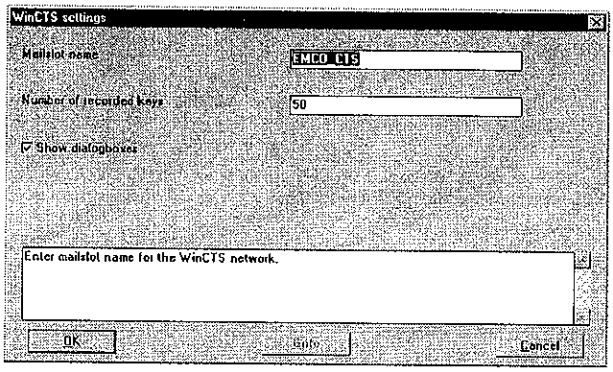
Selection menu for control type



Selection menu for Ini data

The screen shows the selection menu for the Ini data.

Select the desired menu point.
The respective function is explained in the text window.



Input window WinCTS settings

WinCTS Settings

Mailslot name

The mailsot 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.

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.

Show dialogboxes

Prevent display of several dialogboxes here.

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.

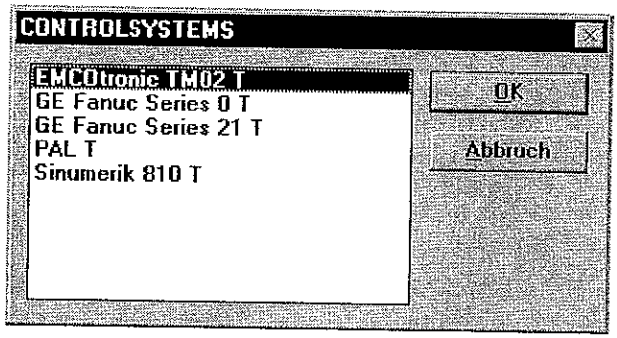
Insert the MSD disk of the machine into drive A or B. The MSD data will be written on the disk. When no disk is inserted you can not store and your alterations will be lost.

Select Window - Msd Data or click on the symbol  .

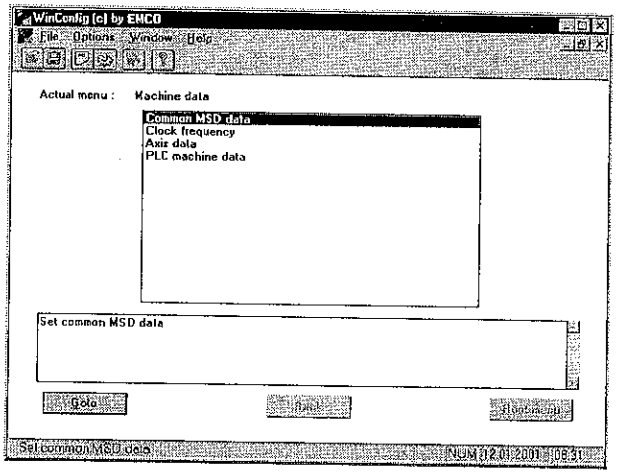
When several control types are installed, the screen shows a selection menu.

Click on the desired control type and on OK.

All following settings are valid for the selected control only.



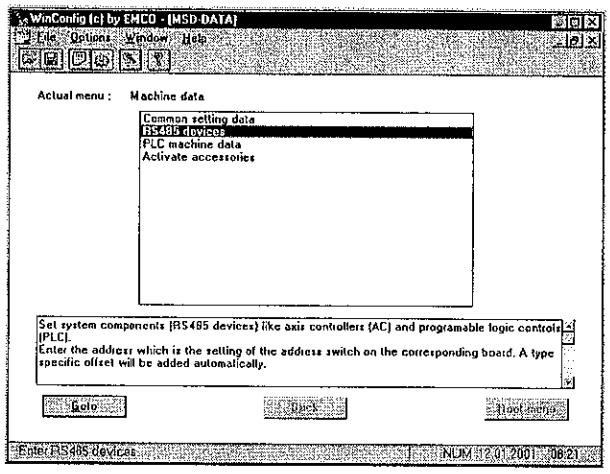
Selection window for control type



Menu for Msd data

The screen shows the menu for the Msd data.

Select the desired menu point.
The respective function is explained in the text window.

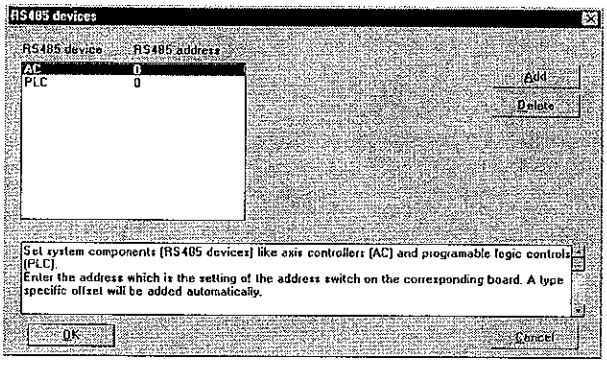


RS485 Device List

For the machines PC TURN 50/120 and PC MILL 50/100 only.

Note

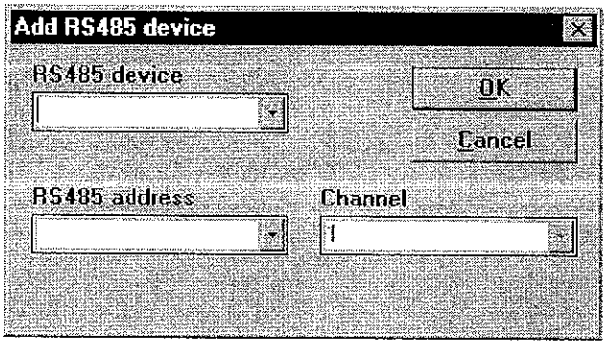
This menu point is needed only for serious alterations on the machine (e.g. mounting a PLC automatisation unit on the milling machine EMCO PC MILL 50).
When the device list that is determined here does not fit to the machine configuration, the machine will not work.



Display of the active RS 485 devices

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

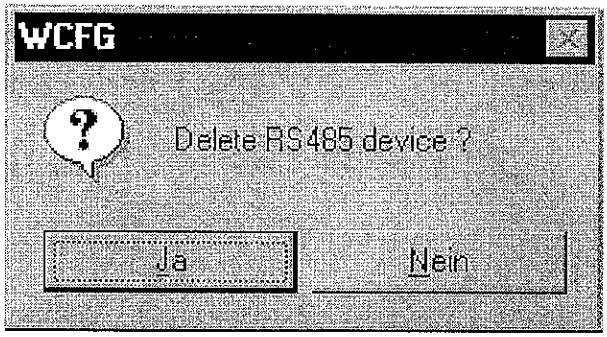
You can add or remove RS485 devices.



Selection window add RS 485 device

Add RS485 device

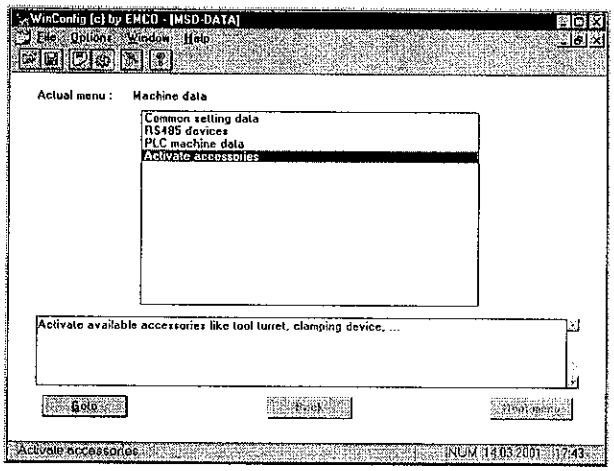
Select the switch button Add. The screen shows a selection window.
Determine which device should be added.
As RS485 address you have to enter that address that is set on the device board at the address switch.



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.

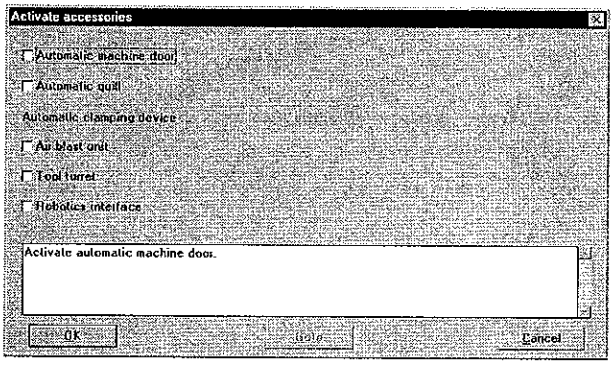


Activate Accessories

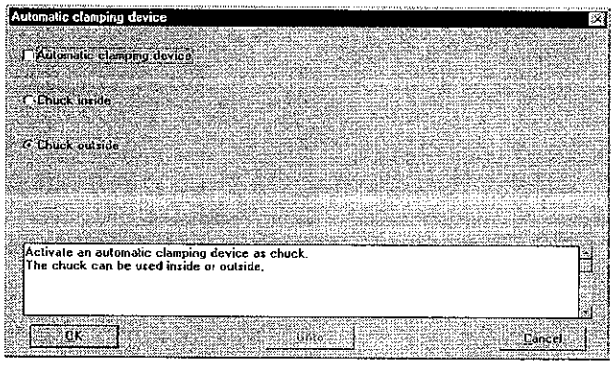
Activate Accessories

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

Select and "Aktivate accessories".



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.

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.

Y: External Input Devices

EMCO Control Keyboard RS232/485

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

The EMCO control keyboard consists of 3 parts:

- Basic acse
- Key module
- Power supply

The basic case is used for all variants of WinNC. Adapting to the used control type occurs by changing the key modules within a few seconds.

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).

Scope of Supply

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

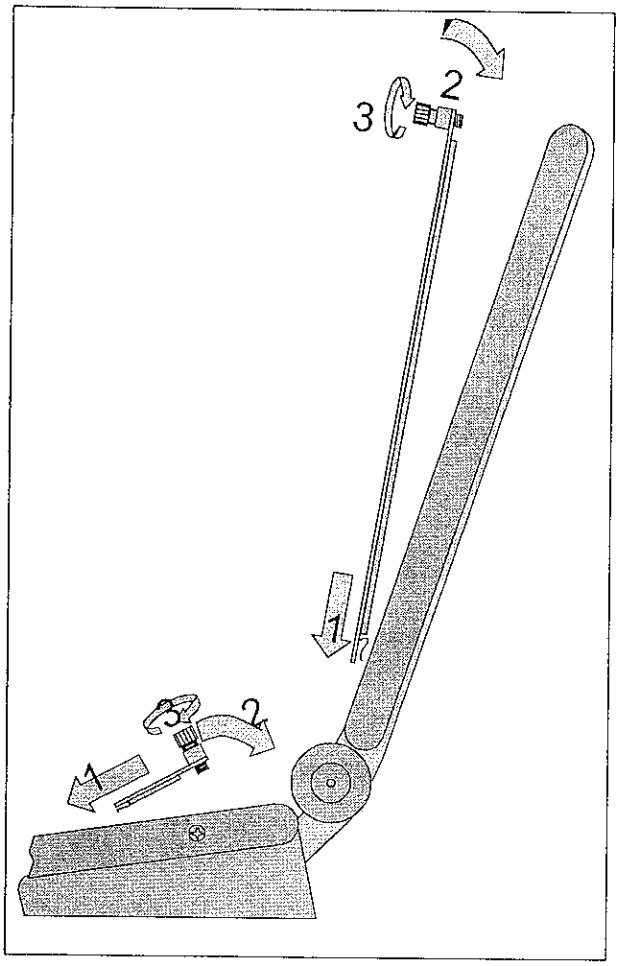
Basic case, Key module, Power supply

Ref. No.	Description
X9A 000	Basic case Including 2 cables (RS 232 and RS 485) for connection to the PC.
X9Z 010N	Key module SINUMERIK 810 2 key sheets with keys 1 package exchange keys
X9Z 020N	Key module SINUMERIK 820 2 key sheets with keys 1 package exchange keys
X9Z 040N	Key module SINUMERIK 840 2 key sheets with keys 1 package exchange keys
X9Z 110N	Key module FANUC 0 2 key sheets with keys 1 package exchange keys
X9Z 130N	Key module FANUC 21 2 key sheets with keys 1 package exchange keys
X9Z 210N	Key module EMCOTRONIC TM02 2 key sheets with keys 1 package exchange keys
X9Z 510N	Key module PAL 2 key sheets with keys 1 package exchange keys
X9Z 520N	Key module HEIDENHAIN 355 2 key sheets with keys 1 package exchange keys
X9Z 426N	Key module HEIDENHAIN 426/430 2 key sheets with keys 1 package exchange keys
795 700	Power supply 230 V
795 710	Power supply 115 V

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.



Assembling

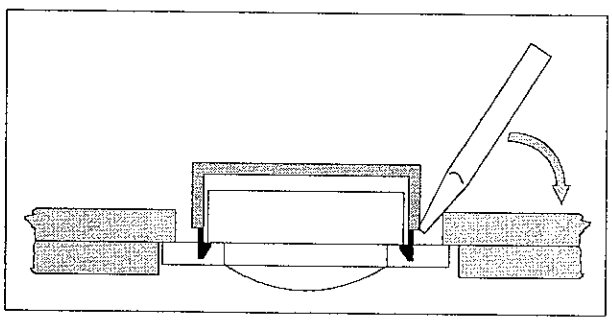
- Place the corresponding key sheet with the clips in the basic case (1).
- Pull the key sheet into the basic case, it must be insertet plainly (2).
- Fix the key sheet with the two knurled screws (3).

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

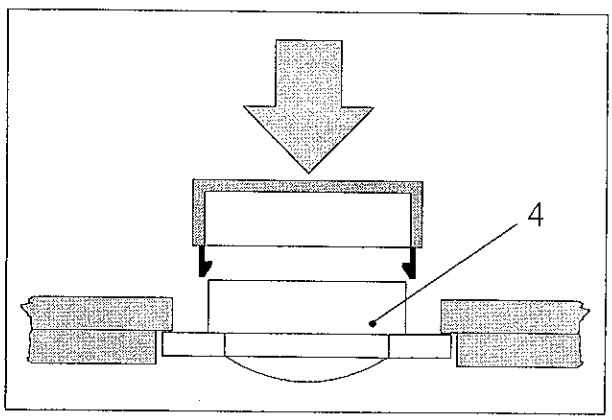
Exchange of single key caps

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.

Note:
For the control type Heidenhain 355 only a milling version is available.
You can select either a Dialog or DIN version an change the corresponding keys.
For the control type Heidenhain 426/430 only a milling version is available.

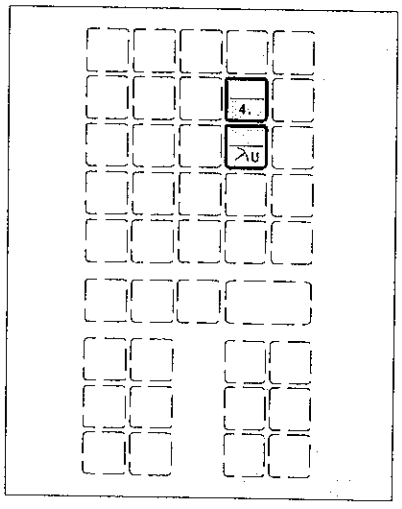


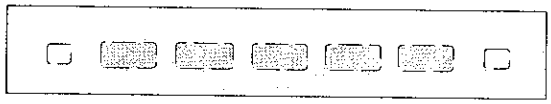
Take off
Pull out carefully the key caps to be exchanged with a fine screw driver or a knife.

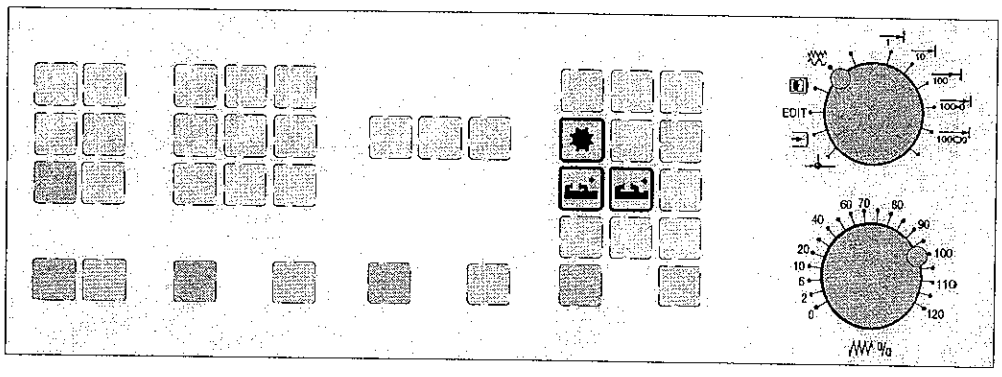


Clip on
Move the key body (4) in the middle of the recess. Push the key cap vertically down onto the key body, until the key cap snaps in tactily.

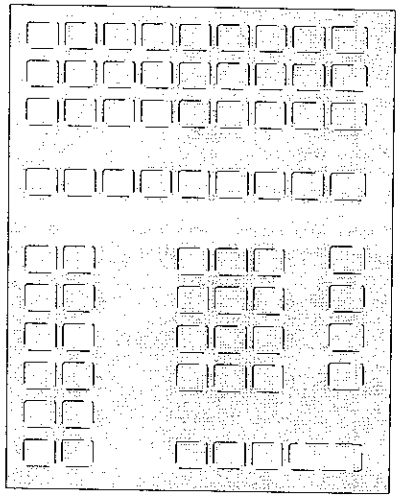
SINUMERIK 810M
Exchange key caps
for milling

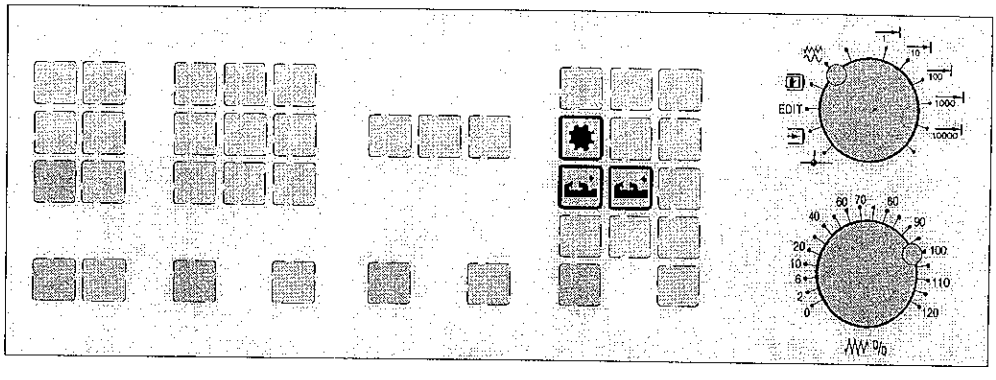




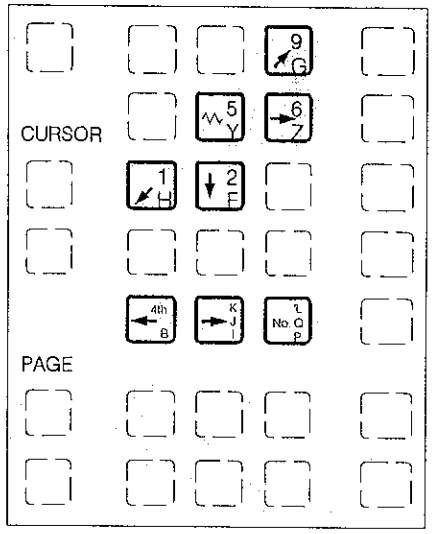


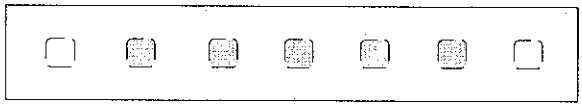
SINUMERIK 820M
and
PAL M
Exchange key caps
for milling

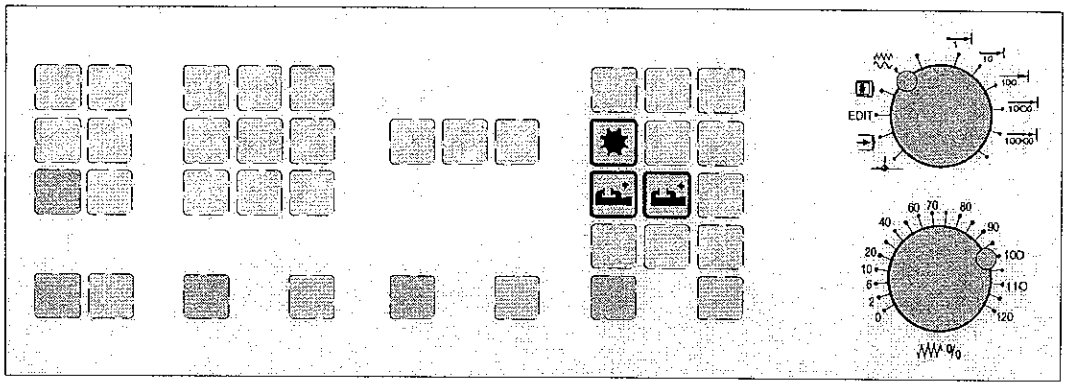




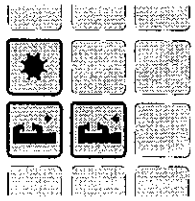
FANUC 0M
Exchange key caps
for milling

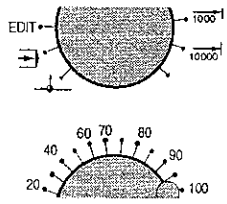




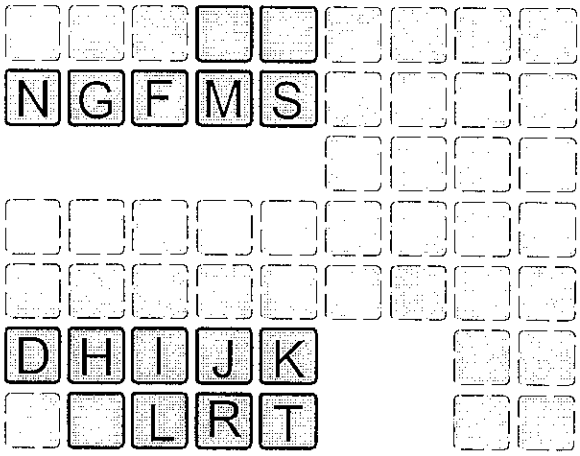


EMCOTRONIC M2
Exchange key caps
for milling

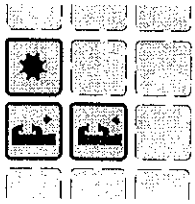


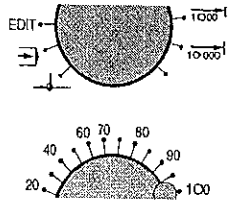


HEIDENHAIN 355
Exchange key caps
for DIN version

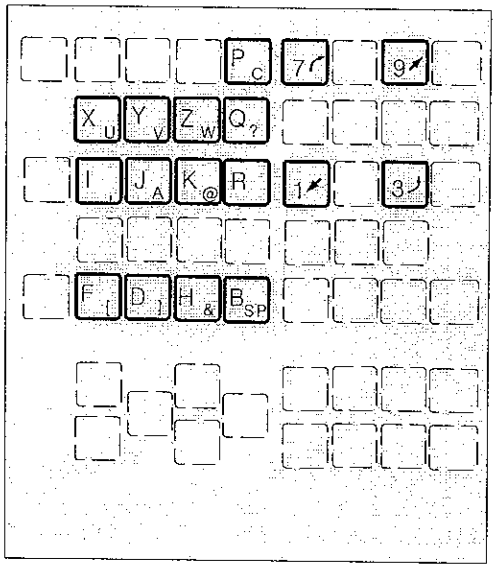


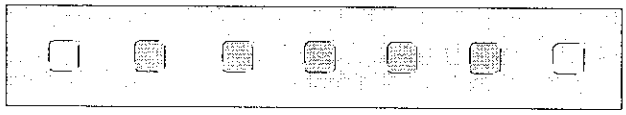
SINUMERIK 810D/840 D Milling
Exchange key caps
for milling

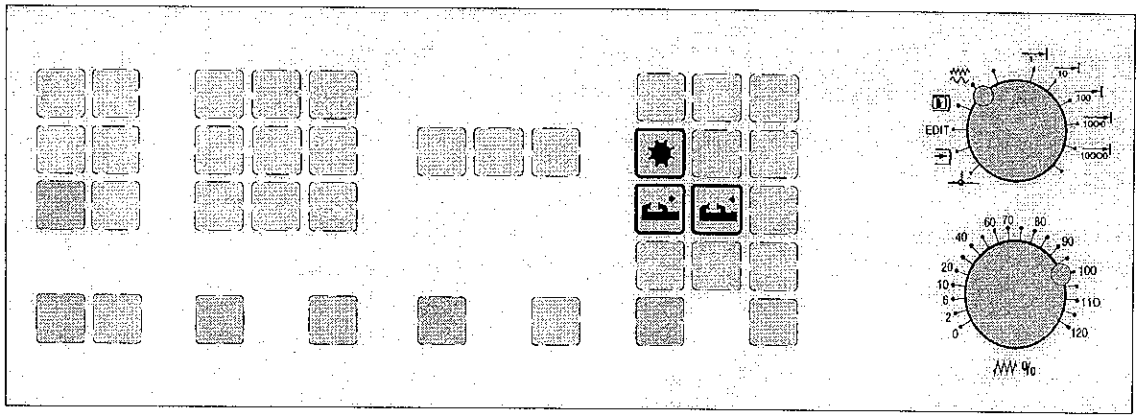


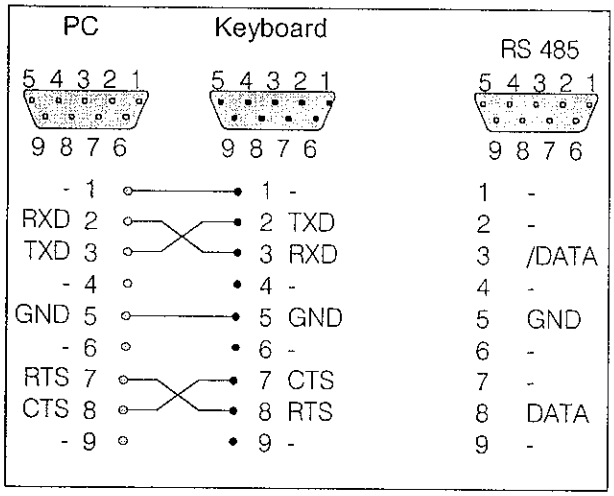


FANUC 21 M
Exchange key caps
for milling

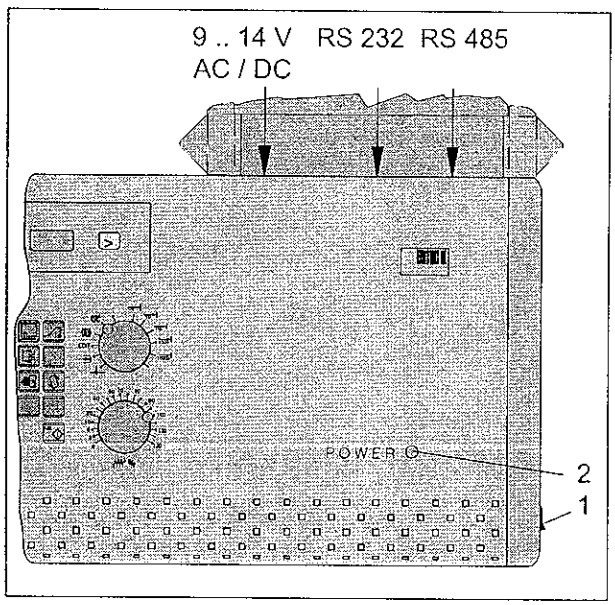


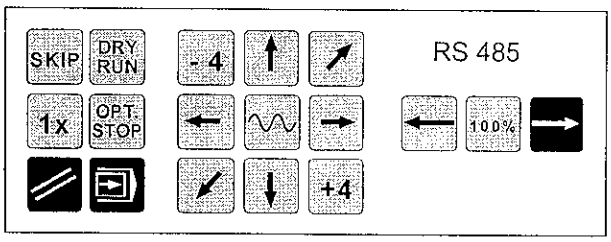


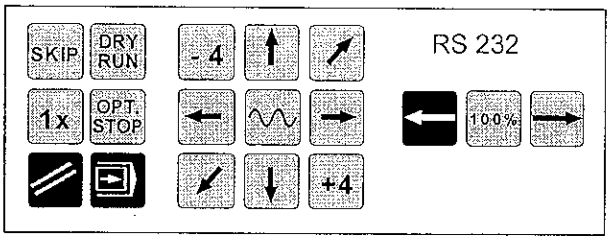




Pin occupation of the interfaces







Connection to the PC

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

Note
When you use the control keyboard in connection to a machine of the series 50/100/120, There is one free socket on your RS 485 PC board. Use this socket for connection to the RS 485 interface of the control keyboard.
When ther is no RS 485 board in your PC (simulation version or MILL/TURN 55/125/155), 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.

Digitizer Tablet

The digitizer tablet and a laid-on overlay (accessory) imitates the EMCO control keyboard.

Order numbers digidizer- overlay:

Control type	Order Nr.
WinNC Sinumerik 810M	ZVP663001
WinNC Sinumerik 810T	ZVP663002
WinNC Sinumerik 820M	ZVP663003
WinNC Sinumerik 820T	ZVP663004
WinNC Sinumerik 810/840D T	ZVP663840
WinNC Sinumerik 810/840D M	ZVP663841
WinNC Fanuc 0M	ZVP663011
WinNC Fanuc 0T	ZVP663012
WinNC Fanuc 21TB	ZVP663210
WinNC Fanuc 21MB	ZVP663220
WinNC Heidenhain 355 Dialog	ZVP663030
WinNC Heidenhain 355 DIN	ZVP663031
WinNC Heidenhain 426/430	ZVP663426
WinNc Emcotronic M02	ZVP663021
WinNc Emcotronic T02	ZVP663022

The digitizer must be calibrated after installation of WinNC or when the overlay was moved.

Every input at the digitizer is indicated by a beep sound. This sound can be switched off by clicking on the name symbol of the control (e.g. Heidenhain).

A digidizer tablet can be connected direct on COM1 - COM4, if it supports the format of the "Summagraphics MM series".
The digidizer tablet must support the original Summagraphics-MM-format, an emulation is not sufficient.
The digidizer tablet is driven direct by the command interface, consequently 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 NEWSketch HR III**

no DIP switches

- **GENIUS New Sketch 1212HR III**

no DIP switches

- **SummaSketch III**

no DIP switches

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).
With software version 4.0 the mode selectro is not active.

Calibration of the digitizer tablet

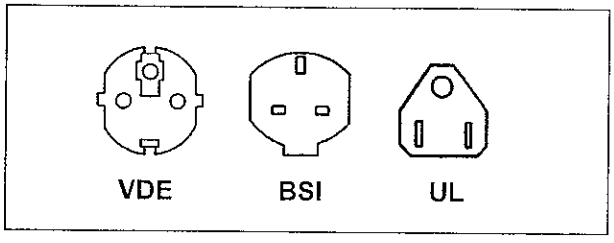
Before the first use of the digitizer the points of the digitizer overlay have to be defined.

- 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 calibrated.

EMCO Control Keyboard USB

Note:
Since the control keyboard is already designed for an optimum operation with a a TFT display, the installation of such a display is recommended. During the use at the machine itself the keys of the control keyboard are illuminated. The control keyboard can be fixed at a machine desk by means of the two front screws.

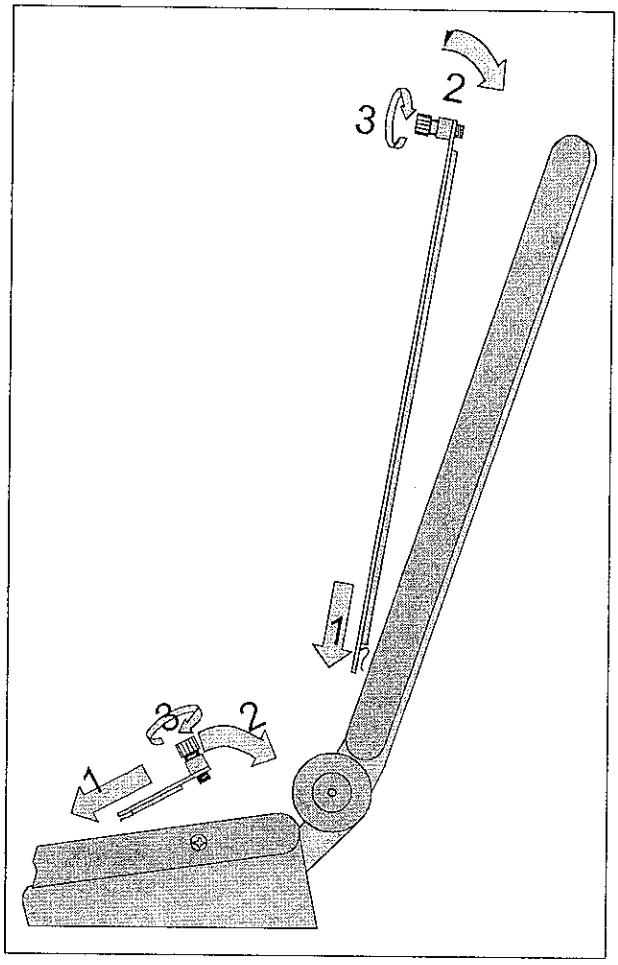


Note:
Installation in Windows 95 and Windows NT4 not possible!

Scope of supply

The scope of supply for a complete control keyboard consists of two parts:
Basic case
Key module

Ref. No.	Description
X9B 000	Basic unit with USB cable
X9Z 600	TFT Display with screen cable and power supply unit
A4Z 010	Mains cable VDE
A4Z 030	Mains cable BSI
A4Z 050	Mains cable UL
X9Z 010N	Key module SINUMERIK 810 2 key sheets with keys 1 package exchange keys
X9Z 020N	Key module SINUMERIK 820 2 key sheets with keys 1 package exchange keys
X9Z 040N	Key module SINUMERIK 840 2 key sheets with keys 1 package exchange keys
X9Z 110N	Key module FANUC 0 2 key sheets with keys 1 package exchange keys
X9Z 130N	Key module FANUC 21 2 key sheets with keys 1 package exchange keys
X9Z 210N	Keymodule EMCOTRONIC TM02 2 key sheets with keys 1 package exchange keys
X9Z 510N	Key module PAL 2 key sheets with keys 1 package exchange keys
X9Z 520N	Key module HEIDENHAIN 355 2 key sheets with keys 1 package exchange keys
X9Z 426N	Key module HEIDENHAIN 426/430 2 key sheets with keys 1 package exchange keys



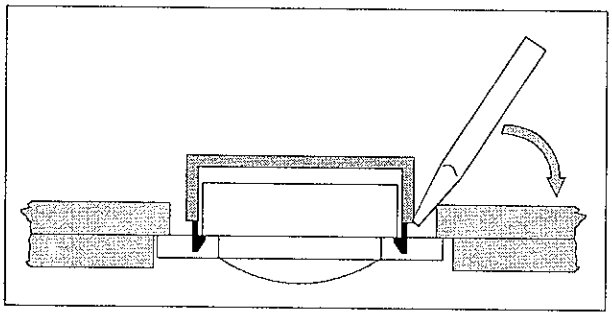
Assembling

- 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.

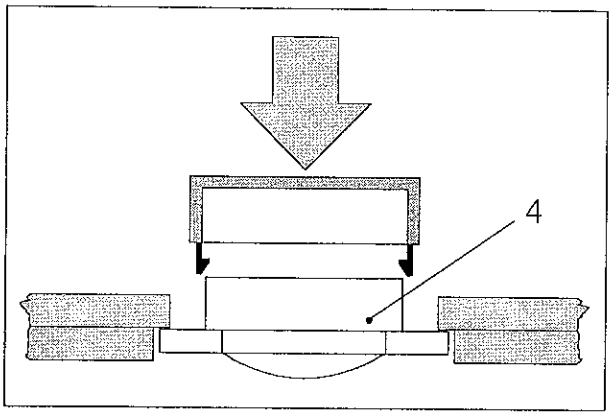
Exchange of single key caps

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.



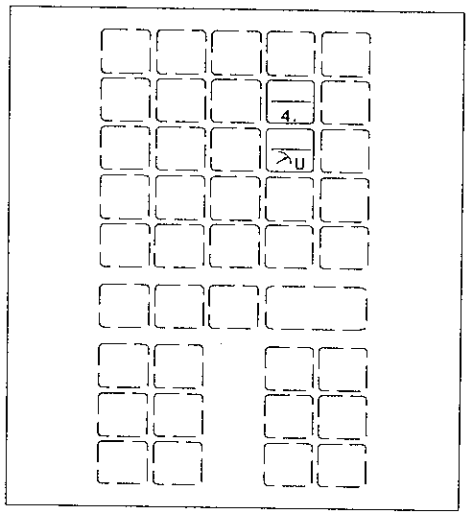
Note:
For the control type Heidenhain 355 only a milling version is available.
You can select either a Dialog or DIN version an change the corresponding keys.
For the control type Heidenhain 426/430 only a milling version is available.

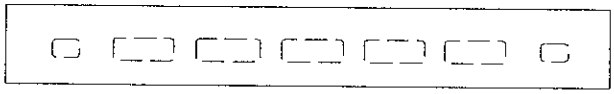
Take off
Pull out carefully the key caps to be exchanged with a fine screw driver or a knife.

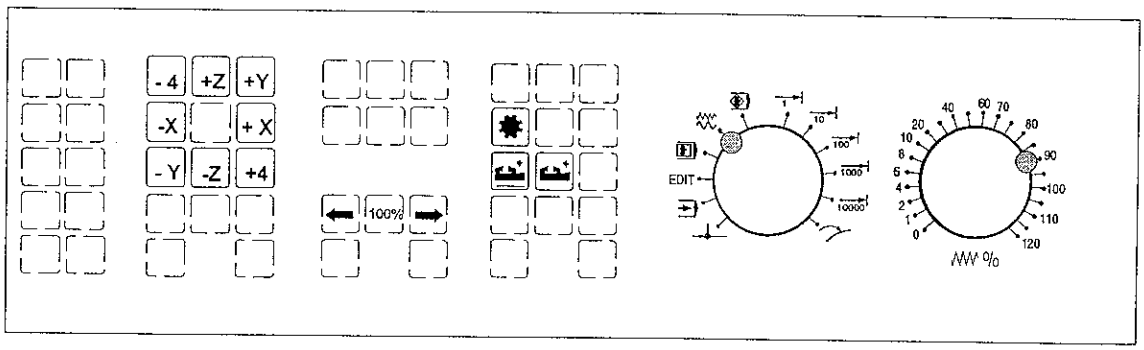


Clip on
Move the key body in the middle of the recess. Push the key cap vertically down onto the key body, until the key cap snaps in tactily.

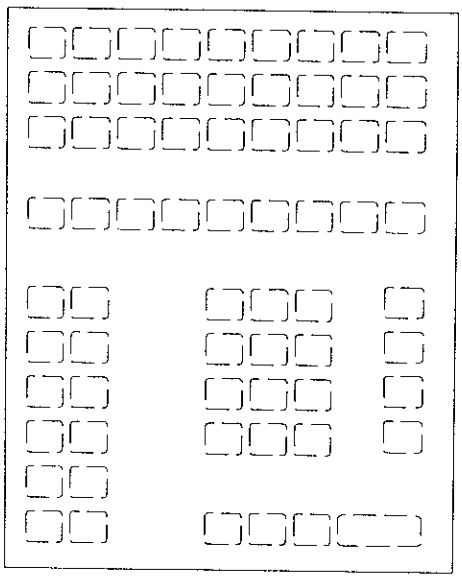
SINUMERIK 810M
Exchange key caps
for milling

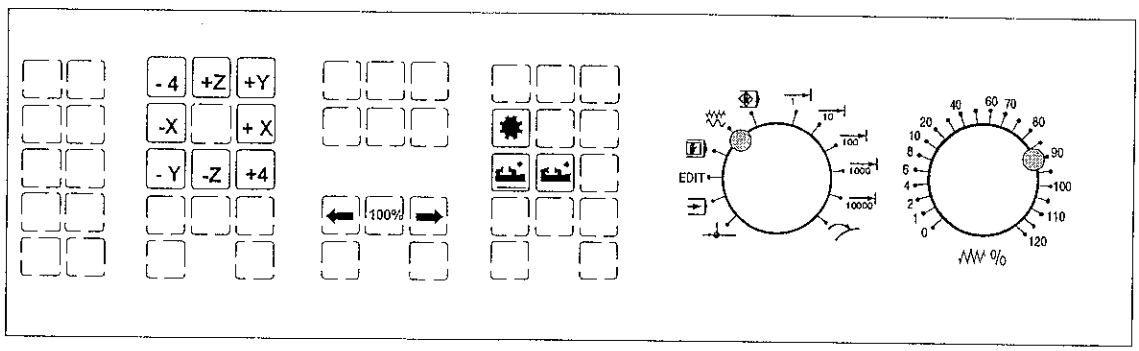




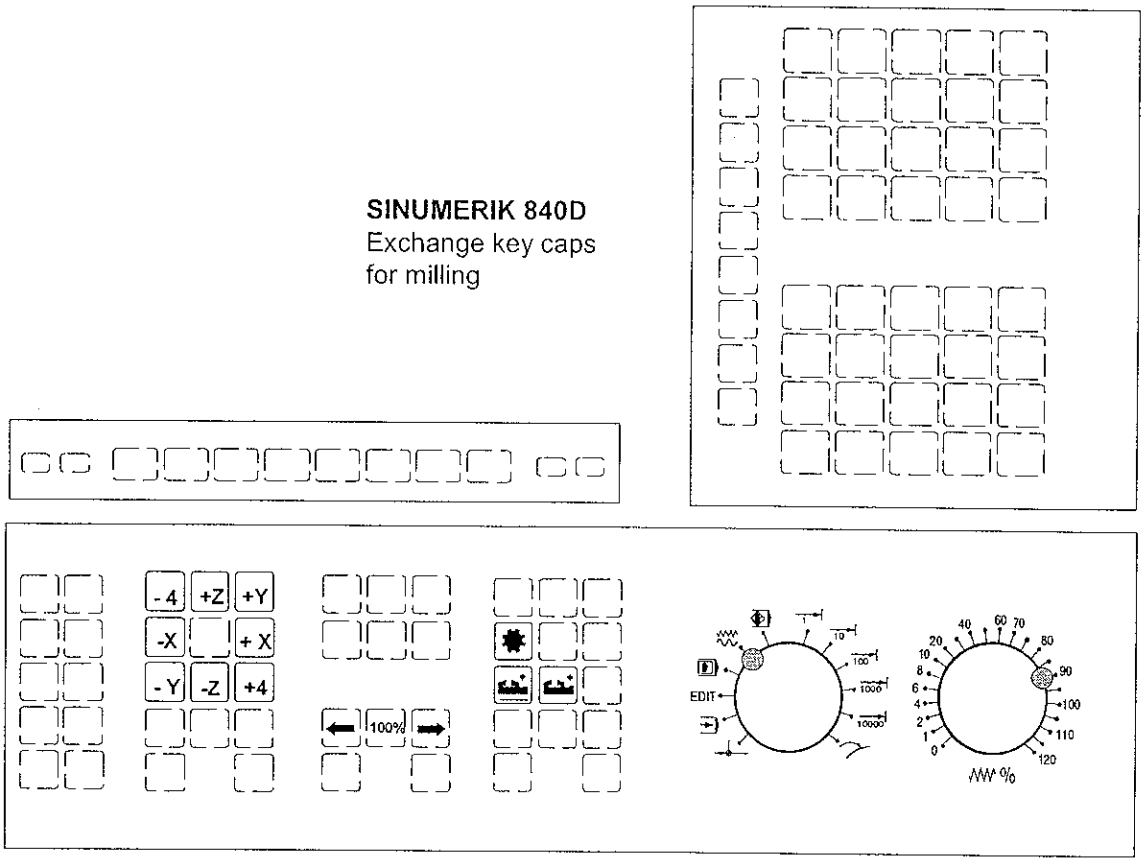


SINUMERIK 820M
Exchange key caps
for milling

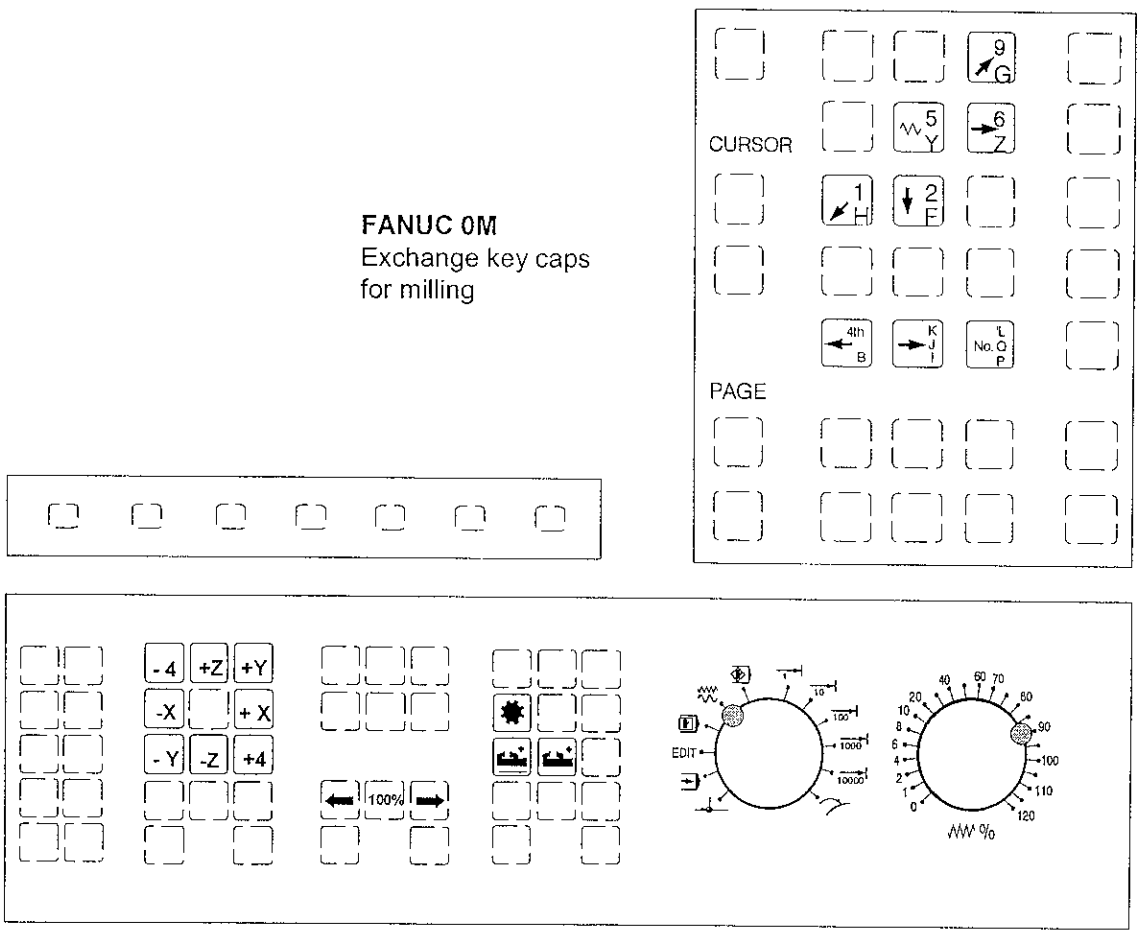


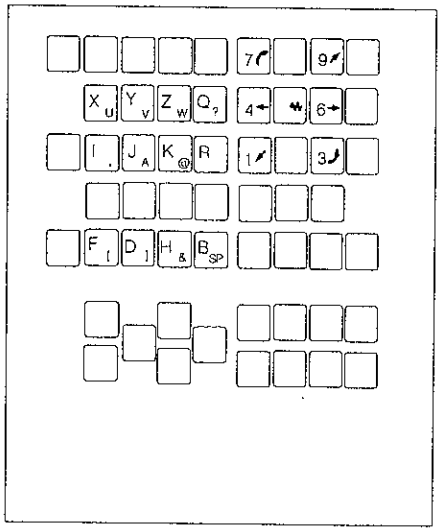


SINUMERIK 840D
Exchange key caps
for milling

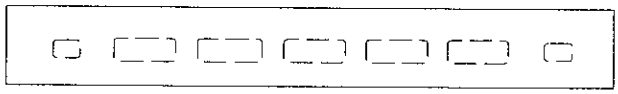


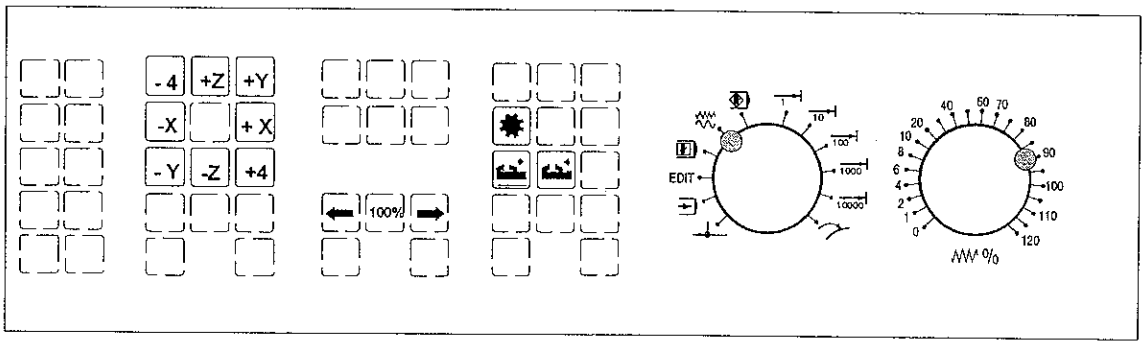
FANUC 0M
Exchange key caps
for milling

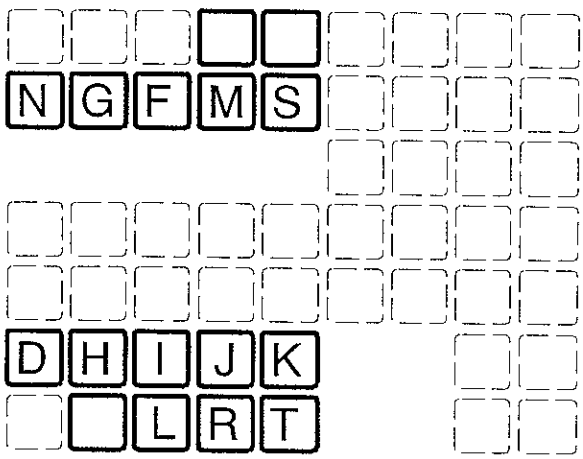




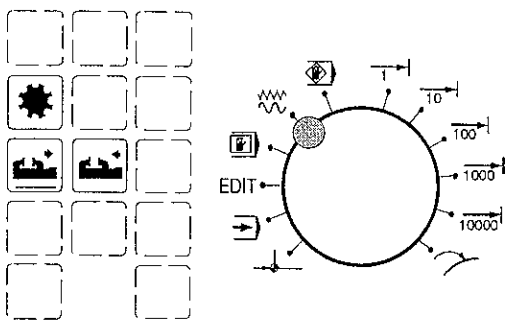
FANUC 21M
Exchange key caps
for milling







HEIDENHAIN 355
Exchange key caps
for DIN-Version



EMCOTRONIC M2
Exchange key caps
for milling

Connection to the PC

The control keyboard is connected via USB interface to the PC.
The connection cable USB taking over at the same time the energy supply of the control keyboard is situated at the rear side of the control keyboard.

Settings at the PC software

Activation of the USB interface

After booting the PC a message appears that a new USB device has been found.

Note:
To enable your PC to recognize the new USB control keyboard, please install the corresponding USB driver from the enclosed disk.

After the installation make sure that your system has recognized the EMCO Control Keyboard (USB).

Setting during new installation of the PC software

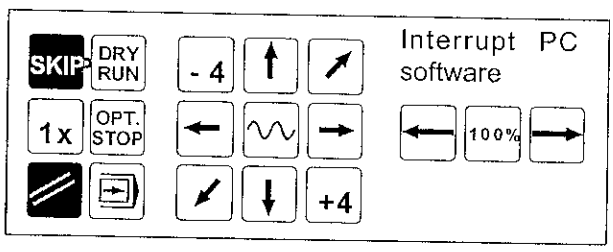
During the installation indicate the control keyboard and the respective USB interface.

Setting in case of PC software already installed

Select in WinConfig at the INI data settings the USB control keyboard as means of entry and the respective interface USB.
Furthermore, set the keyboard type to "New". Don't forget to memorize the settings.

Interruption of the PC software

The PC software can be interrupted by pressing simultaneously the two keys represented in black for at least one second.
This corresponds to Alt+Esc on the PC keyboard.



Z: Software Installation

System Requirements

For running WinNC the following minimum configuration is required:

- PC Celeron or Pentium (II or III) 433 MHz IBM compatible
- 64 MB RAM
- 8 MB VGA colour graphics card
- 3½" floppy drive
- CD ROM drive
- MF2 keyboard
- Colour screen 14"
- Hard disk 4GB
- 1 free ISA or EISA slot for the installation of the interface board
- 10 MB free hard disk space for every installed control type
- WINDOWS 95/98/ME/XP/NT/2000

Machine installations only possible under Windows 95/98.

Variants of WinNC

EMCO WinNC can be installed for the following CNC control types:

- SINUMERIK 810 T and M
- SINUMERIK 820 T and M
- Fanuc Series 0-TC and 0-MC
- EMCOTRONIC TM02 T and M
- PAL T and M
- Heidenhain TNC 355
- SINUMERIK 810D/840 D Milling and Turning
- FANUC 21 TB and MB

When several versions of WinNC are installed on the PC, WinNC starts with a meno to select a version. The following versions can be installed of every WinNC control type:

- Machine licence version:
  With this version a PC-controlled machine (PC TURN, PC MILL) can be controlled by WinNC like a conventional CNC control.
- Single licence version:
  The operation and programming of a CNC control will be simulated at the PC.
- Multiple licence version:
  WinNC is installed on the server of a PC network. Working stations can be installed on any number of network connected PC's.
  On this working stations the operation and programming of a CNC control will be simulated. Multi licence versions can be installed as local version.

Software Installation

- Start Windows 95/98/ME/XP/NT/2000
- AC95: Machine installations under XP/NT/2000 are not possible
- Insert installation CD ROM in CD- drive.
- The installation program will be started (CDStart.exe)
- The installation is menu driven. Study thoroughly the several points.

Notes for Network Installation

Notes:
Network installation is possible only with the multiple licence version of the software.
The network installation must be done by the networks administrator (Supervisor) only.

WinNC is installed once on the server.
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 single user.

First install WinNC on the server and then on all working stations.
In the installation program select "network installation" and indicate whether installing on the server or on a working station.

1. Way

Indication of a local directory for the local data of every working station.
(e.g.: C:\WINNC)

2. Way

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

- Set up directory for user:
  If this is not yet done (as usual in most networks), for every user a directory must be set up in which only the user has acces.
  e.g.: SYS\USERS\USER1
  SYS\USERS\USER2 (Novell)

- 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.

Settings of the Interface Board

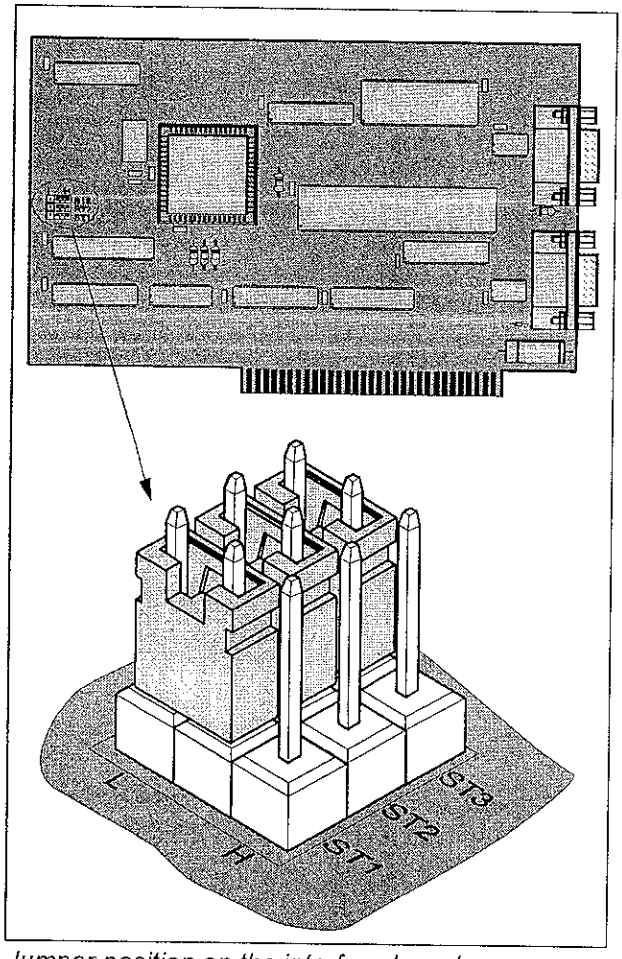
RS 485 Board

(PC TURN 50, PC MILL 50,
PC TURN 120, PC MILL 100)

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, an alarm appears.

After this alarm act as following:



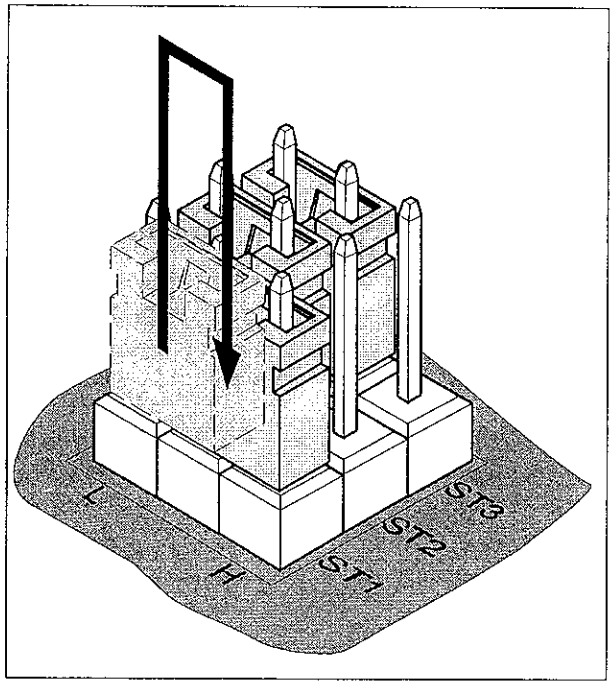
Jumper position on the interface board

Alter jumper positions

By altering the jumpers the interface card can be switched to another memory area.
The following memory areas are available:

RS 485						
Nr.	Jumper ST1	Jumper ST2	Jumper ST3	Hexadecimal Memory Area		
1*	L	L	L	CC000	to	CC7FF
2	H	L	L	CC800	to	CCFFF
3	L	H	L	CD000	to	CD7FF
4	H	H	L	CD800	to	CDFFF
5	L	L	H	CE000	to	CE7FF
6	H	L	H	CE800	to	CEFFF
7	L	H	H	CF000	to	CF7FF
8	H	H	H	CF800	to	CFFFF

*) Basic position



Alter jumper positions

Sequence:

Danger

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

- Remove the interface board from the PC.
- Place the jumpers ST1 - ST3 in the required position (positions 1 - 8 see table on previous page).
- Mount the interface board in the PC.
- Connet the PC to line and switch on.
- Retry installation of the software.
  When the alarm occurs again, try the next jum per position for installation.

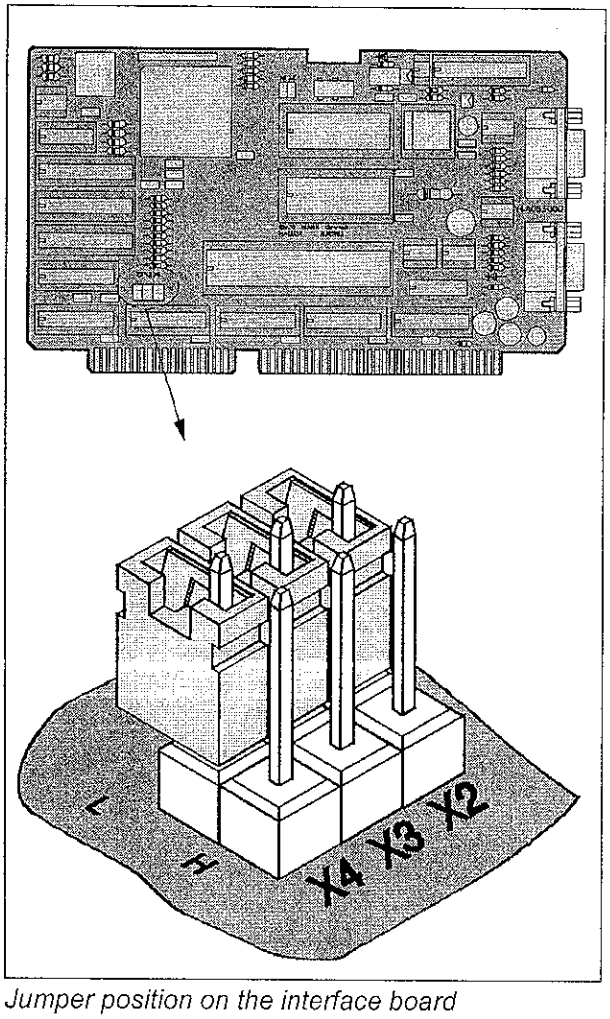
PCCOM (RS 422) Board

(PC TURN 55, PC MILL 55
PC TURN 105, PC MILL 105
PC TURN 125, PC MILL 125
PC TURN 155, PC MILL 155)

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

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

After this alarm act as following:



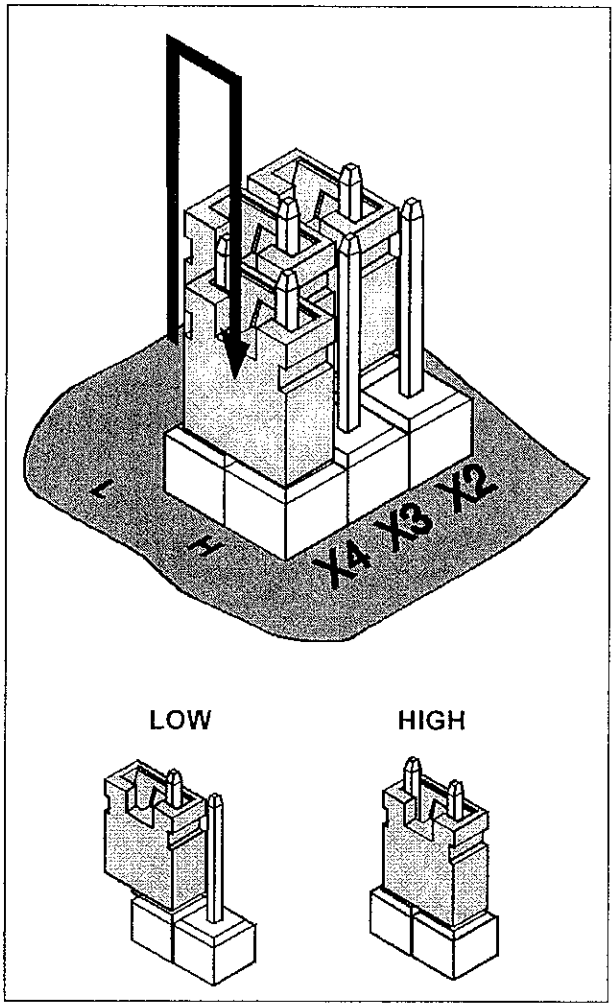
Jumper position on the interface board

Alter jumper positions

By altering the jumpers the interface card can be switched to another memory area.
The following memory areas are available:

RS 422						
Nr.	Jumper			Hexadecimal Memory Area		
	X2	X3	X4			
1*	L	L	L	CC000	to	CC7FF
2[1)]	L	L	H	D8000	to	D87FF
3	L	H	L	CF800	to	CFFFF
4[1)]	L	H	H	E0000	to	E07FF
5	H	L	L	CE000	to	CE7FF
6[1)]	H	L	H	DF800	to	DFFFF
7[1)]	H	H	L	D0000	to	D07FF
8[1)]	H	H	H	E8000	to	E87FF

*) Basic position
1) From PCCOM- Board- version 1



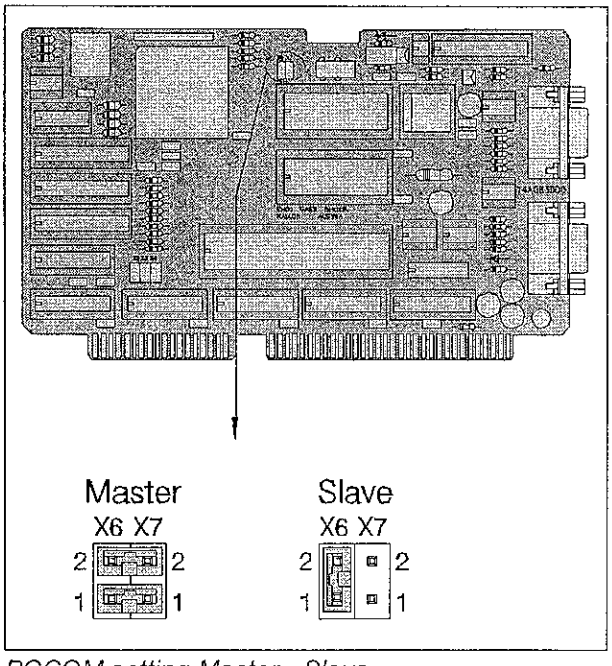
Alter jumper positions

Sequence:

Danger

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

- Remove the interface board from the PC.
- Put the jumpers X2 - X4 in the requierd position (positions 1 - 8 see table on previous page).
  Pins not connected: L
  Pins connected: H
- Mount the interface board in the PC.
- Connet the PC to line and switch on.
- Retry installation of the software.
  When the alarm occurs again, try the next jumper position for installation.



PCCOM setting Master - Slave

PCCOM Master-Slave Setting

Several PCCOM boards can be installed in the PC, e.g. to control more than four axes on a machine.

In this case one of the boards must be set as Master, all other boards must be set as Slaves.

When only one board is installed it also must be set as master.

At delivery all cards are set as Master.

Setting occurs with the jumpers X6 and X7.

Network card

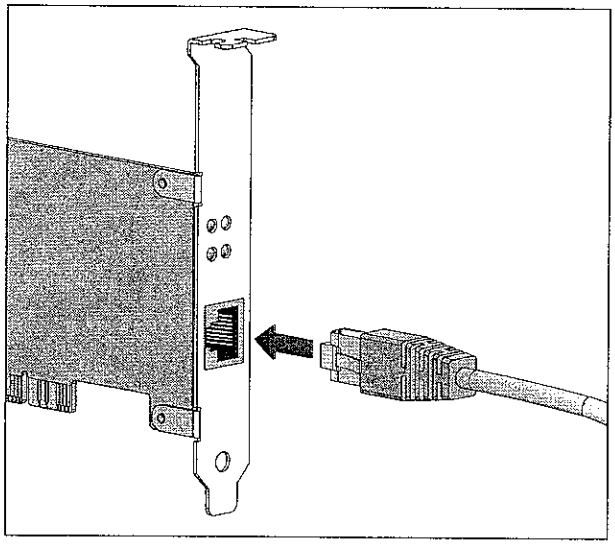
for:
Concept Turn 55
Concept Mill 55
Concept Turn 105
Concept Mill 105

Note:
With a machine installation one network card must be reserved exclusively for the control of the machine.

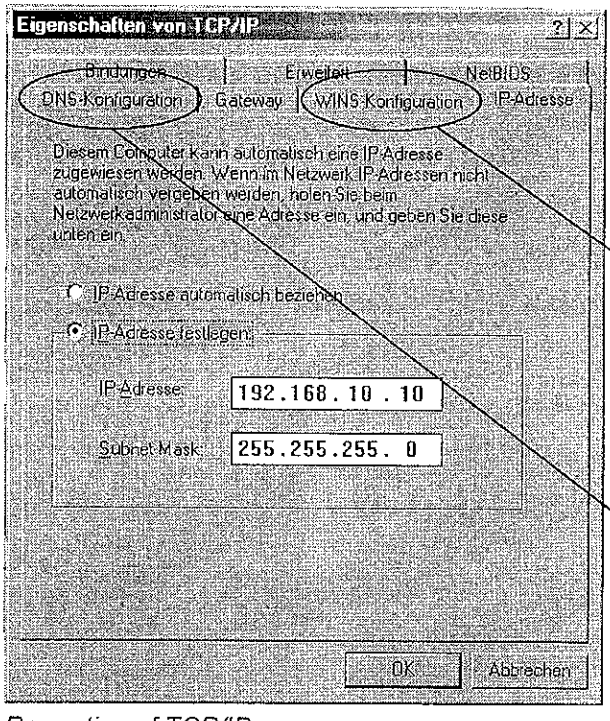
Network card type: TCP/iP enabled network card

Setting of the network card for the local connection to the machine:

IP- Adress: 192.168.10.10
Subnetmask 255.255.255.0

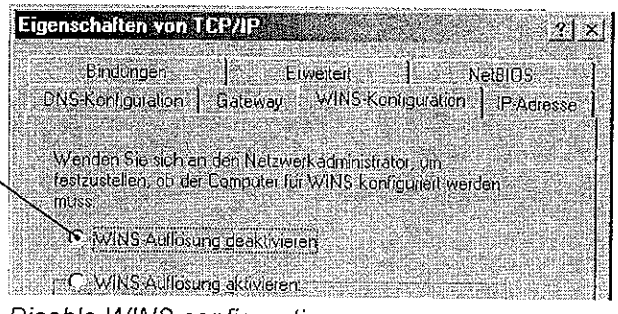


Connection of the machine to the PC

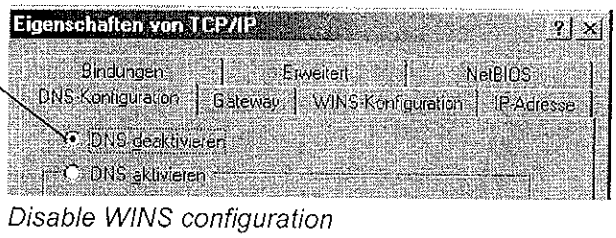


Properties of TCP/IP

Furthermore, in the registers "DNS configuration" and "WINS configuration" are to be disabled.



Disable WINS configuration

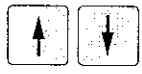


Disable WINS configuration

Starting WinNC

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

Otherwise act as following:

- Switch on the PC and start Windows 95 (resp. automatic start).
- Click on the start symbol in the bottom line.
- Select Programs, EMCO and click on WinNC.
- The screen shows the start picture. In the start picture the version number of WinNC and the licensee are displayed.
- 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.
- Select the desired control type (cursor keys or mouse) 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  .

Notes:
With ACC the machine must be switched on first. Check by means of the LEDs if the machine is ready. After starting the surface a short waiting time results during the transmission of data.

Closing WinNC

Switch off auxiliary drives with  .
By similar pressing the keys "Alt" and "F4" (PC keyboard) or the keys  and  (accessory 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.

With the mouse you can close WinNC by clicking on the symbol in the headline.

Note:
When you have installed WinNC 4 or higher and version lower 3.50 in the same directory, there is no way back into the start menu after closing V<3.50.