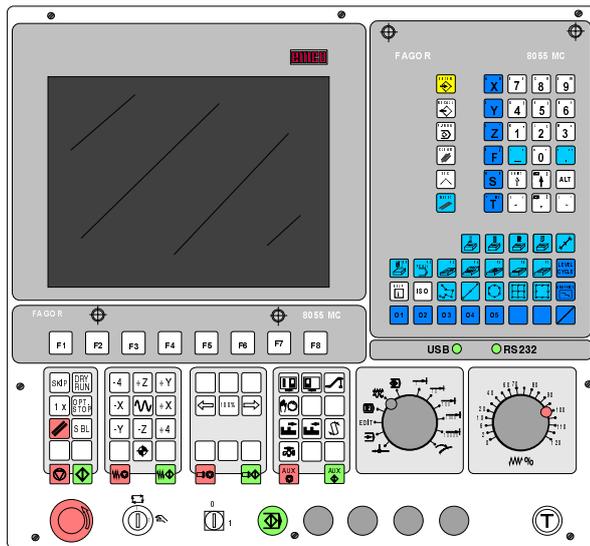


EMCO WinNC Fagor 8055 MC

Software Description/ Software version from 1.16



Software Description EMCO WinNC Fagor 8055 MC Milling

Ref.No. EN 1818

Edition A2005-03

This manual is electronically available (.pdf) upon request
at any time on the EMCO homepage.

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.at

emco
industrial training systems

Notice

This software description contains all functions that may be carried out with WinNC. However, the availability of functions is dependent on the machine you operate with WinNC.



Preface

The EMCO WinNC Fagor 8055 MC Milling Software is part of the EMCO training concept on PC-basis.

This concept aims at learning the operation and programming of a certain machine control on the PC.

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

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

Apart of this software description and the machine description a teaching software CD-ROM "WinTutorial" (CNC examples, operation, description of instructions and cycles) is in preparation.

This manual does not include the whole functionality of the control software Fagor 8055 MC Milling, however emphasis was laid on the simple and clear illustration of the most important functions so as to achieve a most comprehensive learning success.

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

EMCO MAIER Gesellschaft m. b. H.
Department for technical documentation
A-5400 Hallein, Austria

Contents

A: Fundamentals

Reference Points of the EMCO Milling Machines	A 1
Zero Offset	A 2
Reference system for milling machines	A 3
Absolute and incremental workpiece positions	A 4
Absolute workpiece positions	A 4
Incremental workpiece positions	A 4
Tool data	A 5

B: Key Description

Control Keyboard, Digitizer Overlay	B1
Address and Numeric Keyboard	B2
Key Functions	B3
Screen layout of standard screen	B5
Screen layout of special screen	B6
Machine Control Keys	B7
German PC keyboard	B9
English PC keyboard	B11

C: Operation

Operation area of the machine	C 1
Manual JOG mode	C 2
AUTOMATIC MODE	C 4
Program management	C 5
Creating a workpiece program	C 6
Deleting a workpiece program	C 6
Copying a workpiece program	C 7
Changing a workpiece program	C 8
Shifting a cycle	C 8
Changing a cycle	C 8
Deleting a cycle	C 9
Profile editor	C 10
Calling the profile editor	C 10
Screen layout profile editor	C 11
Working with the profile editor	C 12
Editing the profile	C 13
Definition of a straight line	C 14
Definition of a circular arc	C 15
CORNERS	C 16
MODIFY	C 17
DISPLAYAREA	C 19
CONFIGURATION	C 19
Saving the subprogram	C 20
Deleting the subprogram	C 20
Graphic simulation	C 22
Simulating a working cycle	C 22
Simulating the whole workpiece program	C 22
Simulating a part of a workpiece program	C 22
Simulating a stored working cycle	C 23
Graphic illustration	C 24

D: Programming

Overview M-commands	D 2
Overview Cycles	D 3
Input of cycle data	D 4
Technological data input for milling cycles	D 5
Roughing, finishing, complete machining	D 7
Cycles	D 9
Machining cycles	D 9
POSITIONING 1	D 10
POSITIONING 2	D 11
SURFACE MILLING	D 12
GROOVING	D 13
PROFILE 1	D 15
PROFILING	D 16
PROFILE POCKET 2D	D 17
RECTANGULAR BOSS	D 18
ROUND BOSS	D 19
SIMPLE POCKET	D 20
RECTANGULAR POCKET	D 21
CIRCULAR POCKET 1	D 22
CIRCULAR POCKET 2	D 23
BORING 1	D 24
BORING 2	D 25
REAMING	D 26
TAPPING	D 27
DRILLING 1	D 28
DRILLING 2	D 29
DRILLING 3	D 30
CENTER PUNCH	D 31
Multiple positioning	D 32
RANDOM POSIT.	D 32
LINEAR POSIT.	D 33
ARC POSIT. 1	D 34
ARC POSIT. 2	D 35
GRID PATTERN POSIT.	D 36
RECTANGULAR POSIT.	D 37

E: Tool programming

Tool related data	E 1
Feed rate F	E 1
Spindle speed S	E 1
Tool control	E 2
Tool calibration	E 3
Tool calibration with the touching method	E 4

F: Program run

Requirements	F 1
Program start, Program stop	F 2

H: Alarms and Messages

Input Device Alarms 3000 - 3999	H1
Machine Alarms 6000 - 7999	H2
Axis Controller Alarms 8000 - 9999	H10

Starting Information

see attachment

A: Fundamentals

Reference Points of the EMCO Milling Machines

M = Machine zero point

An unchangeable reference point established by the machine manufacturer.

Proceeding from this point the entire machine is measured.

At the same time "M" is the origin of the coordinate system.

R = Reference point

A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the „R“.

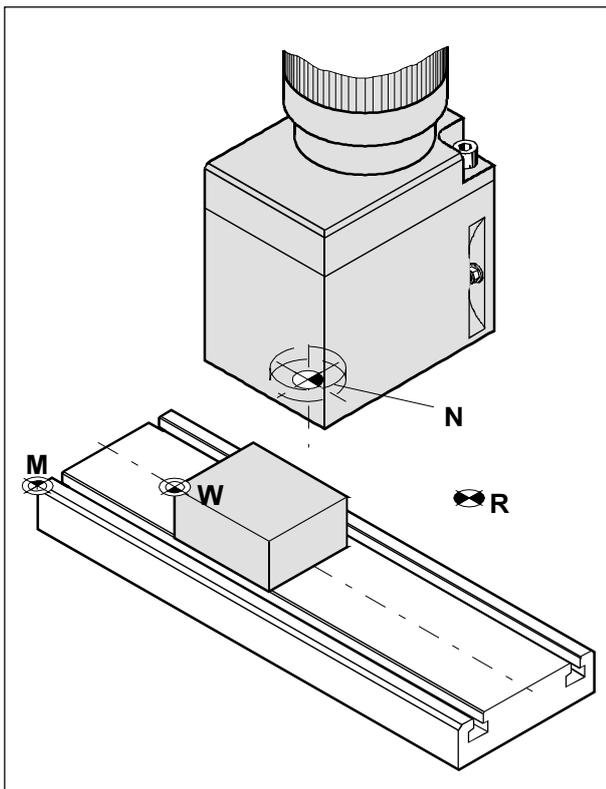
Required after every power failure.

N = Tool mount reference point

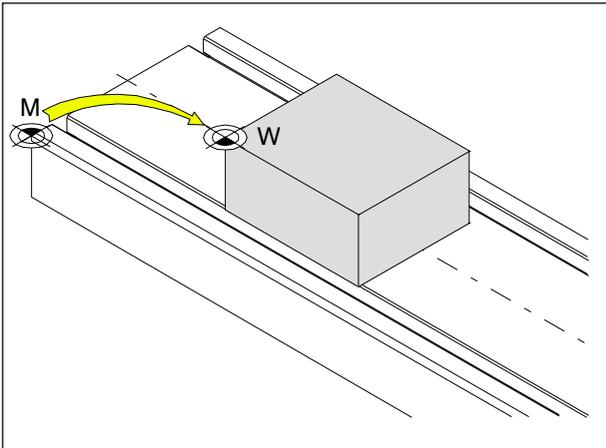
Starting point for the measurement of the tools. „N“ lies at a suitable point on the tool holder system and is established by the machine manufacturer.

W = Workpiece zero point

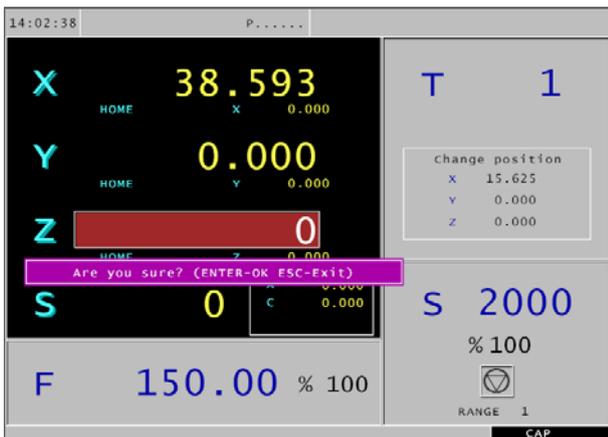
Starting point for the dimensions in the part program. Can be freely established by the programmer and moved as desired within the part program.



Reference points in the working area



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



Definition of the axis value in Z-direction

Zero Offset

The machine datum "M" of EMCO milling machines is located at the left front edge of the machine table. This position is not suited as starting point to programming. With the so called zero offset the coordinate system can be moved to a suitable point within the working area of the machine.

The zero offset is carried out axis by axis.

- Press the button for the axis desired: **X**, **Y** or **Z**.
- Enter the value, at which you want to preset the axis.
- To confirm the entry press **ENTER**, to abort press **ESC**. The WinNC opens the dialogue box "Sure?" (see illustration on the left).
To confirm press **ENTER**, to abort press **ESC**.

Example:

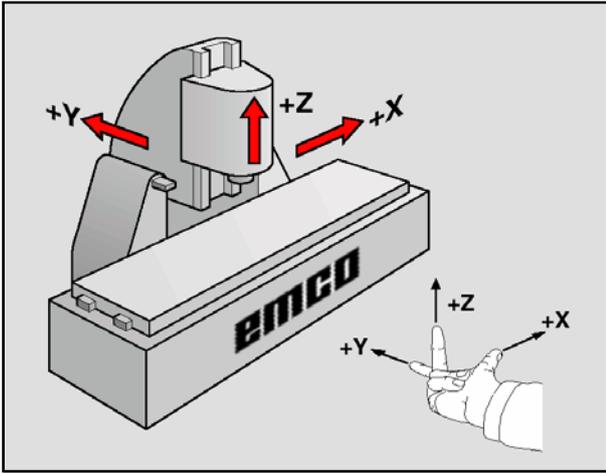
The workpiece zero point should be defined on the upper side of the milling part. Execution: Use a tool to scratch at the upper side of a workpiece trimmed by surface milling.

Press **Z**.

Enter the value "0" and confirm with **ENTER**.

Press **ENTER** again.

The workpiece zero point W is now presetted at the value "0" in Z-axis direction.



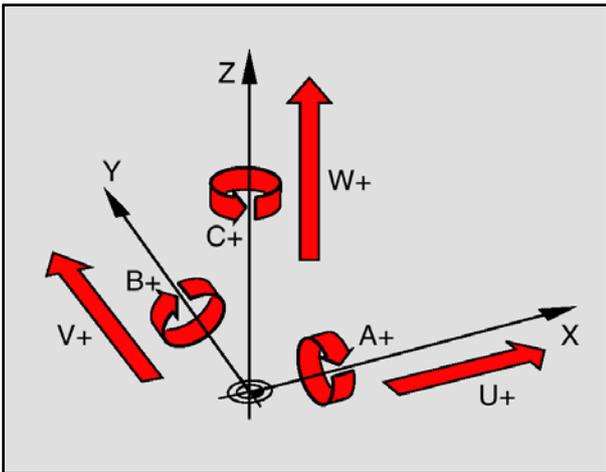
Reference system for milling machines

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.

The Cartesian coordinate system (a rectangular coordinate system) is based on three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate

indicates the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

Coordinates that are referenced to the datum are called **absolute coordinates**. Relative coordinates are referenced to any other known position (reference point) within the coordinate system. Relative coordinate values are also referred to as **incremental coordinate** values.



When working a workpiece on a milling machine you generally orient tool movement to the Cartesian coordinate system. The illustration on the left shows how the Cartesian coordinate system describes the machine axes. The "right-hand-rule" helps to remember the three axes directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece towards the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.

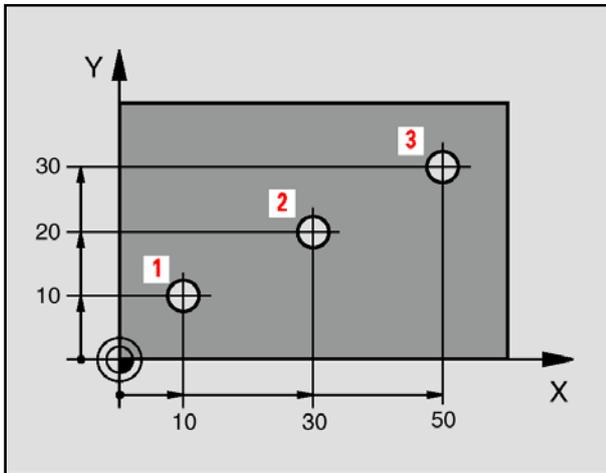
The WinNC is able to control up to 5 axes. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z. Rotary axes are designated as A, B and C. The illustration on the lower left shows the assignment of secondary axes, respectively rotary axes to the main axes.

Notice:

EMCO PC-machines do not endue at secondary axes.



Absolute and incremental workpiece positions



Absolute workpiece positions

Coordinates of a position that refer to the workpiece zero point are called absolute coordinates.

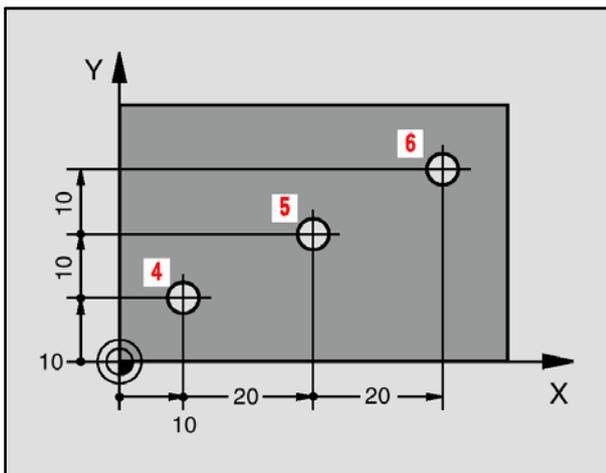
Every position of a workpiece is clearly defined by absolute coordinates (top left-hand illustration).

The origin of the coordinate system is located in the machine datum "M", respectively in the workpiece zero point "W" after a programmed zero offset.

All target points are defined by the respective X-, Y- and Z-distances, leading from the origin of the coordinate system.

Example 1: Holes dimensioned in absolute coordinates

Hole 1	Hole 2	Hole 3
X = 10 mm	X = 30 mm	X = 50 mm
Y = 10 mm	Y = 20 mm	Y = 30 mm



Incremental workpiece positions

Incremental coordinates refer to the position of the tool being programmed last. This position serves as relative zero point. Incremental coordinates define the actual travels of the tool. That is why they are also called incremental dimensions.

Every position of a workpiece is clearly defined by incremental coordinates (illustration in the center left).

The origin of the coordinate system is located in the tool mount reference point "N", respectively in the tool tip after a tool call.

The incremental value programming defines the actual travels of the tool (from point to point).

To program a position in incremental coordinates, enter the prefix "I" before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

IX = 10 mm

IY = 10 mm

Hole **5**, referenced to **4**

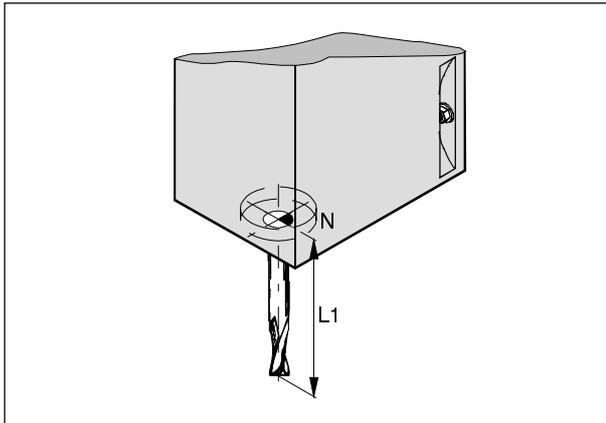
IX = 20 mm

IY = 10 mm

Hole **6**, referenced to **5**

IX = 20 mm

IY = 10 mm



Correction of length

Tool data

The aim of the tool data recording is that the software uses the tool tip, respectively the tool center for positioning and not the reference point for the toolholding fixture.

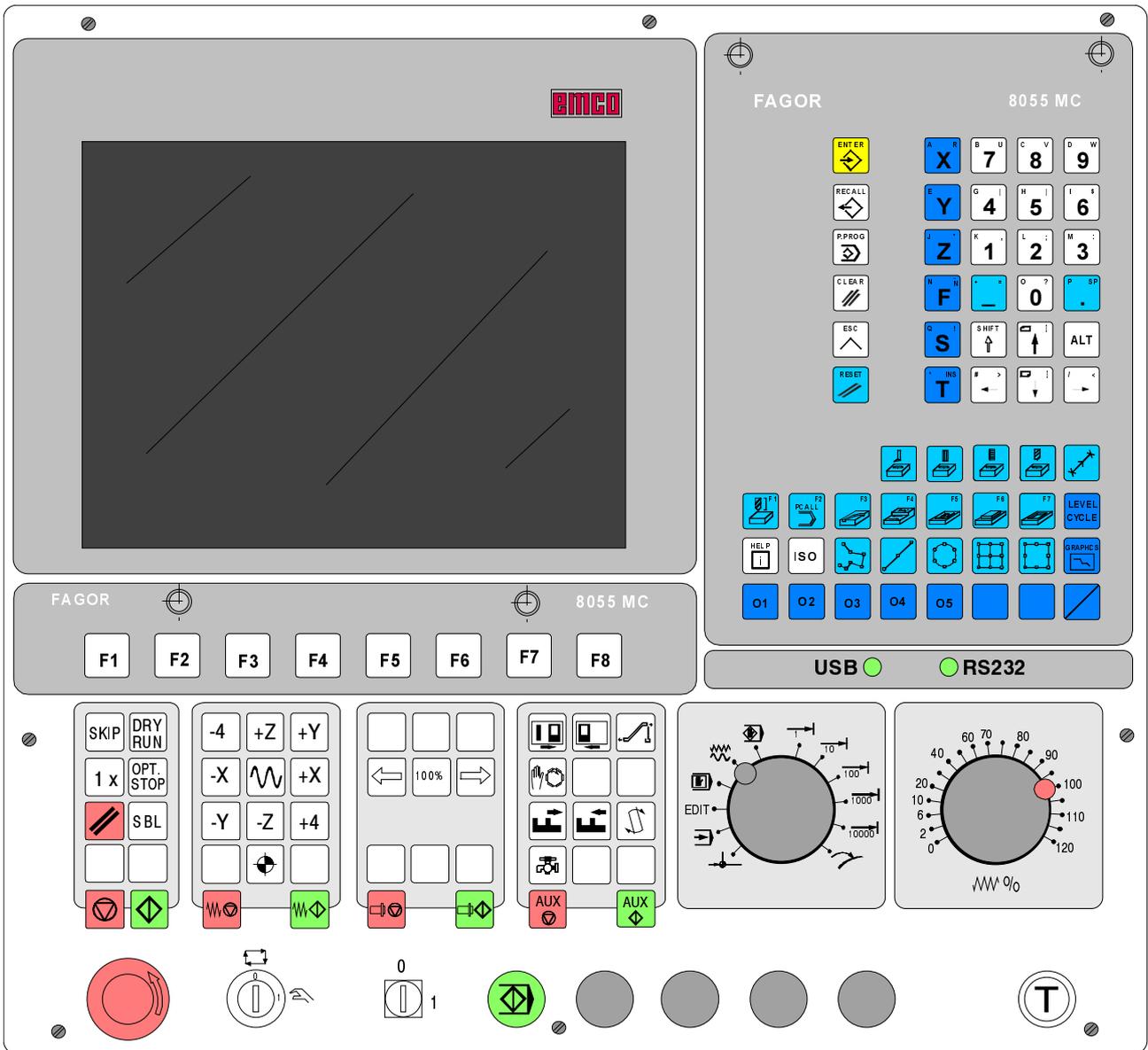
Every tool that is used in the working process must be measured. Therefore, it is necessary to calculate the distance between the tool tip and the toolholding-
fixture reference point "N".

In the so-called tool data store the measured length compensation values and the mill radius can be stored.

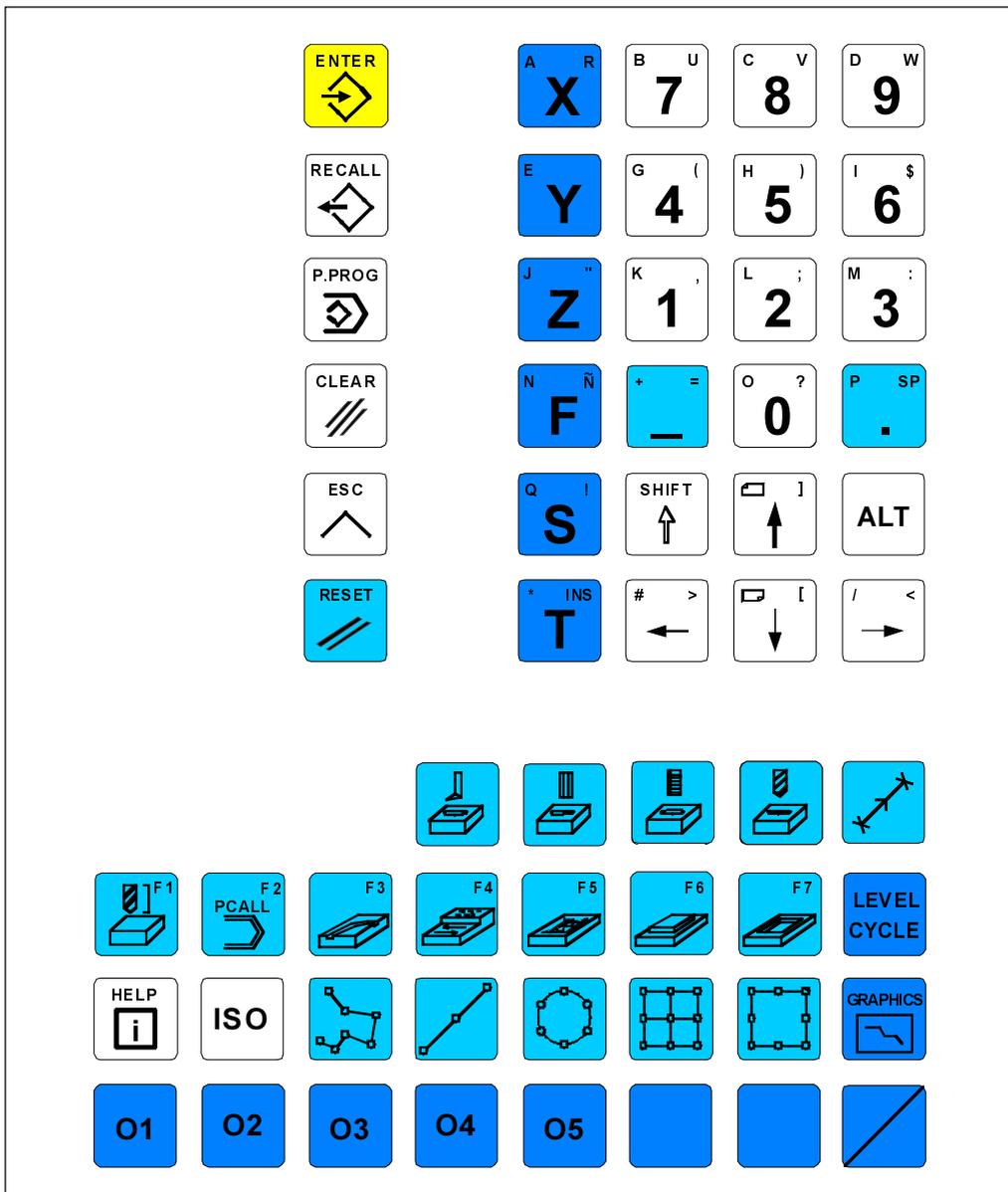
The specification of the mill radius is **only** necessary if a **mill radius compensation** or a milling cycle is selected for the corresponding tool!
(see chapter E tool programming)

B: Key Description

Control Keyboard, Digitizer Overlay



Address and Numeric Keyboard



Key Functions

Input by means of alphanumeric keyboard

 = 7

,  = B

,  = U

Move highlight

    Cursor down / up / left / right

Coordinate axes, feed rate, spindle speed, input of tool number

, ,  Selecting coordinate axes

 Programming the feed rate

 Programming the spindle speed

 Programming the tool number

Enter numbers

Numbers



 Decimal point

Editing

 Selection key

 Confirm input and continue the dialogue

 Take over coordinate values or call current cycle

 Delete input

 Abort dialogue or leave menu

 Delete error message in the graphics mode

 Delete error message in the machine mode

Managing programs, files, WinNC functions

	Create programs, copy or call file management
	ISO-programming
	Show help
	Graphic illustration
	Switch to cycle menu

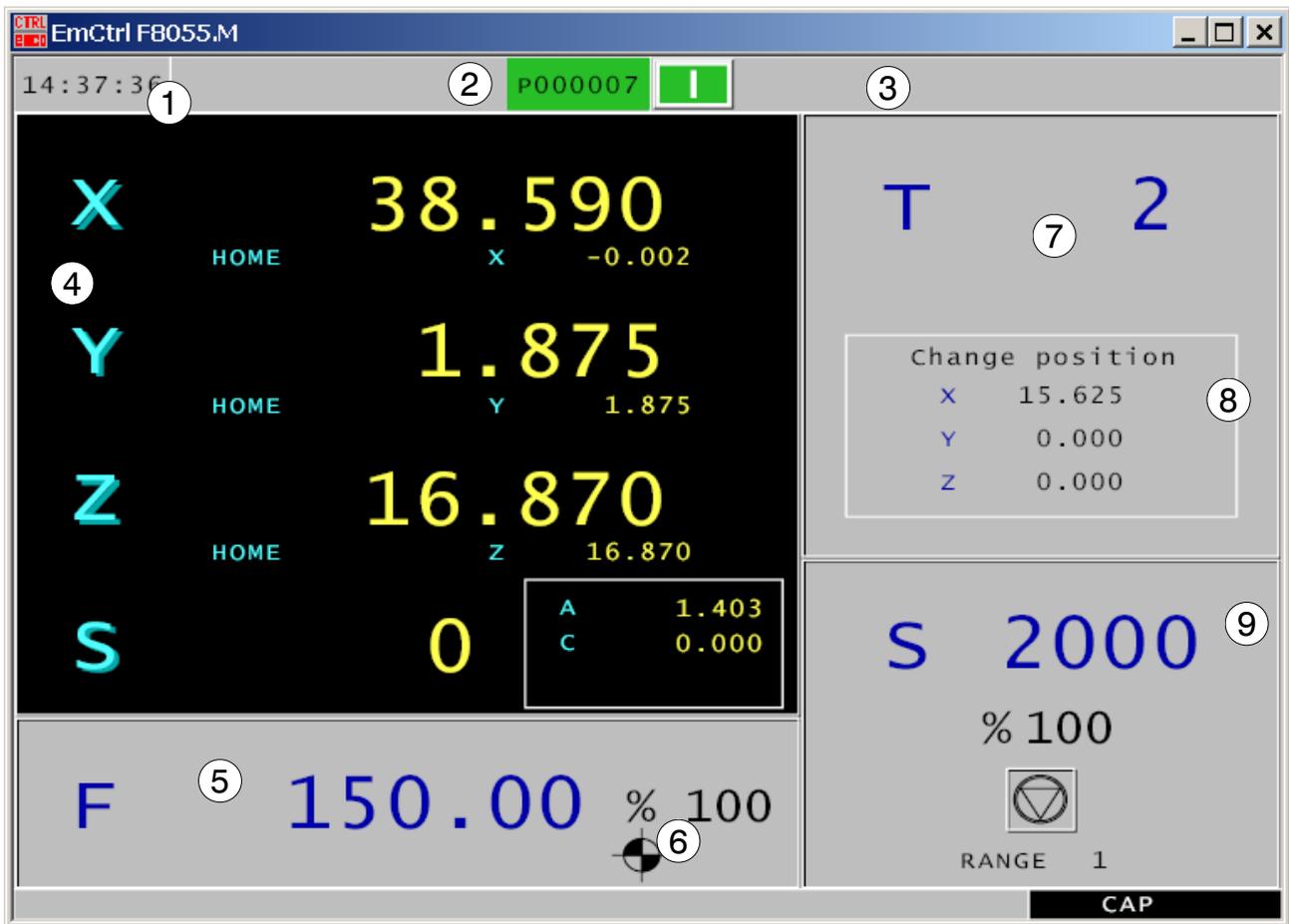
Cycles

	Boring
	Reaming
	Tapping
	Drilling
	Profile milling
	Surface milling
	Profile pocket
	Bosses
	Pockets
	Positioning

Positioning keys

	Positioning from point to point
	Positioning along a line
	Positioning along a circular arc
	Positionieren along a grid pattern
	Positioning along a rectangle

Screen layout of standard screen



- | | |
|---|---|
| <p>1 Time</p> <p>2 This window can show the following data:
SBK in case the mode blockwise execution has been selected.
P.... number of the selected program
Text display: "Positioning", "Execution", "Interruption", "RESET"</p> <p>3 The text displays of the WinNC appear in this window.</p> <p>4 Working window, NC-displays</p> <p>5 Display of the feed rate</p> <p>6 Display of the operating mode</p> <p>7 This window shows the number of the selected tool "T" as well as the number of the correction value "D".</p> | <p>8 Coordinates of the tool changing point related to the workpiece zero point.</p> <p>9 This window shows all information regarding the spindle:
The selected nominal spindle speed "S", during operation in RPM/min.
Symbol for spindle condition (clockwise rotation, counterclockwise rotation or standstill).
Used % of the spindle speed
Active gear position</p> <p>You can switch between standard and special screen by means of the  key.</p> |
|---|---|

Screen layout of special screen



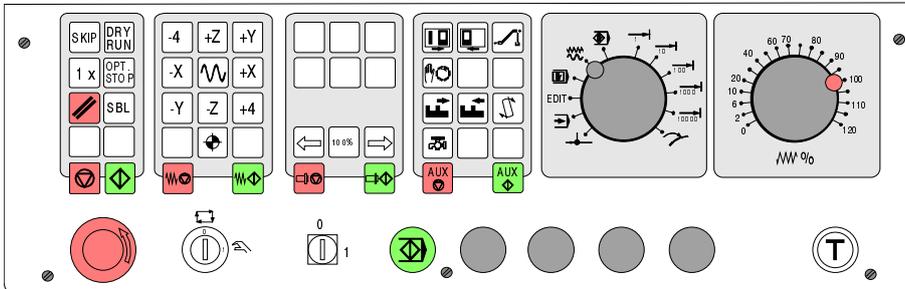
- 1 Time
- 2 This window can show the following data:
SBK in case the mode blockwise execution has been selected.
P.... number of the selected program
Text display: "Positioning", "Execution", "Interruption", "RESET"
- 3 The text displays of the WinNC appear in this window.
- 4 This window shows the command lines of the selected program.
- 5 Each axis disposes of the following fields:
ACTUAL POSITION shows the actual or current axis position.
PATH TO GO shows the remaining distance by which the axis has to be moved in order to reach the programmed coordinate value.
The spindle disposes of the following fields:
NOMINAL VALUE programmed nominal spindle speed S.
RPM spindle speed in rotations per minute.
M/MIN cutting speed in meters/ minute.
- 6 This window shows the status of the activated G-functions and of the auxiliary functions M.
PARTC shows the number of workpieces that have been machined in sequence with the same programm.
CYTIME shows the time that has gone by during the machining of the workpiece.

You can switch between standard and special screen by means of the  key.

Machine Control Keys

The machine keys are in the lower part of the control keyboard or digitizer overlay.

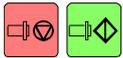
Depending on the used machine and accessory not all of these functions are active.



Machine control keyboard of the EMCO Concept-Mill Serie

Description of keys

	SKIP (skip blocks will not be executed)
	DRY RUN (test run of programs)
	Single Piece Mode
	OPT STOP (program stop at M01)
	RESET
	Single block machining
	Program stop / program start
	Manual axis movement
	Rapid
	Approaching the reference point in all axes
	Feed stop / feed start
	Spindle override lower / 100% / higher



Spindle stop/ spindle start; spindle start in the operating modes "manual operation", respectively "electronic handwheel". Clockwise rotation: press the  key for a short time only; counterclockwise rotation: press the  key for at least 1 sec.



Permissive button open / close door



Open / close door



Swivel dividing head



Open / close clamping device



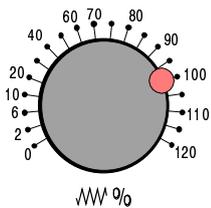
Swivel tool turret



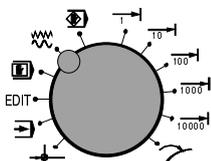
Coolant on/off



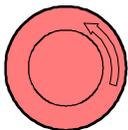
AUX OFF / AUX ON (auxiliary drives off / on)



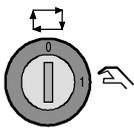
Feed / rapid feed override switch



Mode selector



EMERGENCY OFF (Unlock: pull out button)



Key switch for special operations (siehe Maschinenbeschreibung)

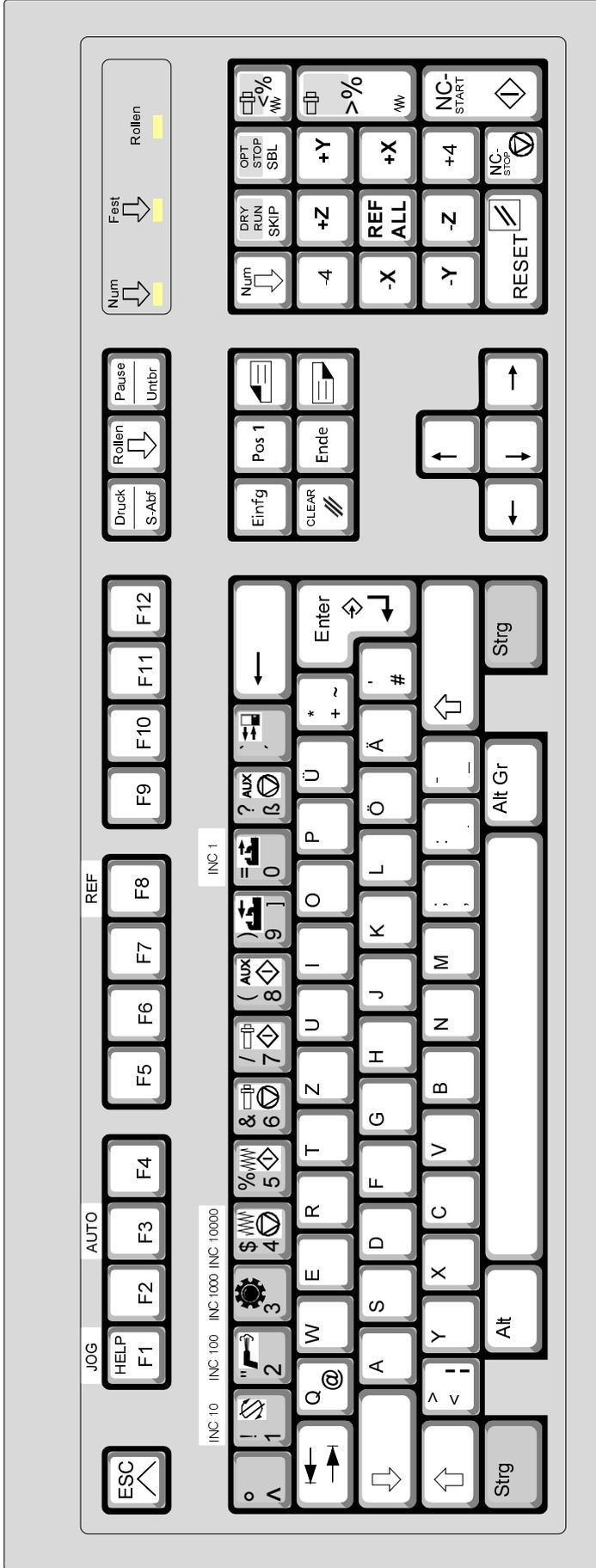


Additional NC start key



No function

German PC keyboard



\$ 4 = 4
 ↑ = \$
 Strg \$ 4 = M = INC 1 0000
 Alt \$ 4 = INC 1 0000

Keys with bold frames represent special functions for the machine and the control; Press the STRG and ALT keys simultaneously in order to activate the patterned key functions.

Several alarms are confirmed with the ESC key.

The meaning of the key combination STRG 2 depends on the machine:

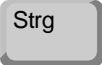
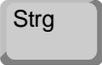
- MILL 55: Puff blowing device ON/OFF
- MILL 105: Coolant ON/OFF
- MILL 125: Coolant ON/OFF

The assignment of the accessories' functions is described in the chapter "functions of accessories".

The machine functions in the numerical block of the keyboard are only active, when NUM-Lock is not active.



Description of keys for German PC keyboard

	Tool calibration cycle			GRAPHICS
	PCALL			ISO
	Profile milling			Switch cycle menu
	Surface milling			P. PROG
	Profile pocket			RECALL
	Bosses			Selection key
	Pockets			Reset during graphic simulation

		HELP
		Boring
		Reaming
		Tapping
		Drilling
		Positioning
		Positioning from point to point
		Positioning along a line
		Positioning along a circular arc
		Positioning along a grid pattern
		Positioning along a rectangle

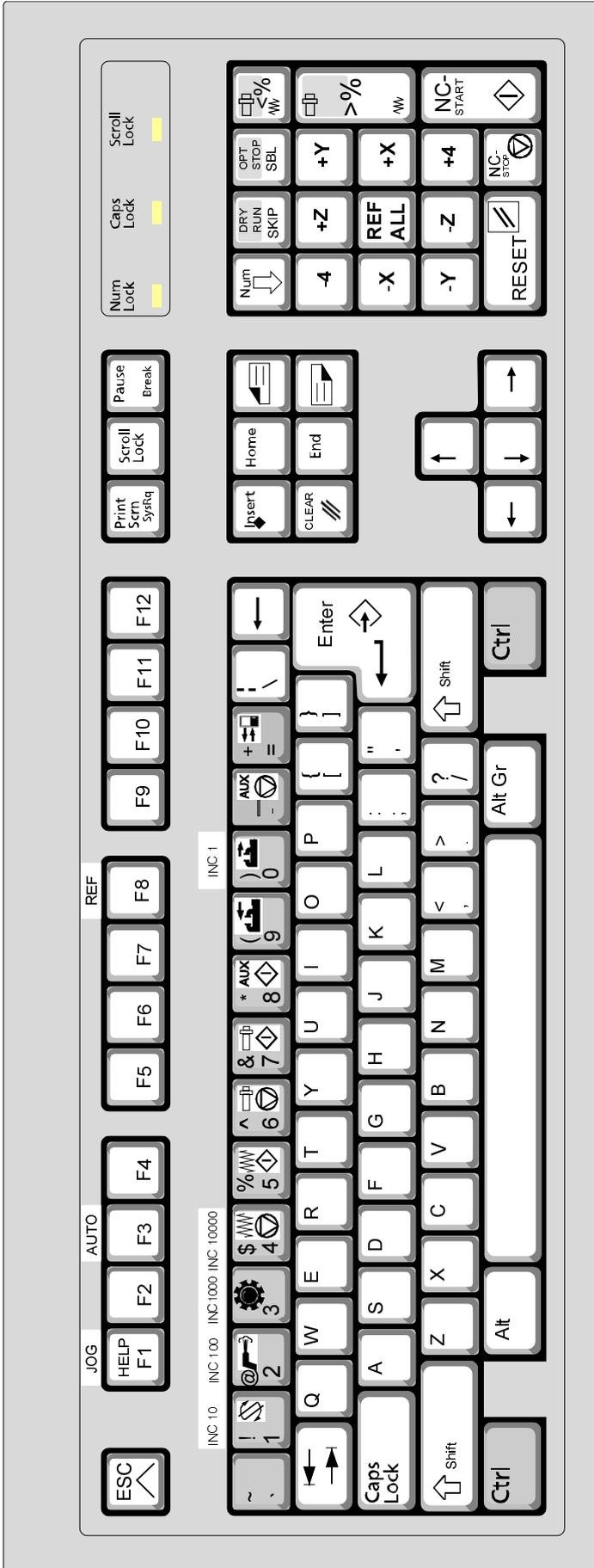
**Notice:**

Selecting machine keys via the PC keyboard:

- 1.) Press and hold  key.
- 2.) Press machine key and then release it.
- 3.) Release  key.



English PC keyboard



- \$ 4** = 4
- ↑ Shift** = \$
- Ctrl** = **\$ 4**
- Alt** = **\$ 4**
- \$ 4** = **INC 1 0000**

Keys with bold frames represent special functions for the machine and the control; Press the STRG and ALT key simultaneously in order to activate patterned key functions.

Several alarms are confirmed with the ESC key.

The meaning of the key combination STRG 2 depends on the machine:

- MILL 55: Puff blowing device ON/OFF
- MILL 105: Coolant ON/OFF
- MILL 125: Coolant ON/OFF.

The assignment of the accessories' functions is described in the chapter "functions of accessories".

The machine functions in the numerical block of the keyboard are only active, when NUM-Lock is not active.



Description of keys for English PC keyboard

	Tool calibration cycle
	PCALL
	Profile milling
	Surface milling
	Profile pocket
	Bosses
	Pockets

		GRAPHICS
		ISO
		Switch cycle menu
		P. PROG
		RECALL
		Selection key
		Reset during graphic simulation

		HELP
		Boring
		Reaming
		Tapping
		Drilling
		Positioning
		Positioning from point to point
		Positioning along a line
		Positioning along a circular arc
		Positioning along a grid pattern
		Positioning along a rectangle



Notice:  Selecting machine keys via the PC keyboard:

- 1.) Press and hold  key.
- 2.) Press machine key and then release it.
- 3.) Release  key.

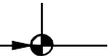
C: Operation

Operation area of the machine

The operation area of the machine comprises all functions and influence factors that lead to actions at the machine tool.

Two modes of operation are distinguished:

- Manual operation mode JOG 
Serves for manual operation and for setting up the machine.
The following functions are available to set up the machine:

Traversing the reference point (Ref) 

Traversing in incremental steps



- AUTOMATIC MODE 
Part programs are worked off fully automatic.

These operating modes can be selected by means of the soft keys (PC keyboard) or with the mode selection switch.

Manual JOG mode

Traversing the reference point

The control is synchronized with the machine by traversing the reference point.

- Set the mode selection switch to .
- Use the direction keys  or  to traverse the reference point in the corresponding axis, analogous to all other axes.
- With the key  or  the reference point is automatically traversed first in the Z-axis and then in the X- and Y-axes.

After having reached the reference point its position is displayed as actual position on the screen. Now the control is synchronized with the machine.

Traversing the slide manually

You can traverse the machine axes manually by means of the direction keys.

- Set the mode selection switch to .
- As long as the keys , , , , , , ,  and so on, are pressed and held the axes are moved in the respective direction.
- The feed rate is set by means of the override switch.
- If you press the key  simultaneously, the slides move at rapid traverse (only for PC Mill 300).

Danger of collision



Look out for obstacles within the working range (clamping devices, clamped workpieces, etc.).

Traversing the slide with presetted coordinates

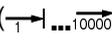
- Set the mode selection switch to .
- Press the keys of the required axis ,  or .
- The WinNC will frame the coordinate value of the corresponding axis.
- Enter the value, at which you would like to preset the axis.
- To confirm, press the  key, so that the WinNC moves the axis to the required coordinate at the set feed rate F.
- To abort, press .

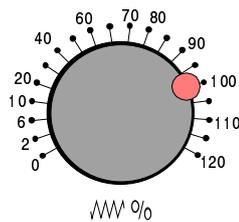
Traversing the slide in incremental steps

During positioning by increments the WinNC moves one machine axis in incremental steps, that you have set before.

You can traverse the machine axes in increments by using the direction keys.

INC 1	1/1000 mm	per depression of key
INC 10	1/100 mm	per depression of key
INC 100	1/10 mm	per depression of key
INC 1000	1 mm	per depression of key

- Set the mode selection switch to INC ( or Alt+0 ... Alt+4 at the PC keyboard, in order to adjust an individual incremental step).
- With each depression of the keys , , , , , , , , and so on, the axes are moved in the corresponding direction by the set incremental step.
- The feed rate is set by means of the override switch.
- If the  key is pressed simultaneously, the slides move at rapid traverse (only for PC MILL 300).



AUTOMATIC MODE

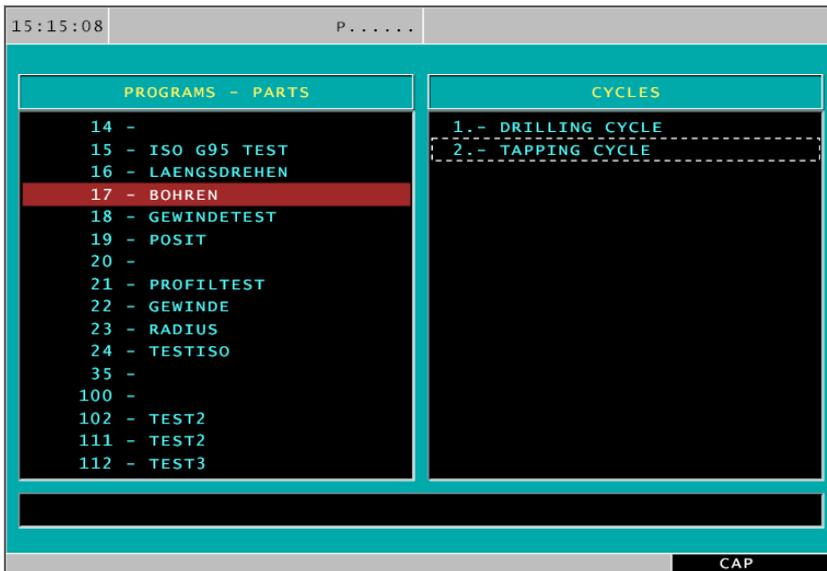
In the AUTOMATIC mode (program run - block mode) you can execute part programs in a fully automatic way.

Preconditions for working off part programs:

- The reference point has been traversed.
- The part program has been loaded into the control.
- The necessary correction values have been checked, respectively entered (e.g. zero offsets, tool corrections).
- The safety lockings are activated (e.g. chip protection door closed).

see chapter F - program run

Program management



A program consists of a sequence of cycles.

Press the  key to get into the program management.

On the left you see the directory of the workpiece programs that are stored in the WinNC. If there exist more programs than the ones displayed in the window, use the keys  and  to leaf through the list. Press the keys

 or the keys  simultaneously to move one page up or down. If one of these programs consists of cycles from the MC mode, these cycles are displayed in the right column.

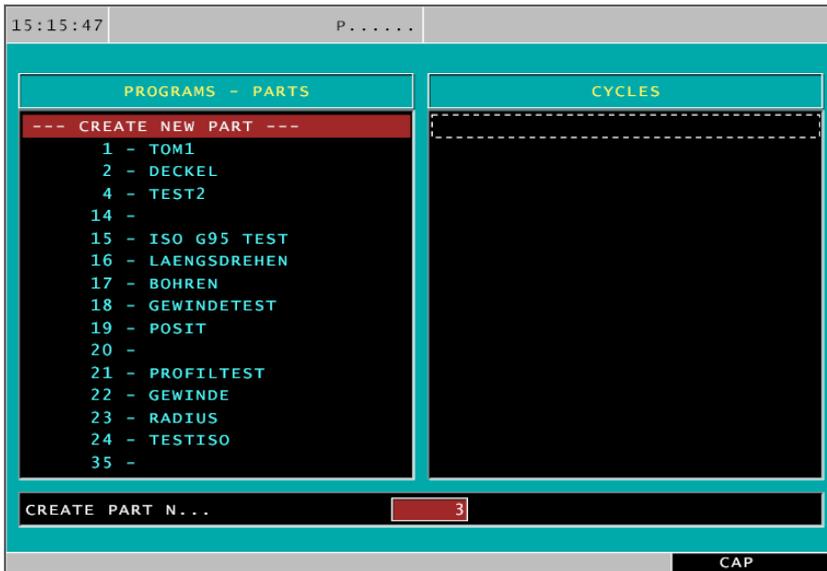
Note:

When the mode "tool calibration" has been selected, there is no direct access to the program management possible. First press  to leave the mode, then press the  key.

The program management enables you to:

- create a workpiece program
- delete a workpiece program
- change a workpiece program
- copy a workpiece program

Creating a workpiece program



- Press the key.
- Use the highlight to select the option "-CREATE NEW PART-" from the left column.
- Press . Enter the program number and press . If there exists already a program that has the same number, the message "PART NUMBER ALREADY EXISTS." appears. With the existing program is overwritten, whereas enables you to select a new program number.

- Enter the program name and press .
- Use the direction keys to switch to the cycle field and define all values of a cycle.
- Press to take the cycle over into the workpiece program.
- Move the highlight to that position within the program where the cycle should be stored and confirm with .
- Enter the next cycle and save it with .

Deleting a workpiece program

- Press the key. Move the highlight to the program that you wish to delete.
- Press . The WinNC opens the dialogue box: "Are you sure?".
- The program is deleted by pressing and it is aborted with the key.

Copying a workpiece program



- Press the  key. Move the highlight to the program you wish to copy.
- Press . The WinNC opens the dialogue box: "COPY A PART N..." (see picture on the left). Enter the program number and confirm with .

The program is now stored under the new number and also remains stored under the old program number.

- If there exists already a program that has the same number, the WinNC opens the dialogue box "PART NUMBER ALREADY EXISTS." (see picture on the bottom left). You can abort by pressing . To overwrite an existing program, press the  key, enter a new program name and confirm with .

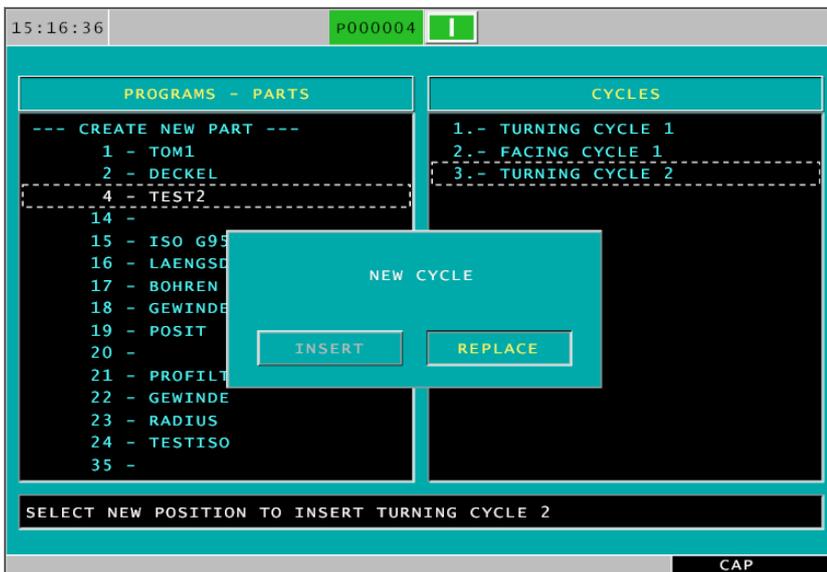


Changing a workpiece program

Shifting a cycle

- Press the  key. Move the highlight to the required program. The cycles are displayed.
- Use the  key to switch to the cycle field and place the highlight on the cycle you wish to shift.
- Press the  key.
- Move the highlight to the place after which the cycle should be inserted and confirm by pressing the  key.

Changing a cycle



- Press the  key. Move the highlight to the required program. The cycles are displayed.
- Use the  key to switch to the cycle field and place the highlight on the cycle you wish to change.
- Press the  key.
- Carry out the required changes, then press  and .
- The WinNC opens the dialogue box "INSERT" or "REPLACE" (see picture on the left).

"INSERT" adds the changed cycle additionally. The old cycle remains stored.

Through "REPLACE" the old cycle is overwritten by the new, changed cycle.

- Select "INSERT" or "REPLACE" and confirm with the  key.

Deleting a cycle

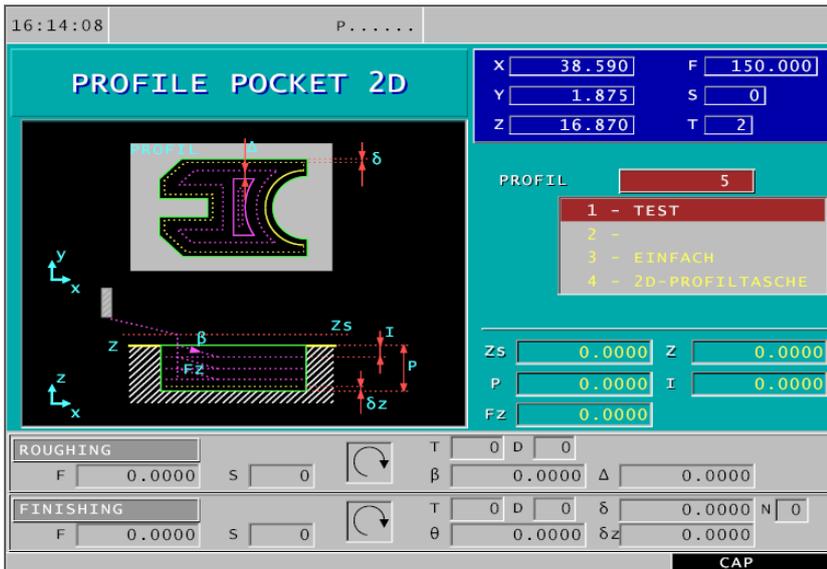
- Press the  key. Move the highlight to the required program. The cycles are displayed.
- Use the  key to switch to the cycle field and place the highlight on the cycle you wish to delete.
- Press the  key.
- The WinNC opens the dialogue box: "Are you sure?". To delete the cycle, press , to abort the dialogue, press .

Profile editor

The profile editor serves to generate any contour desired using any number of points.

The defined contours are saved as subprograms. Before being able to work with the profile editor, a subprogram has to be created.

Calling the profile editor



- Move the highlight to "PROFILE".

- Use the cursor key  to open the selection window of the subprogram (see left-hand illustration).

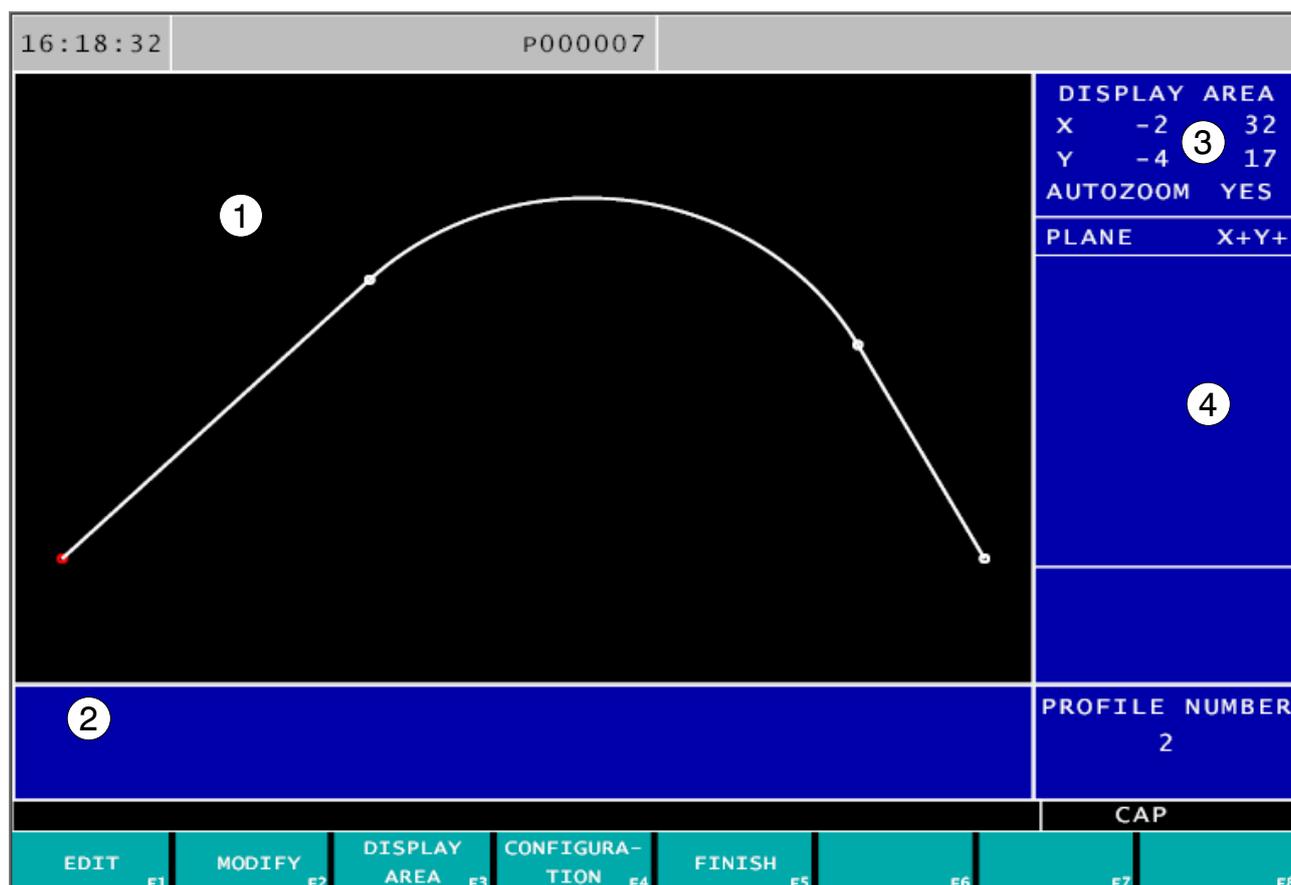
If there exist more subprograms than the ones being displayed in the selection window, use the

keys  and  to leaf through the list.

To quit, press the  key.

- A new subprogram can be edited by entering a number that has not been allocated yet and by pressing the  key.
- In order to edit an already existing subprogram, move the highlight to the requested subprogram number and press the key .

Screen layout profile editor



- 1 Graphic display of the profile, that is currently created.
- 2 This window displays the command lines of the selected element.
- 3 Display area. Defines the area that is shown in the graphic display of the profile by means of the maximum and minimum values of the axes.
- 4 Window to create or to modify the selected profile section.

Working with the profile editor

- Press the soft key **EDIT** F1.
- Select a point of the profile as starting point.
- Divide the profile into straight lines and curves. If the profile has roundings, chamfers, a tangential entry or a tangential exit, treat them like individual sections when sufficient information is available to define them. After the profile definition you may also select the points with the characteristics mentioned and enter the value of the respective radius.

PROFILE F1

PROFILE
Every profile requires a starting point.

CIRCLE F2

CIRCLE
Allows the definition of a circular profile.

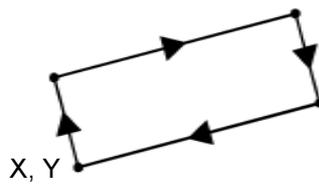
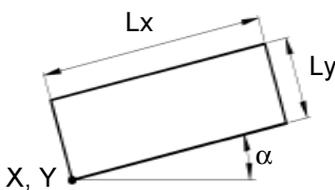
The soft key **PROFILE DIRECTION** F1 indicates, whether the profile is programmed in clockwise or in counter-clockwise direction.

RECTANGLE F3

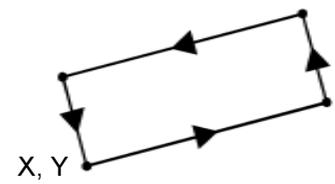
RECTANGLE
Allows the definition of a rectangular profile.

The soft key **PROFILE DIRECTION** F1 indicates, whether the profile is programmed in clockwise or in counter-clockwise direction.

A rectangular profile is defined by one single command. The WinNC divides the profile into 4 linear sections.



in clockwise direction



in counter-clockwise direction

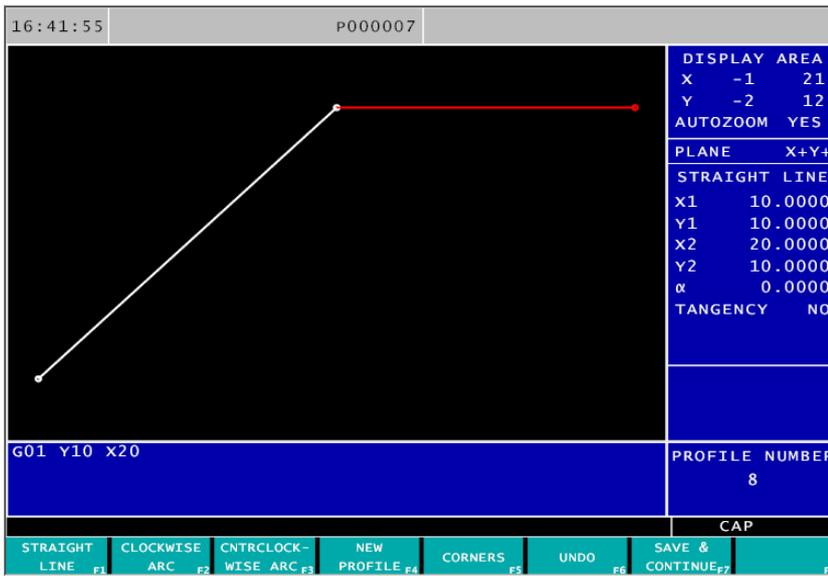
ENLARGE PROFILE F4

ENLARGE PROFILE
Adds an element at the end of the current profile.

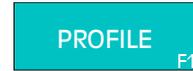
Note:
At the moment the functions "CIRCLE" and "RECTANGLE" cannot be programmed.



Editing the profile



- Press the soft key



- Enter the starting point of the profile. Confirm all value entries with



- Press the soft key



The starting point of the profile is displayed by means of a thick point in the graphic display.



For creating a straight line.



For creating a circular arc in clockwise direction.



For creating a circular arc in counter-clockwise direction.



Finish the editing of the created profile and go back to the main menu.



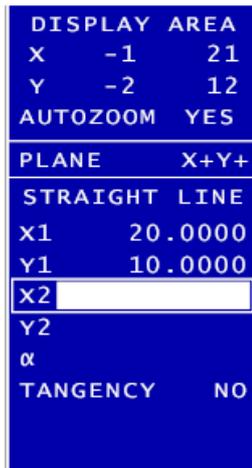
Allows to add roundings, chamfers, tangential entries and tangential exits.



Cancels modifications.



Saves the created profile.



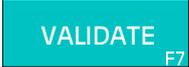
Definition of a straight line

- Press the soft key .

The following information will be displayed:

- X1, Y1** Coordinate values of the starting point of the straight line. These coordinate values cannot be changed, since they are identical to the last point of the previous element.
- X2, Y2** Coordinate values of the end point of the straight line.
- α** Angle between straight line and absciss.
- TANGENCY** Indicates, whether the straight line to draw runs tangentially to the previous section.

Confirm all value entries with .

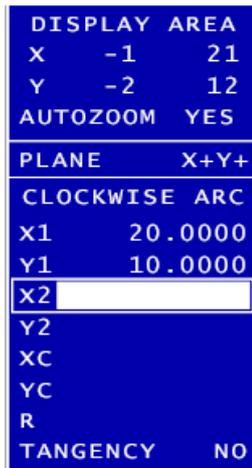
As soon as all parameters known have been defined, press the soft key . The WinNC will display the defined section graphically.

Workpiece contours often cannot be clearly defined, in spite of complete coordinate data. In this case the WinNC will show the different solutions in the profile editor, from which you may choose the one you require. The profile editor displays the workpiece contour by means of different colours:

- white** The contour element is clearly defined.
- red** Marks the current profile element.
- green** No element has been selected from several solutions yet.
- yellow** The element has not been completely defined yet.

In case the data lead to several solutions and the contour element is displayed in yellow colour, please select the correct contour and press the soft keys





Definition of a circular arc

- Press the soft key **CLOCKWISE ARC F2** or **CNTRCLOCKWISE ARC F3**.

The following information will be displayed:

- X1, Y1** Coordinate value of the starting point of the circular arc. These values cannot be changed since they are identical to the last point of the previous element.
- X2, Y2** Coordinate values of the end point of the circular arc.
- XC, YC** Coordinate values of the arc's center.
- R** Radius of the circular arc.
- TANGENCY** Indicates, whether the arc to draw runs tangentially to the previous section.

Confirm all value entries with **ENTER**.

As soon as all parameters known have been defined,

press the soft key **VALIDATE F7**. The WinNC will display the defined section graphically.

Workpiece contours often cannot be clearly defined, in spite of complete coordinate data. In this case the WinNC will show the different solutions in the profile editor, from which you may choose the one you require. The profile editor displays the workpiece contour by means of different colours:

- white** The contour element is clearly defined.
- red** Marks the current profile element.
- green** No element has been selected from several solutions yet.
- yellow** The element has not been completely defined yet.

In case the data lead to several solutions and the contour element is displayed in yellow colour, please select the correct contour and press the soft keys



CORNERS

- Press the soft key



To insert a rounding in any profile corner. Enter the rounding radius R.



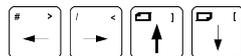
To insert a chamfer in any profile corner. Enter the chamfer radius C.



To define a tangential entry. Enter the radius that the tool requires to execute the tangential entry.



To define the tangential exit. Enter the radius that the tool requires to execute the tangential exit.



When selecting one of the options, a profile element will be marked red. If you wish to select another corner, please press the cursor keys.

Confirm all value entries with .

To deselect the CORNER mode, please press the

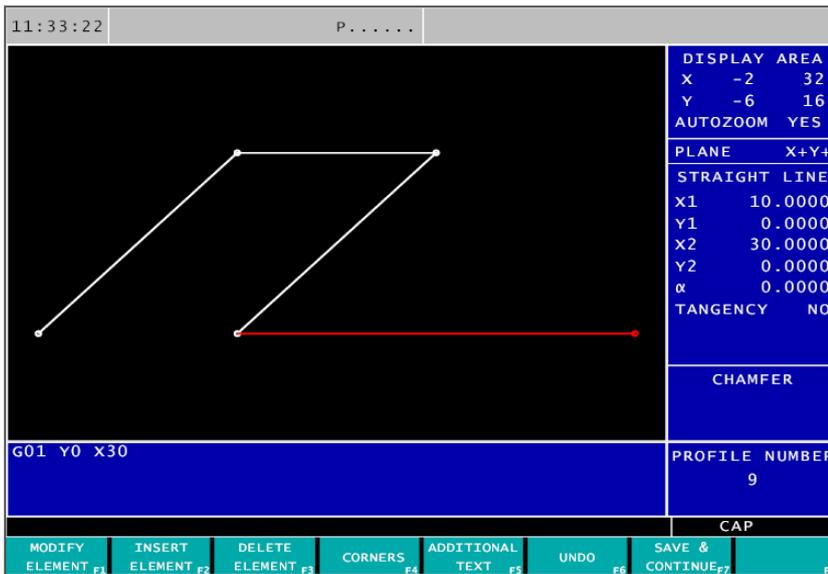
 key.

Note:

At the moment the functions "TANGENTIAL ENTRY" and "TANGENTIAL EXIT" cannot be programmed.



MODIFY



- Press the soft key



To modify any profile element.

- Select the required element. The WinNC will display the definition values of the selected element. Confirm with .
- Change the data input.
- Capture the input by means of .

Abort the input with .



To insert a new element (straight line or arc) at any profile position.

- Select the point or the element after which the new element should be inserted. The WinNC will display the definition values of the selected element. Confirm with .
- Create the element by means of the profile soft keys.
- Capture the input with .

Abort the input with .



To delete any profile element.

- Select the point or the element that should be deleted. The WinNC will display the definition values of the selected element. Confirm with .

CORNERS

F4

ADDITIONAL
TEXT

F5

see CORNERS

To insert any additional text at any profile position.

- Select the point or the element where a text should be added. The WinNC will display the ISO-

Code of the selected element. Confirm with 

- Enter the additional text. You may enter the functions F, S, T, D, M or program comments. These will be effective later in the program.

- Capture the input by means of 

Abort the input with 

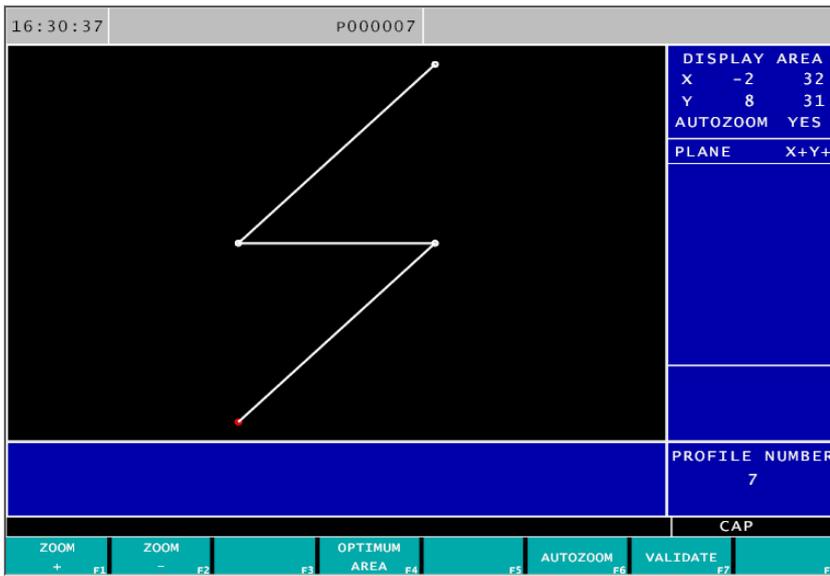
When selecting one of the options, a profile element will be marked red. If you wish to select another element, please press the cursor keys.

Confirm all value entries with To deselect the MODIFY mode, please press **Note:**

At the moment the function "ADDITIONAL TEXT" cannot be programmed.



DISPLAYAREA



- Press the soft key



This soft key enables to zoom the display area manually in



or out



With the soft key the display area is automatically zoomed in or out to window size.

The simulation window can be moved by means of the cursor



CONFIGURATION

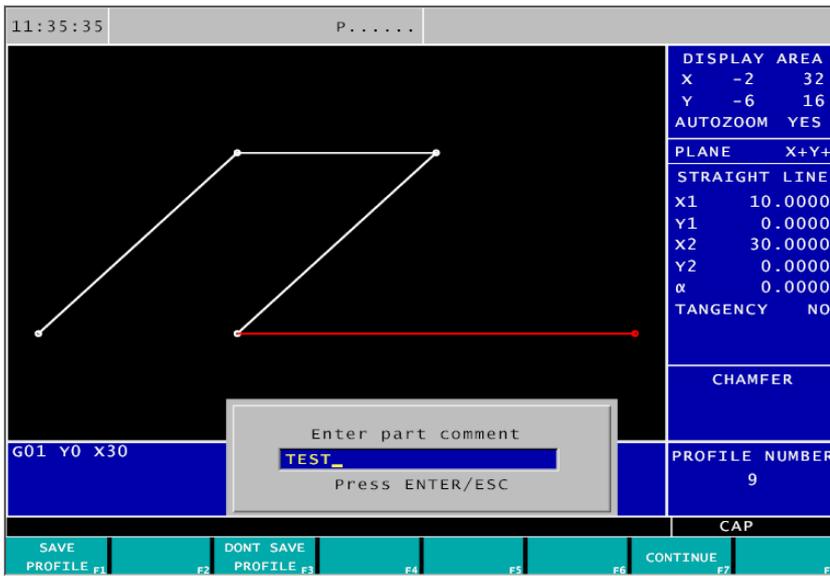
- Press the soft key



With these soft keys you can select the configuration area.

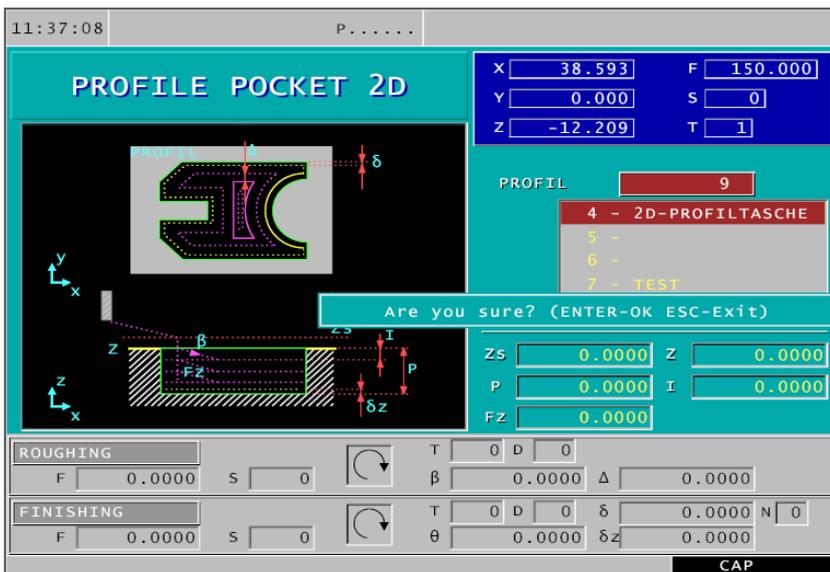


Saving the subprogram



- After having completed the profile definition, press the soft key **FINISH** $F5$.
 - Then press the soft key **SAVE PROFILE** $F1$.
- An input window will appear. Now you may give the subprogram a name.
- Confirm the input of the text with **ENTER**.

Deleting the subprogram



- Use the cursor keys to move the highlight to the subprogram you wish to delete.
 - Press the **CLEAR** key.
 - The WinNC opens the dialogue box: "Are you sure?".
- The subprogram will be deleted when you press **ENTER**.
- The dialogue can be aborted by means of **ESC**.

Graphic simulation

After having been entered, every cycle or working process can be tested by means of the graphic simulation.

The graphic simulation enables to detect geometric faults like e.g. wrong positions, contour violations, wrong tools, etc..

Technological errors like, e.g. a wrong spindle speed or wrong feed rates are not detected.

Simulating a working cycle

- Enter the required cycle completely.
- Press the  key.

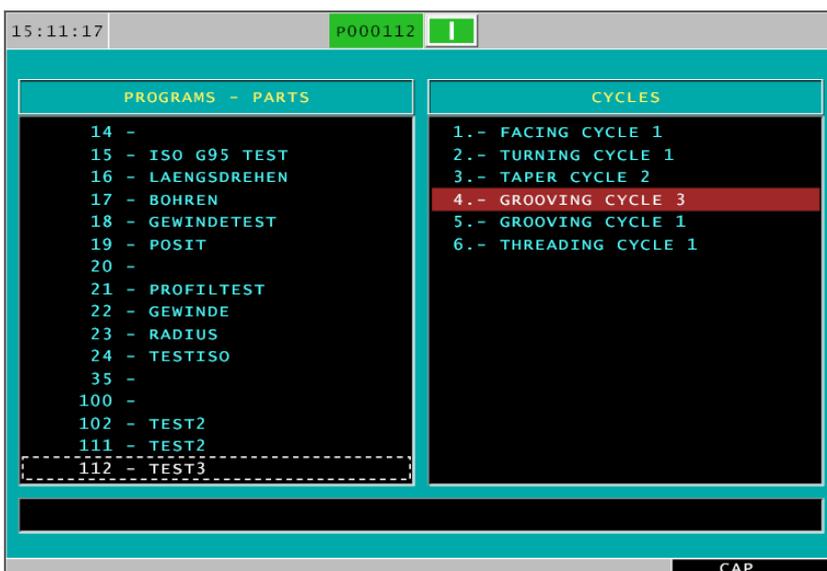


Simulating the whole workpiece program

- Press the  key to call the directory of the stored workpiece programs.
- Use the cursor to select the workpiece program, that should be simulated, from the left column (see illustration in the center left).
- Press the  key.

Simulating a part of a workpiece program

- Press the  key to call the directory of the stored workpiece programs.
- Use the cursor to select the program from the left column and the working cycle, from which the simulation of the workpiece program should be started, from the right column, (see illustration in the center left).
All cycles to follow are simulated as well.
- Press the  key.





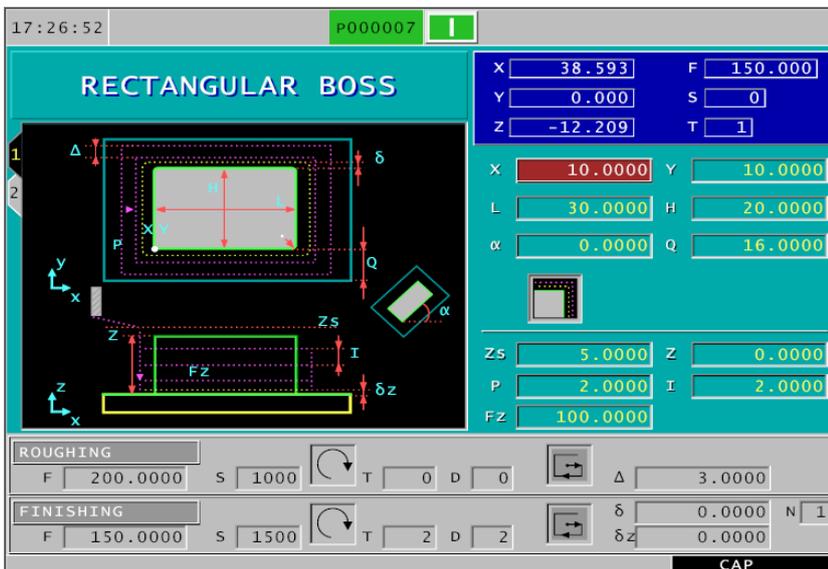
Simulating a stored working cycle

- Press the  key to call the directory of the stored workpiece programs.
- Use the cursor to select the workpiece program including the stored working cycle from the left column, and the working cycle that should be simulated from the right column (see picture at the top left).

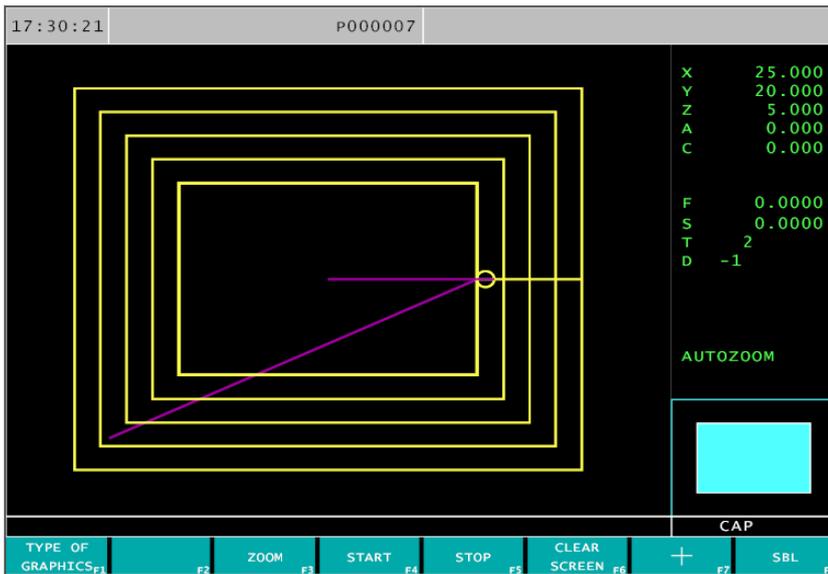
- Press the  key.

The stored cycle is displayed with all entered values (see picture in the center left).

- Press the  key.



Graphic illustration



When you press the  key, the WinNC shows the graphic illustration (see picture on the left). To quit the graphic illustration press the keys  or .

The WinNC displays the following soft keys:

-  Type of graphic illustration
-  Zoom
-  Starts the simulation
-  Stops the simulation
-  Clears the screen
-  Switch between the soft-key rows
-  Switch to the cycle menu, program menu or standard menu that has been selected before
-  Single block

- X-Y F1
- X-Z F2
- Y-Z F3
- 3D F5

Type of graphics

TYPE OF GRAPHICS F1

X-Y, X-Z, Y-Z

This type of graphic illustration uses colourful lines to display the tool movements in the selected planes (XY, XZ, YZ).

The screen only displays the machined side of the part.

3D

This type of illustration can only be selected with the 3D-graphic simulation **3D-View** (optional accessory).

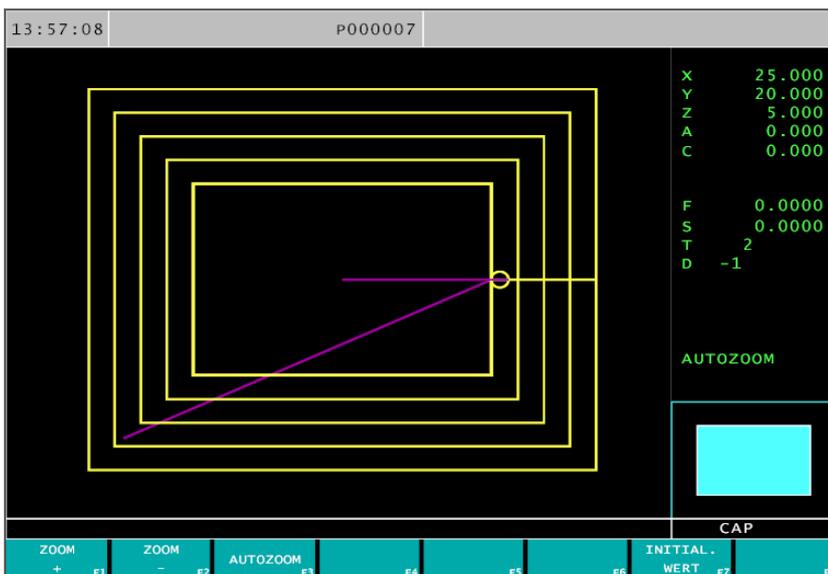
The simulation can be started by pressing the



key.

Zoom

ZOOM F3



With the soft key



the illustration can be manually

zoomed in, with



it is manually zoomed out.

With the soft key

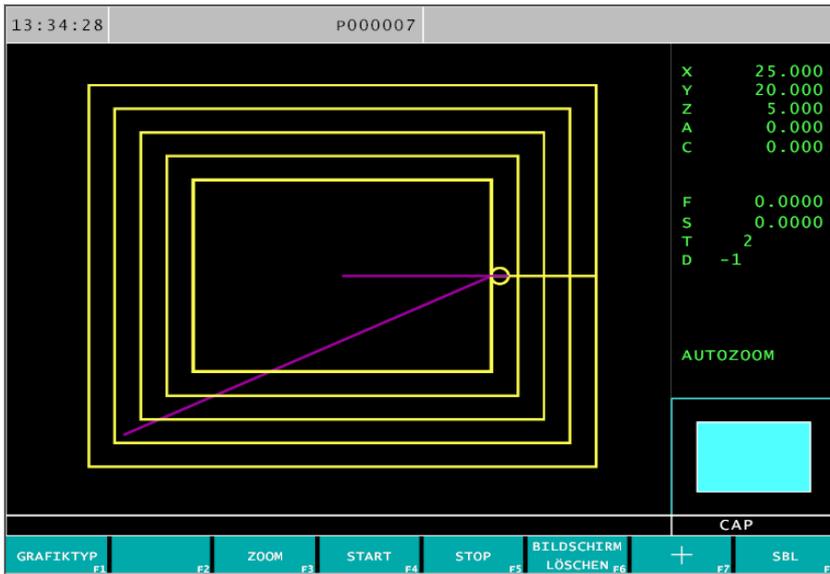


the illustration is automatically magnified or reduced to the size of the window.

The simulation window can be shifted by means of the direction

keys

Single block mode



By means of this soft key the simulation is interrupted after every block. To continue the simulation, press the soft key



The single block mode is activated, when the symbol



appears in the simulation window.

To deactivate the single block mode, press



again.

D: Programming

Notice:

This programming chapter describes all functions that can be done with WinNC. Depending on the machine that is operated with WinNC Fagor 8055 MC Milling not all of these functions may work.

Example:

The lathe Concept MILL 55 has no position controlled main spindle, therefore no spindle position can be programmed.



Overview M-commands

Command	Description
M0	Programmed stop
M1	Optional stop (program stop only with OPT. STOP)
M2	End of program
M3	Spindle ON clockwise
M4	Spindle ON counterclockwise
M5	Spindle OFF
M6	Tool change
M7	Minimum lubrication ON
M8	Coolant ON
M9	Coolant OFF
M10	Spindle brake ON
M11	Spindle brake OFF
M25	Clamping device OPEN
M26	Clamping device CLOSE
M27	Swivel dividing head
M30	End of main program
M70	Spindle positioning
M71	Blowing out ON
M72	Blowing out OFF

Overview Cycles



Calling the cycle selection



Boring



Reaming



Tapping



Drilling



Profiling



Surface milling



Profile pocket



Bosses

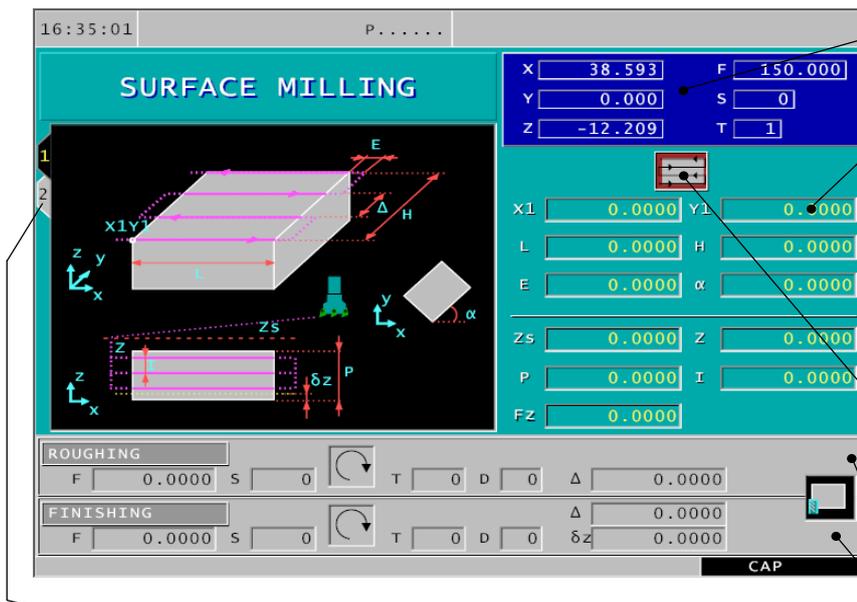


Pockets



Positioning

Input of cycle data



Actual values

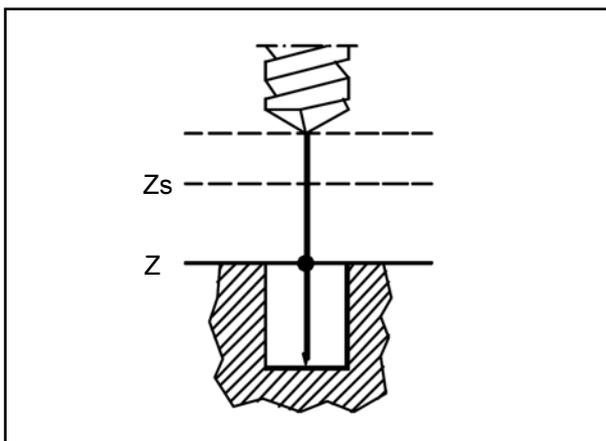
Input boxes: the values are captured by pressing the  key.

Use the direction keys , , ,  to select the boxes.

The  key enables you to switch between the fields (e.g. surface milling types).

Fields for technological data

The main cycle window shows the available planes. The currently active cycle is highlighted. Press the  key to switch from one plane to another within the cycle group.

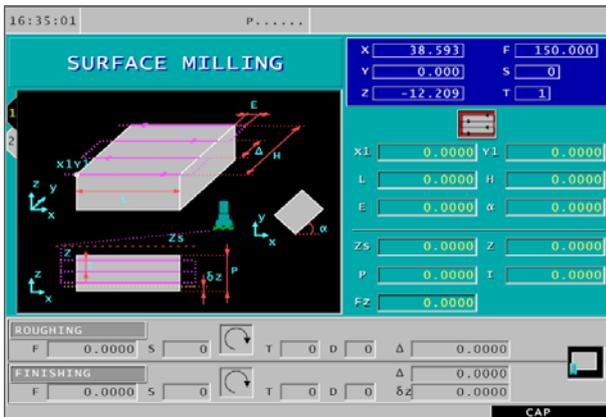


Safety distance

In order to avoid collisions with the workpiece during machining cycles, you can define an approaching height that is traversed before the cycle starting point.

The safety plane Z_s indicates the approaching height in relation to the cycle starting point.

Technological data input for milling cycles



- **Spindle speed / cutting speed**

The spindle speed /cutting speed is entered under the parameter S.

Place the cursor on the field S, key in the new value

and confirm with .

For the roughing and finishing pass of processing cycles (e.g. Surface Milling) you can programm various spindle speeds.

- **Direction of rotation**

The direction of rotation is displayed by the field

, respectively by the field .

Place the cursor on the field that shows the direction of rotation and adjust the direction by means of



- **Feed rate**

The working feed rate is entered under the parameter F.

Place the cursor on the field F, key in the new value

and confirm with .

For the roughing and finishing pass of processing cycles (e.g. Surface Milling) you can program various feed rates.

- **Tool**

Enter the respective tool and the tool correction data under T and D (several tool correction data are possible per tool).

For the roughing and finishing pass of processing cycles (e.g. Surface Milling) you can program various tools (see chapter E tool programming).

• **Cutting depth**

For roughing the lateral roughing infeed has to be defined under Δ . For finishing the lateral finishing infeed has to be defined under Δ .

The finishing infeed, that should remain during roughing, is indicated by means of δ .

The number of finishing passes in Z is indicated by means of **N**.

The finishing infeed in Z, that should remain during roughing, is indicated by means of δz .

• **Machining direction**

Down-cut milling , respectively

up-cut milling 

Place the cursor on the soft key for the machining direction and change it by means of .

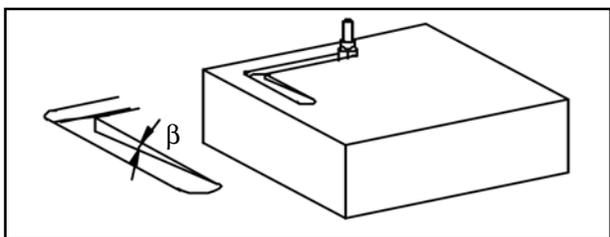
Note:
The angles β and Θ enable to mill a pocket by means of cutting tools. If the angles β and Θ are not defined, it is possible that the non-cutting tool surfaces will contact the blank workpiece. This could damage the non-cutting tool surfaces.



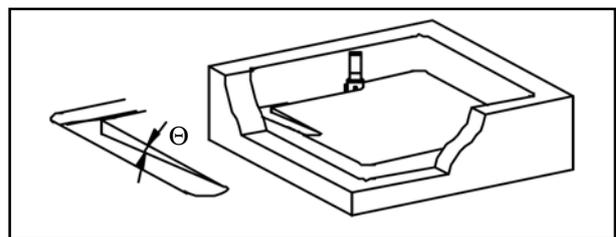
Angle of lateral cutting depth (β , Θ)

In the rectangular pocket the impression is machined from the center of the pocket. This is done by following the first machining path, which is repeated as often as necessary until it ends in the center of the pocket.

In the circular pocket the impression is machined from the center of the pocket following a helical path. The radius of the path is the same as the tool radius. The machining direction remains the same. The impression always ends in the center of the pocket.



Angle of the lateral cutting depth during roughing (β)



Angle of the lateral cutting depth during finishing (Θ)

Roughing, finishing, complete machining

The cycles

- profile 1, profiling
- milling, groove
- profile pocket 2D
- rectangular boss, circular boss
- rectangular pocket milling, circular pockets 1 and 2

can be programmed as roughing cycle, finishing cycle or complete machining cycle (roughing and finishing).

For roughing and finishing you can select various feed rates, spindle speeds and tools.

In case different tools are selected for roughing and finishing, the toolholder automatically approaches a tool changing point.

Complete machining

Define a tool for roughing and finishing. Both cycles are carried out in sequence with the corresponding settings and tools.

Roughing

Select the tool T0 as finishing tool, which means that the finishing cycle will not be executed. A defined finishing clearance is taken into consideration during roughing.

Finishing

Select the tool T0 as roughing tool, which means that the roughing cycle will not be executed.

All roughing and finishing values must be defined anew in every cycle display.

Cycles

Machining cycles

The following cycles can be directly called by means of a key:



Calling the cycle selection



Boring



Reaming



Tapping



Drilling



Profiling



Surface milling



Profile pocket



Bosses



Pockets



Positioning

Multiple positionings

The cycles Boring 1, Boring 2, Reaming, Tapping, Drilling 1, Drilling 2, Drilling 3 and Centering can be programmed by means of multiple positionings:



from point to point



along a line



along a circular arc



along a rectangle



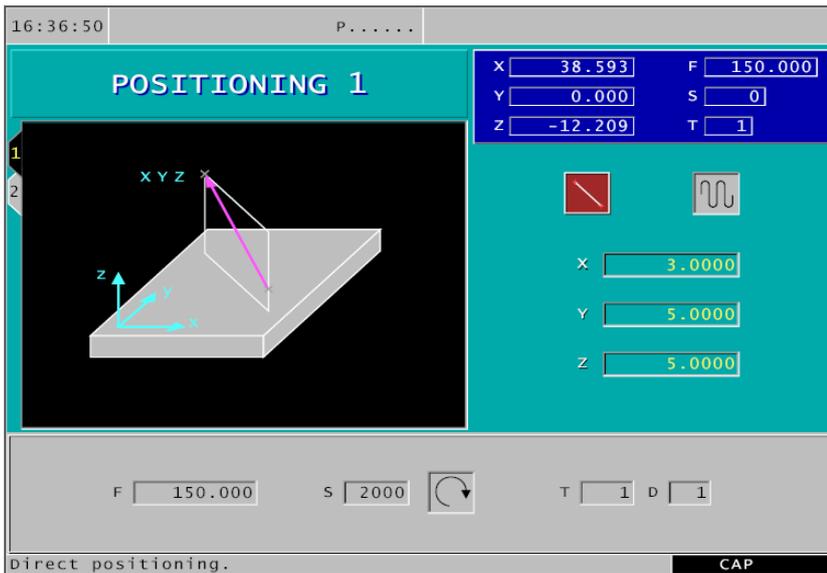
along a grid pattern

After having defined the data in a cycle, press the respective key for multiple positioning and define further data.

You can leaf through the individual variants of cycles which are available in several variants (e.g. Positioning

1, Positioning 2) by means of the  key.

POSITIONING 1



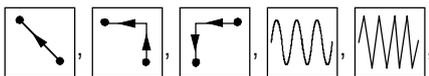
With Positioning Cycle 1 you can position the tool in a linear movement.

Application:

- to position the tool before the actual machining starts
- to traverse the tool between two cycles

Call the positioning cycles with the  key, then select

Positioning Cycles 1-2 with .



Selection with ,
confirmation with .



The tool traverses from its current position to the target position in a linear movement .



From its current position the tool first traverses in Z, then in XY to the target position.



From its current position the tool first traverses in XY, then in Z to the target position.



The tool moves at rapid traverse.



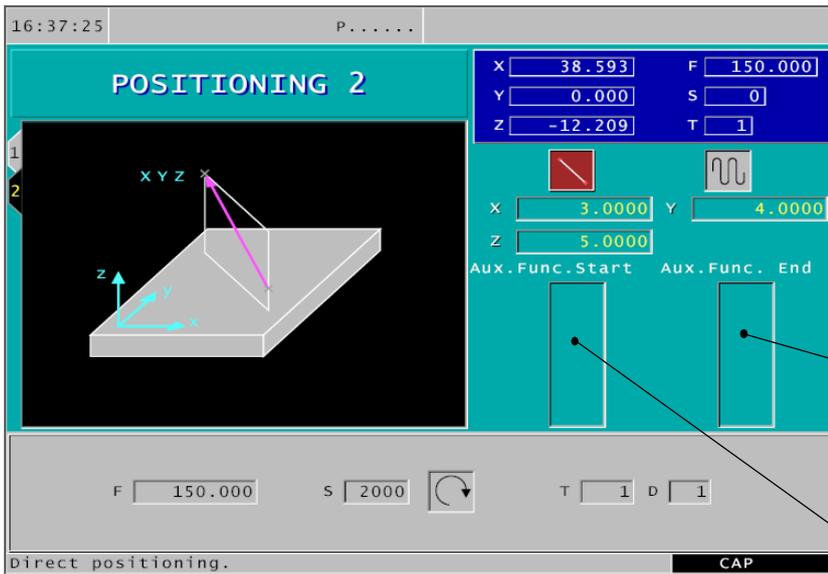
The tool traverses at a programmed feed rate F.

Coordinates

The target position is defined by X, Y and Z.

POSITIONING 2

For input refer to Positioning Cycle 1.

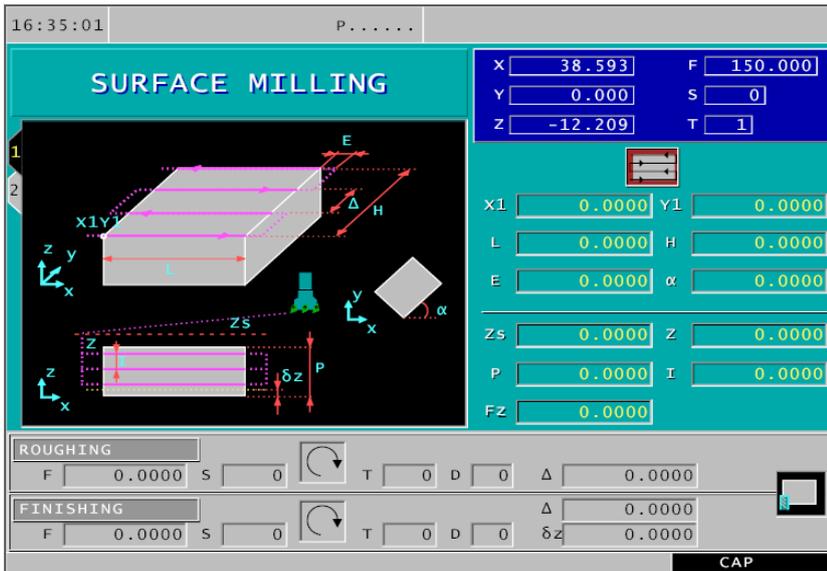


M-commands (auxiliary functions, switch functions), that are executed before and after the cycle, can be entered additionally in the Positioning Cycle 2 (e.g. coolant on / off). (see overview M-commands in chapter D)

M-functions that are programmed in the column "M after", refer to the functioning of the control directly AFTER the positioning process.

M-functions that are programmed in the column "M before", refer to the functioning of the control directly BEFORE the positioning process.

SURFACE MILLING

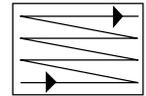
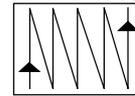
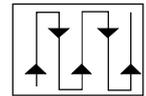
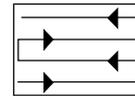


Call the cycle by pressing the



key.

Select Surface Milling with .



Selection of surface milling types



with

Note:

The exceeding E must be at least the same size as the tool radius R.

All value inputs have to be confirmed with .



Coordinates (X1, Y1)

Coordinates of the entering point

Length of the surface milling area in X (L)

Length of the surface milling area in Y (H)

Exceeding (E)

Angle in relation to the X axis (α)

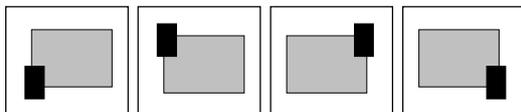
Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)



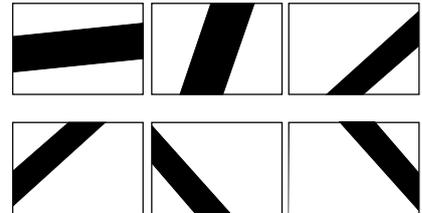
Selects the corner point, where the cutter starts working.

GROOVING



Call the cycle by pressing the  key.

Select Grooving with .



Selection of grooving types with .

Note:
The exceeding E must be at least the same size as the tool radius R.
All value inputs have to be confirmed with .

Coordinates (X, Y)

Coordinates of the entering point

Length of the groove (L)

Width of the groove (H)

Exceeding (E)

Angle of the groove towards the X axis (α)

Safety axis position in Z (Zs)

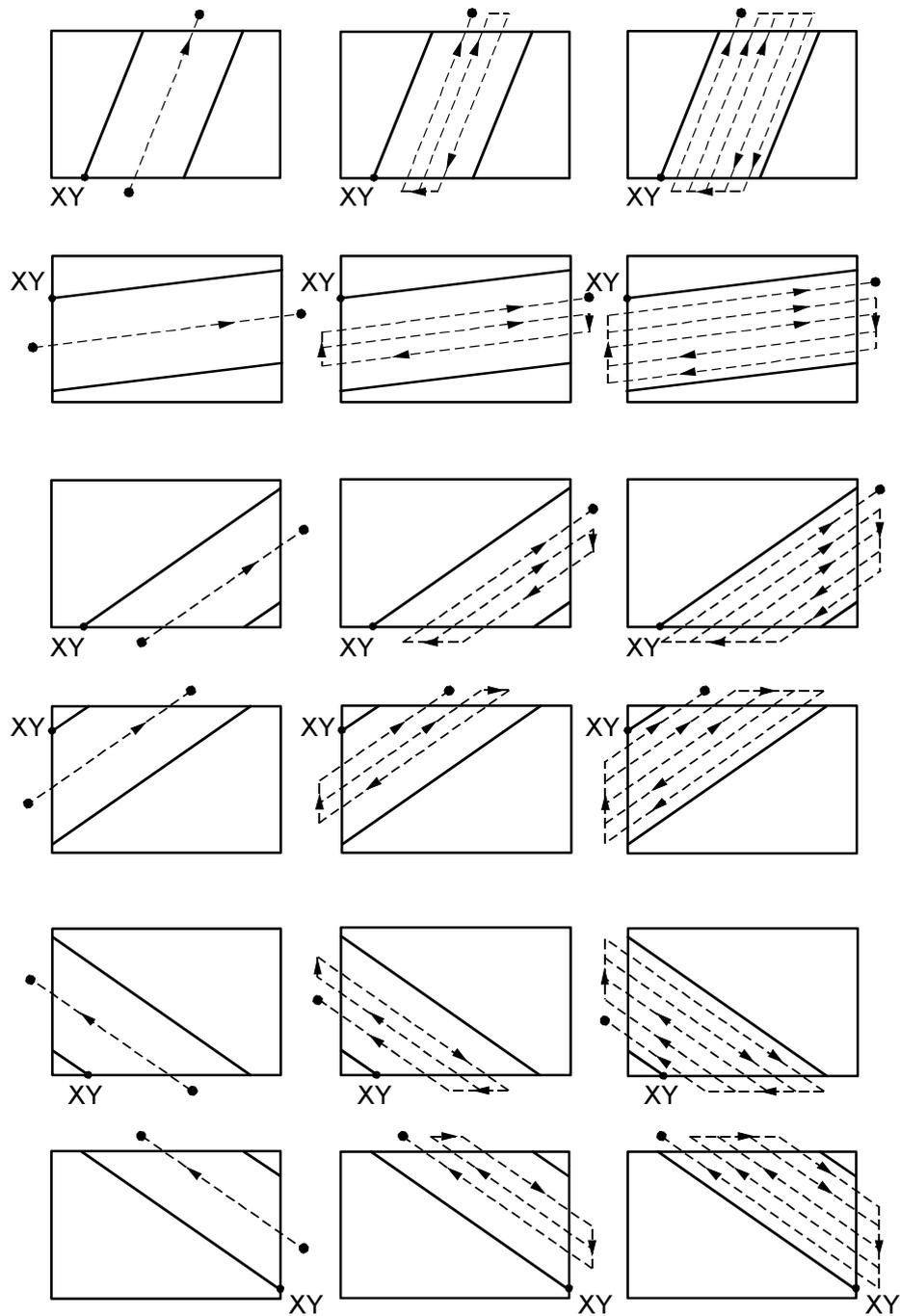
Coordinate value of the workpiece in Z (Z)

Depth (P)

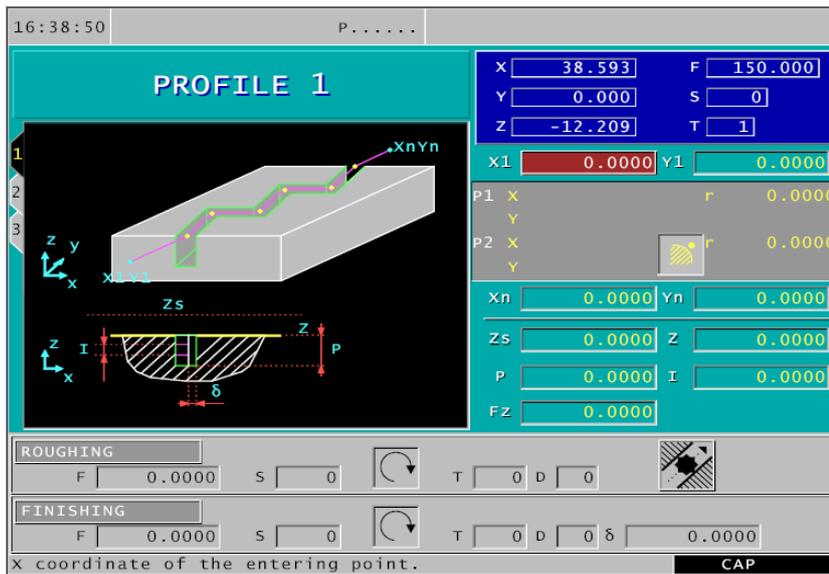
Cutting depth in Z (I)

Feed rate in Z (Fz)

Machining of the different grooving variants in clockwise direction.



PROFILE 1



Profile 1 enables you to create any groove contour. However, this contour must only consist of 12 contour points at maximum.

Call the cycle by pressing the  key.

Select Profile 1 with .

Use the cursor keys or the enter key to move within the cycle program.

Tool radius compensation:



without compensation,



with tool radius compensation on the right



with tool radius compensation on the left

Selection with .

Coordinates (X1, Y1)

Coordinates of the entering point

Coordinates of the points P

Coordinates in X and Y

For all position points you may choose from the following options:



sharp-edged



rounded (with radius definition)



chamfered at 45° (with size definition C)

Coordinates (Xn, Yn)

Coordinates of moving out

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

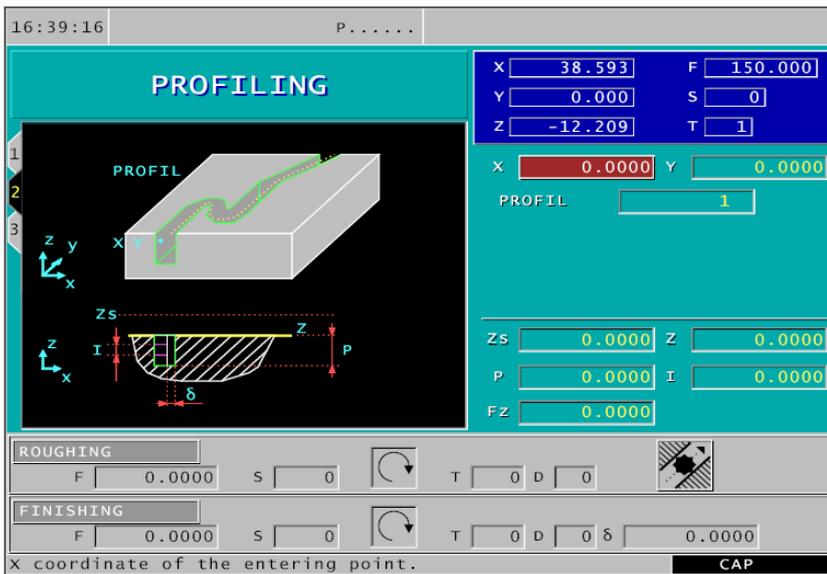
After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

The last contour point has to be programmed twice, if less than 12 contour points are defined.

PROFILING



Profiling enables you to create any groove contour with any number of points. The contour is saved in a subprogram.

Call the cycle by pressing the  key.

Select Profiling with .

Use the cursor keys or the enter key to move within the cycle program.

Tool radius compensation:



without compensation,



with tool radius compensation on the right



with tool radius compensation on the left

Selection with .

Coordinates (X1, Y1)

Coordinates of the entering point

Program number for the contour P

Subprogram, where the contour that should be edited, is described (see chapter C profile editor).

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

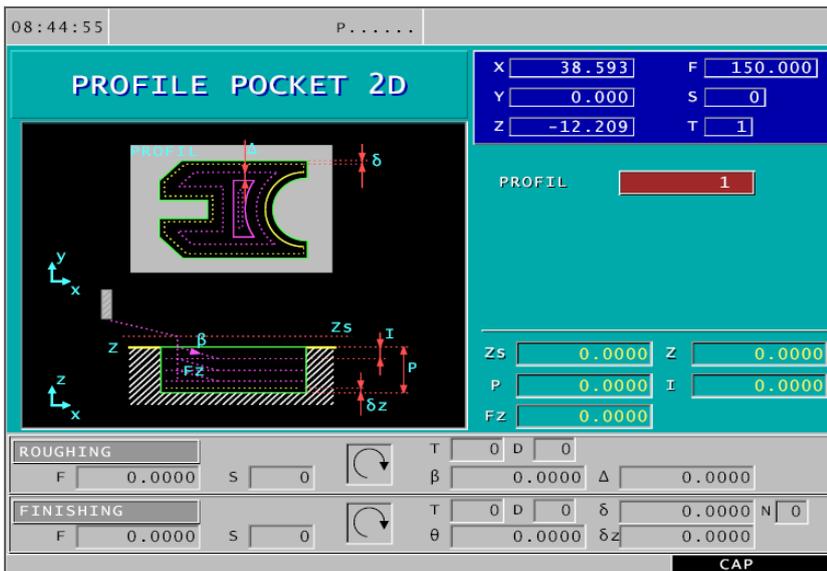
After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

For profile milling the subprogram P1 is saved under the program number **998001**. It is possible to program 1000 profile milling-subprograms.

PROFILE POCKET 2D



Profile pocket 2D enables you to create any pocket contour with any number of points. The contour is saved in a subprogram.

Call the cycle by pressing the



key.

Use the cursor keys or the enter key to move within the cycle program.

Tool radius compensation:



without compensation,

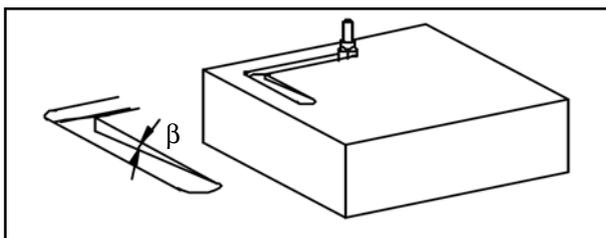


with tool radius compensation on the right

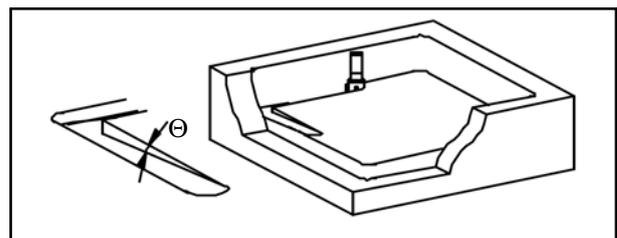


with tool radius compensation on the left

Selection with .



Angle of lateral infeed during roughing (β)



Angle of lateral infeed during finishing (θ)

Note:

All value inputs have to be confirmed with .

For profiling the subprogram P1 is saved under the program number **998001**. It is possible to program 1000 profiling-subprograms.

Program number for the contour P

Subprogram, where the contour that should be edited, is described (see chapter C profile editor).

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

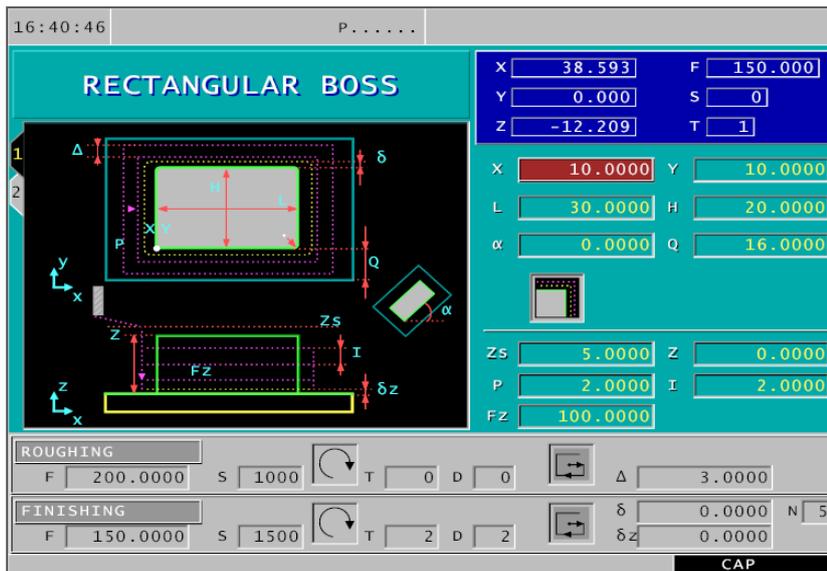
Angle of the lateral infeed (β , θ)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

RECTANGULAR BOSS



Call the the cycle by pressing the



key.

Select Rectangular Boss with



Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the starting point

Length of the boss in X (L)

Length of the boss in Y (H)

Length of the boss towards the X axis (α)

Finishing offset (Q)

Corner rounding

For all corner points of the boss you may choose from the following options:



sharp-edged



rounded (with radius definition)



chamfered at 45° (with size definition C)

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the boss (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

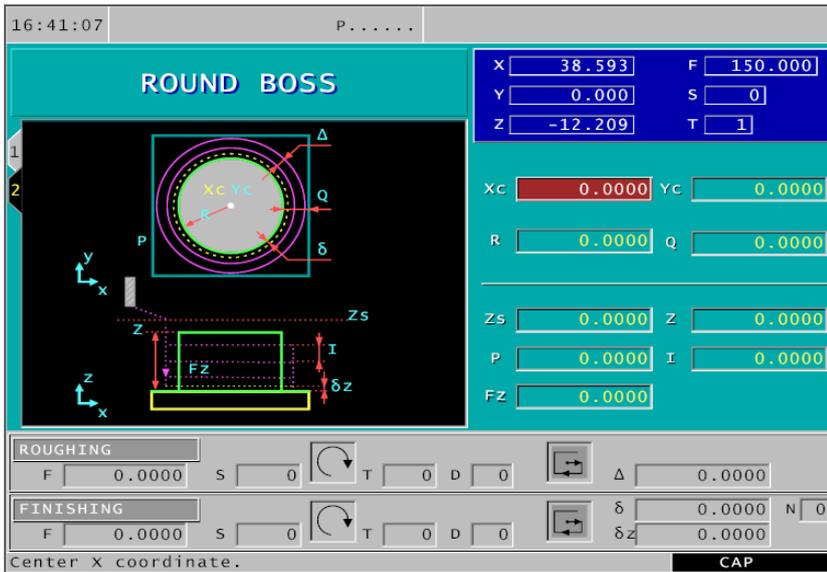
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

ROUND BOSS



Call the cycle by pressing the  key.

Select Round Boss with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (Xc, Yc)

Coordinates of the center of the circle

Radius of the boss (R)

Finishing offset (Q)

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the boss (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

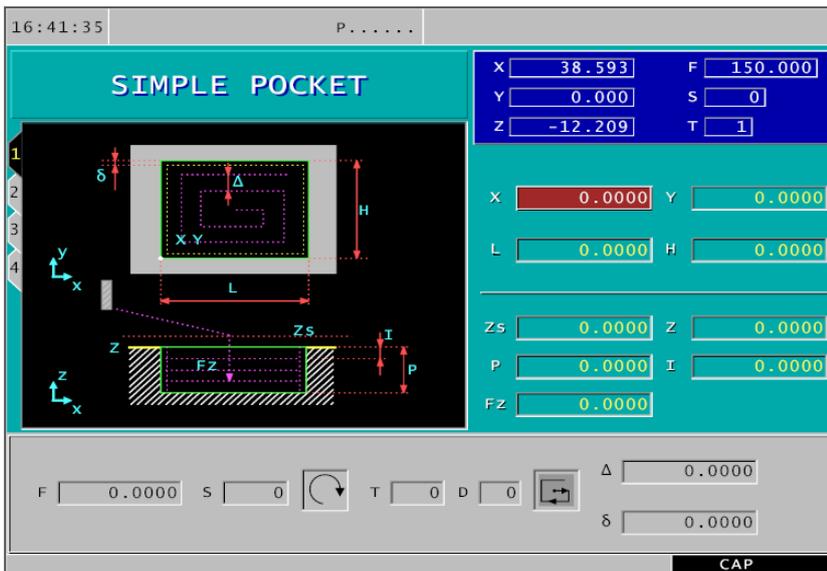
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

SIMPLE POCKET



Call the cycle by pressing the  key.

Select Simple Pocket with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the starting point

Length of pocket in X (L)

Length of pocket in Y (H)

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the pocket (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

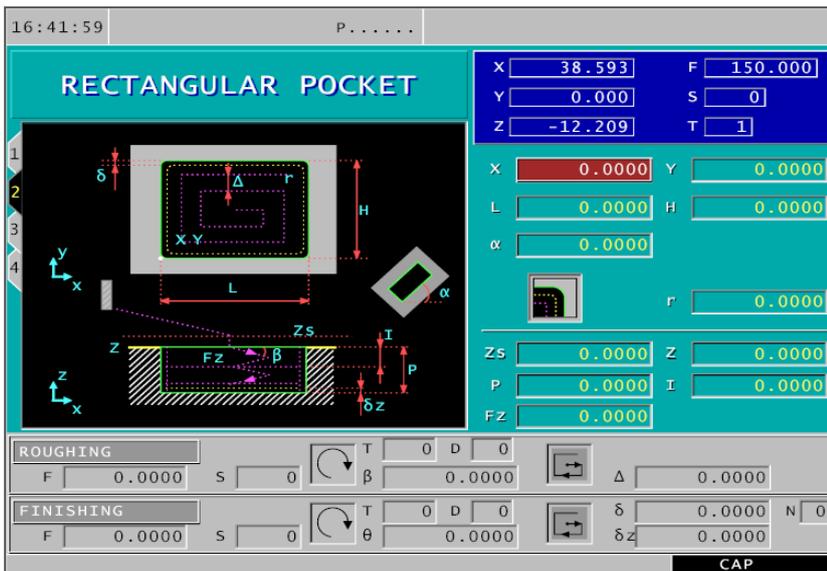
Note:

All value inputs have to be confirmed with .

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing function).

After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

RECTANGULAR POCKET



Call the cycle by pressing the  key.

Select Rectangular Pocket Milling mit .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the starting point

Length of the pocket in X (L)

Length of the pocket in Y (H)

Angle of the pocket towards the X axis (α)

Corner rounding

For all corner points of the pocket you may choose from the following options:



sharp-edged



rounded (with radius definition)



chamfered at 45° (with size definition C)

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the boss (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

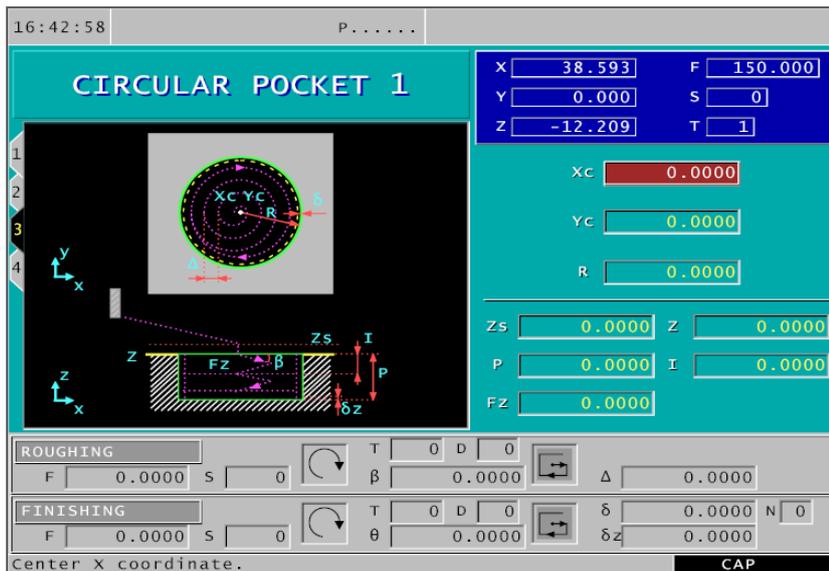
After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

CIRCULAR POCKET 1



Call the cycle by pressing the  key.

Select Circular Pockets 1 with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (Xc, Yc)

Coordinates of the center of the circle

Radius of the pocket (R)

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the pocket (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

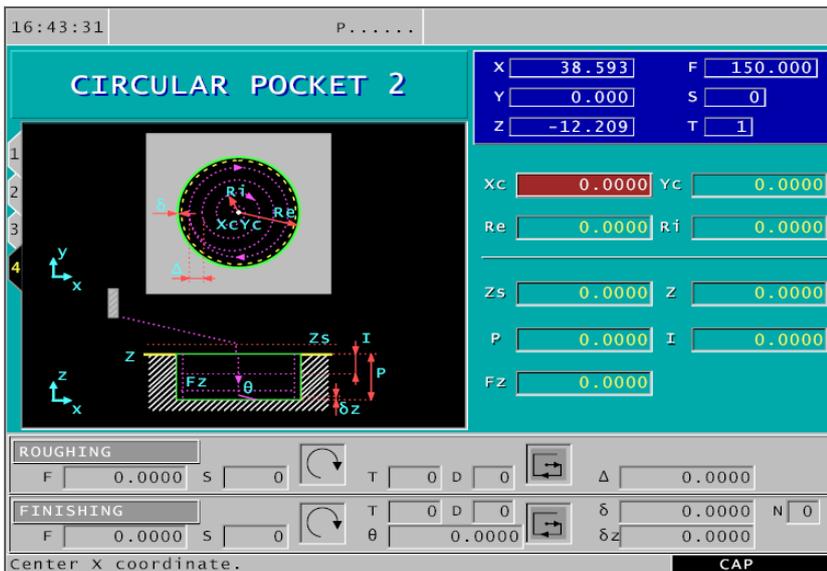
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

CIRCULAR POCKET 2



Call the cycle by pressing the



key.

Select Circular pockets 2 with



Use the cursor keys or the enter key to move within the cycle program.

Coordinates (Xc, Yc)

Coordinates of the center of the circle

Outer radius (Re)

Inner radius (Ri)

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the pocket (P)

Cutting depth in Z (I)

Feed rate in Z (Fz)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation, data for roughing and finishing function).

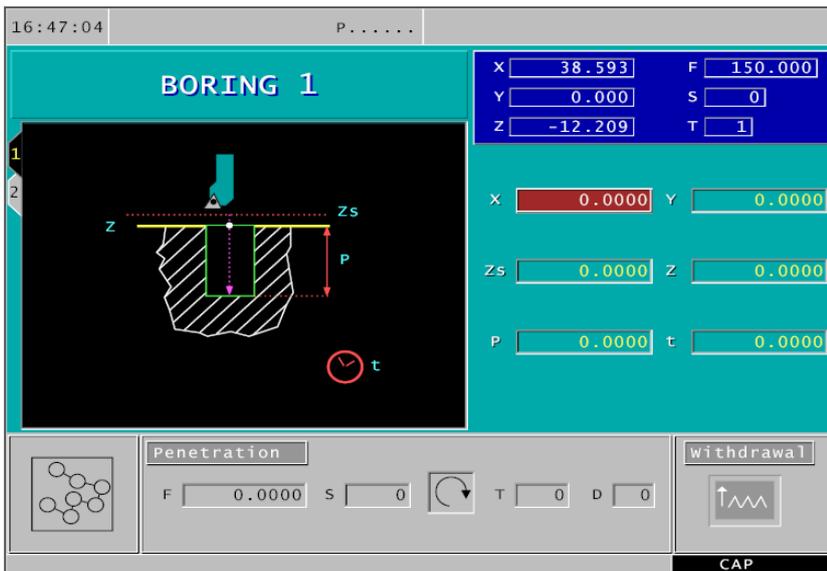
After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with



BORING 1



Call the cycle by pressing the



key.

Select Boring 1 with



Use the cursor keys or the enter key to move within the cycle program.

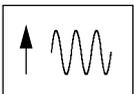
The tool remains at the bottom of the bore until the dwell time has elapsed.

You can choose between 2 withdrawal movements:



move back to the safety axis position at

the programmed feed rate and spindle speed.



move back to the safety axis position at

maximum feed rate and with idle spindle (rapid traverse).

Selection with .

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the bore (P)

Dwell time at the bottom in seconds (t)

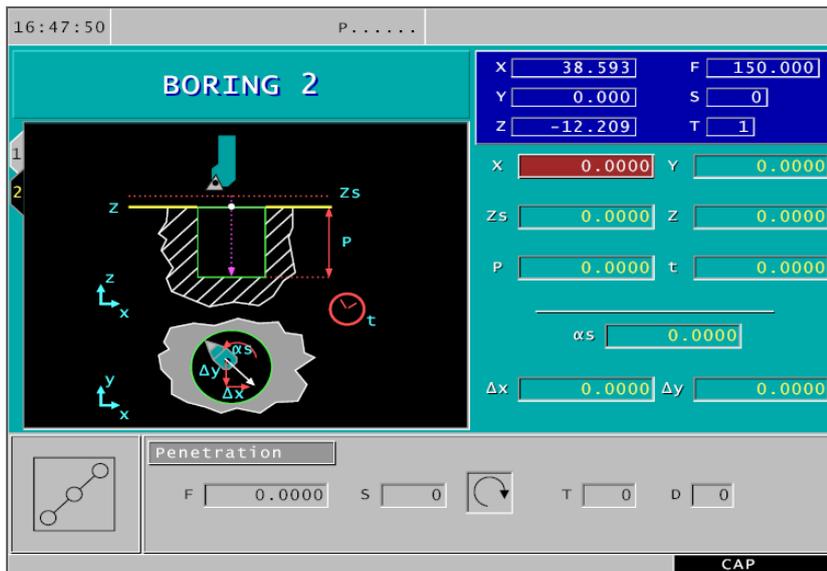
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

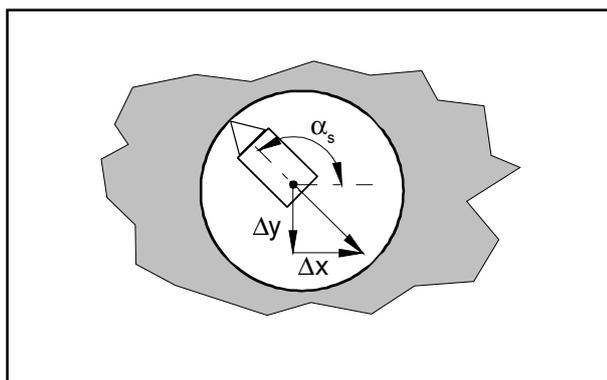
BORING 2



Call the cycle by pressing the  key.

Select Boring 2 with .

Use the cursor keys or the enter key to move within the cycle program.



Tool alignment before the retraction movement ($\alpha_s, \Delta x, \Delta y$):

The surface of the bore will be damaged when an idle tool withdraws to the safety axis position. When using drilling tools of a diameter being smaller than the required drilling diameter, damage can be avoided, when the tool is positioned towards the boring center prior to moving out. The positioning is rendered possible by the angle α_s towards the X axis and the incremental traverse rates Δx und Δy .



Note:

All value inputs have to be confirmed with .

Please mind the correct arithmetic sign for the incremental traverse rates. Negative arithmetic sign means movement towards the negative coordinate axis.

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Z_s)

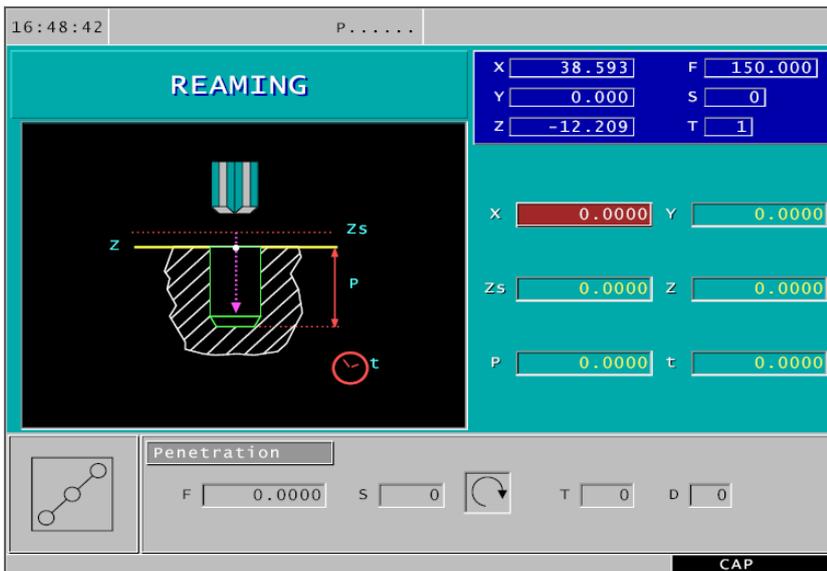
Coordinate value of the workpiece in Z (Z)

Depth of the bore (P)

Dwell time at the bottom in seconds (t)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

REAMING



Select Reaming with the  key.

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the bore (P)

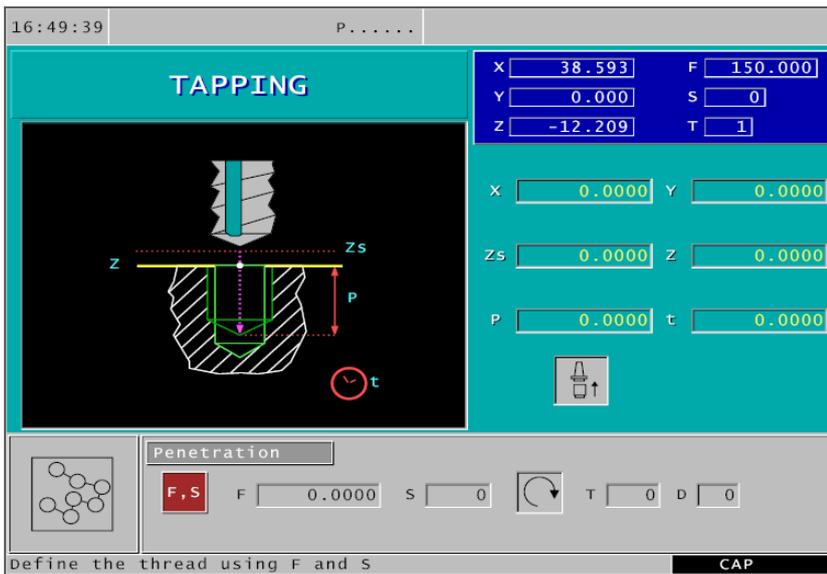
Dwell time at the bottom in seconds (t)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

TAPPING



Select Tapping with the  key.

Use the cursor keys or the enter key to move within the cycle program.

Length compensation

Tapping with length compensation

Tapping without length compensation

Selection with .

Thread definition

Define the thread by means of feed rate (F) and spindle speed (S)

 Define the thread by means of thread pitch (p) and spindle speed (S)

Selection with .

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the thread bore (P)

Dwell time at the bottom in seconds (t)

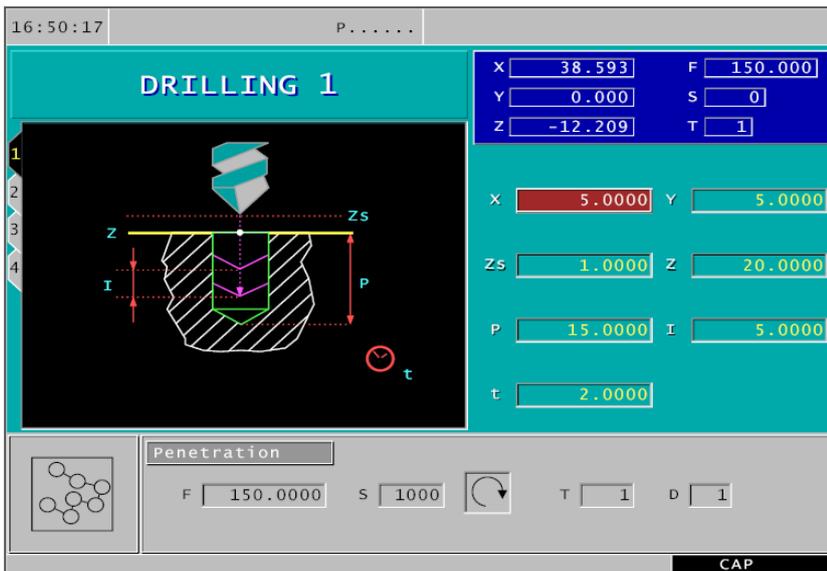
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

Tapping only with length compensation holder.

All value inputs have to be confirmed with .

DRILLING 1



Call the cycle by pressing the  key.

Select Drilling 1 with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the bore (P)

Drilling steps (I)

Plunging depth that is drilled in one step.

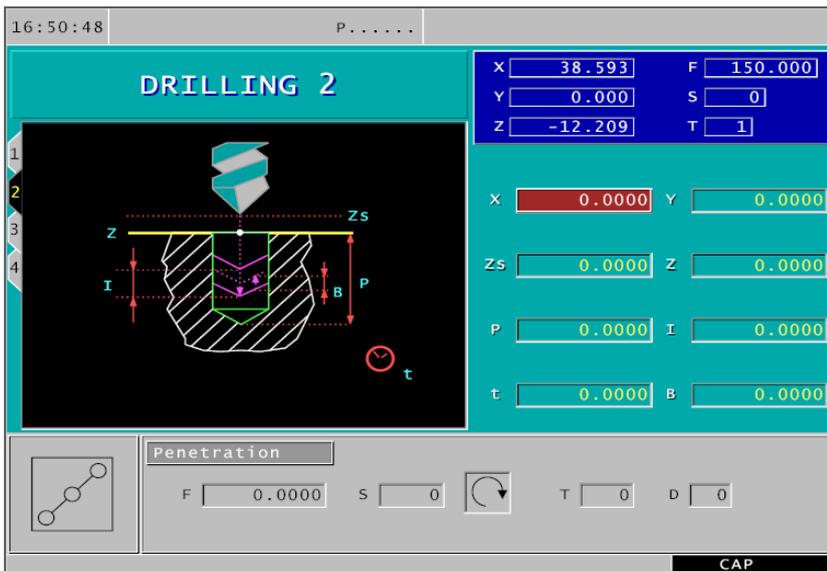
Dwell time at the bottom in seconds (t)

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

DRILLING 2



Call the cycle by pressing the  key.

Select Drilling 2 with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Depth of the bore (P)

Drilling steps (I)

Plunging depth that is drilled in one step.

Dwell time at the bottom in seconds (t)

Retracting distance for chip breaking (B)

Distance by which the tool is retracted for chip breaking.

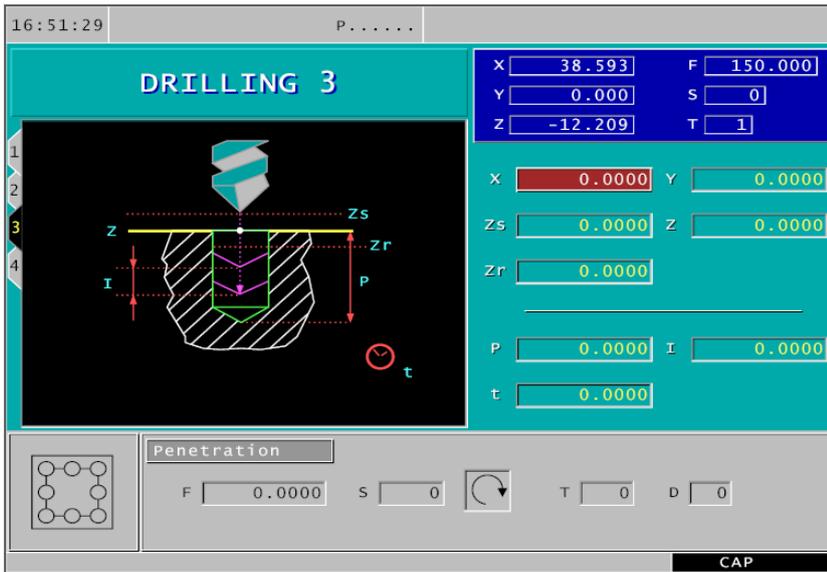
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

DRILLING 3



Call the cycle by pressing the  key.

Select Drilling 3 with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Coordinate value of the retraction movement for chip breaking (Zr)

Depth of the bore (P)

Drilling steps (I)

Plunging depth that is drilled in one step.

Dwell time at the bottom in seconds (t)

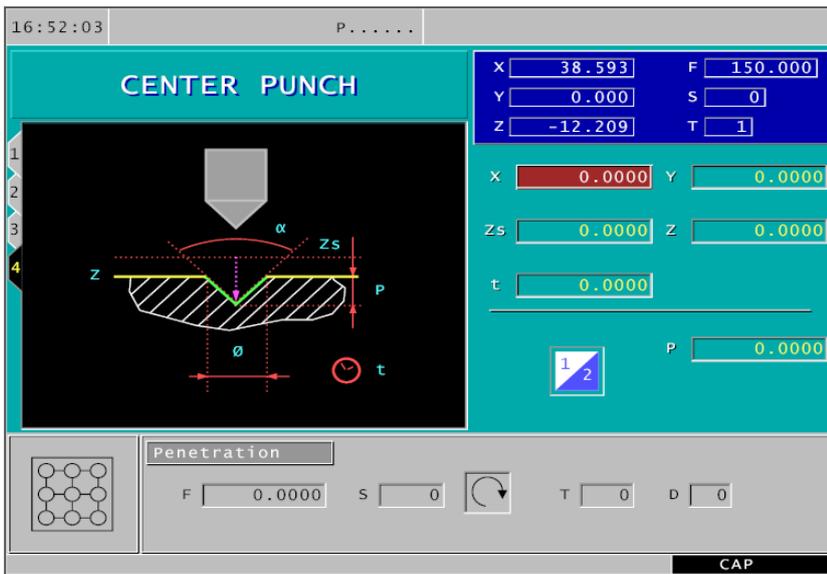
In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

CENTER PUNCH



Call the cycle by pressing the  key.

Select Center Punch with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the center of the bore

Safety axis position in Z (Zs)

Coordinate value of the workpiece in Z (Z)

Dwell time at the bottom in seconds (t)

Types of centering:



Definition by machining depth (P)



Definition by angle (α) and diameter (\varnothing) of center punch

Selection with .

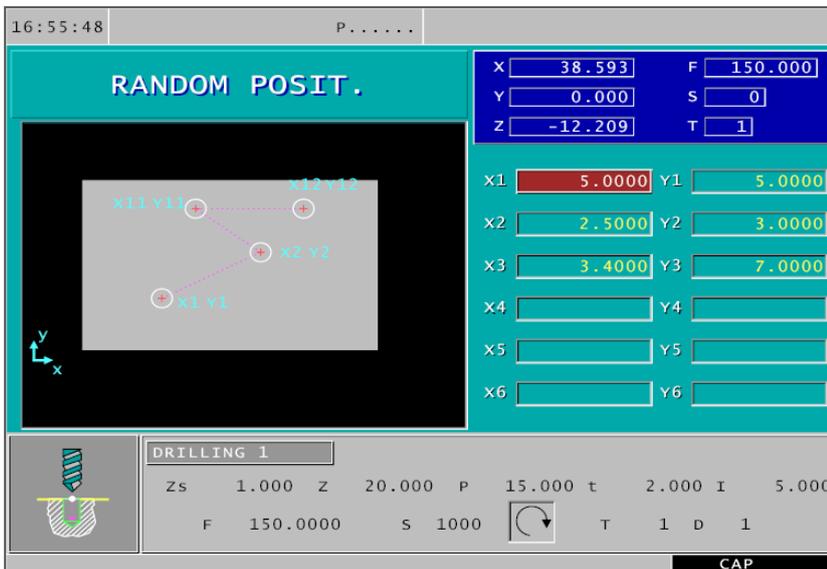
Note:

All value inputs have to be confirmed with .

In addition please enter all necessary technological data (feed rate, spindle speed, direction of rotation). After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Multiple positioning

RANDOM POSIT.



Random positioning enables you to define multiple positionings from point to point. A maximum of 12 position points can be defined.

Selection with the  key.

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X, Y)

Coordinates of the position points

The technological data (feed rate, spindle speed, direction of rotation) that have been captured by the cycle, can be checked in the bottom screen window.

After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

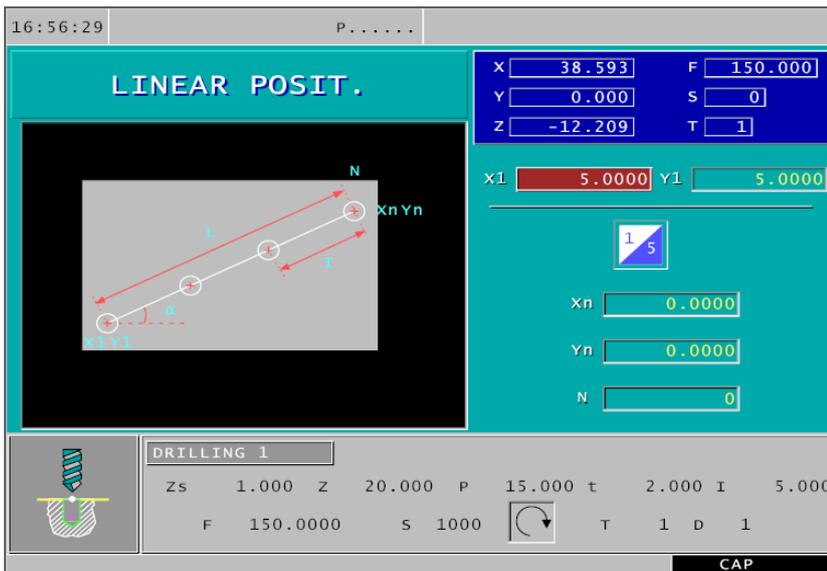
All value inputs have to be confirmed with .

All position points that are not required have to be deactivated with  and  (see illustration).

All position points are preset with the coordinate values $X_n=0.0000$ and $Y_n=0.0000$ at the factory.



LINEAR POSIT.



Linear positioning enables you to define multiple positionings of several points along a line.

Selection with the  key.

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X1, Y1)

Coordinates of the starting point

Type of point definition:



Coordinates (Xn, Yn)

Coordinates of the end point

Number of positionings (N)



Angle distance of the starting point in relation to the X axis (α)

Length of the line (L)

Number of positionings (N)



Angle distance of the starting point in relation to the X axis (α)

Number of positionings (N)

Distance between the positionings (I)



Coordinates (Xn, Yn)

Coordinates of the end point

Distance between the positionings (I)



Angle distance of the starting point in relation to the X axis (α)

Length of the line (L)

Distance between the positionings (I)

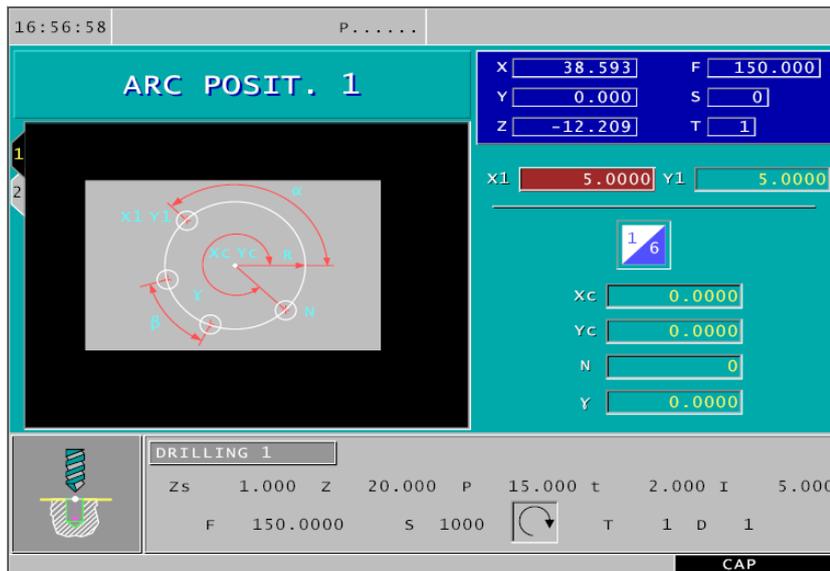
Selection with .

The technological data (feed rate, spindle speed, direction of rotation) that have been captured by the cycle, can be checked in the bottom screen window. After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

ARC POSIT. 1



Arc positioning 1 enables you to define multiple positionings of several points along a circular arc.

Call the cycle by pressing the



key.

Select Arc posit. 1 with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X1, Y1)

Coordinates of the starting point

Type of point definition:



Coordinates (Xc, Yc)

Coordinates of the center of the circle

Number of positionings (N)

Angle distance of the end point in relation to the X axis (γ)



Coordinates (Xc, Yc)

Coordinates of the center of the circle

Number of positionings (N)

Angle between the positionings (β)



Arc radius (R)

Angle distance of the starting point in relation to the X axis (α)

Number of positionings (N)

Angle distance of the end point in relation to the X axis (γ)



Arc radius (R)

Angle distance of the starting point in relation to the X axis (α)

Number of positionings (N)

Angle between the positionings (β)



Coordinates (Xc, Yc)

Coordinates of the center of the circle

Angle distance of the end point in relation to the X axis (γ)

Angle between the positionings (β)



Arc radius (R)

Angle distance of the starting point in relation to the X axis (α)

Angle distance of the end point in relation to the X axis (γ)

Angle between the positionings (β)

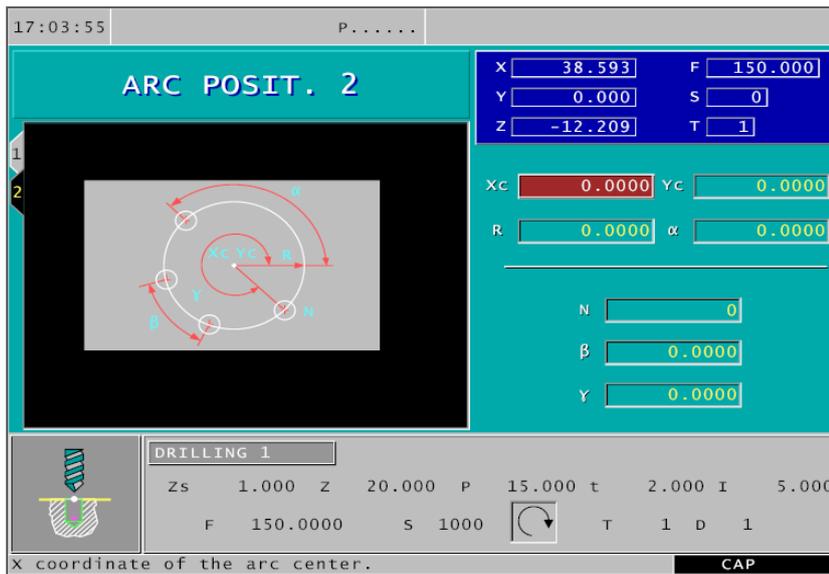
Selection with .

Note:

All value inputs have to be confirmed with .

The technological data (feed rate, spindle speed, direction of rotation) that have been captured by the cycle, can be checked in the bottom screen window. After having completed the input, the cycle can be checked by means of the graphic function  (see graphic simulation).

ARC POSIT. 2



Arc positioning 2 enables you to define multiple positionings of several points along a circular arc.

Call the cycle by pressing the



key.

Select Arc posit. 2 with .

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (Xc, Yc)

Coordinates of the center of the circle

Arc radius (R)

Angle distance of the starting point in relation to the X axis (α)

You have to make at least 2 of the following definitions:

Number of positionings (N)

Angle between the positionings (β)

Angle distance of the end point in relation to the X axis (γ)

Note:

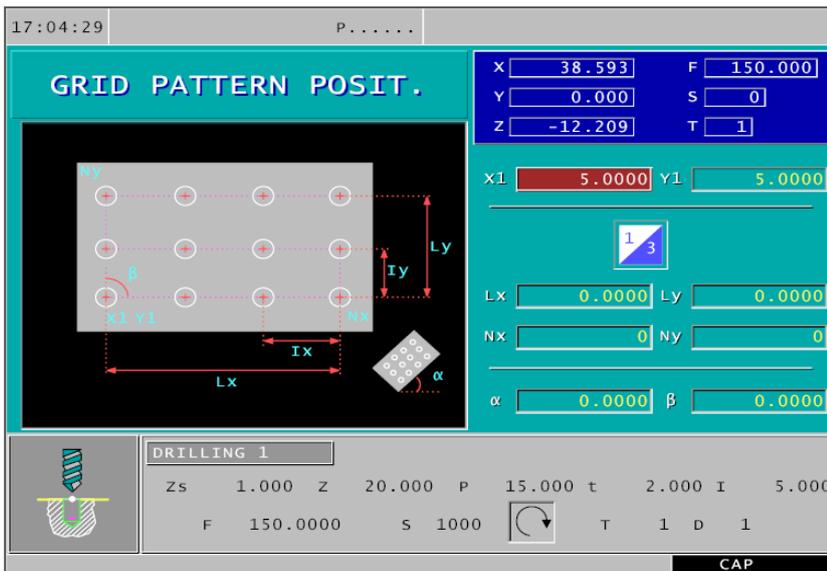
All value inputs have to be confirmed with



The technological data (feed rate, spindle speed, direction of rotation) that have been captured by the cycle, can be checked in the bottom screen window. After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

GRID PATTERN POSIT.



Grid pattern positioning enables you to define multiple positionings of several points along a grid pattern.

Selection with the  key.

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X1, Y1)

Coordinates of the starting point

Type of point definition:



Length in X (Lx)

Length in Y (Ly)

Number of positionings in X (Nx)

Number of positionings in Y (Ny)



Distance between the positionings in X (Ix)

Distance between the positionings in Y (Iy)

Number of positionings in X (Nx)

Number of positionings in Y (Ny)



Length in X (Lx)

Length in Y (Ly)

Distance between the positionings in X (Ix)

Distance between the positionings in Y (Iy)

Selection with .

Angle in relation to the X axis (α)

Angle between the X- and the Y axis of the positioning (β)

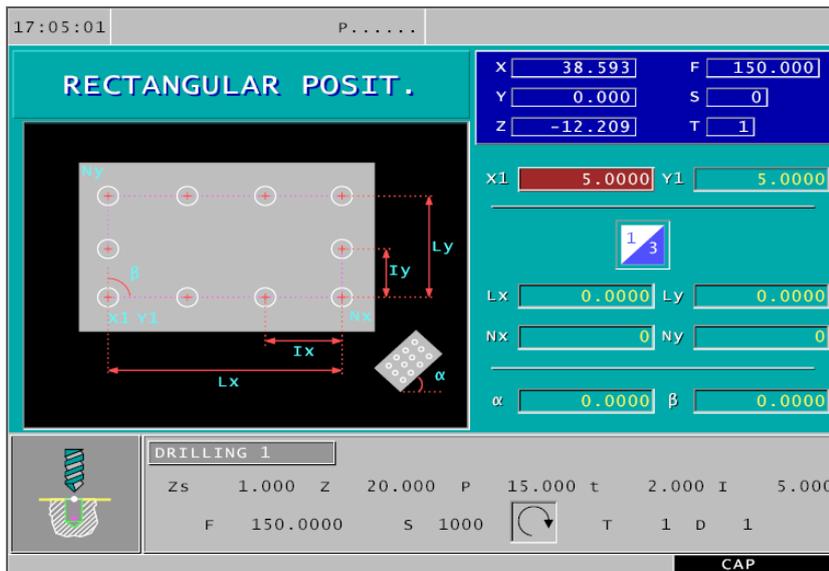
The technological data (feed rate, spindle speed, direction of rotation) that have been captured by the cycle, can be checked in the bottom screen window. After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

RECTANGULAR POSIT.



Rectangular positioning enables you to define multiple positionings of several points along a rectangle.

Selection with the  key.

Use the cursor keys or the enter key to move within the cycle program.

Coordinates (X1, Y1)

Coordinates of the starting point

Type of point definition:



Length in X (Lx)

Length in Y (Ly)

Number of positionings in X (Nx)

Number of positionings in Y (Ny)



Distance between the positionings in X (Ix)

Distance between the positionings in Y (Iy)

Number of the positionings in X (Nx)

Number of the positionings in Y (Ny)



Length in X (Lx)

Length in Y (Ly)

Distance between the positionings in X (Ix)

Distance between the positionings in Y (Iy)

Selection with .

Angle in relation to the X axis (α)

Angle between the X- and the Y axis of the positioning (β)

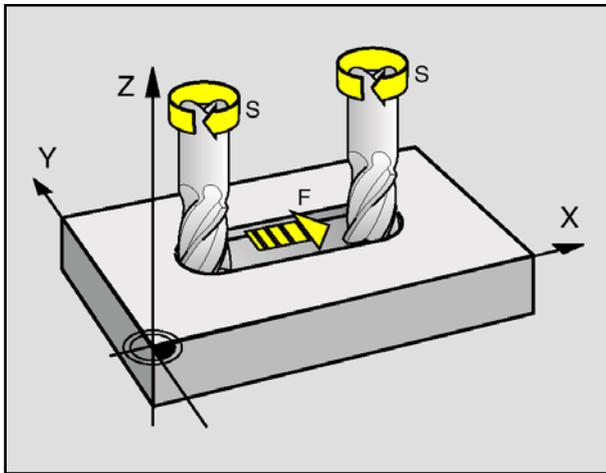
The technological data (feed rate, spindle speed, direction of rotation) that have been captured by the cycle, can be checked in the bottom screen window. After having completed the input, the cycle can be

checked by means of the graphic function  (see graphic simulation).

Note:

All value inputs have to be confirmed with .

E: Tool programming



Tool related data

Feed rate F

The feed rate **F** is the speed in mm/min (inch/min), at which the tool center moves on its path. The maximum feed rate can be different for each machine axis and is defined by machine parameters.

Input

- Press the **F** key.
- Enter the required feed rate.
- Press **ENTER** so that the WinNC captures the new value for the axis feed.

To abort, press **ESC**.

Rapid traverse

For rapid traverse enter F99999.

Adjustment during program run

During program run you can adjust the feed rate by means of the override turning knob **F** for the feed rate.

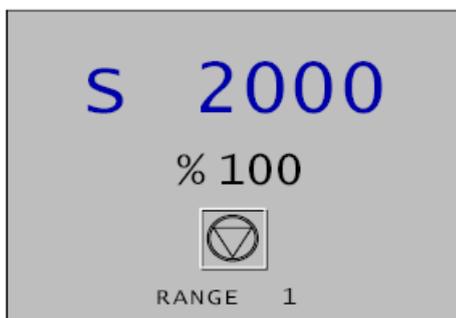
Spindle speed S

The spindle speed **S** has to be entered in rotations per minute (rpm).

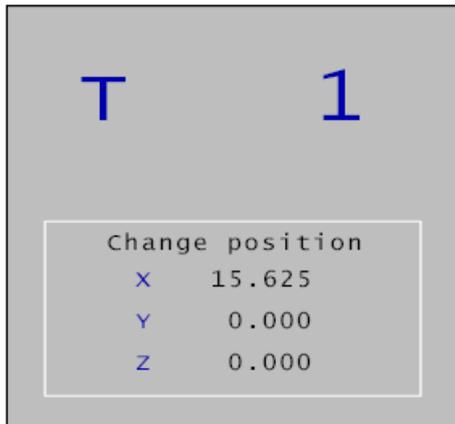
Input

- To select another spindle speed press the **S** key. The highlight jumps on the current value.
- Enter the new value and press the **DOWN** key.
- Spindle condition: clockwise rotation: press the key **CW** for a short time, counterclockwise rotation: press the key **CCW** for at least 1 sec.
- Percentage (%) of the nominal spindle speed that is applied. If you wish to change it, press **←**,

100% or **→**.



Input box for spindle speed in RPM



Tool control

The standard screen displays the following information regarding the tool:

- In large figures the number "T" of the selected tool and a graphic display of its tip.
- The coordinate values of the tool changing point.

Selecting a new tool

- Press the  key. The highlight jumps on the current tool number.
- Enter the number of the required tool.
- After you have pressed the  key, the WinNC will execute the tool change.

Tool changing point

There is only one tool changing point for the whole workpiece program.

The tool changing point that was programmed last, remains active until a new point will be defined.

- Press the  key. The highlight jumps on the current tool number.
- Press the keys ,  and .

Now you have the possibility to:

a.) enter the required value manually and confirm with .

b.) move the axes to the required point by means of the JOG keys and enter the current position of the machine. Press the  key. The WinNC assigns the coordinate value to the selected field.

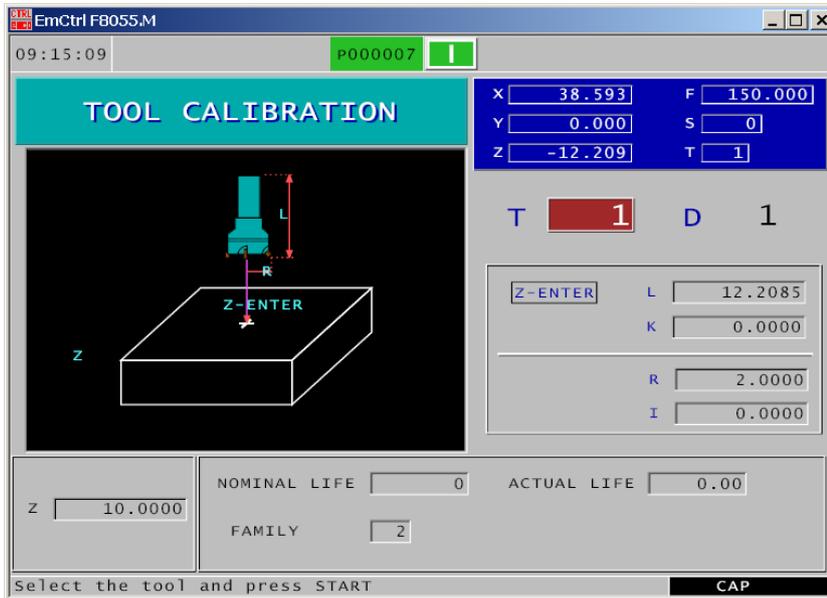
Confirm with .



Danger of collision

Look out for possible collisions between workpiece, clamping device and tool when approaching the tool changing point.

Tool calibration



Press the  key to access the tool calibration mode.

Clamping the workpiece
Clamp a workpiece of known height into the workpiece holder. Define the workpiece height Z in the calibration window (at the bottom left).

Defining tool data

Define the tool number "T":

(tool number) and press .

Define the number of the tool offset "D":

(number of the tool offset) and press .

Define the tool dimensions:

- L...length
- R...radius
- I...correction value radius wear
- K...correction value length wear

Define the remaining data that refer to the tool:

- Nominal working life
- Actual working life
- Code of the tool family

Note:

All inputs have to be confirmed with .

The data for nominal working life, actual working life or the code of the tool family are not taken into consideration by the WinNC.



Tool calibration with the touching method

- Define the Z value of the calibration control part.
- The tool must touch the face of the workpiece, then press  .
- Now the tool is measured in longitudinal direction. The WinNC updates the value of the tool length L and sets the values of I and K to 0.
- Define the corresponding tool radius R.

Note:

The data L and R indicate the dimensions of the tool. The data I and K indicate the correction value that the WinNC has to consider in order to compensate for the tool wear.

The WinNC adds the value of the correction value (K) to the length (L) and the correction value (I) to the radius (R), so as to get the actual tool length (L+K) and the tool radius (R+I) that has to be used.

To leave the tool calibration mode, press .

F: Program run

Requirements

Datum setting

The used datums must be measured and entered.

Tools

The used tools must be measured and entered.
The tools must be located at the corresponding position (T) in the tool changer.

Reference point

The reference point must be traversed in all axes.

Machine

The machine must be ready for operation.
The workpiece must be clamped safely.
Loose parts (clamping keys, etc.) must not be in the working place in order to avoid collisions.
The machine door must be close before the program is started.

Alarms

There must not be any alarms activated.

Program start, Program stop

Select a program for machining.

Press the  key.

Stop program with , continue with .

Abort program with .

H: Alarms and Messages

Input Device Alarms 3000 - 3999

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

Missing digitizer calibration

Cause: A digitizer tablet has been installed but not calibrated

Remedy: Calibrate digitizer tablet (set corner points), see External Input Devices

3001 General RS232 communication error

Remedy: Correct settings of serial interface.

3002 Control keyboard missing

Remedy: Connect control keyboard, switch on, ...

3003 Digitizer missing

Remedy: Connect digitizer, switch on, ...

3004 Check sum error in control keyboard

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

3005 Error in control keyboard

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

3006 Error with initializing control keyboard

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

Machine Alarms 6000 - 7999

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

PC MILL 50 / 55, PC TURN 50 / 55

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

6000: EMERGENCY OFF

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

6001: CYCLE TIME EXCEEDS LIMIT

Contact EMCO Service.

6002: NO PLC PROGRAM LOADED

Contact EMCO Service.

6003: DB NOT EXISTENT

Contact EMCO Service.

6004: RAM ERROR ON PLC BOARD

Contact EMCO Service.

6009: FAILURE SAFETY CIRCUIT

Defective door limit switch or main contactor. Operating the machine is not possible.
Contact EMCO Service.

6010: X-AXIS NOT READY

Step motor board defective, 24 V or 30 V fuse defective. Check fuses and switch box fan filter.
Contact EMCO Service.

6011: Y-AXIS NOT READY

see alarm 6010.

6012: Z-AXIS NOT READY

see alarm 6010.

6013: MAIN DRIVE NOT READY

Main drive power supply defective, cable defective, main drive overload.
Check fuse, reduce load.
Contact EMCO service.

6014: NO SPEED FOR MAIN SPINDLE

This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).

6019: VICE TIMEOUT

24 V fuse defective, hardware defective.
Contact EMCO service.

6020: VICE FAILURE

24 V fuse defective, hardware defective.
Contact EMCO service.

6024: DOOR NOT CLOSED

The door was opened while a machine movement.
The program will be aborted.

6025: GEARBOX COVER NOT CLOSED

The gearbox cover was opened while a machine movement. A running CNC program will be aborted.
Close the cover to continue.

6027: DOOR LIMIT SWITCH DEFECTIVE

The limit switch of the automatic door is displaced, defective, wrong cabled.
Contact EMCO service.

6028: DOOR TIMEOUT

The automatic door sticks, the pressured air supply is insufficient, the limit switch is displaced.
Check door, pressured air supply, limit switch or contact EMCO service.

6030: NO PART CLAMPED

No workpiece inserted, vice cheek displaced, control cam displaced, hardware defective.
Adjust or contact EMCO service.

6031: QUILL FAILURE**6037: CHUCK TIMEOUT****6039: CHUCK PRESSURE FAILURE****6041: TOOL CHANGE TIMEOUT**

Tool turret sticks (collision?), 24 V fuse defective, hardware defective.

A running CNC program will be stopped.
Check for a collision or contact EMCO service.

6042: TOOL CHANGE TIMEOUT

see alarm 6041.

6043: TOOL CHANGE TIMEOUT

see alarm 6041.

6044: TOOL TURRET SYNC ERROR

Hardware defective.
Contact EMCO service.

6046: TOOL TURRET SYNC MISSING

Hardware defective.
Contact EMCO service.

6048: DIVIDING TIME EXCEEDED

Dividing head sticks, insufficient pressured air supply, hardware defective.

Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED

see alarm 6048

6050: FAILURE DIVIDING DEVICE

Hardware defective.
Contact EMCO service.

7000: INVALID TOOL NUMBER

The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7007: FEED HOLD

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

7017: GO FOR REFERENCE POINT

Approach the reference point.

7040: DOOR OPEN

The main drive can not be switched on and NC-Start can not be activated.

Some accessories can be operated only with open machine door.

Close the machine to run a program.

7043: PIECE COUNT REACHED

A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7050: NO PART CLAMPED

After switching on or after an the vice is neither at the open position nor at the closed position.

NC-Start is locked.

Traverse the vice manually on a valid end position.

7051: DIVIDING DEVICE NOT INTERLOCKED

After switching on or after an the dividing head is not in a lock position. NC-Start is locked.

PC MILL 100/105/125/155

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

6000: EMERGENCY OFF

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

6001: PLC-CYCLE TIME EXCEEDING

Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED

Contact EMCO Service.

6003: PLC - NO DATA UNIT

Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE

Contact EMCO Service.

6005: OVERHEAT BRAKEMODUL

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

6006: OVERLOAD BRAKE RESISTOR

see 6005

6007: SAFETY CIRCUIT FAULT

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

6009: SAFETY CIRCUIT FAULT

Defective step motor system.

A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.

Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY

The step motor board is defective or too hot, a fuse or cabling is defective.

A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.

Check fuses or contact EMCO service.

6011: DRIVE Y-AXIS NOT READY

see alarm 6010.

6012: DRIVE Z-AXIS NOT READY

see alarm 6010.

6013: MAIN DRIVE NOT READY

Main drive power supply defective, main drive too hot, fuse defective.

A running program will be stopped, the auxiliary 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 auxiliary drives will be stopped.

6024: MACHINE DOOR OPEN

The door was opened while a machine movement. The program will be aborted.

6040: TOOL TURRET INDEX FAILURE

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

6041: TOOL CHANGE TIMEOUT

Tool drum stuck (collision?), main drive not ready, fuse defective, hardware defective.

A running CNC program will be stopped.

Check for collisions, check fuses or contact EMCO service.

6043-6046: TOOL DISK POSITION FAULT

Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.

The Z axis could have been slipped out of the toothing while the machine was switched off.

A running CNC program will be stopped.

Contact EMCO service.

6047: TOOL DISK UNLOCKED

Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.

A running CNC program will be interrupted.
Contact EMCO service.

When the tool drum is turned out of locked position (no defect), act as following:

Turn the drum into locking position manually

Change into MANUAL (JOG) mode.

Turn the key switch. Traverse the Z slide upwards, until the alarm disappears.

6050: M25 AT RUNNING MAIN SPINDLE

Cause: Programming mistake in NC program.

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY

Cause: pressure failure automatic door
automatic door stucks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN

When opening the clamping pressure switch does not fall within 400ms. Pressure switch defective or mechanical problem. E22.3

6070: PRESSURE SWITCH FOR TANI MISSING

When closing the clamping pressure switch does not respond. No compressed air or mechanical problem. E22.3

6071: DIVIDING DEVICE NOT READY

Servo Ready Signal from frequency converter missing. Excess temperature drive TANI or frequency converter not ready for operation.

6072: VICE NOT READY

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

Vice stucks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.

Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY

Cause: locking switch defective
cabling defective
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic dividing device
lock the dividing device

6074: DIVIDING TIME EXCEEDED

Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic dividing device

6075: M27 AT RUNNING MAIN SPINDLE

Cause: Programming mistake in NC program.

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: Correct NC program

7000: INVALID TOOL NUMBER PROGRAMMED

The tool position was programmed larger than 10.

The CNC program will be stopped.

Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES

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

7017: REFERENCE MACHINE

Approach the reference point.

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

7018: TURN KEY SWITCH

With NC-Start the key switch was in position "setting operation".

NC-Start is locked.

Turn the key switch in the position "automatic" to run a program.

7020: SPECIAL OPERATION MODE ACTIVE

Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.

Manual traversing the axes is possible with open door. Swivelling the tool turret is not possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.

For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: INITIALIZE TOOL TURRET

The tool turret operating was interrupted.

No traversing operation is possible.

Press tool turret key in JOG operation. Message occurs after alarm 6040.

7022: INITIALIZE TOOL TURRET !

see 7021

7038: LUBRICATION SYSTEM FAULT

The pressure switch is defective or gagged.

NC-Start is locked. This can be reset only by switching off and on the machine.

Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT

Not enough lubricant, the pressure switch is defective.

NC-Start is locked.

Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN

The main drive can not be switched on and NC-Start can not be activated (except special operation mode)

Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR

Every movement and NC-Start are locked.

Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED

A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7054: VICE OPEN

Cause: the workpiece is not clamped

When switching on the main spindle with M3/M4 alarm 6073 (vice not ready) will be released.

Remedy: Clamp

7055: DIVIDING DEVICE NOT LOCKED

Cause: the dividing device is not locked

When switching on the main spindle with M3/M4 alarm 6073 (dividing device not ready) will be released.

Remedy: lock dividing device

7270: OFFSET COMPENSATION ACTIVE !

Only with PC-MILL 105

Offset compensation activated by the following operation sequence.

- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4

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

7271: COMPENSATION FINISHED, DATA SAVED !

see 7270

PC TURN 105/120/125/155

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

6000: EMERGENCY OFF

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

6001: PLC-CYCLE TIME EXCEEDING

The auxiliary drives will be switched off.
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED

The auxiliary drives will be switched off.
Contact EMCO Service.

6003: PLC - NO DATA UNIT

The auxiliary drives will be switched off.
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE

The auxiliary drives will be switched off.
Contact EMCO Service.

6008: MISSING CAN SUBSCRIBER

Check fuses or contact EMCO Service.

6009: SAFETY CIRCUIT FAULT

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

6010: DRIVE X-AXIS NOT READY

The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6012: DRIVE Z-AXIS NOT READY

see 6010.

6013: MAIN DRIVE NOT READY

Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.
A running program will be stopped, the auxiliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED

This alarm will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxiliary drives will be switched off.

6015: NO DRIVEN TOOL SPINDLE SPEED

see 6014.

6024: MACHINE DOOR OPEN

The door was opened while a machine movement.
The program will be aborted.

6040: TOOL TURRET INDEX FAILURE

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

6041: TOOL CHANGE TIMEOUT

Tool drum stuck (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6042: TOOL TURRET OVERHEAT

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

6043: TOOL CHANGE TIMEOUT

Tool drum sticks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6046: TOOL TURRET ENCODER FAULT

Fuse defective, hardware defective.
Check fuses or contact EMCO service.

6048: CHUCK NOT READY

Attempt to start the spindle with open chuck or without clamped workpiece.
Chuck sticks mechanically, insufficient pressured air supply, fuse defective, hardware defective.
Check fuses or contact EMCO service.

6049: COLLET NOT READY

see 6048

6050: M25 DURING SPINDLE ROTATION

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

6055: NO PART CLAMPED

This alarm occurs when with rotating spindle the clamping device or the tailstock reach the end position. The workpiece has been pushed out of the chuck or has been pushed into the chuck by the tailstock.
Check clamping device settings, clamping forces, alter cutting data.

6056: QUILL NOT READY

Attempt to start the spindle or to move an axis or to swivel the tool turret with undefined tailstock position. Tailstock is locked mechanically (collision), insufficient pressured air supply, fuse defective, magnetic switch defective.
Check for collisions, check fuses or contact EMCO service.

6057: M20/M21 DURING SPINDLE ROTATION

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

6058: M25/M26 DURING QUILL FORWARD

To actuate the clamping device in an NC program with M25 or M26 the tailstock must be in back end position.

6059: C-AXIS SWING IN TIMEOUT

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

6060: C-AXIS INDEX FAILURE

When swivelling in the C-axis the limit switch does not respond.
Check pneumatics, mechanics and limit switch.

6064: AUTOMATIC DOOR NOT READY

Door sticks mechanically (collision), insufficient pressured air supply, limit switch defective, fuse defective.
Check for collisions, check fuses or contact EMCO service.

6065: LOADER MAGAZINE FAILURE

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

6066: CLAMPING DEVICE FAILURE

No compressed air at the clamping device
Check pneumatics and position of the clamping device proximity detectors.

7000: INVALID TOOL NUMBER PROGRAMMED

The tool position was programmed larger than 8.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES

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

7017: REFERENCE MACHINE

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

7018: TURN KEY SWITCH

With NC-Start the key switch was in position "setting operation".

NC-Start is locked.

Turn the key switch in the position "automatic" to run a program.

7019: PNEUMATIC LUBRICATION MONITORING!

Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE

Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.

Manual traversing the axes is possible with open door. Swivelling the tool turret is possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.

For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: TOOL TURRET NOT LOCKED

The tool turret operating was interrupted.

NC start and spindle start are locked. Press the tool turret key in the RESET status of the control.

7038: LUBRICATION SYSTEM FAULT

The pressure switch is defective or gagged.

NC-Start is locked. This alarm can be reset only by switching off and on the machine.

Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT

Not enough lubricant, the pressure switch is defective.

NC-Start is locked.

Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN

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

7042: INITIALIZE MACHINE DOOR

Every movement and NC-Start are locked.

Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED

A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7048: CHUCK OPEN

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

7049: CHUCK - NO PART CLAMPED

No part is clamped, the spindle can not be switched on.

7050: COLLET OPEN

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

7051: COLLET - NO PART CLAMPED

No part is clamped, the spindle can not be switched on.

7052: QUILL IN UNDEFINED POSITION

The tailstock is in no defined position.

All axis movements, the spindle and the tool turret are locked.

Drive the tailstock in back end position or clamp a workpiece with the tailstock.

7053: QUILL - NO PART CLAMPED

The tailstock reached the front end position. Traverse the tailstock back to the back end position to continue.

7054: NO PART CLAMPED

No part clamped, switch-on of the spindle is locked.

7055: CLAMPING DEVICE OPEN

This message indicates that the clamping device is not in clamping state. It disappears as soon as a part is clamped.

AC ALARMS

Axis Controller Alarms 8000 - 9999

8000 Fatal Error AC

8004 ORDxx Failure main-drive unit

8005 - 8009 ORDxx Internal error AC

Remedy: report to EMCO if repeatable

8010 ORDxx Syncr. error main drive

Cause: synchronisation mark missing for main drive

Remedy: report to service technician, if repeatable

8011 - 8013 ORDxx Internal error AC

Remedy: report to EMCO, if repeatable

8014 ORDxx Decel.-time of axis too high

Remedy: report to service technician, if repeatable

8018 ORDxx Internal error AC

Remedy: report to EMCO, if repeatable

8021 ORDxx Internal error AC

Remedy: report to EMCO, if repeatable

8022 ORDxx Internal error AC

Remedy: report to EMCO, if repeatable

8023 ORDxx Invalid Z value for helix

Cause: The Z value of the helix must be smaller than the length of the arc to be traversed

Remedy: Program correction

8100 Fatal init error AC

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8101 Fatal init error AC

see 8101.

8102 Fatal init error AC

see 8101.

8103 Fatal init error AC

see 8101.

8104 Fatal system error AC

see 8101.

8105 Fatal init error AC

see 8101.

8106 No PC-COM card found

Cause: PC-COM board can not be accessed (ev. not mounted).

Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working

see 8106.

8108 Fatal error on PC-COM card

see 8106.

8109 Fatal error on PC-COM card

see 8106.

8110 PC-COM init message missing

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8111 Wrong configuration of PC-COM

see 8110.

8113 Invalid data (pccom.hex)

see 8110.

8114 Programming error on PC-COM

see 8110.

8115 PC-COM packet acknowledge missing

see 8110.

8116 PC-COM startup error

see 8110.

8117 Fatal init data error (pccom.hex)

see 8110.

8118 Fatal init error AC

see 8110, ev. insufficient RAM memory

8119 PC interrupt no. not valid

Cause: The PC interrupt number can not be used.

Remedy: Find out free interrupt number in the Windows95 system control (allowed: 5,7,10,11,12,3,4 and 5) and enter this number in WinConfig.

8120 PC interrupt no. unmaskable

see 8119

8121 Invalid command to PC-COM

Cause: Internal error or defective cable

Remedy: Check cables (screw it); Restart software or reinstall when necessary, report to EMCO, if repeatable.

8122 Internal AC mailbox overrun

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8123 Open error on record file

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8124 Write error on record file

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8125 Invalid memory for record buffer

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

8126 AC Interpolation overrun

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

8127 Insufficient memory

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

8128 Invalid message to AC

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

8129 Invalid MSD data - axisconfig.

see 8128.

8130 Internal init error AC

see 8128.

8130 Internal init error AC

see 8128.

8132 Axis accessed by multiple channels

see 8128.

8133 Insufficient NC block memory AC

see 8128.

8134 Too much center points programmed

see 8128.

8135 No centerpoint programmed

see 8128.

8136 Circle radius too small

see 8128.

8137 Invalid for Helix specified

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

8140 Maschine (ACIF) not responding

Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

8141 Internal PC-COM error

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

8142 ACIF Program error

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

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

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

8146 Multiple request for axis
see 8142.

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

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

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

8150 Fatal ACIF error
see 8142.

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

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

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

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

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

8157 Data record done
see 8142.

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

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

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

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:

- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm) or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost

see 8161

8163 Z-Axis synchronization lost

see 8161

8164 Software limit switch max axis 3..7

Cause: Axis is at traverse area end

Remedy: Retract axis

8168 Software limit overtravel axis 3..7

Cause: Axis is at traverse area end

Remedy: Retract axis

8172 Communication error to machine

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running**8174 INC not allowed****8175 MSD file could not be opened**

Cause: Internal error

Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened

see 8175.

8177 PLS file could not be accessed

see 8175.

8178 PLS file could not be written

see 8175.

8179 ACS file could not be opened

see 8175.

8180 ACS file could not be accessed

see 8175.

8181 ACS file could not be written

see 8175.

8182 Gear change not allowed**8183 Gear too high****8184 Invalid interpolaton command****8185 Forbidden MSD data change**

see 8175.

8186 MSD file could not be opened

see 8175.

8187 PLC program error

see 8175.

8188 Gear command invalid

see 8175.

8189 Invalid channel assignement

see 8175.

8190 Invalid channel within message**8191 Invalid jog feed unit****8192 Invalid axis in command****8193 Fatal PLC error**

see 8175.

8194 Thread without length**8195 No thread slope in leading axis**

Remedy: Program thread pitch

8196 Too many axis for thread

Remedy: Program max. 2 axes for thread.

8197 Thread not long enough

Cause: Thread length too short.

With transition from one thread to the other the length of the second thread must be sufficient to produce a correct thread.

Remedy: Longer second thread or replace it by a linear interpolation (G1).

8198 Internal error (to many threads)

see 8175.

8199 Internal error (thread state)

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8200 Thread without spindle on

Remedy: Switch on spindle

8201 Internal thread error (IPO)

see 8199.

8201 Internal thread error (IPO)

see 8199.

8203 Fatal AC error (0-ptr IPO)

see 8199.

8204 Fatal init error: PLC/IPO running

see 8199.

8205 PLC Runtime exceeded

Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation

see 8199.

8207 Invalid PLC machine data

see 8199.

8208 Invalid application message

see 8199.

8211 Feed too high (thread)

Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed

Remedy: Program correction, lower pitch or lower spindle speed for thread

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

