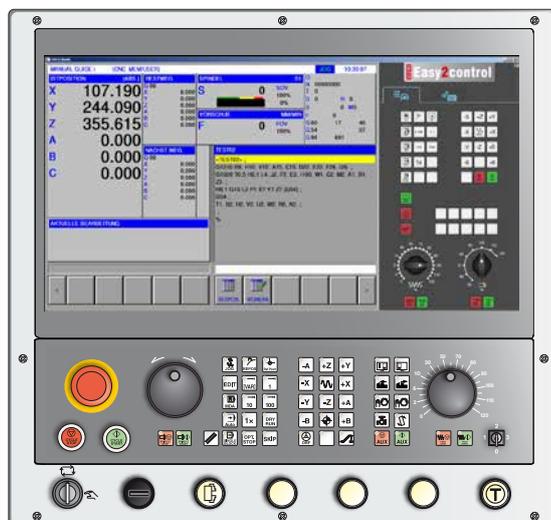


# EMCO WinNC for Fanuc 31i Mill

## Software description, software release effective from 01.02



### Software description

### WinNC for Fanuc 31i Mill

Ref. no. EN 1846  
Edition C 2016-02

These instructions are also available at any time  
on request as electronic copy (.pdf).

Original operating instructions

EMCO GmbH  
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-world.com](http://www.emco-world.com)  
E-Mail: [service@emco.at](mailto:service@emco.at)



Designed for your profit

**Instructions:**

Some of the functionality of the EMCO WinNC for Fanuc 31i control software is beyond the scope of these instructions. Emphasis is placed instead on representing the most important functions simply and clearly, in order to achieve the greatest possible learning success. Depending on the machine you are operating with EMCO WinNC for Fanuc 31i, not all functions may be available.

# Preface

The EMCO WinNC for Fanuc 31i software is a component of the EMCO training concept. EMCO WinNC for Fanuc 31i makes it easy to operate CNC lathes/milling machines. Neither is prior knowledge of ISO programming needed.

An interactive contour programming routine means you can define workpiece contours with linear and circular contour elements.

A cycle is programmed interactively, with graphics support. A large number of processing cycles, which can be combined freely with each other to form a single program, is available to the user.

Individual cycles or the NC programs produced can be simulated graphically on screen.

Some of the functionality of the EMCO WinNC for Fanuc 31i control software is beyond the scope of these instructions. Emphasis is placed instead on representing the most important functions simply and clearly, in order to achieve the greatest possible learning success.

If you have queries about or suggestions for improving these operating instructions, please make direct contact with [name needed!]

**EMCO GmbH**  
**Technical Documentation Department**  
**A-5400 HALLEIN, Austria**



Designed for your profit

## EC compliance



The CE symbol, together with the EC compliance declaration, certifies that the machine and instructions comply with the directives under which the products are covered.

All rights reserved. Copying only with permission from EMCO GmbH  
© EMCO GmbH., Hallein

# Table of contents

Preface ..... 3  
 Table of contents ..... 4

## A: Principles

EMCO milling machine reference points ..... A1  
     N (T) = Tool zero point ..... A1  
     M = Machine zero point ..... A1  
     W = Workpiece zero point ..... A1  
     R = Reference point ..... A1  
 Milling machine reference system ..... A2  
 Polar coordinates ..... A3  
 Absolute and incremental workpiece positions ..... A4  
 Zero point offset ..... A5  
 Milling operations ..... A7  
     Down-cut Milling ..... A7  
     Up-cut Milling ..... A7  
     Down-cut/up-cut milling ..... A7  
 Tool radius compensation ..... A8  
 Tool data ..... A9

## B: Key description

EMCO WinNC for Fanuc 31i control panel ..... B1  
 Address and number pad ..... B2  
 Button functions ..... B3  
 Key description ISO functions ..... B5  
 Screen layout Manual Guide *i* ..... B10  
 PC keyboard ..... B11  
     Overview button assignment control keyboard ..... B12  
     Overview Button Assignment Machine Operating Elements B13  
 Machine control panel ..... B15  
 Key description ..... B15  
     Skip (block mask) ..... B15  
     Dry run (test run feed) ..... B15  
     Individual piece mode ..... B16  
     Optional stop ..... B16  
     Edit ..... B16  
     Handwheel mode (optional) ..... B16  
     Reset key (Reset) ..... B16  
     Feed Stop ..... B16  
     Feed Start ..... B16  
     Single block ..... B17  
     Cycle stop ..... B17  
     Cycle start ..... B17  
     Arrow keys ..... B17  
     Rapid traverse ..... B17  
     Reference point ..... B17  
     Chip conveyor (Option) ..... B17  
     Swing tool drum ..... B18  
     Manual tool change ..... B18  
     Clamping devices ..... B18  
     Coolant ..... B18  
     Feed Stop ..... B18  
     Feed Start ..... B18  
     Types of operation ..... B19  
     Auxiliary OFF ..... B20  
     Auxiliary ON ..... B20  
     Override switch (feed rate override) ..... B21  
     EMERGENCY SHUTDOWN ..... B21  
     Key Switch Special Operations Mode ..... B21

Multifunction switch for operating modes ..... B22  
 Key switch ..... B25  
 Additional clamping device button ..... B25  
 USB connection (USB 2.0) ..... B25  
 Enable button ..... B25

## C: Operation

Feed F [mm/min] ..... C1  
 Spindle speed S [rev/min] ..... C2  
 Types of operation ..... C3  
 Approach reference point ..... C5  
 Move slides manually ..... C6  
 Move slides in step-movement ..... C6  
 Program management ..... C8  
     Create program ..... C9  
     Where programs are saved ..... C9  
     Programming list ..... C10  
     Copy program ..... C12  
     Delete program ..... C12  
     Enter comment ..... C13  
     Search program ..... C13  
     Delete a number of programs simultaneously ..... C14  
     Change the sorting sequence ..... C15  
     Open program ..... C15  
     Change program name ..... C16  
     Program properties ..... C16  
     Program protection ..... C17  
     Program input and output on a memory card ..... C17  
     Search for text from the input line in the program ..... C18  
     Search for and open the program ..... C18  
     Copy marked text into a buffer file ..... C19  
     Move marked text into the buffer file ..... C19  
     Insert text ..... C20  
     Delete highlighted text ..... C20  
     Insert marked text into an input line ..... C21  
     Revocation and withdrawal ..... C22  
     Search and replace ..... C22  
     End program listing ..... C23  
 Program mode ..... C24  
     Background Editing ..... C25  
 Semi-automatic mode ..... C26  
 Zero point table ..... C28  
     Setting up workpiece coordinates data ..... C28  
     Measuring ..... C29  
     Compute ..... C29  
 Graphical simulation ..... C31  
     Screen layout graphical simulation ..... C32  
     Softkey functions ..... C33  
     3D configuration ..... C37  
     Shift graphics ..... C38

## D: Programming with MANUAL GUIDE *i*

Overview ..... D1  
     M-commands ..... D1  
     Overview ..... D2  
 Produce MANUAL GUIDE *i* program ..... D2  
     Programming set-up ..... D3  
 Raw part definition ..... D6

Cycle overview .....	D7
Working with cycles .....	D10
Data Entry for Machining Cycles .....	D12
Default values for cycle parameters .....	D14
Ignore plausibility check for saving .....	D15
Select dimensions system .....	D16
Hole machining .....	D17
Centre drilling G1100 .....	D18
Drilling G1001 .....	D20
Tapping G1002 .....	D24
Reaming G1003 .....	D26
Boring G1004 .....	D28
Face machining .....	D31
Facing (rough) G1020 .....	D32
Facing (finish) G1021 .....	D34
Contouring .....	D37
Outer wall (rough) G1060 .....	D38
Outer wall (bottom finish) G1061 .....	D44
Outer wall (side finish) G1062 .....	D48
Outer wall (chamfer) G1063 .....	D52
Inner wall (rough) G1064 .....	D56
Inner wall (bottom finish) G1065 .....	D58
Inner wall (side finish) G1066 .....	D60
Inner wall (chamfer) G1067 .....	D62
Partial (rough) G1068 .....	D64
Partial (bottom finish) G1069 .....	D66
Partial (side finish) G1070 .....	D68
Partial (chamfer) G1071 .....	D70
Pocketing .....	D73
Pocketing (rough) G1040 .....	D74
Pocketing (bottom finish) G1041 .....	D78
Pocketing (side finish) G1042 .....	D80
Pocketing (chamfer) G1043 .....	D82
Figure: Hole position .....	D85
Random Points G1210 .....	D86
Linear Points	
(same interval) G1211 .....	D88
XY Grid points G1213 .....	D89
XY Rectangle points G1214 .....	D90
XY Circle points G1215 .....	D91
XY Arc points	
(same interval) G1216 .....	D92
XA A-axis Hole Arc	
G1772 .....	D93
XA A-axis Hole Free	
G1773 .....	D94
Figure: Front Surface Contour .....	D95
XY Square facing G1220 .....	D97
Figure: Contouring figure .....	D99
XY Square convex G1220 .....	D100
XY Circle convex G1221 .....	D101
XY Track convex G1222 .....	D102
XY Polygon convex G1225 .....	D103
XY Free figure convex .....	D104
Overview of the input elements for free contour programming.	
D105	
Input items of line (XY plane) G1201 .....	D106
Input items for arc (XY plane) G1202, 1203 .....	D107
Input items of chamfer (XY plane) G1204 .....	D108
Input items of corner (XY plane) G1205 .....	D108
Input items of end of arbitrary figures G1206 .....	D109
Symbol representation of the contour elements .....	D110
XA plane free convex figure for cylinder G1700 .....	D112
XY Square concave G1220 .....	D113
XY Circle concave G1221 .....	D114
XY Track concave G1222 .....	D115
XY Polygon concave G1225 .....	D116
XY Free concave figure .....	D117
XA plane free concave figure for cylinder G1700 .....	D118

XY Free open figure .....	D119
XA plane free open figure for cylinder G1700 .....	D120
Figure: Pocket figure .....	D121
XY Square concave G1220 .....	D122
XY Circle concave G1221 .....	D123
XY Track concave G1222 .....	D124
XY Polygon concave G1225 .....	D125
XY Free concave figure .....	D126
XA plane free concave figure for cylinder G1700 .....	D127
Sub-routines .....	D130
Fixed forms .....	D131
Creating fixed forms .....	D132
M-code menu .....	D134

## E: G code programming

Overview .....	E1
M-commands .....	E1
Command abbreviations overview .....	E2
Calculation operators for NC program .....	E3
Overview of machine G-commands .....	E4
Brief description of G-commands .....	E7
G00 Rapid traverse .....	E7
G01 Straight interpolation .....	E8
Entering chamfers and radii .....	E8
Direct drawing dimension input .....	E9
G02 Circular interpolation, clockwise .....	E11
G03 Circular interpolation, counterclockwise .....	E11
Helical interpolation .....	E11
G04 Dwell time .....	E12
G09 Precision stop (block-by-block) .....	E12
G17-G19 Plane selection .....	E13
G20 Dimensional data in inches .....	E13
G21 Dimensional data in millimetres .....	E13
G28 Return to reference point .....	E14
Mill radius offset .....	E15
G40 Deselection mill radius offset .....	E15
G41 Mill radius offset left .....	E15
G42 Mill radius offset right .....	E15
G43 Tool length offset positive .....	E18
G44 Tool length offset negative .....	E18
G49 Deselection tool length offset .....	E18
G50 Deselection scaling factor .....	E18
G51 Scaling factor .....	E18
G51.1 Mirroring a contour .....	E19
G50.1 Deselecting mirroring .....	E19
G52 Local coordinates system .....	E20
G53 Machine coordinates system .....	E20
G54-G59 Zero point offsets 1-6 .....	E20
G61 Precision stop (modally effective) .....	E21
G64 Cutting mode .....	E21
G65 Macro call .....	E22
G66 Macro call (modal) .....	E23
G67 Macro call (modal) End .....	E23
G68 Coordinate system rotation .....	E24
Drilling Cycles G73 - G89 .....	E25
G73 Chip Break Drilling Cycle .....	E26
G74 Left Tapping Cycle .....	E26
G76 Fine Drilling Cycle .....	E27
G80 Cancel Drilling Cycles .....	E27
G81 Drilling Cycle .....	E28
G82 Drilling Cycle with Dwell .....	E28
G83 Withdrawal Drilling Cycle .....	E29
G84 Tapping Cycle without length compensation .....	E29
G84 Tapping Cycle with length compensation .....	E30
G85 Reaming Cycle .....	E30
G89 Reaming cycle with dwell time .....	E30
G90 Absolute value programming .....	E31

G91 Incremental value programming.....	E31
G94 Feed per minute.....	E31
G95 Feed per revolution.....	E31

## F: Tool management

Tool settings.....	F1
Tool length offset entry.....	F2
Tool radius offset entry.....	F2
Tool wear offset.....	F3
Tool data.....	F4
Select tool.....	F5
Tool setting number.....	F6
Setting angle, corner angle.....	F7
Tool correction and tool data input and output.....	F9
Simulation data.....	F11
3D tool.....	F11
Color select.....	F12
Measuring tool manually.....	F13

## G: Program sequence

Preconditions.....	G1
NC start.....	G2
NC reset.....	G2
NC stop.....	G2
Program start, program stop.....	G2
Repositioning.....	G3
Continue program execution.....	G3
Block scan.....	G4

## H: Alarms and Messages

Machine Alarms 6000 - 7999.....	H1
Inputunit alarms 1700 - 1899.....	H17
Axis Controller Alarms.....	H19
8000 - 9000, 22000 - 23000, 200000 - 300000.....	H19
Axis Controller Messages.....	H26
Control alarms 2000 - 5999.....	H27

## I: Fanuc 31i controller alarms

Controller alarms 0001 - 88000.....	I1
-------------------------------------	----

## W: Accessory Functions

Activating accessory functions.....	W1
Robotic Interface.....	W1
Automatic doors.....	W1
Win3D View.....	W1
DNC interface.....	W2

## X: EMConfig

General.....	X1
How to start EMConfig.....	X2
How to activate accessories.....	X3
High Speed Cutting.....	X3
Easy2control on screen operation.....	X4
Settings.....	X4
Machine room camera.....	X5
How to save changes.....	X6
How to create machine data floppy disk or machine data USB flash drive.....	X6

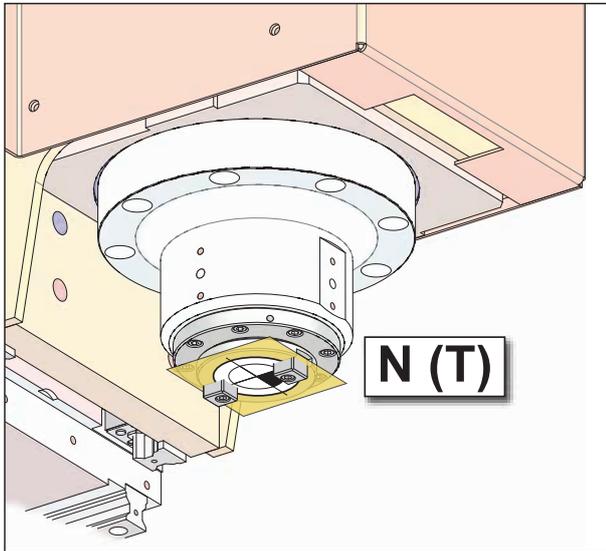
## Y: External Input Devices

EMCO Control Keyboard USB.....	Y1
Scope of supply.....	Y1
Assembling.....	Y2
Connection to the PC.....	Y3
Settings at the PC software.....	Y3
Easy2control On Screen operation.....	Y4
Scope of supply.....	Y4
Operating areas.....	Y5
Machine room camera.....	Y8
Installing the camera.....	Y8
Operating the camera.....	Y9

## Z: Software Installation Windows

System prerequisites.....	Z1
Software installation.....	Z1
Variants of WinNC.....	Z1
Starting WinNC.....	Z3
Terminating WinNC.....	Z3
Checks by EmLaunch.....	Z4
Licence input.....	Z6
Licence manager.....	Z6

# A: Principles



Points on the machine

## EMCO milling machine reference points

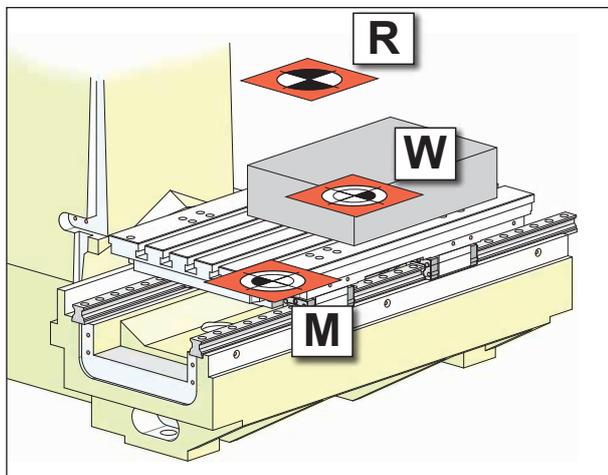
### ⊕ N (T) = Tool zero point

The tool zero point N (T) lies exactly at the cutting point of the spindle axis with the front part of the milling spindle.

The tool zero point is the start point for measuring the tools.

#### Instructions:

The actual reference points may have been defined at other positions, depending on the machine type. In any case, the information in the operation manual for the relevant machine applies!



Machine reference points

### ⊕ M = Machine zero point

The machine zero point M is an unchangeable reference point defined by the manufacturer.

The entire machine is measured from this point. The machine zero point M is the origin of the coordinate system.

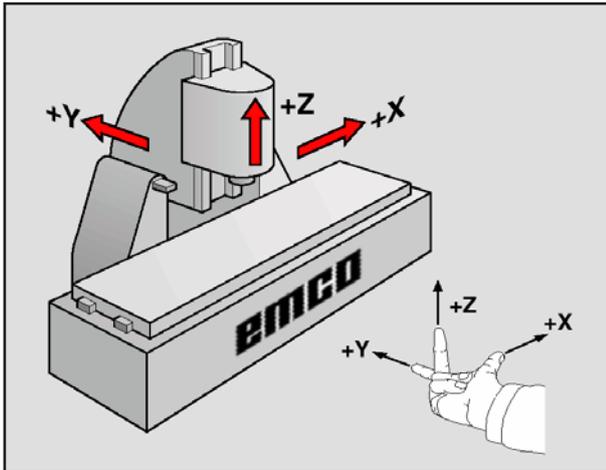
### ⊕ W = Workpiece zero point

The workpiece zero point W can be freely programmed by the user. By programming a workpiece zero point, the origin of the coordinate system will be shifted from the machine zero point M to the workpiece zero point W.

The workpiece zero point W is the start point for the measurements in the parts program.

### ⊕ R = Reference point

The reference point R is a firmly defined point on the machine which is used to calibrate the measurement system. The reference point must be approached each time after the machine is switched on, in order to make the exact distance between the points M and N (T) known to the control system.



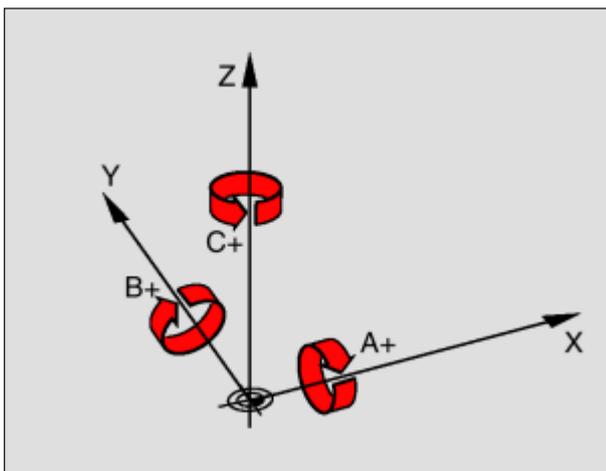
Coordinates system

## Milling machine reference system

With a reference system, you define positions in a plane or in space unambiguously. Positional information always relates to a given point and is described by means of coordinates.

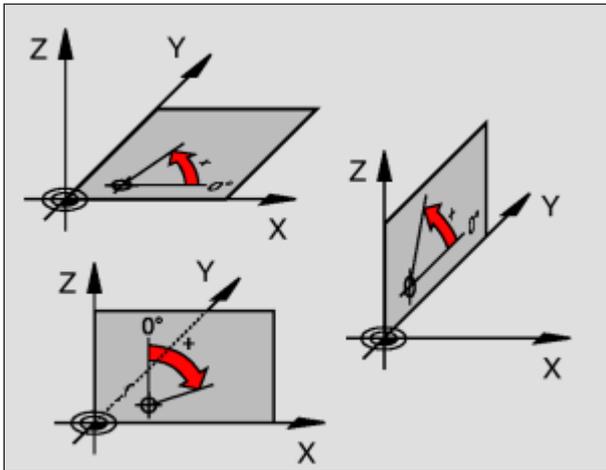
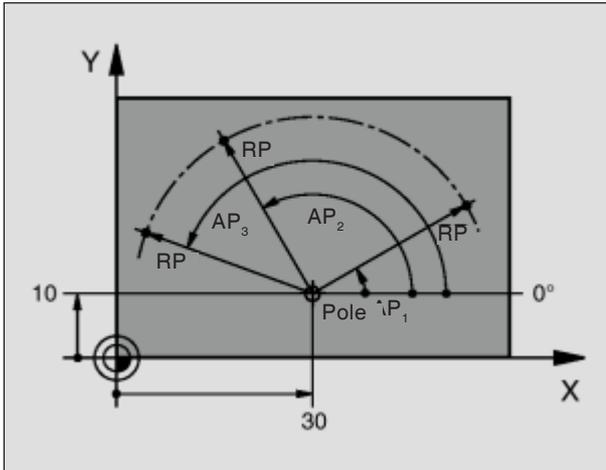
In the right-angled system (Cartesian system), three directions are defined as X, Y and Z axes. The axes are perpendicular respectively to one another and intersect in one point, the zero point. One coordinate gives the distance to the zero point in one of these directions. This is how you describe a position in a plane by means of two coordinates and in space by means of three coordinates.

Coordinates that relate to the zero point are described as **absolute coordinates**. Relative coordinates are related to any other random position (reference point) in the coordinates system. Relative coordinate values are also described as **incremental coordinate** values.



Assignment of the rotating axes to the main axes

When machining a workpiece on a milling machine, you refer in general to the Cartesian coordinates system. The picture on the left shows how the Cartesian coordinates system is assigned to the machine axes. The three-finger right hand rule serves as an aide memoire: If the middle finger is pointing in the direction of the tool axis from workpiece to tool, then it is pointing in the Z+ direction, the thumb in the X+ direction and the index finger in the Y+ direction.



## Polar coordinates

If the production drawing is dimensioned orthographically, you create the machining program with Cartesian coordinates also. For workpieces with arcs or angles, it is often easier to determine positions with polar coordinates.

In contrast to the Cartesian X, Y and Z coordinates, polar coordinates describe only positions in a plane. Polar coordinates have their zero point in the pole.

This is how a position in a plane is unambiguously determined by means of:

- Polar coordinates radius (RP): the distance from the pole to the position.
- Polar coordinates angle (AP): The angle between the angle reference axis and the path that connects the pole with the position.

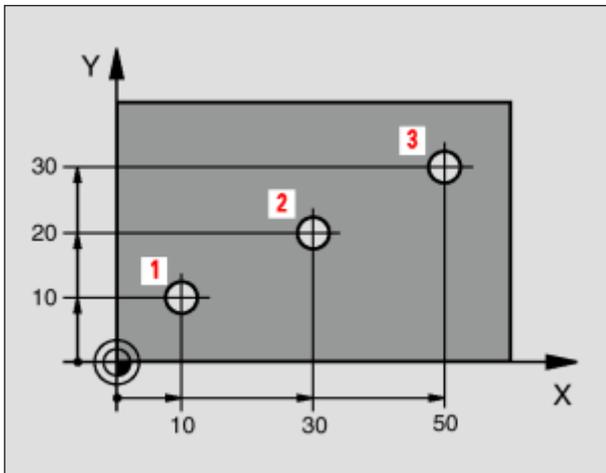
(See picture above left)

### Determining the pole and the angle reference axis

Determine the pole by means of two coordinates in the Cartesian coordinate system in one of the three planes. This is also how the angular reference axis for the polar coordinates angle (AP) is assigned unambiguously.

Polar coordinates (plane)	Angular reference axis
X/Y (G17)	+X
Y/Z (G19)	+Y
Z/X (G18)	+Z

## Absolute and incremental workpiece positions



### Absolute workpiece positions

If the coordinates of a position are related to the coordinates zero point (origin), these are described as absolute coordinates. Each position on a workpiece is determined unambiguously by its absolute coordinates.

Example 1: Drillings with absolute coordinates

Drilling <b>1</b>	Drilling <b>2</b>	Drilling <b>3</b>
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 last programmed position of the tool, which serves as the relative (conceptual) zero point. Incremental coordinates describe the actual paths of the tool. Consequently, it is also described as chain measurement.

You mark an incremental dimension by means of an "I" in front of the axis designation.

Example 2: Drillings with incremental coordinates

Absolute coordinates of the drilling **4**

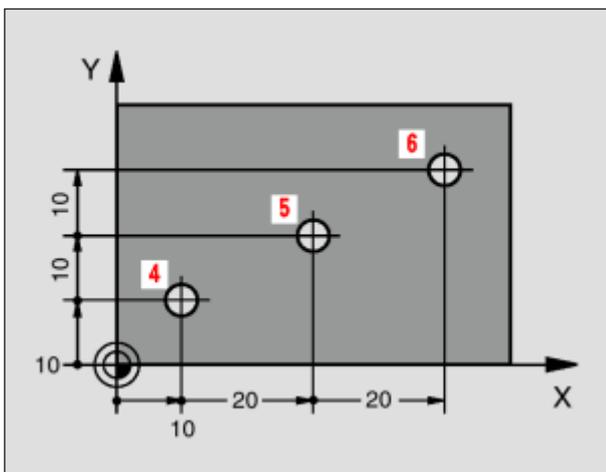
IX = 10 mm  
IY = 10 mm

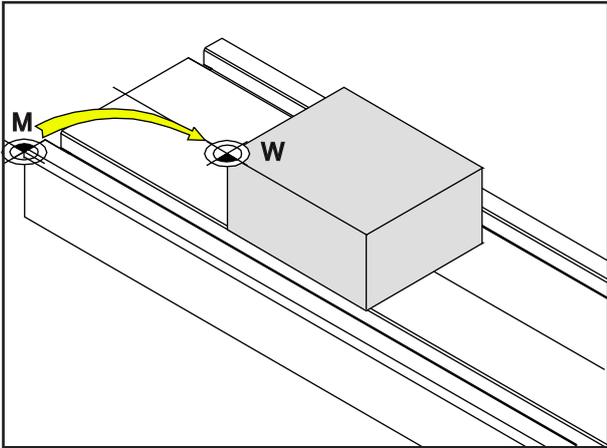
Drilling **5**, related to **4**

IX = 20 mm  
IY = 10 mm

Drilling **6**, related to **5**

IX = 20 mm  
IY = 10 mm





Zero point shift of machine zero point M to tool zero point W

## Zero point offset

On EMCO milling machines, the machine zero point "M" is on the left front edge of the machine table. This position is unsuitable as a start point for programming. Using the so-called zero point offset, the coordinate system can be shifted to a suitable point in the machine working area.

A distinction is made between the following zero point offsets:

- Machine coordinate systems (MCS) with the machine zero point M
- Basic zero point system (BNS)
- Configurable zero point system (CZS)
- Workpiece coordinate system (WCS) with workpiece zero point W.

### Machine coordinate system (MCS)

After approaching the reference point the NC position displays of the axis coordinates are relative to the machine zero point (M) of the machine coordinate system (MCS).

Workpiece change points are defined in the machine coordinate system.

### Base zero point (BZS).

If a base shift is performed in the machine coordinates system (MCS), a base zero point offset is the result (BZS). With this, for example, a palette zero point can be defined.

### Configurable zero point system (CZS)

#### Configurable zero point offset

If a configurable zero point system (G54-G599) is performed from the base zero point system (BZS), a configurable zero point system (CZS) is the result.

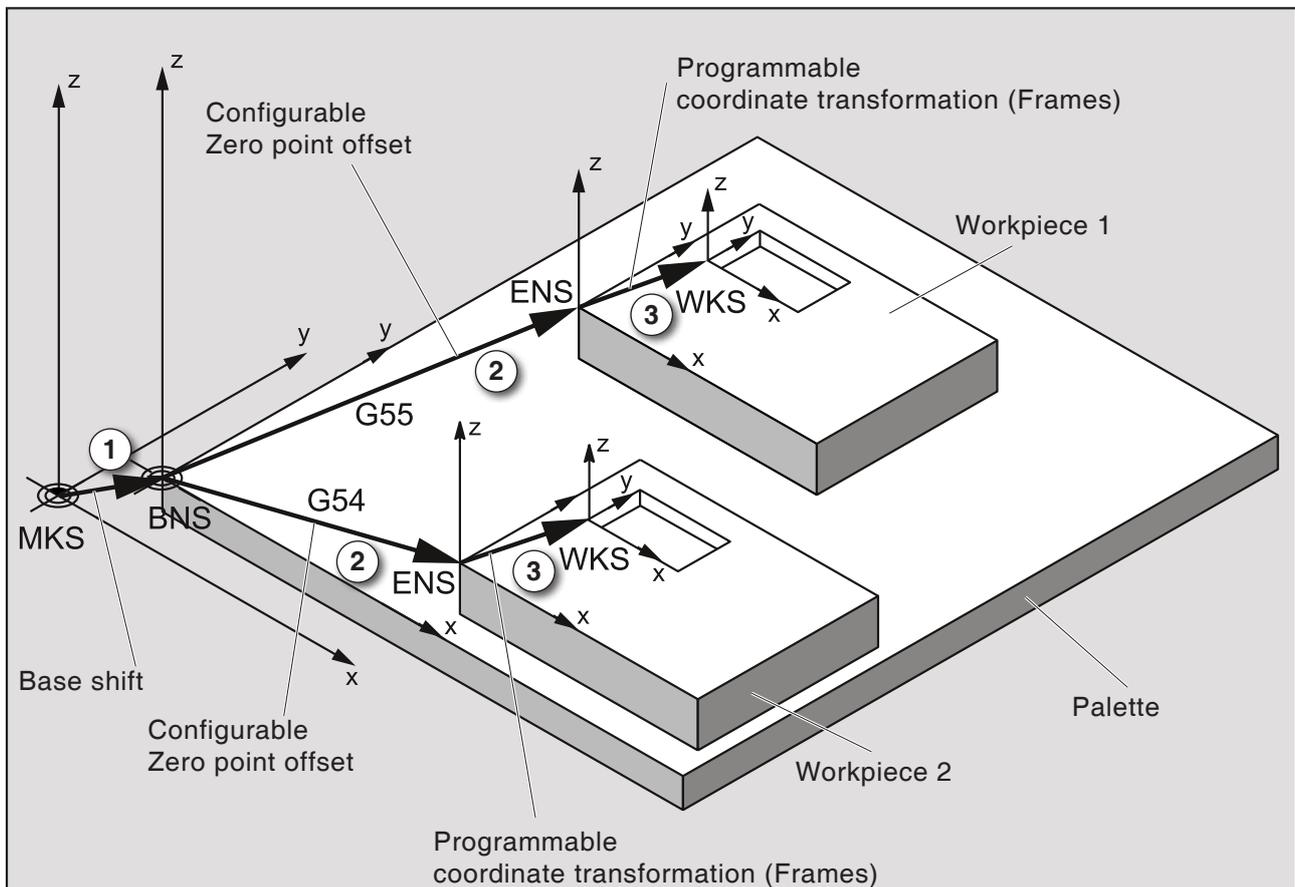
#### Programmable coordinate transformation (Frames)

Programmable coordinate transformations (frames) shift the originally selected workpiece coordinate system to another position, to rotate, scale or mirror it.

### Workpiece coordinate system (WCS)

The program for executing the workpiece is relative to the workpiece zero point (W) of the workpiece coordinate system (WCS).

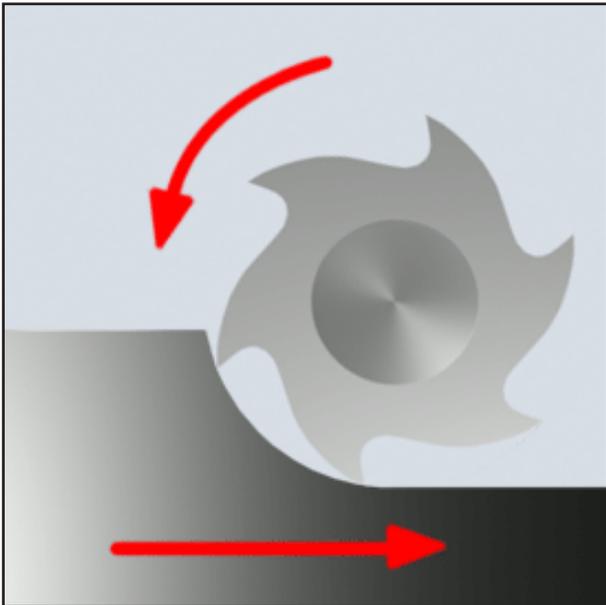
Machine zero point and workpiece zero point are usually not identical. The distance between the points is the entire zero point offset and is made up of various shifts:



- ① With basic offset, the basic zero point offset (BNS) is generated with the range zero point.
- ② With variable zero point offset (G54-G599) and with frames, zero point systems are defined for workpiece 1 or workpiece 2.
- ③ With programmable coordinate transformation (frames), workpiece coordinate systems (WKS) are defined for workpiece 1 or workpiece 2.

## Milling operations

### Down-cut Milling



*Down-cut Milling*

In down-cut milling, feed direction and cutting direction of the mill are identical.

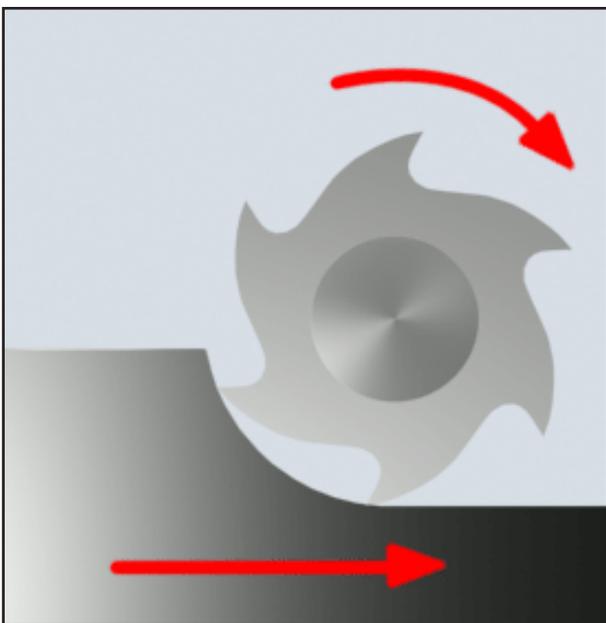
The cut penetrates the surface of the raw part in the material initially.

It is advantageous that a larger chamfer angle enables the immediate penetration of the blade into the material. It is not as with up-cut milling, in which a specific sliding cutting path is left behind under pressure and friction.

In down-cut milling, the feed force supports the feed drive in the same direction of rotation. With machines with play in the feed drive, jerky movements which lead to damage to the cut occur.

Down-cut milling is generally to be preferred if the machine permits it (backlash-free table drive in EMCO CNC machines).

### Up-cut Milling



*Up-cut milling*

In down-cut milling, feed direction and cutting direction of the mill are identical.

The tool's cuts meet in a very sharp angle ( $\varphi = 0$ ) on the material.

Before the blades penetrate into the material, they slide with increasing contact force a little piece on the surface. After penetration, the chip cross-section slowly increases and then falls away quickly.

Up-cut milling is preferred with unstable machine conditions (machines constructed conventionally) and should be used with higher rigidity materials.

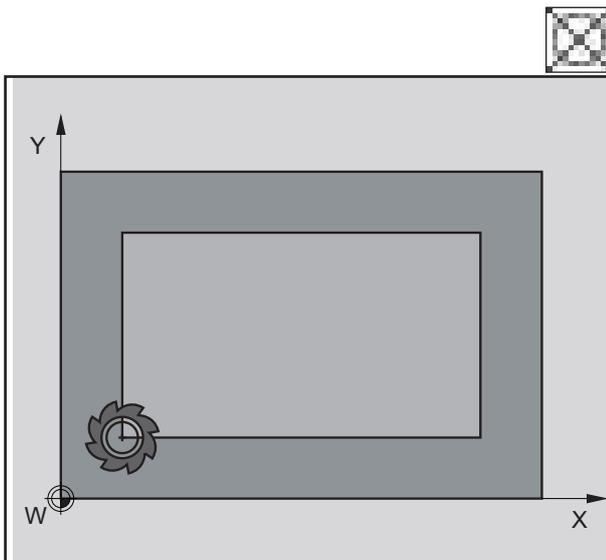
### Down-cut/up-cut milling

Down-cut/up-cut milling is a combination of down-cut and up-cut milling.

## Tool radius compensation

### Without tool radius compensation

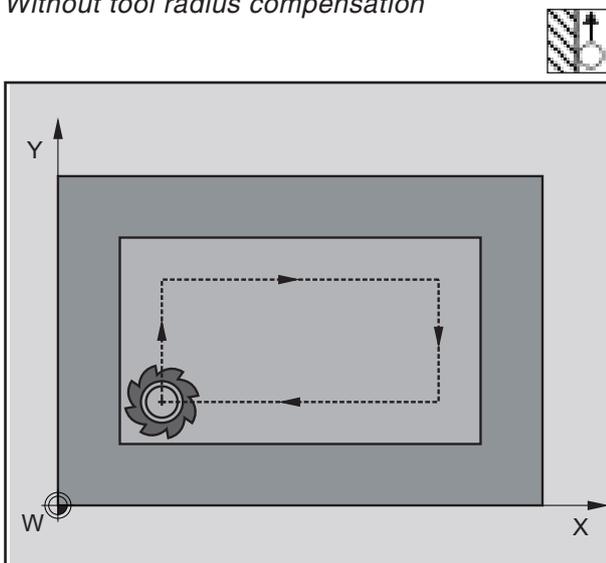
If tool radius compensation is switched off, the tool traces the contour's central track.



Without tool radius compensation

### Right-hand tool radius compensation

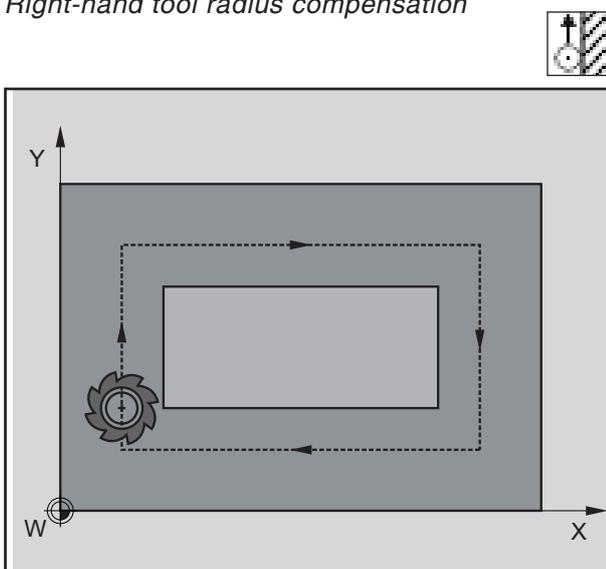
With right-hand tool radius compensation, the control system automatically calculates the equidistant tool distance right of the contour in each case for the various tools.



Right-hand tool radius compensation

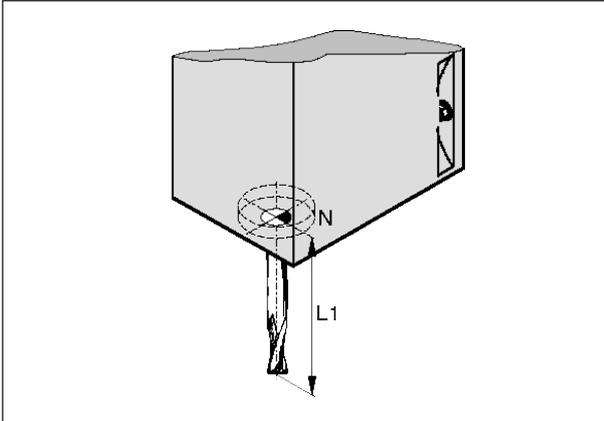
### Left-hand tool radius compensation

With left-hand tool radius compensation, the control system automatically calculates the equidistant tool distance left of the contour in each case for the various tools.



Left-hand tool radius compensation

## Tool data



*Tool length*

The object of tool data capture is that the software uses the tool tip or the tool centre point and not the tool housing reference point for positioning.

Each tool used for machining must be measured. The distance from the cutting tip to the tool housing reference point must also be determined.

The measured lengths and mill radius can be stored in the tool list.

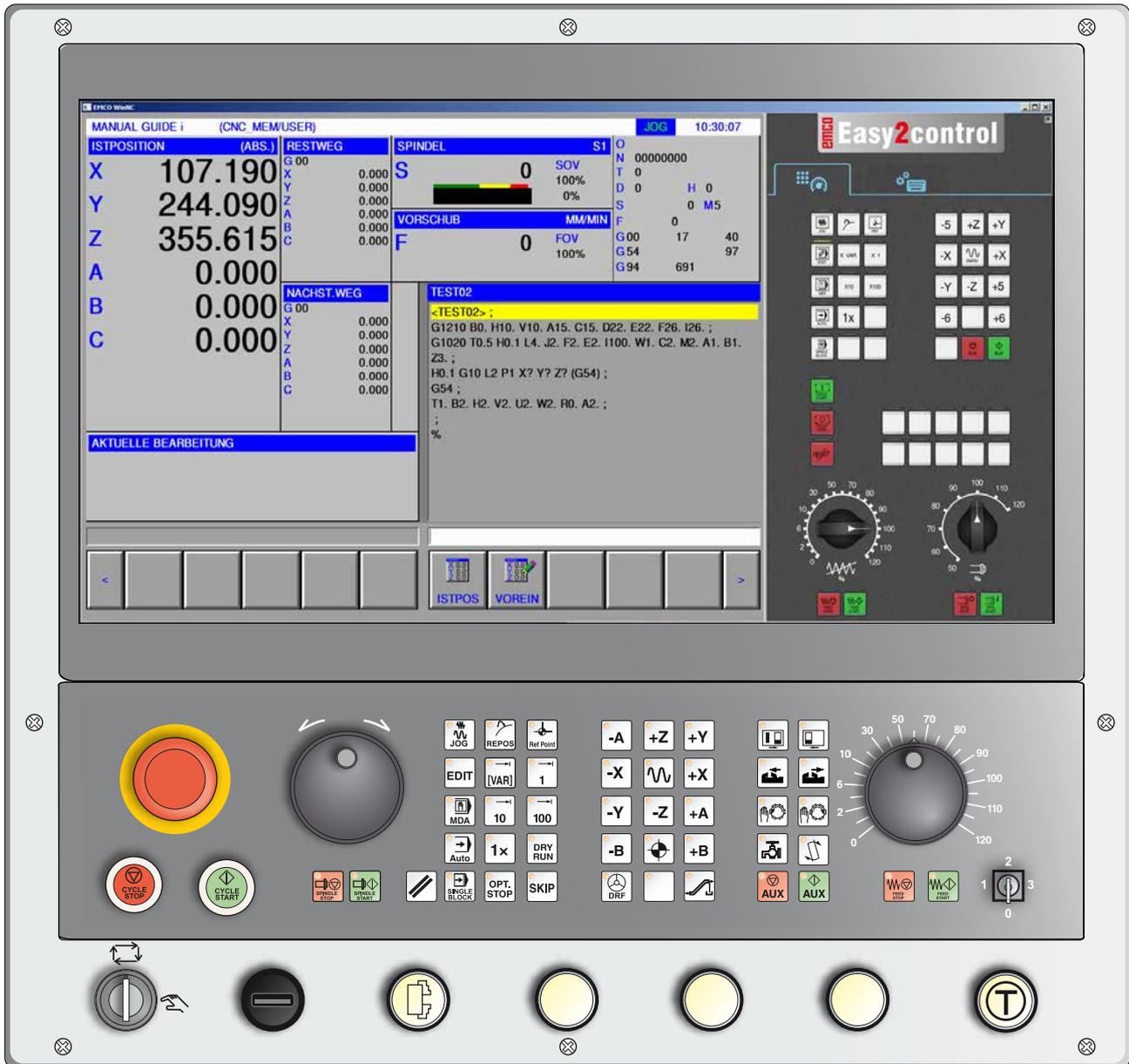
The mill radius information is **only** necessary if a **mill radius compensation** or a milling cycle is selected for the relevant tool!

(See chapter F Tool Management)



# B: Key description

## EMCO WinNC for Fanuc 31i control panel



**Instructions:**  
 Depending on the machine you are operating with EMCO WinNC for Fanuc 31i, not all functions and machine keys may be available.

## Address and number pad

The Shift key switches to the second key function (shown in the top left corner of the button).

Example:

 Q

  Queries



## Button functions

	End Of Block.
	Delete input
	Delete alarm messages, reset CNC (e.g. interrupt program).
	Call context-sensitive help.
	Alphanumeric input.
	Shift key
	Replaces the marked text with the text from the input textbox.
	Insert the text from the input textbox after the cursor.
	Delete (program, block, word).
	Enter word, accept data.
 	Scroll backwards/forwards
 	Cursor left/right.
 	Cursor up/down.
	Indicates the current position.
	Program functions
	Setting and indication of the zero point shift, the tool offsets, wear offsets and variables.
 	Not populated.



Parameter settings and indications, and troubleshooting indications.



Alarm and message display.



Switch to Manual Guide mode.

# Key description ISO functions



Actual position display

ACTUAL POSITION TEST **BOHREN\_G1101 N00000**

ABSOLUTE			MACHINE		DISTANCE TO GO	
X		0.000	X	107.190	X	0.000
Y		44.090	Y	244.090	Y	0.000
Z		55.615	Z	355.615	Z	0.000
A		0.000	A	0.000	A	0.000
B		0.000	B	0.000	B	0.000

MODAL			F	
G0	G15	F 0 M 5	F	0 MM/MIN
G17	G98	H 0	S	0/MIN
G90		D 0	SOV	100% SLM 0%
		T 0	DRY RUN F	7200 MM/MIN
G94	G97	S 0		
G71	G54			
G40	G61			
G49	G69			

A-

JOG \*\*\*\* \* 07:59:19

ABSOLUTE RELATIVE ALL

1 2 3

Actual position

- 1 Absolute position
- 2 Relative position
- 3 Indicate both positions together



Release overview display

ACTUAL POSITION TEST **BOHREN\_G1101 N00000**

ABSOLUTE			F	
X		0.000	F	0 MM/MIN
Y		44.090		
Z		55.615		
A		0.000		
B		0.000		

MODAL			VERSIONLIST	
G0	G15	F 0 M 5	name	act. Version
G17	G98	H 0	Version:	SERIES 31i G41Z-07.0
G90		D 0	WinNC Control:	1.02
		T 0	AC:	10.20
G94	G97	S 0	PLC:	
G71	G54		MachineCvt:	2.08.0002
G40	G61		Keyboard:	
G49	G69		Easy2control:	
			3DView:	14.40.0001
			DNC:	

S 0 OV 100 LM 0

A-

JOG \*\*\*\* \* 08:03:29

ABSOLUTE RELATIVE ALL

VERSION

Release overview

Displays the current WinNC software release



Alarm and message display.



*Alarm and report overview*

Displays all alarms and reports

# KEY DESCRIPTION



Zero point offset, tool correction and customer macro variables setting and display

ACTUAL POSITION TEST **BOHREN\_G1101 N00000**

ABSOLUTE **F** 0 MM/MIN

X 0.000  
Y 0.000  
Z 4.000  
A 0.000  
B 0.000

MODAL  
G0 G15 F 0 M 5  
G17 G98 H 0  
G90 D 0  
T 0  
G94 G97 S 0  
G71 G54  
G40 G61  
G49 G69

S 0 OV 100 LM 0

WORK COORDINATES (G54)

NO.	DATA	NO.	DATA
000 X	7.190	001 X	100.000
EXT Y	0.000	G54 Y	200.000
Z	0.000	Z	300.000
A	0.000	A	0.000
B	0.000	B	0.000
C	0.000	C	0.000

MEM \*\*\*\* \* ALM 08:13:00

ABSOLUTE RELATIVE ALL OFFSET WORK (OPRT) +

< ABSOLUTE RELATIVE ALL NO.SRH MEASUR +INPUT INPUT +

## Zero point offset

- Use the "OPRT" softkey and the "+" expansion button to display the file input and output, data entry and measure and search softkeys.
- The data is saved to the file EXT\_WKZ.TXT.
- The path to save and read the data is set in EMConfig under the "Directory exchange" heading.

OFFSET TEST **BOHREN\_G1101 N00000**

NO.	(LENGTH)		(RADIUS)	
	GEOM	WEAR	GEOM	WEAR
01	70.000	0.000	5.000	0.000
02	72.000	0.000	10.000	0.000
03	55.615	0.000	5.000	0.000
04	0.000	0.000	0.000	0.000
05	0.000	0.000	0.000	0.000
06	0.000	0.000	0.000	0.000
07	0.000	0.000	0.000	0.000
08	0.000	0.000	0.000	0.000
09	0.000	0.000	0.000	0.000
10	0.000	0.000	0.000	0.000
11	0.000	0.000	0.000	0.000
12	0.000	0.000	0.000	0.000
13	0.000	0.000	0.000	0.000
14	0.000	0.000	0.000	0.000
15	0.000	0.000	0.000	0.000
16	0.000	0.000	0.000	0.000
17	0.000	0.000	0.000	0.000
18	0.000	0.000	0.000	0.000
19	0.000	0.000	0.000	0.000

ABSOLUTE  
X 0.000  
Y 0.000  
Z 4.000  
A 0.000  
B 0.000

RELATIVE  
X 0.000  
Y 0.000  
Z 4.000  
A 0.000  
B 0.000

MACHINE  
X 107.190  
Y 200.000  
Z 304.000  
A 0.000  
B 0.000

MEM \*\*\*\* \* ALM 08:16:23

OFFSET WORK (OPRT) +

## Tool correction

OFFSET TEST **BOHREN\_G1101 N00000**

NO.	(LENGTH)		(RADIUS)		Z
	GEOM	WEAR	GEOM	WEAR	
01	70.000	0.000	5.000	0.000	X
02	72.000	0.000	10.000	0.000	Y
03	55.615	0.000	5.000	0.000	Z
04	0.000	0.000	0.000	0.000	A
05	0.000	0.000	0.000	0.000	B
06	0.000	0.000	0.000	0.000	
07	0.000	0.000	0.000	0.000	
08	0.000	0.000	0.000	0.000	
09	0.000	0.000	0.000	0.000	
10	0.000	0.000	0.000	0.000	
11	0.000	0.000	0.000	0.000	
12	0.000	0.000	0.000	0.000	
13	0.000	0.000	0.000	0.000	
14	0.000	0.000	0.000	0.000	
15	0.000	0.000	0.000	0.000	
16	0.000	0.000	0.000	0.000	
17	0.000	0.000	0.000	0.000	
18	0.000	0.000	0.000	0.000	
19	0.000	0.000	0.000	0.000	

ABSOLUTE	
X	0.000
Y	0.000
Z	4.000
A	0.000
B	0.000

RELATIVE	
X	0.000
Y	0.000
Z	4.000
A	0.000
B	0.000

MACHINE	
X	107.190
Y	200.000
Z	304.000
A	0.000
B	0.000

A->

NO.SRH	MEASUR	INP.C.	INPUT	ERASE	MEM	ALM	08:19:11
<					MEM	ALM	08:19:11

- 1 Search for tool number
- 2 Measure tool
- 3 Coordinates entry
- 4 Computes current value + entry from entry line
- 5 Accept value from entry line
- 6 Delete
- 7 Tool length data
- 8 Tool radius data



## Screen layout Manual Guide *i*

MANUAL GUIDE *i* (CNC MEM/USER) STR ALM MEM 13:42:59

ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE		S1	
X	107.190	G 00		S	6640	SOV	
Y	244.090	X	0.000			100%	
Z	355.615	Y	0.000			0%	
A	0.000	Z	0.000	FEED		MM/MIN	
B	0.000	A	0.000	F	0	FOV	
C	0.000	B	0.000			100%	
		C	0.000				

BOHREN G1101 8

9 10 11 12 13 14

15

16 6

17

18 3514 WRONG FIGURE DATA

19

20

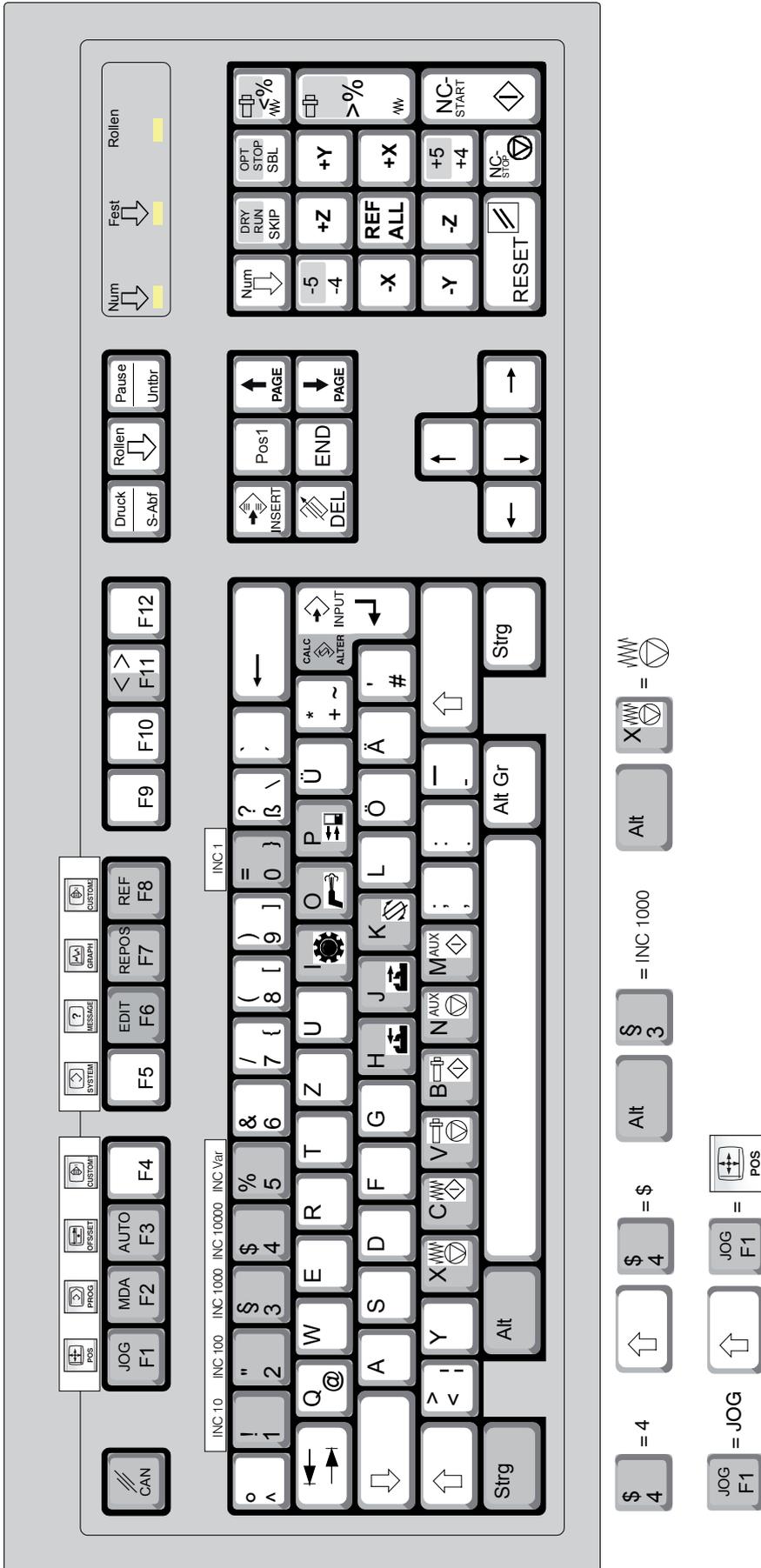
- |                   |  |
|-------------------|--|
| 1 Mode            | 11 Spindle speed                             |
| 2 Alarm status    | 12 M-commands                                |
| 3 Program mode    | 13 Feed display                              |
| 4 Axis position   | 14 Display of active G-functions             |
| 5 Residual travel | 15 Program window                            |
| 6 Spindle speed   | 16 Current line number in ISO program window |
| 7 Feed            | 17 Graphical simulation                      |
| 8 Program name    | 18 Message window                            |
| 9 Block number    | 19 Keypad buffer                             |
| 10 Tool number    | 20 Softkey list                              |

### Instructions:

For axis position and residual travel:  
The number of axes varies according to the configuration of the machine.

For detailed descriptions see chapter "C Operation"

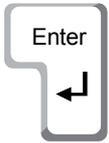
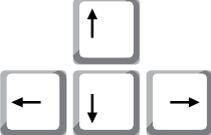
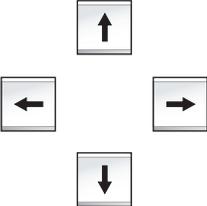
# PC keyboard



To activate patterned key functions, the CTRL or ALT keys must be pressed at the same time.

**Instructions:**  
The machine functions in the numerical key-pad are only active if NUM Lock is not active.

## Overview button assignment control keyboard

PC button	Control system button	Check
		Delete input
		Conclude input and continue dialog.
		Move mark
		Upper/lower case
		Single block (SBL)
		Skip (block mask)
		Reset key (Reset)
		Dry run (test run feed)
		Optional stop
		Indicates the current position
		Program functions
		Setting and indication of the zero point shift, tool offsets, wear offsets and variables
		Not populated.
		Parameter settings and indications, and troubleshooting indications.
		Alarm and message display.
		Manual Guide mode
		Not populated.
		Call context-sensitive help

## Overview Button Assignment Machine Operating Elements

PC button	Control elements	Check
Alt I		Swivel divider
Alt O		Coolant / Purge on / off
Alt P		Door open / closed
Alt H		Clamp device closed
Alt J		Clamp device open
Alt K		Swivel tool holder
Alt X		Feed Stop
Alt C		Feed Start
Alt V		Spindle Stop
Alt B		Spindle Start
Alt N		Switch auxiliary drives on AUX ON
Alt M		Switch auxiliary drive off AUX OFF
Enter		NC start
,		NC stop
5		Approach reference point

### Instructions:

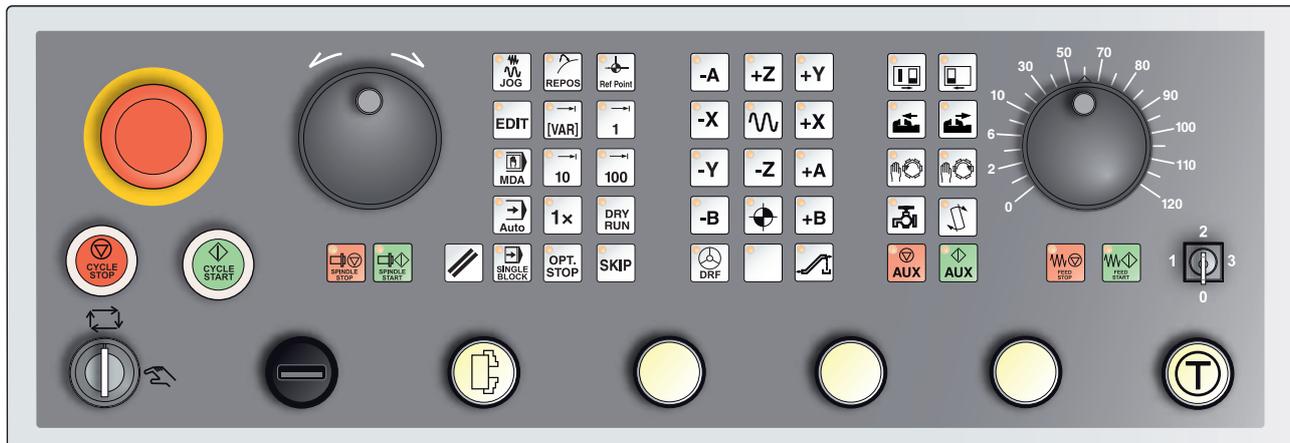
Selecting the machine buttons via the PC keyboard:

- 1.) Hold "Alt" button down.
- 2.) Press and then release machine button.
- 3.) Release "Alt" button.



PC button	Control elements	Check
 		Spindle speed correction
 		Override (feed rate override)

## Machine control panel



Depending on machine configuration, the control panel can differ slightly from what is shown here.

## Key description

### Skip (block mask)



In Skip mode, program blocks that are marked with an oblique stroke "/" in front of the block number are skipped during the program run (e.g.: /N100).

Active when the LED is ON.

### Dry run (test run feed)



In Dry run mode, positioning movements will be carried out with the feed value stipulated in the "Dry run feed" setting datum.

The dry-run feed works instead of the programmed movement commands.

Spindle commands are ignored.

Active when the LED is ON.



#### Caution:

The dry run feed is higher than the programmed feed. Ensure that no workpiece is clamped before starting dry run mode.

When machining parts, ensure that dry run mode is switched off before you start the machine (the LED in the button is OFF).

## Individual piece mode



This button makes individual piece mode or constant operation in conjunction with automatic loading equipment available for selection. Individual piece state is the default state when switched on. Active individual piece mode is indicated by illumination of the corresponding LED on the machine control panel.

## Optional stop



When this function is active (key held down), program processing will stop at blocks in which the M01 special function has been programmed.

You start processing again with the NC Start button.

If the function is not active, the M01 special function (from the parts program) will be ignored.

## Edit



Toggle into Edit mode.

## Handwheel mode (optional)



This key enables and disables the connected handwheel (optional).

## Reset key (Reset)



Pressing the Reset key:

interrupts the machining of the current parts program.

- deletes monitoring messages unless these are Power On or Recall alarms.
- Moves the channel into "Reset" status; this means:
  - The NC controller remains in synchronism with the machine.
  - All interim and work stores are deleted (but the content of the parts program memory is retained).
  - The control system is in the default position and ready for a new program sequence.

## Feed Stop



This button interrupts a programmed carriage movement.

## Feed Start



This button continues a programmed but interrupted carriage movement.

If the main spindle motion was also interrupted, this must be switched on again first.

## Single block

This functions allows you to run a parts program block by block. You can enable the Individual Block feature in Automatic mode.



### When single block processing is active:

- the current block of the parts program is only machined when you press the NC Start key.
- machining stops after a block is executed.
- the following block is executed by pressing the NC Start key again. The function can be deselected by pressing the Single Block key again.

## Cycle stop



After pressing the Cycle stop button after the controller has taken over the function, machining of the current parts program is suspended

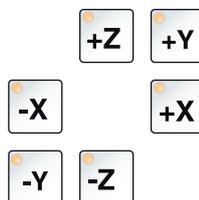
Machining can then be continued by pressing the Cycle start key.

## Cycle start



After pressing the Cycle start key, the selected parts program will be started with the current block.

## Arrow keys



Use these buttons to move the NC axes in JOG mode.

Depending on the machine configuration, various arrow keys are available

## Rapid traverse



If this button is pressed in addition to the arrow keys, the axes concerned move in rapid traverse.

## Reference point



Pressing this button causes the reference points to be approached in the spindle and tool turret axes.

## Chip conveyor (Option)



Switch on chip conveyor:

Forwards: Hold the key down for less than 1 second.

Backwards: Hold the key down for longer than 1 second.

The chip conveyor will be switched off after a defined time (approx. 35 seconds).

This value is set in the factory.

## Swing tool drum



Pressing this button causes the tool drum to swivel by one position:  
Cycle in the clockwise direction (one position further)



Cycle in the counterclockwise direction (one position back)

### Preconditions:

- Machine doors closed
- "JOG" operating mode
- Key switch in "Hand" position

## Manual tool change



Pressing this button starts a manual tool change.  
The tool clamped in the milling spindle will be removed and replaced with the tool from the currently swivelled-in tool drum.

### Preconditions:

- Machine doors closed
- "JOG" operating mode
- Key switch in "Hand" position

### Information:

- Interrupt the change process by moving the override switch below 4%.
- Cancellation of the change procedure by pressing the reset button.

## Clamping devices



These functions activate the clamping device.

## Coolant



This function switches the coolant equipment on or off.

## Feed Stop



This button interrupts a programmed carriage movement.

## Feed Start



This button continues a programmed but interrupted carriage movement.

If the main spindle motion was also interrupted, this must be switched on again first.

## Types of operation

### JOG



Standard movement of the machine by continuous movement of the axes via the arrow keys or by incremental movement of the axes via the arrow keys or the hand wheel.

### MDA - Manual Data Automatic



Control the machine by executing a block or a sequence of blocks. Blocks are entered via the control panel.

### Automatic



Control the machine by automatically executing programs.

### REF - Reference mode



Approaching the reference point (Ref) in the JOG operating mode.

### Inc 1 - Incremental feed



Move step by step a predefined distance of 1 increment in handwheel/jog mode.

Metrical measurement system: Inc 1 corresponds to 1µm  
Imperial (inch-based) measurement system: Inc 1 equates to 0.1 µinch

### Inc 10 - Incremental feed



Move step by step a predefined distance of 10 increments.

Metrical measurement system: Inc 10 corresponds to 10µm  
Imperial (inch-based) measurement system: Inc 10 equates to 1 µinch

### Inc 100 - Incremental feed



Move step by step a predefined distance of 100 increments.

Metrical measurement system: Inc 100 corresponds to 100µm  
Imperial (inch-based) measurement system: Inc 100 equates to 10 µinch

### Inc [VAR]



Move in steps with a continuously variable increment.

### REPOS - Repositioning



Back-positioning, approach contour again in the JOG operating mode

**Information:**

- The operating modes can be selected via softkeys (PC keyboard) or with the operating mode selector switch = multifunction switch.
- Switching between the metrical measurement system and the imperial (inch-based) measurement system is carried out with the EmConfig utility software (see Chapter X EmConfig).
- Metric is assigned in the imperial measuring system as follows:

**Feed:**

Millimeters in inches:  
 mm/min => inches/min  
 mm/U => inches/U

**Constant cutting speed:**

Meters in feet:  
 m/min => feet/min

## Auxiliary OFF



This button switches off the machine's auxiliary units. Only effective if spindle and program are off.

## Auxiliary ON



This button makes the machine's auxiliary units ready for operation (hydraulics, feed drives, spindle drives, chip conveyor lubricant, coolant).

The button must be pressed for around 1 second.

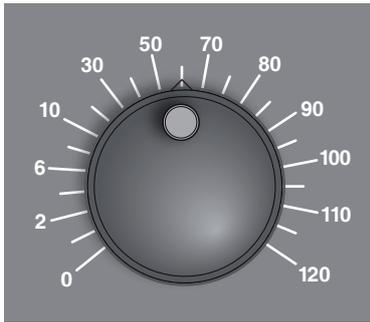
Tapping the AUX ON key is a clear function and causes the central lubrication system to perform a lubrication squirt.

## Freewheeling before referencing

If the carriage has to be freewheeled before referencing (e.g. from a position where it is at risk of colliding), press the button  and then the corresponding arrow key.

## Free swivelling of the tool turret

If the tool turret has to be swivelled freely after an alarm is present, press the  buttons and then .



### Override switch (feed rate override)

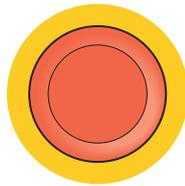
The rotary switch with notch positions enables you to change the programmed feed value F (corresponds to 100%). The set feed value F in % will be shown on the screen.

Adjustment range:

0% to 120% of the programmed feed.

In rapid traverse 100% will not be exceeded.

No effect with thread commands G33, G63



### EMERGENCY SHUTDOWN

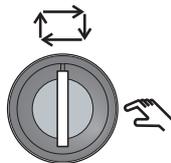
Press the red button only in an emergency.

Effects:

As a rule, the EMERGENCY SHUTDOWN button will lead to all drives being stopped with the greatest possible braking torque.

To continue working, press the following keys:

RESET, AUX ON, doors OPEN and CLOSED.



### Key Switch Special Operations Mode

The key switch can be switched to the "AUTOMATIC" or "SET-UP" (hand) position.

With this key switch it is possible to perform movements in Jog Mode when the sliding door is open.



#### Danger:

When Special Operations mode is active, the danger of accidents is increased.

The key for this switch should only be held by persons who have the required knowledge about the dangers and exercise appropriate care.

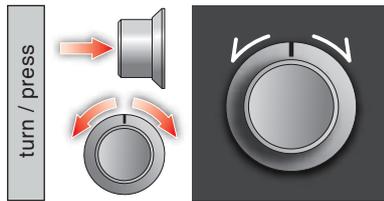
Keep the chip guard door closed even in Set-up mode.

Keys should only be used by authorized persons.

After work is carried out in Special Operations mode, always withdraw key (accident danger).

Observe country-specific safety instructions (e.g.: SUVA, BG, UVV ...).

## Multifunction switch for operating modes

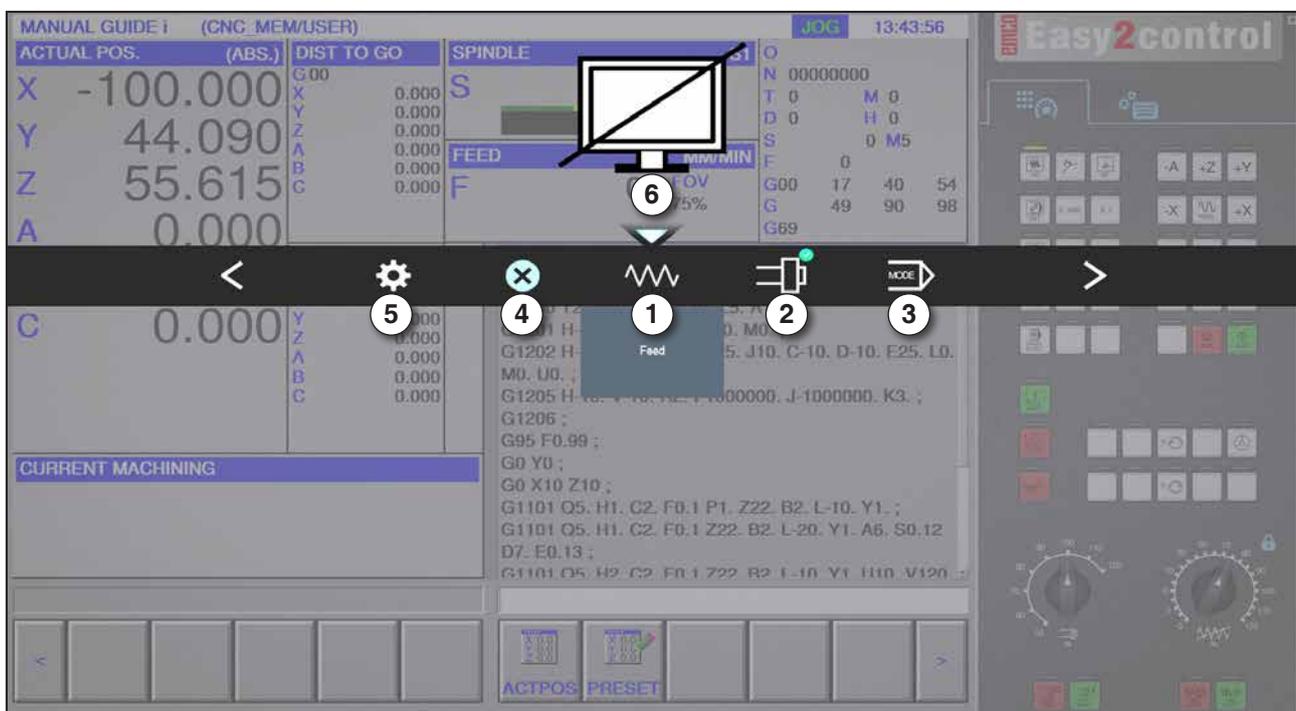


The multi-function switch is designed as a rotary switch with a press feature.

### Populated function

- The user interface is opened by pressing the multifunction operation. The active function is indicated by a green check-box.
- Turning the switch allows you to switch between the functions. The black bar with the symbols moves to the left or to the right.
- Activating a function or a change to a sub-menu is executed by pressing the button.

The interface offers the following functions:



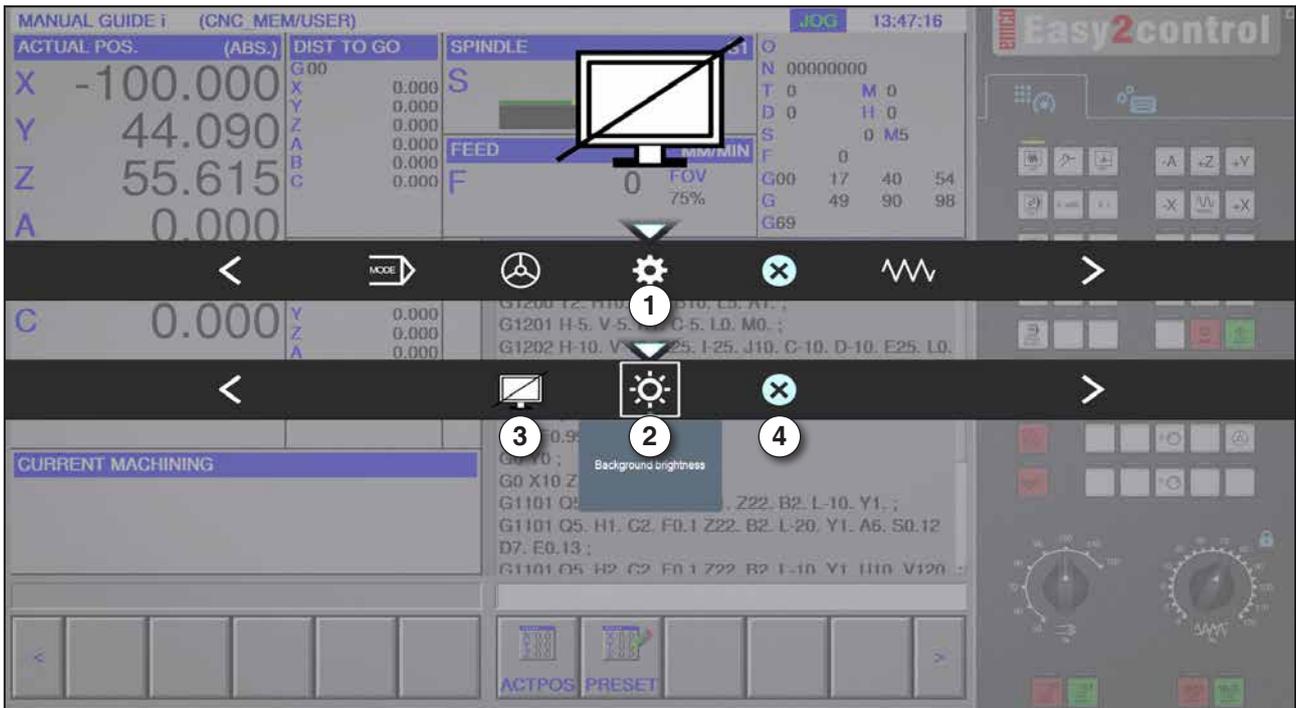
### Overview

- |  |   |
|--|---|
| 1 Feed override: controls the feed rate equivalent to conventional feed override           | 4 Close: The user interface is closed. The menu disappears, return to the control surface |
| 2 Spindle override: controls the spindle speed equivalent to conventional spindle override | 5 Settings: opens another level with settings   |
| 3 Modes: allows you to select the operating mode using the multifunction operation         | 6 Cursor: shows the actual position in the menu   |

### Note:

The functionality of the multifunction operation is depending on the installed software version.





Settings the background brightness

1 Settings

2 Background brightness: adjusts the transparency of the background

3 Lock screen: pressing again unlocks the screen.

4 Close: closes the submenu and returns to a higher-level menu.

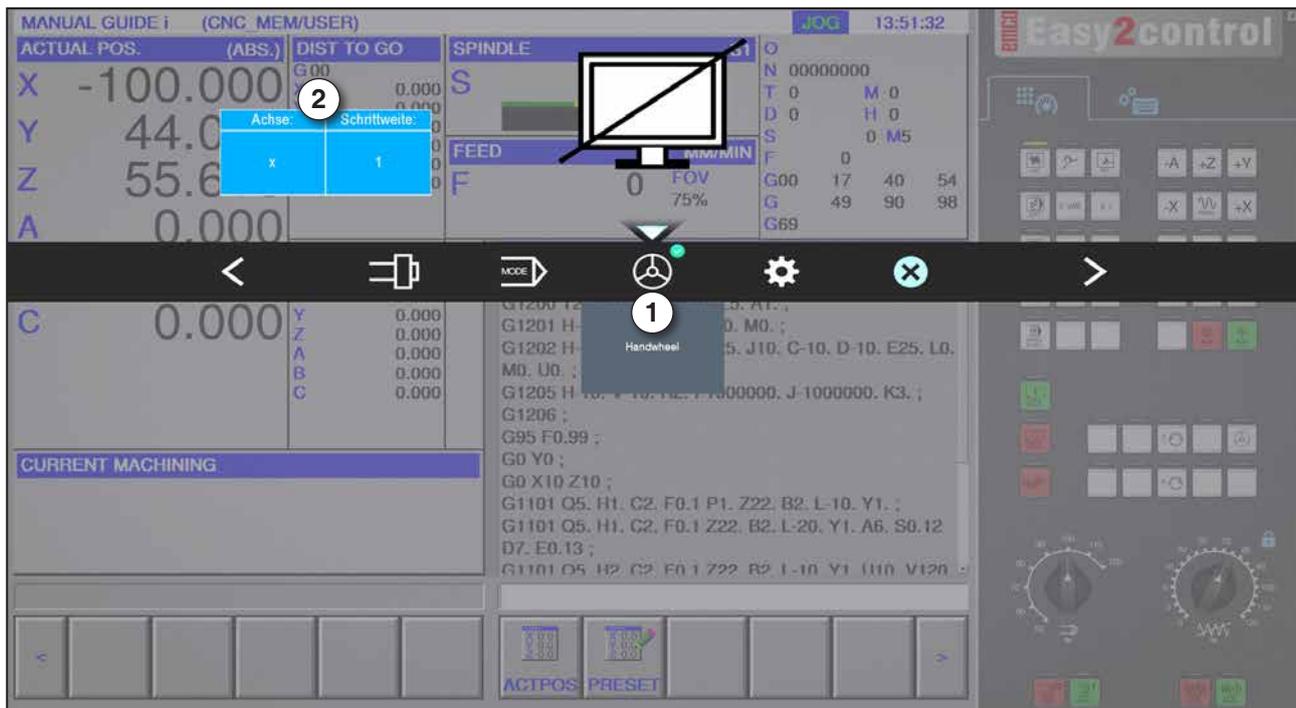
### Setting the background transparency



- By pressing once, a white border around the icon appears. The menu item is activated.



- Now, the transparency of the background can be changed by turning the rotary switch:  
Turn to the left: brighter  
Turn to the right: darker
- Pressing again closes the menu and the white border disappears.



Handwheel function

The handwheel (1) activates the handwheel mode. The parameters for axis and step width (2) are set with the axis- and operating -mode buttons on the machine keyboard.

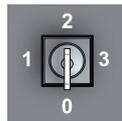
### Operation

- The electronic handwheel is used to traverse the slides at a defined step width.
- The step width depends on the selected Inc mode: Inc 1, Inc 10, Inc 100.
- There must be one pre-selected Inc mode and an axis defined by a direction key.
- Also refer to "types of operation" und "arrow keys" in chapter B.

### Note:

In the mode "Inc 1000" the slides cannot be moved with the handwheel. "Inc 1000" operates with "Inc 100".





### Key switch

The key switch function is machine-specific.



### Additional clamping device button

The additional button has the same function as on the machine control panel.  
(Double movement because of better operation).



### USB connection (USB 2.0)

Data is exchanged with the machine (data copying, software installation) via this USB connection.



### Enable button

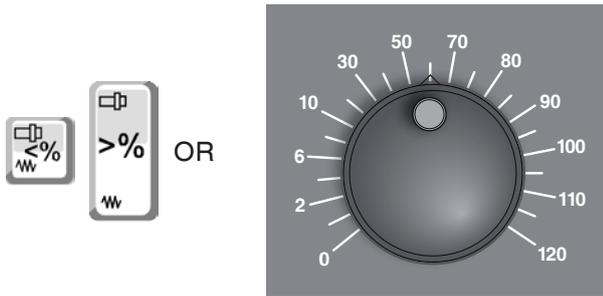
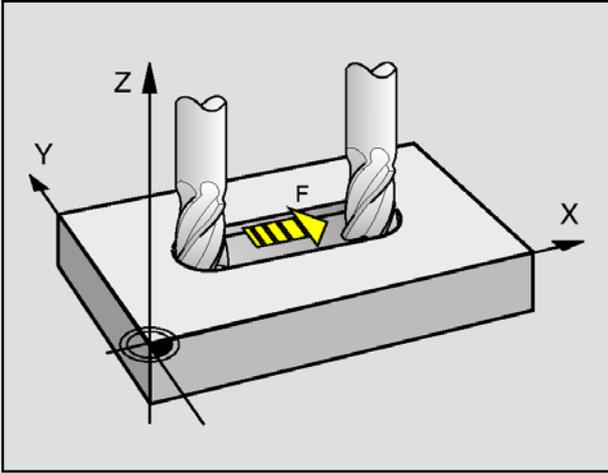
When the door is open, axis movements via the arrow keys and tool turret movements are authorized by pressing the enable button (precondition key switch in SET-UP position).  
In machines with automated doors (option) pressing the enable switch opens the machine doors.



# C: Operation

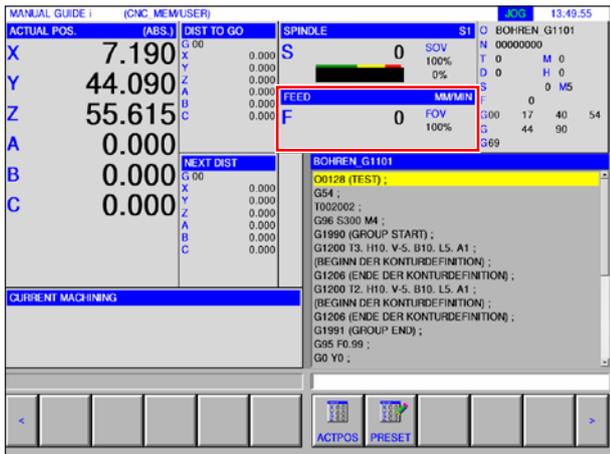
## Feed F [mm/min]

The feed F is the speed in mm/min (feet/min) with which the tool centre point moves on its path. The maximum feed can be different for each machine axis and is defined by machine parameters.

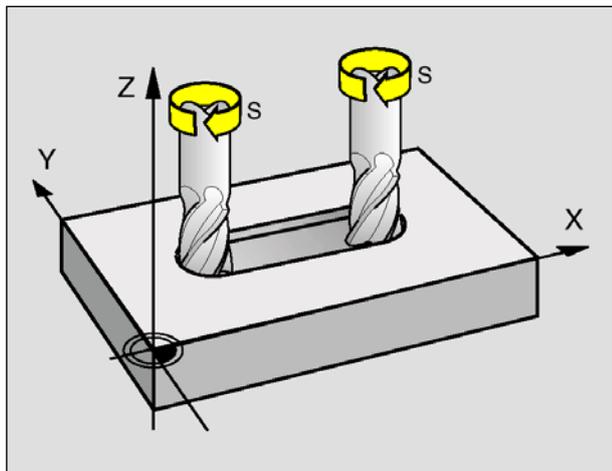


### Feed influence

The feed value F programmed by you corresponds to 100%. With this button or with the feed override the set feed value F can be changed in %.

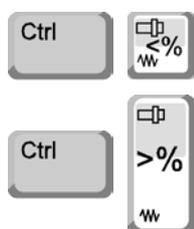


Adjustment range:  
0% to 120% of the programmed feed.  
Only the changed percentage and not the resulting effective value will be shown.  
In rapid traverse 100% of the maximum rapid traverse feed will not be exceeded.



## Spindle speed S [rev/min]

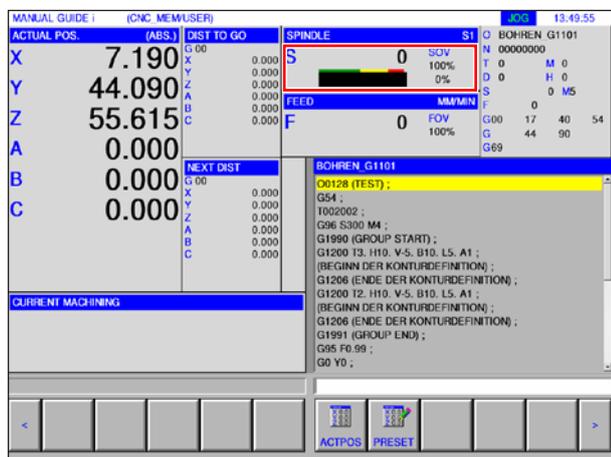
Indicate the spindle speed S in revolutions per minute (rpm).



### Spindle speed correction

The spindle speed S programmed by you corresponds to 100%.

These button combinations or the spindle speed override changes the set spindle speed S in %.



Adjustment range:

0% to 120% of the programmed spindle speed.

Only the changed percentage and not the resulting effective value will be shown.

## Types of operation



### **JOG** - Jogging

Standard movement of the machine by continuous movement of the axes via the arrow keys or by incremental movement of the axes via the arrow keys or the hand wheel.

JOG is used in manual mode as well as for set-up of the machine.



### **MDA** - Semi-automatic mode (Manual Data Automatic)

Controlling the machine by executing a block or a sequence of blocks. Block input is performed via the operating panel or the PC keyboard.



### **AUTO** - Automatic mode

Controlling the machine by automatically executing programs.

Here part programs are selected, started, adjusted, deliberately influenced (e.g. individual set) and executed.



### **REF** - Reference mode

Approaching the reference point (Ref) in the JOG operating mode.



### **REPOS** - Repositioning

Back-positioning, approach contour again in the JOG operating mode



#### **Instructions:**

The modes can be selected via softkeys (PC keyboard) or with the mode selector switch (multifunction switch).

**Inc 1 - Incremental feed**

Move step by step a predefined distance of 1 increment in handwheel/jog mode.

Metrical measurement system: Inc 1 equals 1 $\mu$ m

Imperial (inch-based) measurement system: Inc 1 equals 0.1  $\mu$ inch

**Inc 10 - Incremental feed**

Move step by step a predefined distance of 10 increments in handwheel/jog mode.

Metrical measurement system: Inc 10 equals 10 $\mu$ m

Imperial (inch-based) measurement system: Inc 10 equals 1  $\mu$ inch

**Inc 100 - Incremental feed**

Move step by step a predefined distance of 100 increments in handwheel/jog mode.

Metrical measurement system: Inc 100 equals 100 $\mu$ m

Imperial (inch-based) measurement system: Inc 100 equals 10  $\mu$ inch

**Inc 1000 - Incremental feed**

Move step by step a predefined distance of 200 increments in handwheel mode, or 1000 increments in jog mode.

Metrical measurement system: Inc 1000 equals 1000 $\mu$ m

Imperial (inch-based) measurement system: Inc 1000 equals 100  $\mu$ inch

**Instructions:**

Metric is assigned in the imperial measuring system as follows:

**Feed:**

Millimetres in inches:  
mm/min => inches/min  
mm/U => inches/U

**Constant cutting speed:**

Meters in feet:  
m/min => feet/min



## Approach reference point

The reference point R is a specified fixed point on the machine. It is used to calibrate the measurement system.



The reference point must be approached each time after the machine is switched on or the EMERGENCY STOP button is unlocked, in order to make the exact distance between the machine zero point M and the tool fixture reference point N or T known to the control system.

- Switch to the reference mode REF.



### Option A:

#### Reference the axes individually

Press the +Z and +X buttons.

The slides move one another to their reference points, after each has reached the collision-free area.

#### Instructions:

- After reaching the reference points the software limit switches are active. The reference point position will be shown as the current position on the screen.
- The tailstock (if available) must be at the right end of the bed when referencing the axes, so that the Z-slides do not collide with the tailstock.



### Option B:

#### Reference automatically

Pressing the button "Reference point" causes the axes to approach their reference points automatically one after another. First the axes then the tool changer will be referenced.

## Move slides manually

The machine axes are moved manually using the arrow keys.



- Switch to JOG mode.

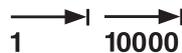


- The arrow keys buttons move the axes in the corresponding direction as long as the key is pressed.
- The feed speed is set with the override switch.



- If the button is pressed simultaneously, the carriages move in rapid traverse.

## Move slides in step-movement

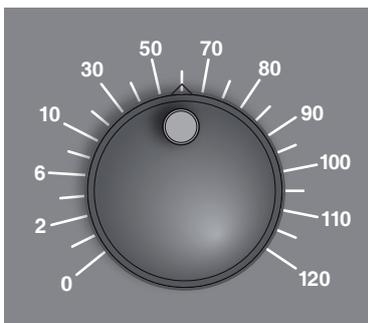


The machine axes can be moved in steps using the direction buttons.

- Switch to INC mode.



- The direction buttons move the axes in the corresponding direction by the set step distance each time the button is pressed.



- The feed speed is set with the override switch.



- If the button is pressed simultaneously, the carriages move in rapid traverse.



**MDA mode** - Semi-automatic mode

Control the machine by executing a block or a sequence of blocks. For this the desired movements can be entered via the control keyboard in the form of part program sets.



The control system executes the blocks entered after the control button is pressed.

For running an MDA program, the same preconditions are necessary as for fully automatic mode.



**AUTO mode** - Automatic mode

Controlling the machine by automatically executing programs.

Here part programs are selected, started, adjusted, deliberately influenced (e.g. individual set) and executed.

Preconditions for executing part programs:

- The reference point was approached
- The part program is loaded into the control system.
- The correction values required were checked or entered (e.g. zero point offsets, tool corrections)
- The safety locks are activated (e.g. chip-guard doors closed).

Options in Automatic mode:

- Program correction
- Block search run
- Overstore
- Program influence

(see Section G program sequence)

# Program management



- Select "Edit" mode.

ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE		S1		
X	7.190	G 00		S	0	SOV	100%	
Y	44.090	X	0.000				0%	
Z	55.615	Y	0.000	FEED		MM/MIN		
A	0.000	Z	0.000	F	0	FOV	100%	
B	0.000	A	0.000					O BOHREN_G1101
C	0.000	B	0.000					N 00000000
								T 0 M 0
								D 0 H 0
								S 0 M5
								F 0
								G00 17 40 54
								G 44 90
								G69

**BOHREN\_G1101**

O0128 (TEST) ;

G54 ;

T002002 ;

G96 S300 M4 ;

G1990 (GROUP START) ;

G1200 T3. H10. V-5. B10. L5. A1 ;

(BEGINN DER KONTURDEFINITION) ;

G1206 (ENDE DER KONTURDEFINITION) ;

G1200 T2. H10. V-5. B10. L5. A1 ;

(BEGINN DER KONTURDEFINITION) ;

G1206 (ENDE DER KONTURDEFINITION) ;

G1991 (GROUP END) ;

G95 F0.99 ;

G0 Y0 ;

O:PROGRAM NUMBER

<	NEWPRG	O LIST	SRCH↑	SRCH↓	O SRCH	COPY	CUT	DELETE	KEYPST	PASTE	>
10	1	2	3	4	5	6	7	8	9	10	

- |   |  |    |                                       |
|---|--|----|---------------------------------------|
| 1 | Creating a new machining program                   | 6  | Move marked text into a buffer file   |
| 2 | Call program listing                               | 7  | Delete marked text                    |
| 3 | Search for text from the input line in the program | 8  | Insert marked text into an input line |
| 4 | Search for and open the program                    | 9  | Insert text from a buffer file        |
| 5 | Copy marked text into a buffer file                | 10 | Forward and Back (extension keys)     |

### Instructions:

Alternatively, the buttons on the softkey toolbar can also be operated with the function keys F1 to F10, e.g.: NEUPRG = F1, EINFUE = F10

## Create program

A program consists of a succession of cycles, commands and/or sub-routines.



1 Select "Edit" mode.

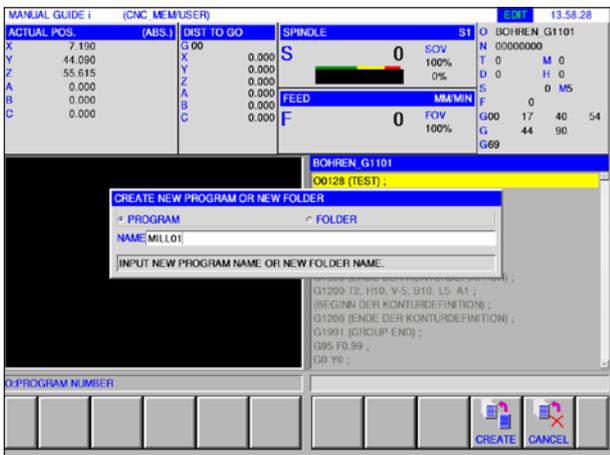


2 Press softkey.



3 Enter program or folder name and confirm with "CREATE".

If a program name was allotted previously, an appropriate message will be displayed.



The Fanuc standard format for program names is as follows:

O1234. If fewer than 4 numbers are entered, those missing are populated with zeroes.

### Instructions:

Program names can consist of at least 1 and a maximum of 32 characters. The following characters are allowed: "Zz", "0 to 9", "\_" "+" and ".".

## Where programs are saved

Programs can be saved in and invoked from the control system's program directory, a local drive or a USB data storage device.

Back folder on top:  
goes back one level



Program control path:  
C:\WinNC32\_Fanuc\Fanuc\_i.M\PRG\LIBRARY\

## Programming list



1 Select "Edit" mode.



2 Press softkey.

MANUAL GUIDE I (CNC MEM/USER) EDIT 14:07:18

ACTUAL POS. (ABS.)	DIST TO GO	SPINDLE	S1	O
X 7.190	G 00	S	0	BOHREN_G1101
Y 44.090	X 0.000		SOV 100%	N 00000000
Z				T 0 M 0

PROGRAM LIST (//CNC\_MEM/USER/LIBRARY/)

NAME	COMMENT	MODIFIED DATE	SIZE(CHAR)
RETURN UPPER FOLDER		<FOLDER>	
BOHREN_G1101		<FOLDER>	
GEWINDEDREHEN_G1140		<FOLDER>	
ISO_DIAMONOF		<FOLDER>	
ISO_FEEDSPEED		<FOLDER>	
ISO_G54_T		<FOLDER>	
ISO_GENAUHALT		<FOLDER>	
ISO_KREIS		<FOLDER>	
ISO_M98		<FOLDER>	
ISO_MACRO		<FOLDER>	
ISO_MCALL		<FOLDER>	
KONTURDREHEN		<FOLDER>	
KONTURDREHEN_ISO		<FOLDER>	
LIBRARY		<FOLDER>	
SCHEIDEN_AUSSEN		<FOLDER>	

SELECT PROGRAM NAME.AND SELECT SOFT KEY.

O:PROGRAM NUMBER

<	NEW	COPY	DELETE	EDTCOM	SEARCH	M CARD	MLTDEL	SRTORD	OPEN	CLOSE	>
11	1	2	3	4	5	6	7	8	9	10	11

- |  |  |
|--|--|
| <ul style="list-style-type: none"> <li>1 Creating a new machining program</li> <li>2 Copy programs</li> <li>3 Delete programs</li> <li>4 Enter a comment for a program</li> <li>5 Search for programs</li> <li>6 Output of a program to an external memory card</li> </ul> | <ul style="list-style-type: none"> <li>7 Delete a number of programs</li> <li>8 Change the sorting sequence</li> <li>9 Open program</li> <li>10 End program listing</li> <li>11 Forward and Back (extension keys)</li> </ul> |
|--|--|

**Instructions:**

The program listing is also available in Automatic and in Simulations modes, albeit with restricted functionality.



MANUAL GUIDE i (CNC MEM/USER) EDIT 14:09:00

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	O BOHREN_G1101
X	7.190	G 00	S	0	N 00000000
Y	44.090	X 0.000		SOV	T 0 M 0
Z		0.000		100%	

**PROGRAM LIST (//CNC\_MEM/USER/LIBRARY/)**

NAME	COMMENT	MODIFIED DATE	SIZE(CHAR)
RETURN UPPER FOLDER		<FOLDER>	
BOHREN_G1101		<FOLDER>	
GEWINDEDREHEN_G1140		<FOLDER>	
ISO_DIAMONOF		<FOLDER>	
ISO_FEEDSPEED		<FOLDER>	
ISO_G54_T		<FOLDER>	
ISO_GENAUHALT		<FOLDER>	
ISO_KREIS		<FOLDER>	
ISO_M98		<FOLDER>	
ISO_MACRO		<FOLDER>	
ISO_MCALL		<FOLDER>	
KONTURDREHEN		<FOLDER>	
KONTURDREHEN_ISO		<FOLDER>	
LIBRARY		<FOLDER>	
CONTOURDREHEN_MISSEN		<FOLDER>	

SELECT PROGRAM NAME.AND SELECT SOFT KEY.

O:PROGRAM NUMBER

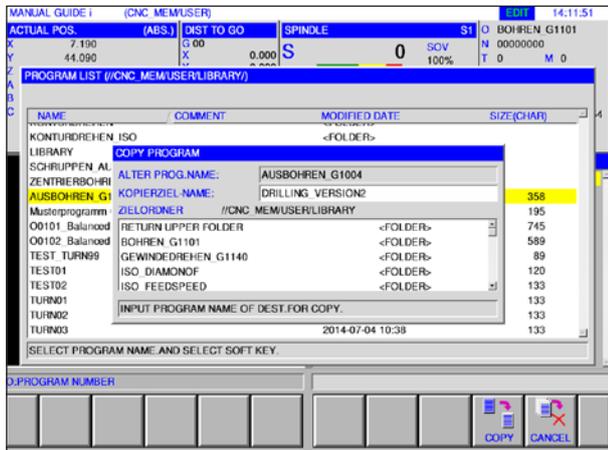
<	ABC		SEARCH	WRPROT		OPEN	CLOSE	>
1	RENAME		2	3		4	5	

- |   |  |
|---|--|
| <p>1 Rename program or folder names</p> <p>2 Program properties</p> <p>3 Program protection</p> | <p>4 Open program</p> <p>5 End program listing</p> |
|---|--|

## Copy program

- 1 Select "Edit" mode.
- 2 Move mark to the desired program.
- 3 Copy program.
- 4 This softkey calls the window for copying programs. After a program name has been entered for the program to be copied (target program name), press the "COPY" softkey to copy the given program.

EDIT



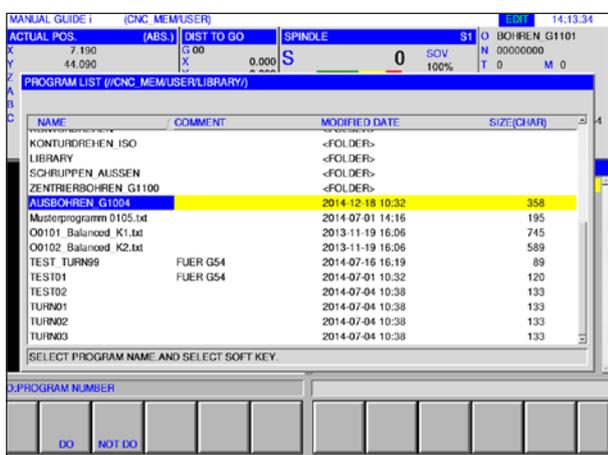
- 5 Press the softkey to copy.



## Delete program

- 1 Select "Edit" mode.
- 2 Move mark to the desired program.
- 3 Delete program.
- 4 This softkey calls the window for deleting programs. "DO" deletes the given program, "NOT DO" interrupts the delete process.
- 5 Press the softkey to delete.

EDIT



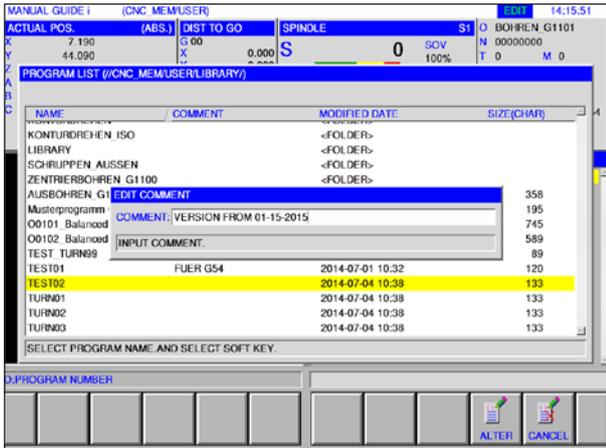
## Enter comment



- 1 Select "Edit" mode.
- 2 Move mark to the desired program.
- 3 Enter comment.



- 4 This softkey calls the comment entry window. After a comment has been entered for the program, press the "ALTER" softkey to add the given comment in the program.



- 5 Press the softkey to change.

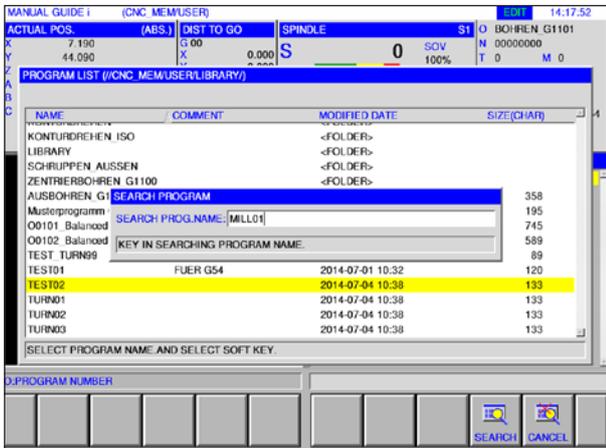
## Search program



- 1 Select "Edit" mode.



- 2 Search program.



- 3 This softkey calls the window for searching programs. After a program name has been entered for the program to be found (target program name), press the "SEARCH" softkey to look for the given program.

- 4 Press the softkey to search.

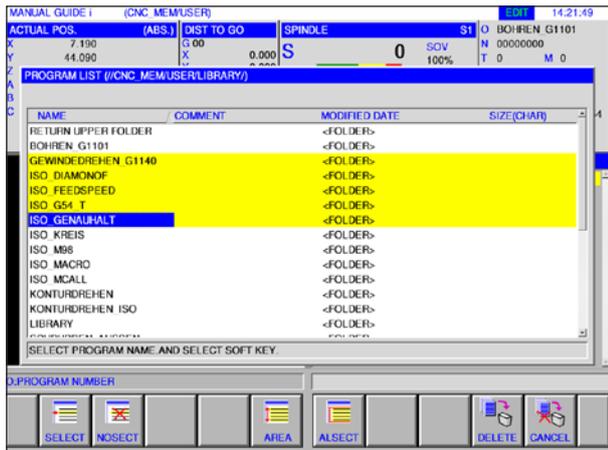
## Delete a number of programs simultaneously



1 Select "Edit" mode.



2 Delete a number of programs.



3 This softkey calls the window for deleting a number of programs.



4 This softkey selects the program to be deleted. Place the cursor on the program to be selected and press "SELECT".



5 This softkey cancels the program selection. Place the cursor on the program whose selection is to be cancelled and press "NOSECT".



6 This softkey selects an area. Place the cursor on the first program in a program sequence that is to be selected, press the "AREA" softkey and then place the cursor on the last program in the program sequence. Now press "SELECT" to define the area.



7 This softkey selects all programs. Pressing the softkey deletes all programs in the relevant folder.



8 Press the softkey to delete.

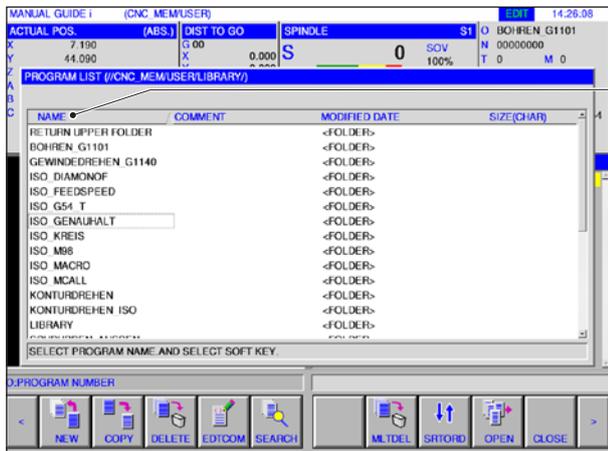
## Change the sorting sequence



1 Select "Edit" mode.



2 Change the sorting sequence.



3 This softkey changes the sorting sequence. You can sort by name, comment, change of date and file size while you click on the area with the mouse.



4 Press the softkey to change.

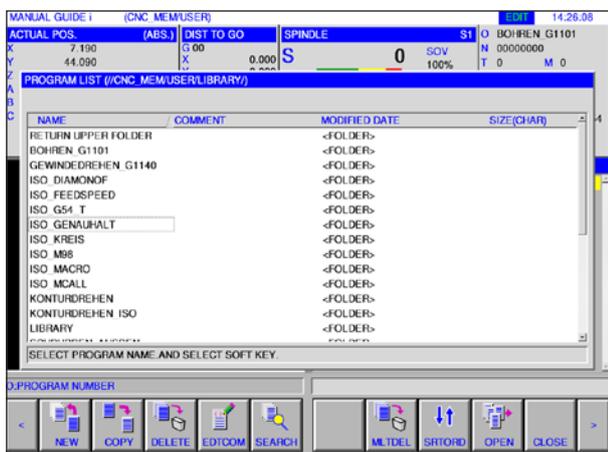
## Open program



1 Select "Edit" mode.



2 Open program.

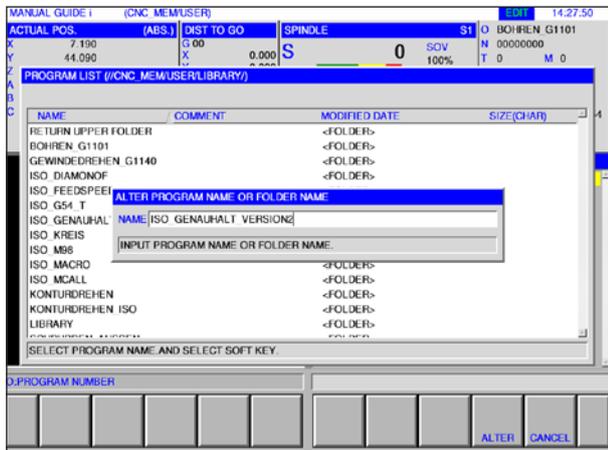
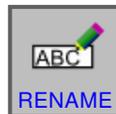


3 This softkey opens the selected program.



## Change program name

- 1 Select "Edit" mode.
- 2 Move mark to the desired program.
- 3 Change the program or folder name.
- 4 This softkey calls the window for renaming programs or folders. After a new program name has been entered for the program, press the "CHANGE" softkey to rename the given program.



- 5 Change the program or folder name.

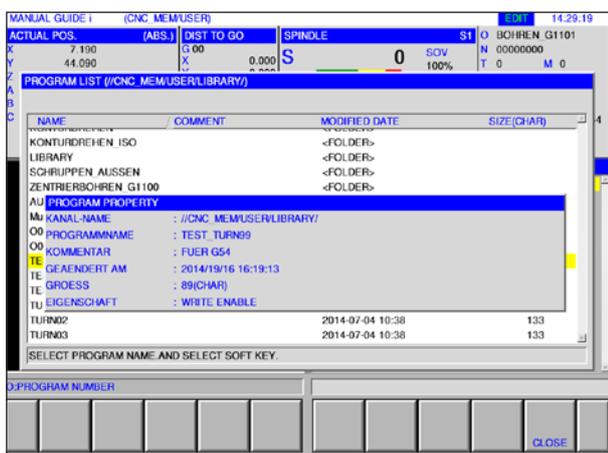


## Program properties

- 1 Select "Edit" mode.
- 2 Move mark to the desired program.
- 3 Show program properties.



This softkey calls the window to show program properties.



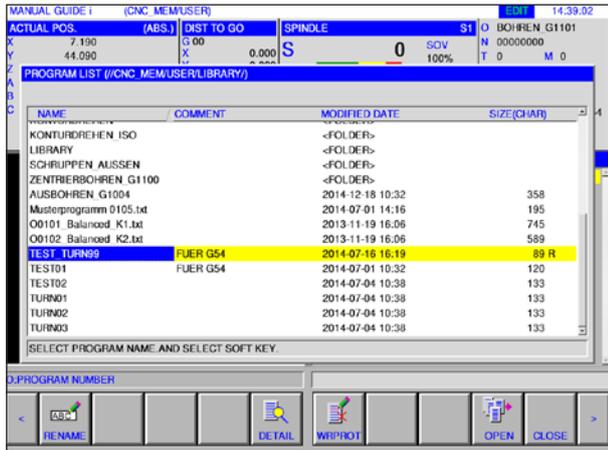
- 4 End program properties display.



## Program protection



- 1 Select "Edit" mode.
- 2 Move mark to the desired program.
- 3 Enable write protection for the program.

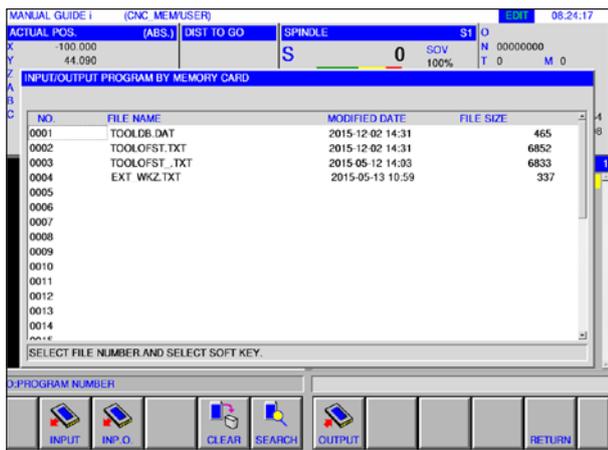


This softkey enables write protection for a program. An "R" is added beside the file size. Press the softkey again to cancel write protection once more.

## Program input and output on a memory card



- 1 Select "Edit" mode.
- 2 Open dialogue for input and output.
- 3 The following softkeys are available:



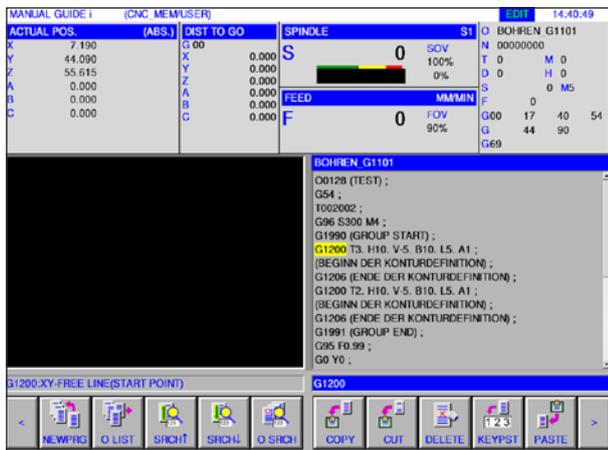
- "INPUT": Upload file from memory card
- "EINP.O": Upload file and change O- number
- "CLEAR": Delete file
- "SEARCH" Search for file
- "OUTPUT": Save file
- "RETURN": End dialogue

**Note:**

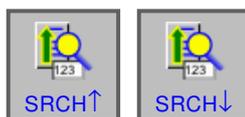
The path of the directory from where the control software was installed is set as default. The path can subsequently be changed in EMConfig.

## Search for text from the input line in the program

EDIT



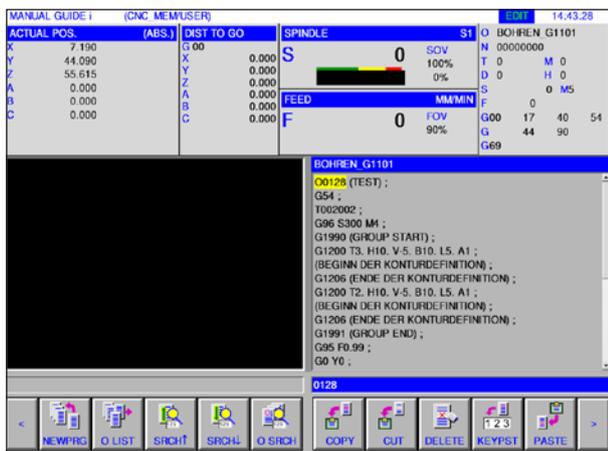
- 1 Select "Edit" mode.
- 2 Enter the text that is to be found in the entry line.



- 3 Press the "SRCH↑" or "SRCH↓" softkey to start the search. The found text will be marked in yellow.

## Search for and open the program

EDIT



- 1 Select "Edit" mode.
- 2 Enter the program name or number that is to be found in the entry line.



- 3 Press the "O SRCH" softkey to start the search. If a program has been found, it is opened automatically.

### Instructions:

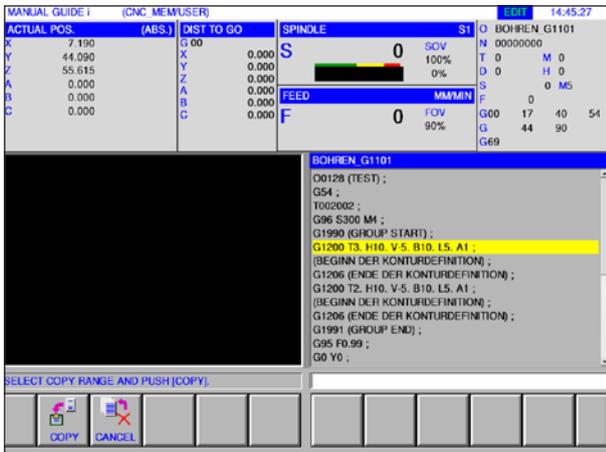
If "O SRCH" is pressed with empty entry line, the next program of the actual folder is opened.

### Instructions:

Only the current folder that was finally selected under "O LIST" is searched.

## Copy marked text into a buffer file

EDIT



- 1 Select "Edit" mode.
- 2 Set the area for copying with the cursor keys (the marked area will be yellow).



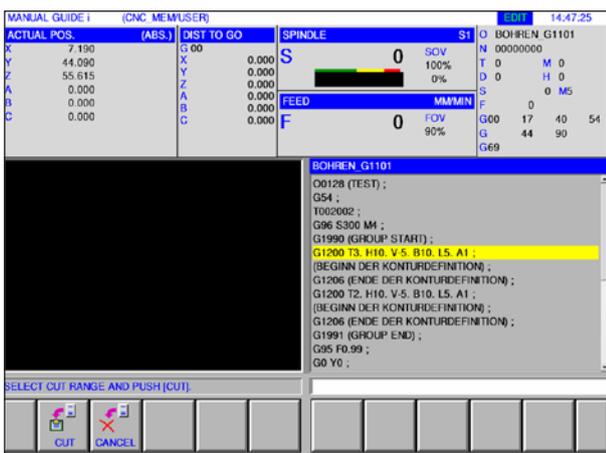
- 3 Press the "COPY" softkey to save the text in the Clipboard.



- 4 The Clipboard content can be added at another point in the program or in a different program.

## Move marked text into the buffer file

EDIT



- 1 Select "Edit" mode.
- 2 Set the area for moving with the cursor keys (the marked area will be yellow).

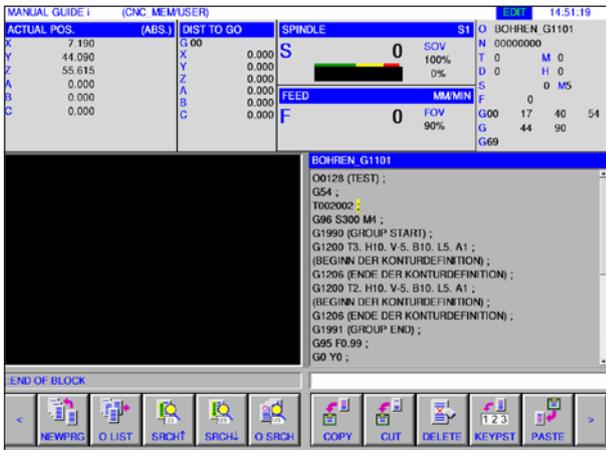


- 3 Press the "CUT" softkey to move the text from the program into the Clipboard.



- 4 The Clipboard content can be added at another point in the program or in a different program.

## Insert text

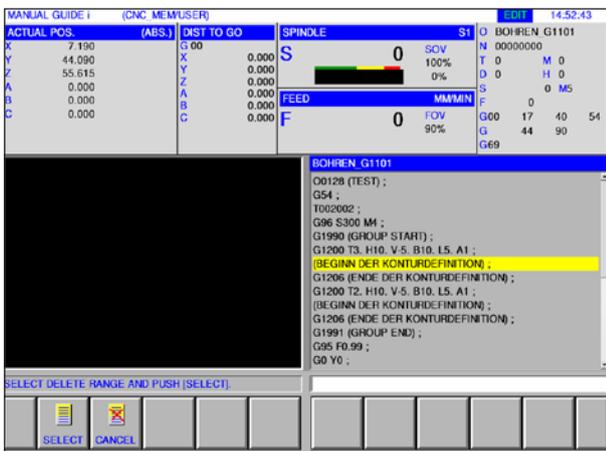


- 1 Select "Edit" mode.
- 2 Move the cursor to the point where the text from the Clipboard is to be inserted.



- 3 This softkey insert the content of the clipboard at the point after the cursor.

## Delete highlighted text



- 1 Select "Edit" mode.
- 2 Set the area for moving with the cursor keys (the marked area will be yellow).



- 3 This softkey calls the window for deleting a number of programs.

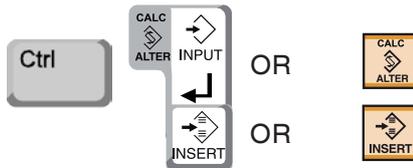
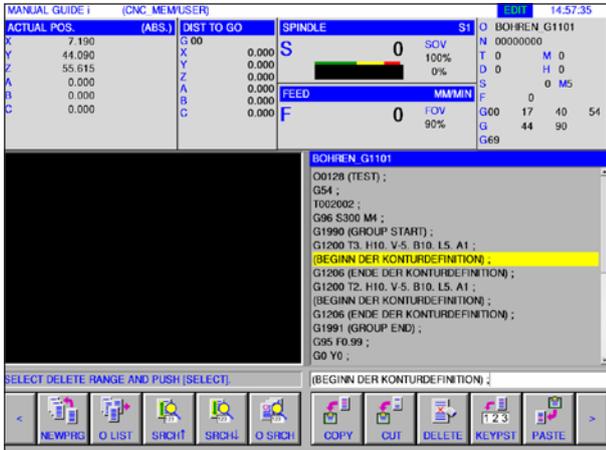


- 4 The "SELECT" softkey determines the area that is to be deleted. Existing highlights can still be changed here if need be.



- 5 Confirm the deletion process with "DO" or cancel it with "NOT DO".

## Insert marked text into an input line



1 Select "Edit" mode.

2 Set the area to be inserted moving with the cursor keys (the marked area will be yellow).

3 Press the "KEYPST" softkey to copy the highlighted text into the input line. The text can now be changed as you wish.

4 CALC/ALTER replaces the highlighted text in the program with that from the input line.

5 INSERT inserts the text from the input line into the program, after highlighting it.

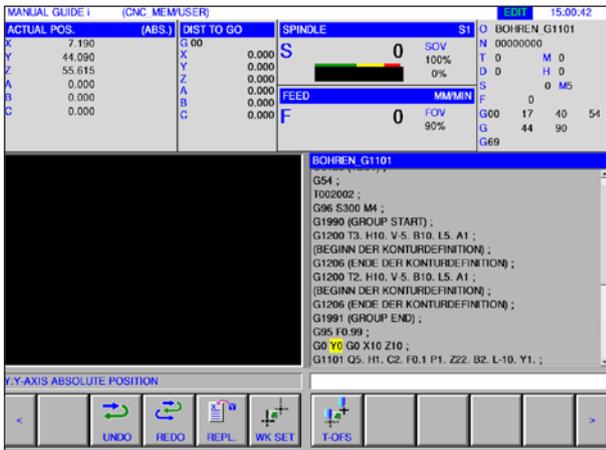
### Instructions:

Use the "KEYPST" keyboard input to change very long program comments and parts of customer macro program blocks with ease.

## Revocation and withdrawal



1 Select "Edit" mode.

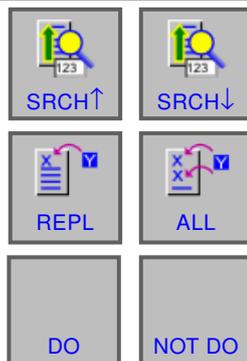
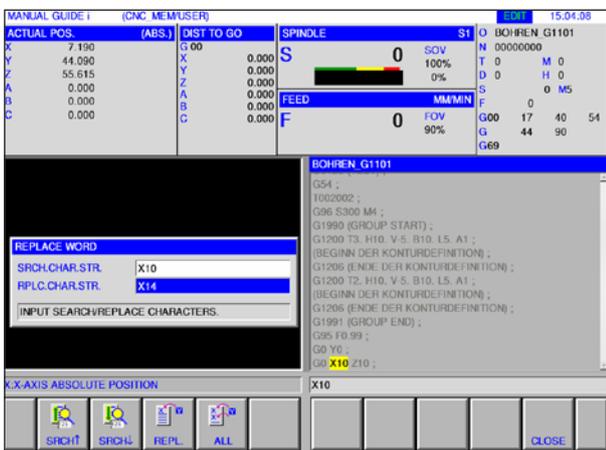


2 Press the "UNDO" softkey to revert to an editing process.  
Press the "REDO" softkey to revoke a change retrospectively (redo).

## Search and replace



1 Select "Edit" mode.



2 Press the "SRCH↑" or "SRCH↓" softkey to search for the string to be replaced.  
Enter the replacement string value and press "REPL" or "ALL".

Confirm query with "DO" to replace all search strings.

## End program listing



1 Select "Edit" mode.



2 Press the "CLOSE" softkey to end the program listing.

## Program mode



Select "Auto" mode.

The following or supplementary functions from "Edit" mode are available for "Auto" mode.

ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE S1		O BOHREN_G1101	
X	7.190	G00	0.000	S	0	SOV	00000000
Y	44.090	X	0.000			100%	T 0 M 0
Z	55.615	Y	0.000			0%	D 0 H 0
A	0.000	Z	0.000	FEED MM/MIN			S 0 M5
B	0.000	A	0.000	F	0	FOV	F 0
C	0.000	B	0.000			90%	G00 17 40 54
		C	0.000				G 44 90
							G69
NEXT DIST				BOHREN_G1101			
		G00	0.000	O0128 (TEST) ;			
		X	0.000	G54 ;			
		Y	0.000	T002002 ;			
		Z	0.000	G96 S300 M4 ;			
		A	0.000	G1990 (GROUP START) ;			
		B	0.000	G1200 T3. H10. V-5. B10. L5. A1 ;			
		C	0.000	(BEGINN DER KONTURDEFINITION) ;			
CURRENT MACHINING				G1206 (ENDE DER KONTURDEFINITION) ;			
				G1200 T2. H10. V-5. B10. L5. A1 ;			
				(BEGINN DER KONTURDEFINITION) ;			
				G1206 (ENDE DER KONTURDEFINITION) ;			
				G1991 (GROUP END) ;			
				G95 F0.99 ;			
				G0 Y0 ;			
O:PROGRAM NUMBER							
<div style="display: flex; justify-content: space-between; align-items: center;"> <div style="display: flex; gap: 10px;"> <div style="text-align: center;">&lt;</div> <div style="text-align: center;">  1123  <b>SETOP</b> </div> <div style="text-align: center;">  O LIST  <b>O LIST</b> </div> <div style="text-align: center;">  BGEDIT  <b>BGEDIT</b> </div> <div style="text-align: center;">  N SRCH  <b>N SRCH</b> </div> <div style="text-align: center;">  O SRCH  <b>O SRCH</b> </div> </div> <div style="display: flex; gap: 10px;"> <div style="text-align: center;">  ACTPOS  <b>ACTPOS</b> </div> <div style="text-align: center;">  PRESET  <b>PRESET</b> </div> <div style="text-align: center;">  SIMLAT  <b>SIMLAT</b> </div> </div> <div style="display: flex; justify-content: space-between; align-items: center;"> <div style="text-align: center;">&lt;</div> <div style="text-align: center;">11</div> </div> </div>							
<div style="display: flex; justify-content: space-between; align-items: center;"> <div style="text-align: center;">&lt;</div> <div style="text-align: center;">  Wk Set  <b>Wk Set</b> </div> <div style="text-align: center;">  T-Ofs  <b>T-Ofs</b> </div> <div style="text-align: center;">  Setting  <b>Setting</b> </div> <div style="text-align: center;">&gt;</div> </div> <div style="display: flex; justify-content: space-between; align-items: center;"> <div style="text-align: center;">9</div> <div style="text-align: center;">10</div> <div style="text-align: center;">12</div> </div>							

- |  |  |
|--|--|
| <p>1 Go to the start of the selected program</p> <p>2 End program listing</p> <p>3 Background edit</p> <p>4 Search for block number</p> <p>5 Search for and open the program</p> <p>6 Display Actual position ABSOLUTE, RELATIVE and MACHINE</p> <p>7 Preset relative position</p> | <p>8 Simulation</p> <p>9 Workpiece coordinates system</p> <p>10 Tool settings</p> <p>11 Forward and Back (extension keys)</p> <p>12 Editor for fixed forms</p> |
|--|--|

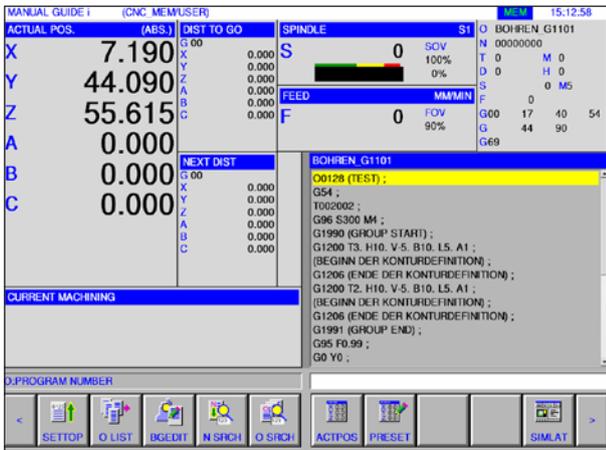
### Instructions:

See chapter F for more information about tool settings: Tool management



## Background Editing

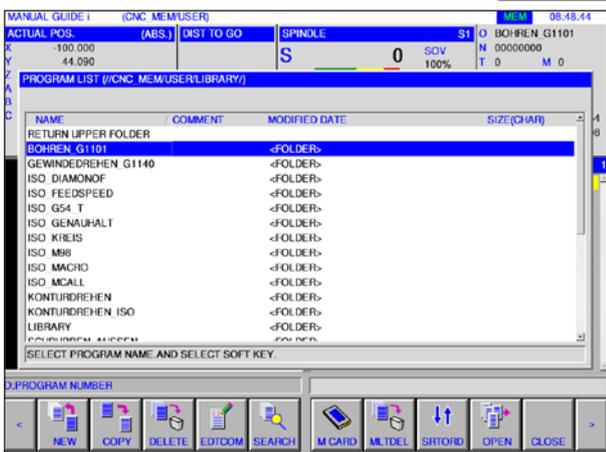
1 Select "Auto" mode.



2 Press "BGEDIT" softkey.

While a part is being machined, contents of other parts programs can be edited.

By clicking the "BGEDIT" softkey in MEM mode, the display switches to the screen for background editing. The program directory for choosing programs is displayed.



3 Choose the program by using the cursor keys and press the "OPEN" softkey to open the program.



4 Press the extension key, until the "BGEND" softkey appears.

By pressing "BGEND" you exit the screen for background editing. The display changes back to automatic operation screen.



**Note:**

Switching to another operating mode during editing will end the background edit mode.







**Instructions:**

See chapter F for more information about tool settings: Tool management

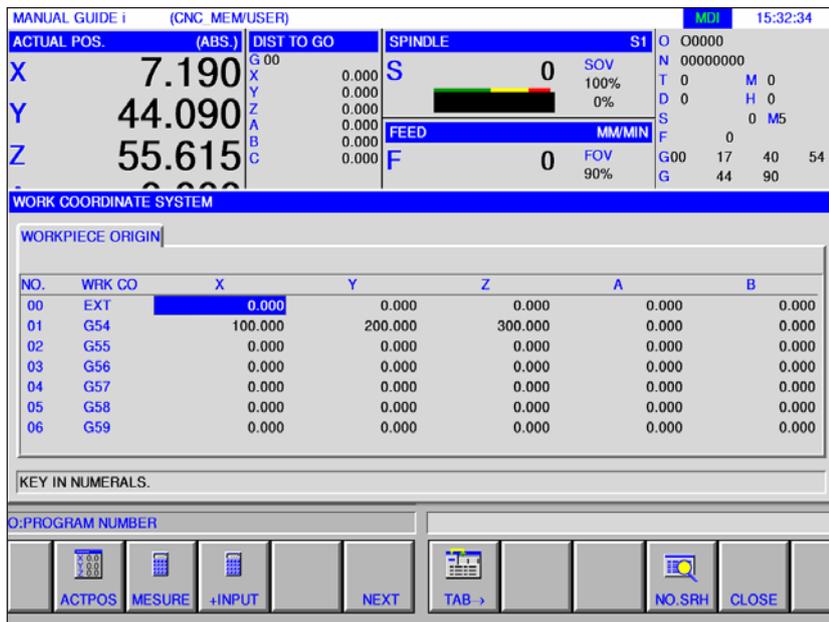
- 1 Go to the start of the selected program
- 2 Search for text from the input line in the program
- 3 Copy highlighted text into a buffer file
- 4 Move highlighted text into a buffer file
- 5 Delete highlighted text
- 6 Insert highlighted text into an input line
- 7 Insert text from a buffer file
- 8 Revoke/Withdraw
- 9 Workpiece coordinates system
- 10 Tool settings
- 11 Editor for fixed forms
- 12 Display Actual position ABSOLUTE, RELATIVE and MACHINE
- 13 Preset relative position
- 14 Forward and Back (extension keys)

## Zero point table

### Setting up workpiece coordinates data



- Select "Edit", "JOG", "MDA" or "AUTO" mode.
- Press softkey.
- "WK SET" opens the workpiece coordinates window in all modes, such as EDIT, JOG, MDA and AUTO.



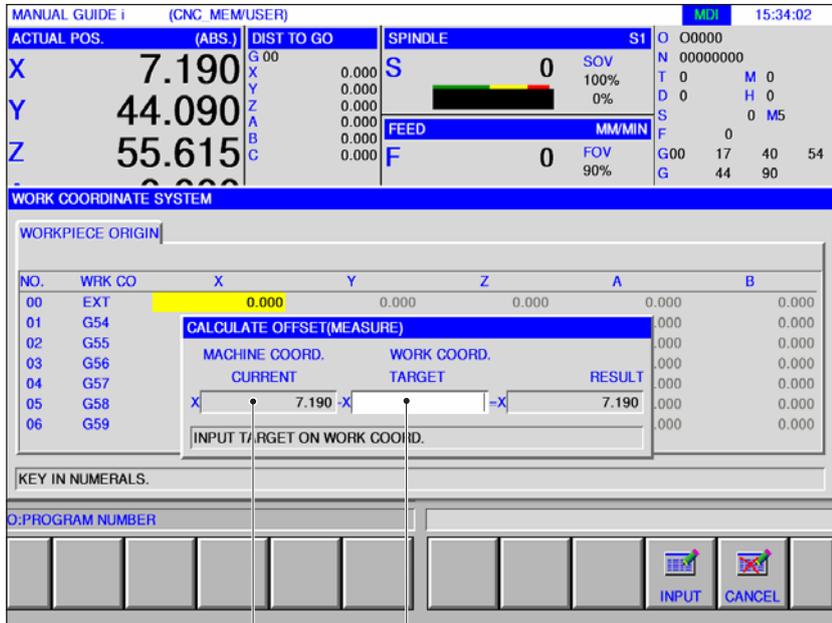
With milling machines, a window to set the workpiece zero point offset is displayed. The data elements to be set and to be displayed are the same as with the usual machine data elements.

Workpiece zero point offset

## Measuring

### Workpiece zero point offset

- Press softkey.



"MEASURE" performs subsequent calculations.

Current value of the machine coordinates (1) - target value of the workpiece coordinates (2).

1

2

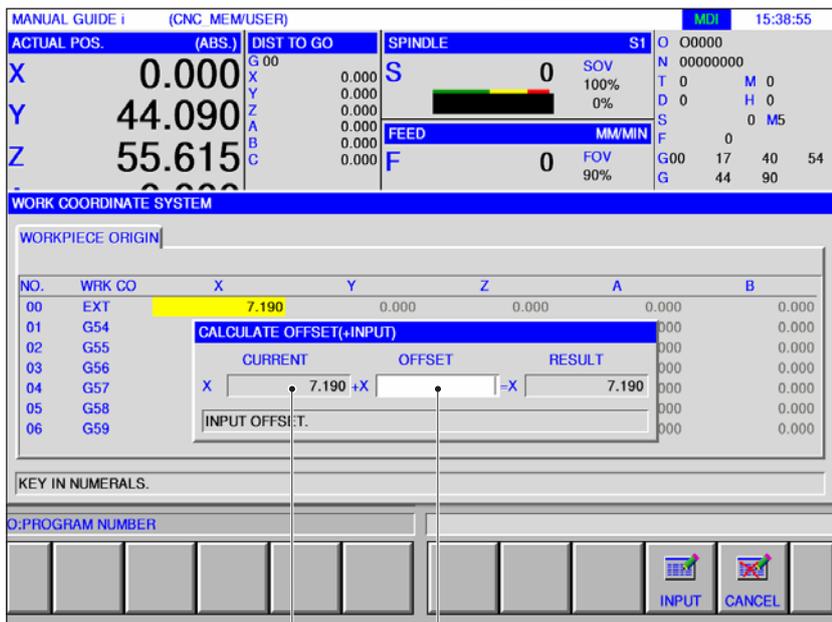
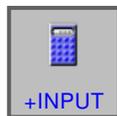


- Conclude measurement with "INPUT".

## Compute

### Workpiece zero point offset

- Press softkey.
- "+INPUT" performs the following calculation.



Current value (1) + offset (2).

1

2



- End computation with "INPUT".

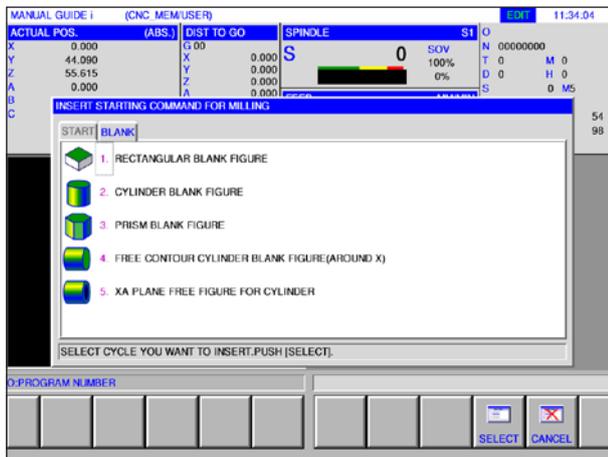


## Graphical simulation

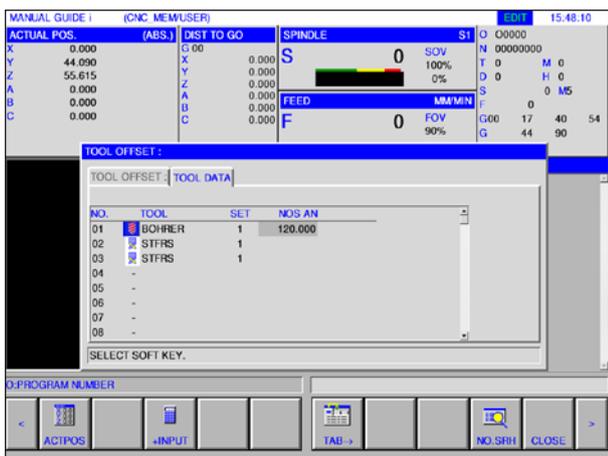
Using the graphical simulation the current program is fully calculated and the result graphically displayed. This monitors the result of the programming without moving the machine axes. Wrongly programmed processing steps are recognized in a timely fashion, preventing the workpiece being mis-processed.

### Unmachined part and tool definition

- Definition of the unmachined parts  
Select the "BLANK". register card in the "START" menu heading
- Define the tools  
Select the "TOOL DATA" register card in the "T-OFS" menu heading.



Raw part definition



Tool definition

There is a precise description of the unmachined parts in Chapter D. Programming Manual Guide i" and the tool definition in chapter "F: Tool Management".

#### Instructions:

If no new unmachined part is defined, the last defined unmachined part is used.



## Screen layout graphical simulation

MANUAL GUIDE i (CNC MEM/USER) 3 STRT 2 ALM MEM 1 13:42:59

ACTUAL POS. (ABS.)		DIST TO GO		SPINDLE S1		O BOHREN_G1101	
X	107.190	G 00		S	6640	SOV	N 00000000
Y	244.090	X	0.000			100%	T 20 M 0
Z	355.615	Y	0.000			0%	D 0 H 0
A	0.000	Z	0.000	FEED MM/MIN			S 300 M4
B	0.000	A	0.000	F	0	FOV	F 0
C	0.000	B	0.000			100%	G00 17 40 54
		C	0.000				G 44 90
							G69

SIMULATE-ANIMATE BOHREN\_G1101 6

G96 S300 M4 ;  
 G1990 (GROUP START) ;  
 G1200 T3. H10. V-5. B10. L5. A1 ;  
 (BEGINN DER KONTURDEFINITION) ;  
 G1206 (ENDE DER KONTURDEFINITION) ;  
 G1200 T2. H10. V-5. B10. L5. A1 ;  
 (BEGINN DER KONTURDEFINITION) ;  
 G1206 (ENDE DER KONTURDEFINITION) ;  
 G1991 (GROUP END) ;  
 G95 F0.99 ;  
 G0 Y0 ;  
 G0 X10 Z10 ;  
 G1101 Q5. H1. C2. F0.1 P1. Z22. B2. L-10. Y1. ;  
 G1101 Q5. H1. C2. F0.1 Z22. B2. L-20. Y1. A6. S0.12 D7.

3514 WRONG FIGURE DATA

REWIND START PAUSE SINGLE STOP INIT INTERF TLPATH GRPOFF

- 1 Mode
- 2 Alarm status
- 3 Program mode
- 4 Simulation report line
- 5 Position display of the axes
- 6 Tool number
- 7 Current NC program block
- 8 Current line number in ISO program window
- 9 Softkey strip to control simulation.
- 10 Coloured positioning movements:
  - Red positioning movement = tool moves in rapid traverse.
  - Green positioning movement = tool moves in machining feed.

### Start simulation mode

Change to Automatic mode.

The "SIMLAT" softkey opens the SIMULATE-ANIMATE screen.



## Softkey functions

### To start the simulation

Use the "REWIND" softkey to reach the NC program.



### Start simulation

The "START" softkey starts the simulation. For the simulation to be started, an NC program must be selected. The name of the open NC program is shown at the top edge of the program window.



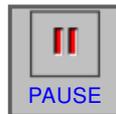
### Single block

The "SINGLE" softkey makes a stop after each individual block if the machining is simulated in continuous mode. If the machining simulation is in the stopped state, this softkey starts the machining simulation in single block mode.



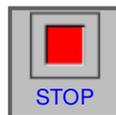
### Stop simulation

This "PAUSE" softkey halts the simulation and the NC program. The simulation can be continued with "START".



### Cancel simulation

The "STOP" softkey interrupts the simulation and the NC program. The simulation can be restarted with "START".



### Initialize unmachined part

The "INIT" softkey initializes the part to be machined, which is used for animation.



### Workpiece collision

The "INTERF" softkey starts the interference check during the simulation. If the tool cutter collides with a workpiece while in rapid traverse, an alarm pops up and the area with which the tool cutter collided is shown in the same colour as the tool.





## Tool path

Press the "TLPATH" softkey to reveal the SIMULATE TOOL PATH strip with the following softkeys:

"DISP", "NODISP" and "DELETE"

You use the "DISP" and "NODISP" softkeys to mark only the needed sections of a tool path.

The "DISP" softkey starts the tool path display from the next block.

The "DISP" softkey suppresses the tool path display from the next block.

The toolpath is not shown until the "DISP" softkey is pressed.

The "DELETE" softkey deletes the former tool path. On continuing the simulation, the toolpath display is resumed.

The "ANIME" softkey selects the mode for the machining simulation (animated graphics).

## Graphics OFF

The "GRPOFF" softkey ends the machining simulation.

## Scaling and moving



The "LARGE" softkey enlarges the drawing.



The "SMALL" softkey reduces the size of the drawing.



The "AUTO" scales the drawing automatically and matches it to the size of the window.



The "REVERS" softkey moves the viewing point into the opposite position.



The "← MOVE" softkey moves the viewing point to the left. Accordingly, the marked tool path is moved to the right.



The "→ MOVE" softkey moves the viewing point to the right. Accordingly, the marked tool path is moved to the left.



The "↑ MOVE" softkey moves the viewing point up. Accordingly, the marked tool path is moved to the left.



The "↓ MOVE" softkey moves the viewing point down. Accordingly, the marked tool path is moved up.

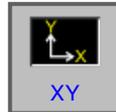


The "CENTER" softkey places the centre of the toolpath in the centre of the window.

## Selecting a graphics coordinates system



The "ROTATE" softkey selects the graphics coordinates system.



This softkey selects the XY plane.



This softkey selects the ZY plane.



This softkey selects the ZY plane.



This softkey selects the ZY plane.



This softkey selects the ZY plane.



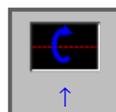
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis.



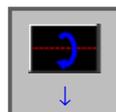
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis. The viewing point is opposite to that mentioned above.



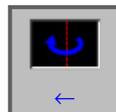
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis.



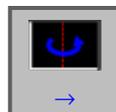
This softkey executes a counterclockwise rotation with the screen as the central axis.



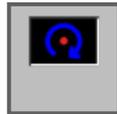
This softkey executes a clockwise rotation with the screen as the central axis.



This softkey executes a counterclockwise rotation with the screen as the central axis.



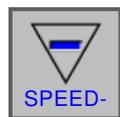
This softkey executes a clockwise rotation with the screen as the central axis.



This softkey executes a clockwise rotation with the screen as the central axis.



This softkey executes a counterclockwise rotation with the screen as the central axis.

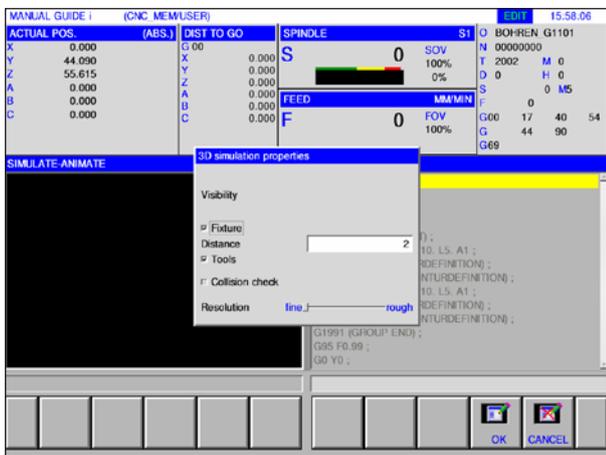


This softkey selects the simulation playback speed. There are 5 steps available for the speed.

### 3D configuration



This softkey starts 3D configuration.

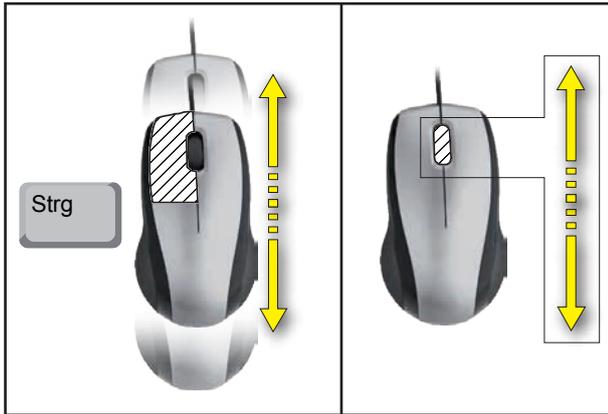


3D configuration

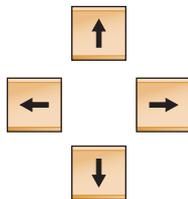
#### Instructions:

The selection of available setting options depends on whether a 3D viewing licence is available or not.



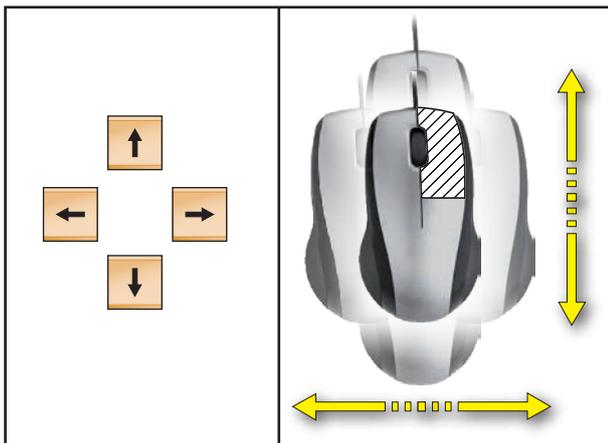


**Zooming with the mouse**



**Shift graphics**

Press cursor button to shift the graphics.



**Shift**

# D: Programming with MANUAL GUIDE *i*

**Instructions:**

This programming manual describes all functions that can be executed with WinNC. Depending on the machine you are operating with WinNC, not all functions may be available.

**Example:**

The Concept MILL 55 milling machine has no position-controlled main spindle, so it is also not possible to program spindle positions.

## Overview

### M-commands

M00	Programmed stop	M25	Open clamping device
M01	Optional stop	M26	Close clamping device
		M27	Swivel divider
M02	End of program	M29	Thread tapping without compensation chuck
M03	Spindle ON clockwise		
M04	Spindle ON counterclockwise	M30	End of main program
M05	Spindle stop	M51	C-axis mode activation
		M52	C-axis mode deactivation
M06	Perform tool change	M71	Exhaust ON
M07	Minimal lubrication ON	M72	Exhaust OFF
M08	Coolant ON		
M09	Coolant OFF, minimal lubrication OFF	M98	Call subroutine
M10	Divider clamp on	M99	Jump back to calling program
M11	Slacken divider clamping		

## Overview

MANUAL GUIDE *i* helps you to use CNC controllers which are installed in lathes and milling machines.

Using a single screen, machining programs can be produced, checked by animation and the machining set up and run.

MANUAL GUIDE *i* uses the ISO code format for machining programs and machining cycles in order to implement extended machining processes.

## Produce MANUAL GUIDE *i* program

1 Select "Edit" mode.

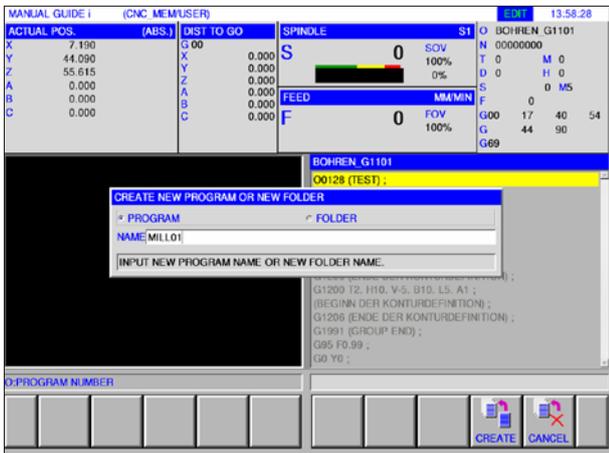


2 Press softkey.



3 Enter program or folder name and confirm with "CREATE".

If a program name was allotted previously, an appropriate message will be displayed.



The Fanuc standard format for program names is as follows:

O1234. If fewer than 4 numbers are entered, those missing are populated with zeroes.

### Instructions:

Program names can consist of at least 1 and a maximum of 32 characters. The following characters are allowed: "Zz", "0 to 9", "\_", "+", and ".".

## Programming set-up

The new program is entered with the softkeys in the following steps:



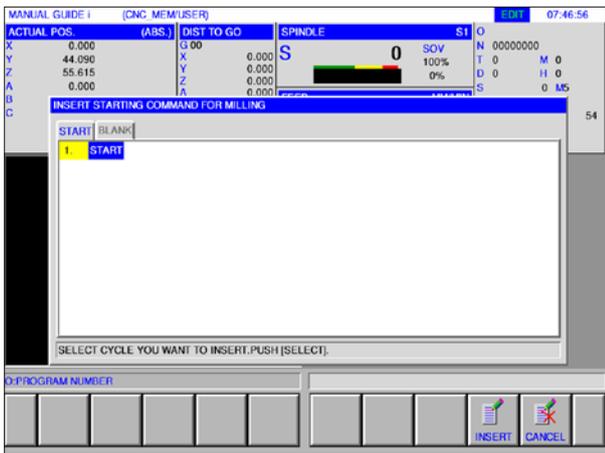
- START
- CYCLE
- END

A cycle is divided into 2 parts:

- Machining conditions
- Geometric data

### Program start START

1 Press "START" softkey.



#### Instructions:

The fixed shapes from form 1 are available for "START".



2 Find the desired selection in the START register card and confirm with "SELECT".





**Select unmachined part**

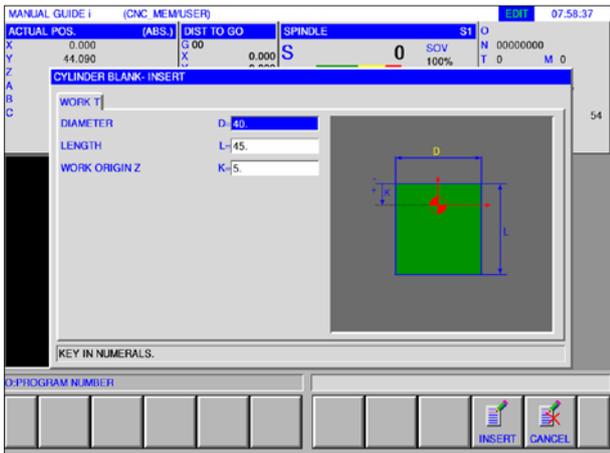
1 Press "START" softkey.



2 Select the register card for an unmachined part.



3 Select the desired unmachined part and confirm with "SELECT".



4 Enter the data for the unmachined part and close the entry with "INSERT".

### Cycle programming

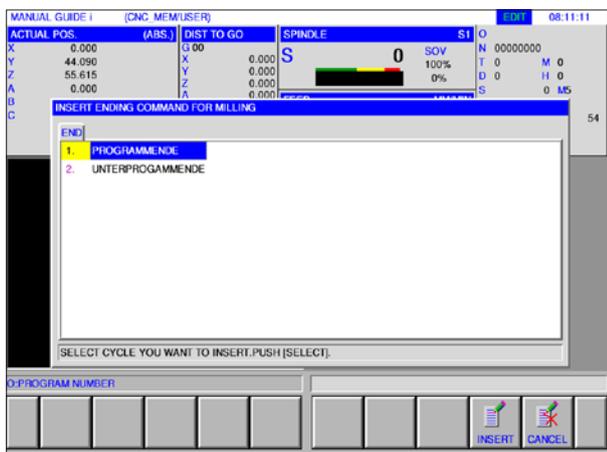
1 Press "CYCLE" softkey.

There is a precise description of entry and programming for cycles later in this chapter under "Working with cycles".



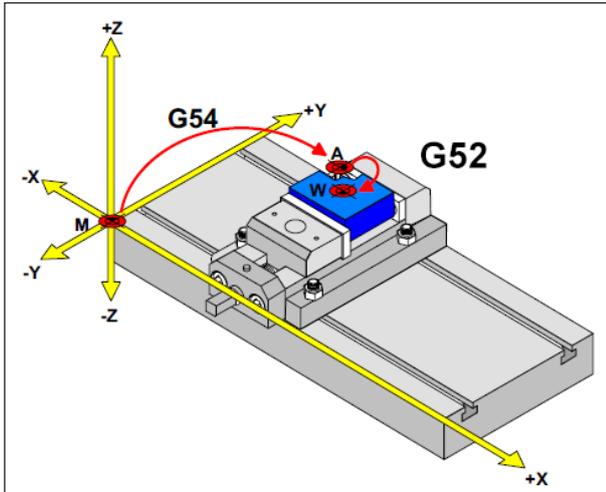
### Program end END

1 Press "END" softkey.



2 Go to the end of program selection and close the entry with "INSERT".





*In the following situation, the unmachined part must be described from A outwards*

## Raw part definition

If you are machining in a program with a stop point (e.g.: G54) and a transformation (G52) to the actual workpiece zero point, the unmachined part must be described from the stop point outwards.

M= Machine zero point

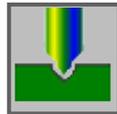
A= Stop point

W= Workpiece zero point

## Cycle overview

The cycle groups with the Fanuc ManualGuide *i* cycles defined within them are listed here.

### Hole machining



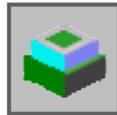
- Centre Drilling G1100
- Drilling G1101
- Tapping G1002
- Reaming G1103
- Boring G1104

### Face machining



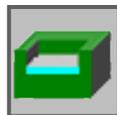
- Facing rough G1020
- Facing finish G1021

### Contouring



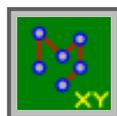
- Outer wall countering rough G1060
- Outer wall countering bottom finish G1061
- Outer wall countering side finish G1062
- Outer wall countering chamfer G1063
- Inner wall countering rough G1064
- Inner wall countering bottom finish G1065
- Inner wall countering side finish G1066
- Inner wall countering chamfer G1067
- Partial contouring rough G1068
- Partial contouring bottom finish G1069
- Partial contouring side finish G1070
- Partial contouring chamfer G1071

### Pocketing



- Pocketing rough G1040
- Pocketing bottom finish G1041
- Pocketing side finish G1042
- Pocketing chamfer G1043

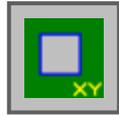
### Figure: Hole position



- Random points G1210
- Linear points G1211
- Grid points G1213
- Rectangle points G1214
- Circle points G1215
- Arc points G1216
- A-axis hole in cylinder (arc) G1772
- A-axis hole in cylinder (free) G1773

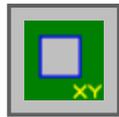


**Figure: Facing figure**



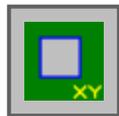
- Square facing G1220

**Figure: Contouring figure**



- Square G1220 convex
- Circle G1221 convex
- Track G1222 convex
- Polygon G1225 convex
- Free figure
- XA-plane free form convex contour for cylinder G1700
- Square G1220 concave
- Circle G1221 concave
- Track G1222 concave
- Polygon G1225 concave
- Free concave figure
- XA-plane free form concave contour for cylinder G1700
- Free open figure
- XA-plane free form open contour for cylinder G1700

**Figure: Pocket figure**



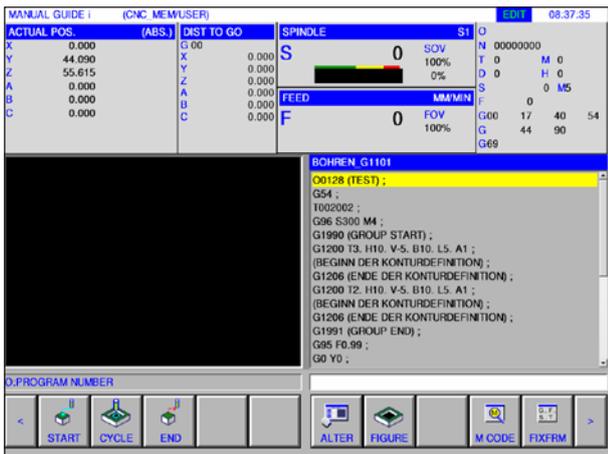
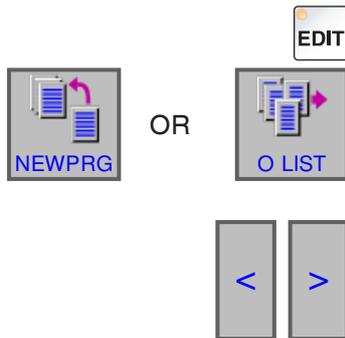
- Square concave G1220
- Circle concave G1221
- Track concave G1222
- Polygon concave G1225
- Free concave figure
- XA-plane free form concave contour for cylinder G1700

## Working with cycles

Frequently repeated processes, which include several machining steps, are saved in the controller as cycles. Some special functions are also available as cycles.

### Define cycle

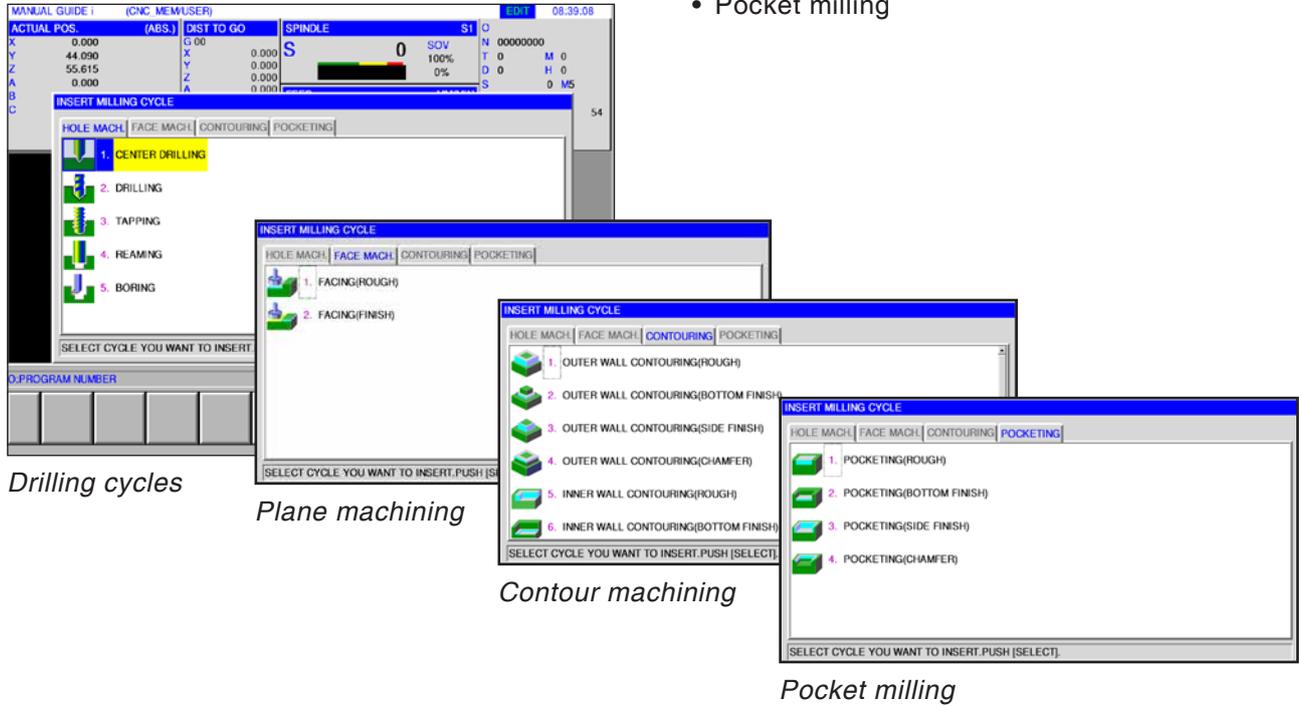
- Select "Edit" mode.
- Create a new program or open an existing program.
- Select the softkey toolbar for turning or milling cycles with the extension buttons.



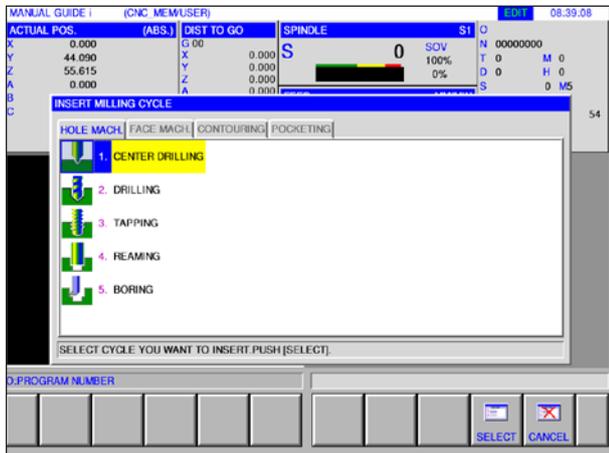
- Press softkey.

The dropdown window shows the different cycle groups.

- Drilling
- Plane machining
- Contour machining
- Pocket milling

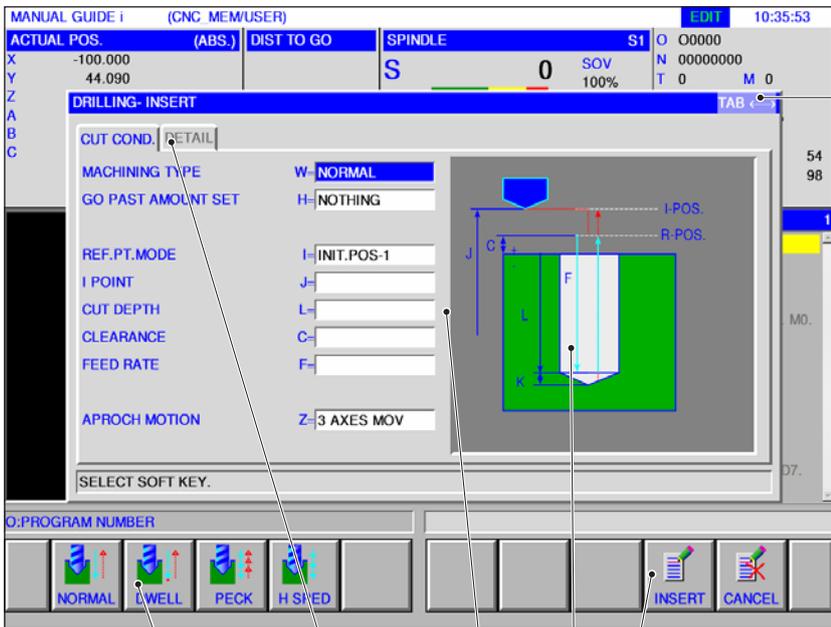


Select the desired cycle and confirm the selection with the "SELECT" softkey or cancel it with "CANCEL".



- "SELECT" softkey to confirm.
- "CANCEL" softkey to cancel.

## Data Entry for Machining Cycles



For switching tabs with the cursor keys, "TAB <->" is displayed in the upper right corner of the window.

Softkey for inserting the entered data into the program or for cancelling the data entry.

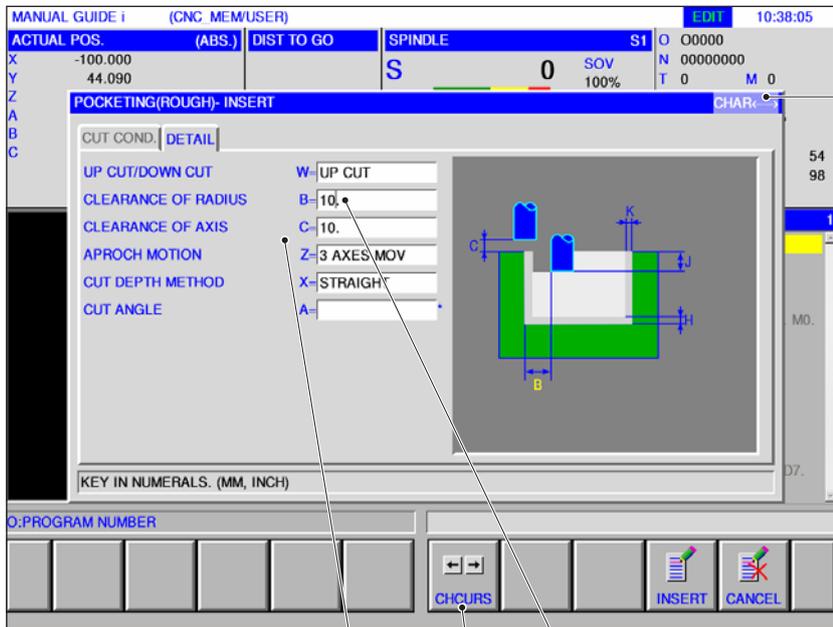
- Coloured positioning movements:
- Red positioning movement = tool moves in rapid traverse.
  - Green positioning movement = tool moves in machining feed.

Entry fields marked with \* are optional and are not required to be populated. These entry fields can remain empty.

Tab for entering machining conditions, tool distance and other details.

With the cursor keys <-> or <->> you can switch from one tab to another. The chosen active tab is depicted in blue. In the upper right corner of the window "TAB" is displayed.

Options menu for data entry via softkey.



For cursor movement inside an entry field "CHAR ←→" is displayed.

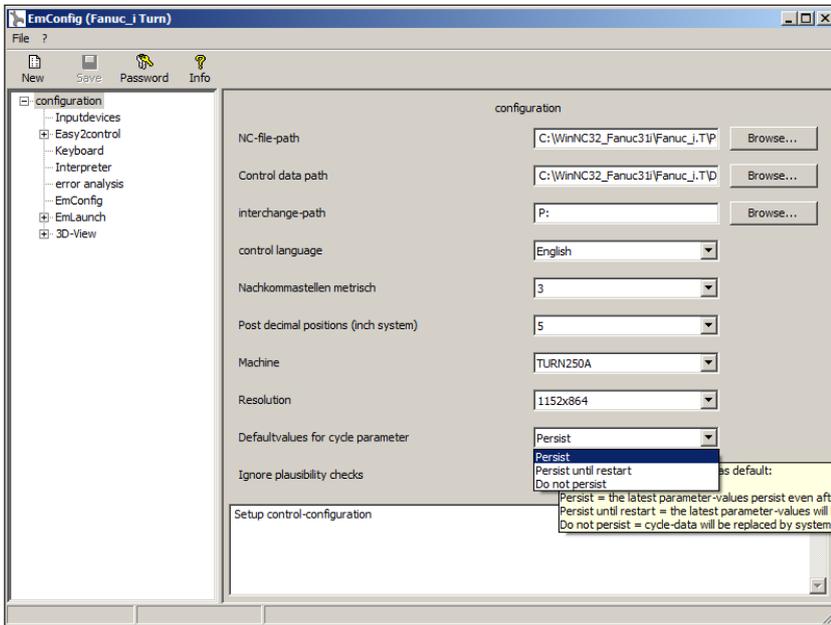
The softkey "CHCURS" (change cursor) is displayed in the data entry windows for the machining cycle, figure and contour programs. This softkey is used to choose, whether the cursor keys <←> and <→> are used for switching between the tabs or for moving the cursor inside the entered data in the entry field.

• **Data entry**

By actuating the cursor keys <↑> or <↓>, you can set the cursor on the desired data entry field.

- There are 2 options for data entry:
  1. Entering data as numerals.  
In the lower part of the window, the message "Enter data" is displayed.
  2. Entering data by using softkeys. For these fields an options menu is provided in the softkey bar.  
The message "Choose a softkey" is displayed.

## Default values for cycle parameters



EMConfig is an auxiliary software to WinNC.

WinNC settings can be changed with EMConfig.

Open EMConfig and select the configuration point:

*Default values for cycle parameters*

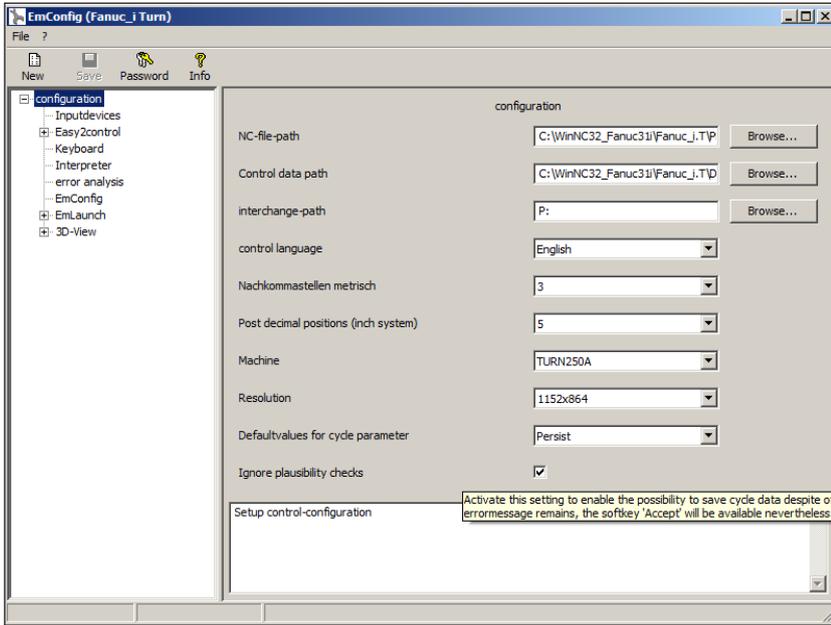
### Instructions:

If cycles have already been programmed once, then these input values are saved and next time are proposed as default values. This can be unfavourable in training and can therefore be configured via EMConfig.

The following settings can be reached under the default values heading:

- **Persist**  
last entered cycle data remains even after receiving a control system restart
- **Persist until restart**  
last entered cycle data remains received while the control system is still running
- **Do not persist**  
Cycle data is reset to the default values immediately after quitting the cycle

## Ignore plausibility check for saving

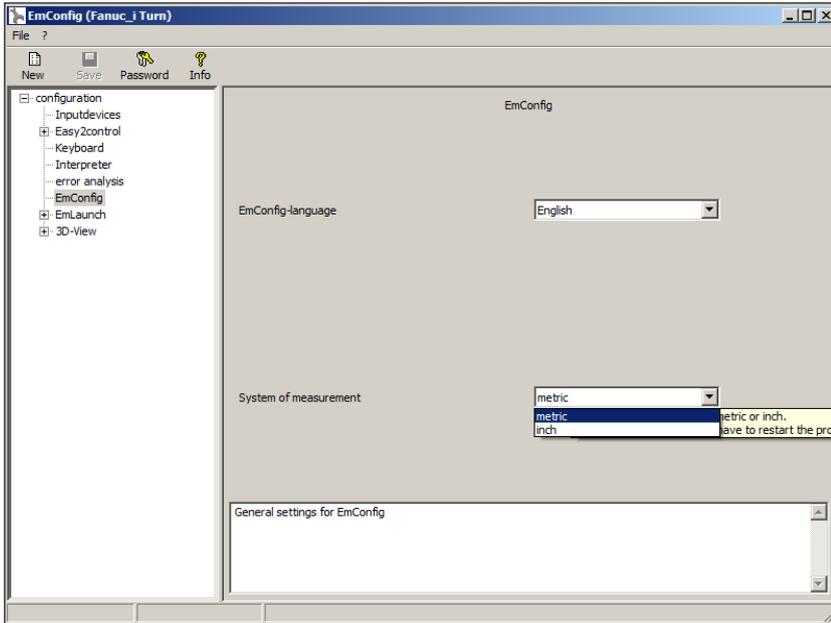


Use this checkbox to activate or deactivate the plausibility check for saving.

Activate this setting to be able to save cycles, despite an error message being present. The corresponding error messages certainly remain in existence, but the "Accept" softkey is available.

*Set plausibility check for saving*

## Select dimensions system



Use this checkbox to select the metric or imperial measurement system for the controller.

Set metric or imperial measuring system

### Instructions:

Imperial programs cannot be used with a metric controller (and vice-versa).

### Units table

Length dimensions imperial			
<b>feet</b> °)	inch	mm	m
<b>1</b>	12	304.5	0.304
<b>inch</b> °)	feet	mm	m
<b>1</b>	0.83	25.4	0.0254

Length dimensions metric			
<b>m</b>	mm	inch	feet
<b>1</b>	1000	39.37008	3.28084
<b>mm</b>	m	inch	feet
<b>1</b>	0.001	0.0393701	0.0032808

\*) **feet**: only at constant cutting speed

°) **inch**: Standard input



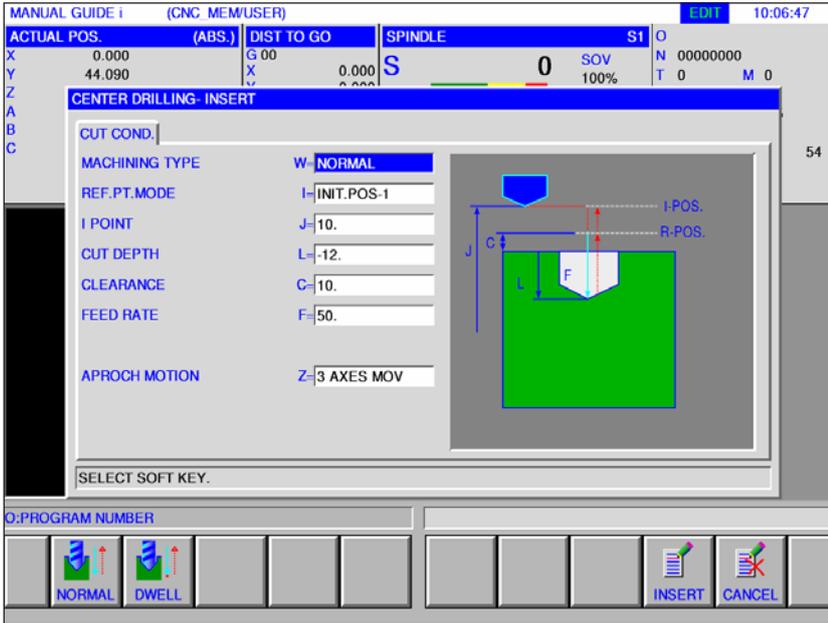
## Hole machining

- Centre Drilling G1100
- Drilling G1101
- Tapping G1002
- Reaming G1103
- Boring G1104

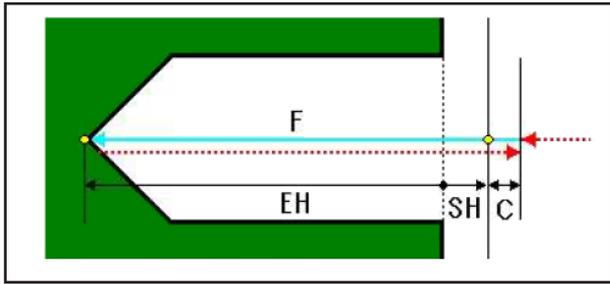


## Centre drilling G1100

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
W	Machining type	<ul style="list-style-type: none"> <li>[NORMAL]: Without dwell time. (Initial value)</li> <li>[DWELL]: With dwell time.</li> </ul>
I	Ref.point mode	<ul style="list-style-type: none"> <li>[INIT-1]: When traversing between borings, the return is made to reference point R. At the end, the return is made to point I. (Initial value).</li> <li>[INIT-2]: All movements between borings, including the last return, are made to point I.</li> <li>[REF]: All movements between borings, including the last return, are made to point R.</li> </ul>
J	I Point	I point coordinates.
L	Cut depth	Drilling depth (radius value, negative value)
C	Clearance	Distance between workpiece surface and R position (radius value, positive value).
F	Feed rate	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the drilling. If this is missing, 0 is assumed. (In seconds, positive value)
Z	Approach motion	<ul style="list-style-type: none"> <li>[3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



Tool track

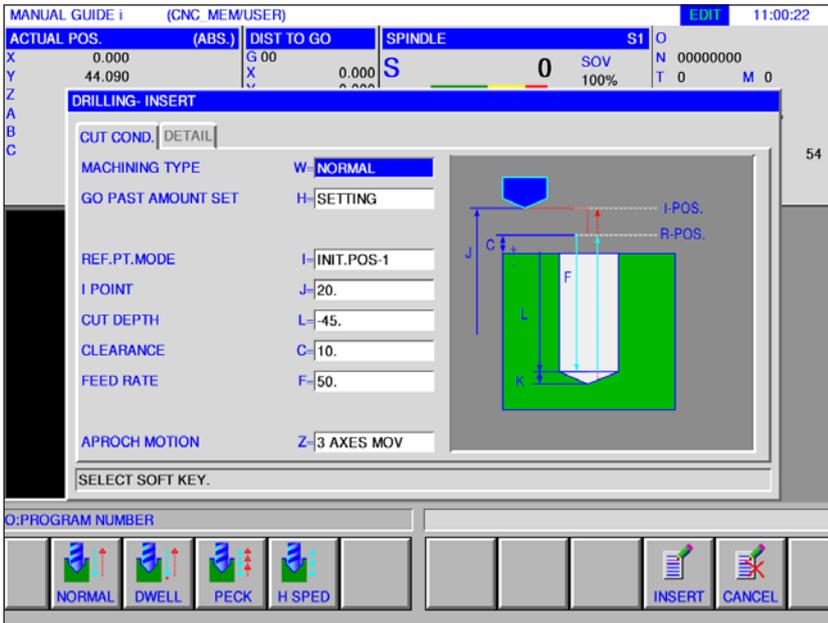
**Cycle description**

- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the machining end position.
- 3 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".



## Drilling G1001

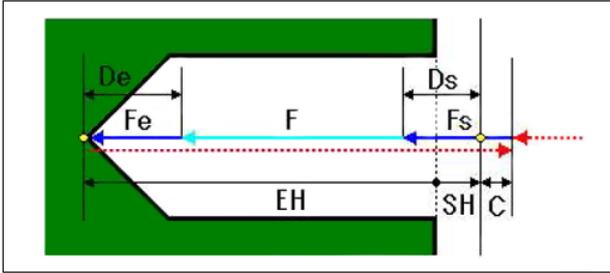
Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
W	Machining type	<ul style="list-style-type: none"> <li>[NORMAL]: Without dwell time. (Initial value)</li> <li>[DWELL]: With dwell time.</li> <li>[PECK]: Deep hole drilling</li> <li>[H SPED]: With dwell time.</li> </ul>
H	Go past amount set	<ul style="list-style-type: none"> <li>[NOTHIN]: Drilling depth relative to the tool shaft</li> <li>[SETING]: Drilling depth relative to the tool tip</li> </ul> For the setting [SETING] the parameters U, V, K and the softkey [CALC] are available under Details.
Q*	Stroke depth (INCR+)	Cut depth executed per cut (radius value, positive value). Only for deep hole drilling and chip breaking.
I	Ref.point mode	<ul style="list-style-type: none"> <li>[INIT-1]: When traversing between borings, the return is made to reference point R. At the end, the return is made to point I. (Initial value).</li> <li>[INIT- 2]: All movements between borings, including the last return, are made to point I.</li> <li>[REF.] : All movements between borings, including the last return, are made to point R.</li> </ul>
J	I Point	I point coordinates.
L	Cut depth	Drilling depth (radius value, negative value)
C	Clearance	Distance between workpiece surface and R-position (Radius value, positive value)
F	Feed rate	Feed speed (positive value)

Cutting condition		
Data item		Meaning
P*	Dwell time	Dwell time at the bottom of the drilling. If this is missing, 0 is assumed. (In seconds, positive value) Only in machining type [DWELL].
Z	Approach motion	• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.

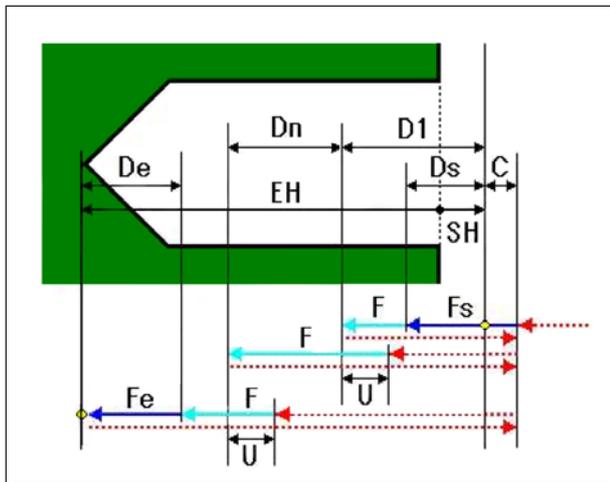
Detail		
Data item		Meaning
A*	Start cut depth	The start feed S applies for the start cut depth A
S*	Start feed rate	Feed speed at start
D*	End cut depth	The end feed E applies for the end cut depth D
E*	End feed rate	Feed speed at end
U	Tool diameter	Tool diameter entry
V	Nose angle	Cutting angle entry
K	Go past amount	Overrun entry



tool track

**Cycle description: with/without dwell time**

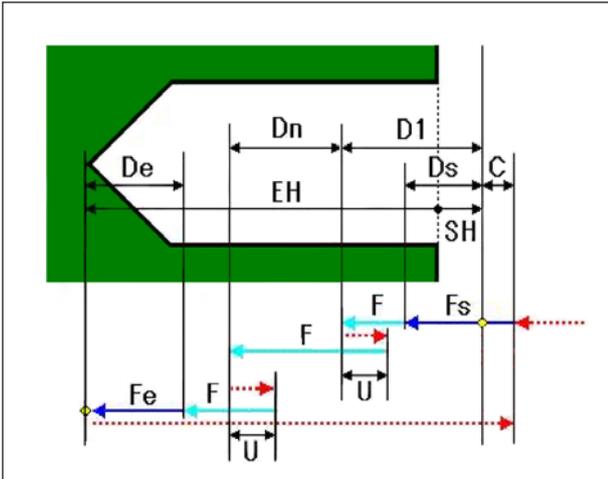
- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the machining end position.
- 3 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".



tool track

**Cycle description: Long-hole drilling**

- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the position "Machining start position - Depth of the first cut (D1)".
- 3 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 4 The tool is moved in rapid traverse into the position "Machining end position of the preceding cut + retraction clearance (U)".
- 5 The tool is moved with feed speed (F) into the position "Machining end position - Cut depth compensation cut (Dn)".
- 6 Steps <3> to <5> are repeated until the final machining end position is reached.
- 7 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".



tool track

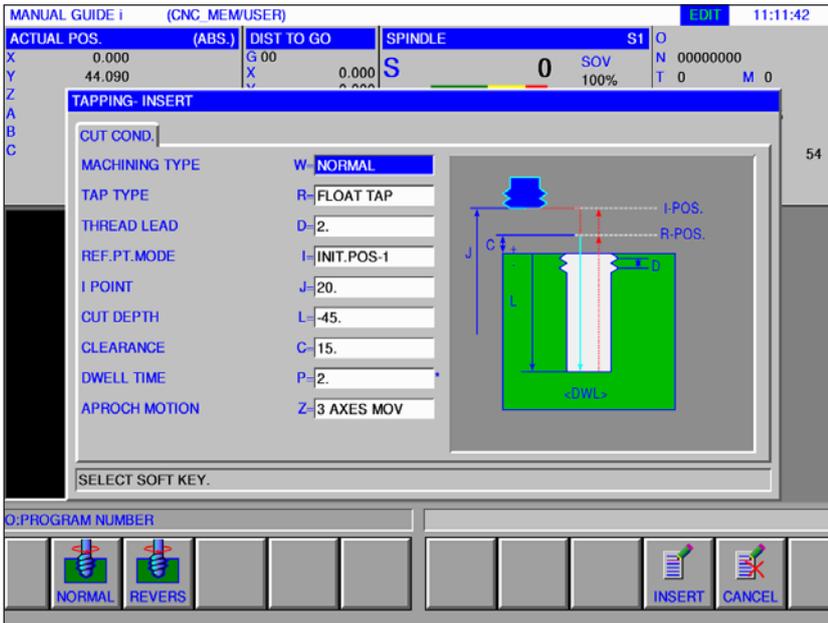
**Cycle description: Chip crushing**

- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the position "Machining start position - Depth of the first cut (D1)".
- 3 The tool is moved in rapid traverse into the position "Instantaneous position + retraction clearance (U)".
- 4 The tool is moved with feed speed (F) into the position "Machining end position - Cut depth compensation cut (Dn)".
- 5 Steps <3> and <4> are repeated until the machining end position is reached.
- 6 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".

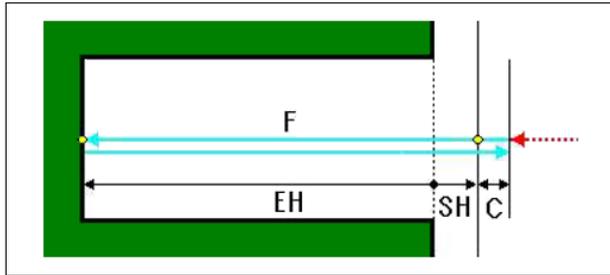


## Tapping G1002

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
W	Machining type	<ul style="list-style-type: none"> <li>[NORMAL]: Clockwise tapping.</li> <li>[REVERS]: Counterclockwise tapping.</li> </ul>
R	Tap type	<ul style="list-style-type: none"> <li>[FLOAT]: Thread tapping with compensating chuck.</li> <li>[RIGID]: Thread tapping without compensating chuck.</li> </ul>
D	Thread lead	Tap pitch (radius value, positive value).
I	Ref.point mode	<ul style="list-style-type: none"> <li>[INIT-1]: When traversing between borings, the return is made to reference point R. At the end, the return is made to point I. (Initial value).</li> <li>[INIT- 2]: All movements between borings, including the last return, are made to point I.</li> <li>[REF]: All movements between borings, including the last return, are made to point R.</li> </ul>
J	I Point	I point coordinates.
L	Cut depth	Drilling depth (radius value, negative value)
C	Clearance	Distance between workpiece surface and R-position (Radius value, positive value)
P*	Dwell time	Dwell time at the bottom of the drilling. If this is missing, 0 is assumed. (In seconds, positive value)
Z	Approach motion	<ul style="list-style-type: none"> <li>[3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>

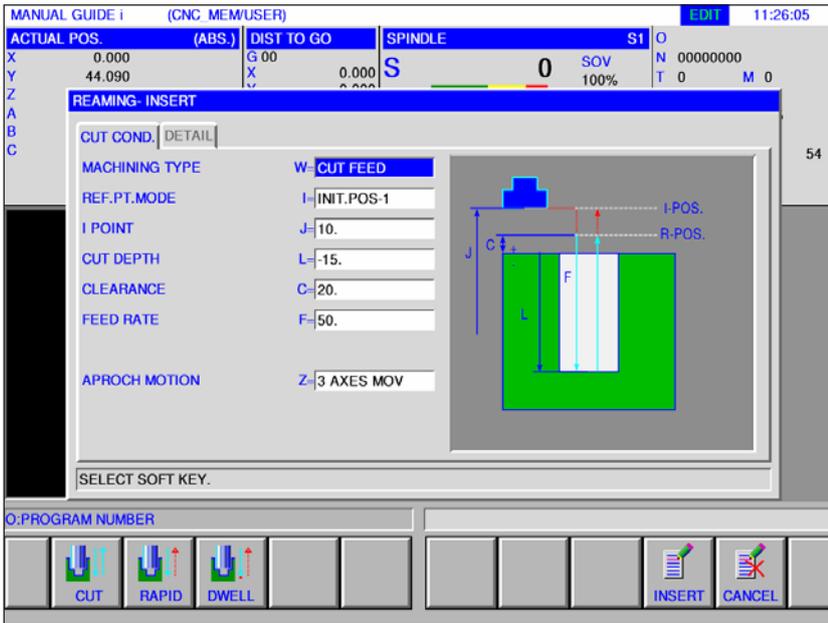
*tool track***Cycle description:**

- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the machining end position.
- 3 Stop spindle.
- 4 The spindle is turned in the opposite direction.
- 5 The tool is moved with feed speed (F) into the position "Machining start position + clearance (C)".
- 6 The spindle changes to the normal direction of rotation.



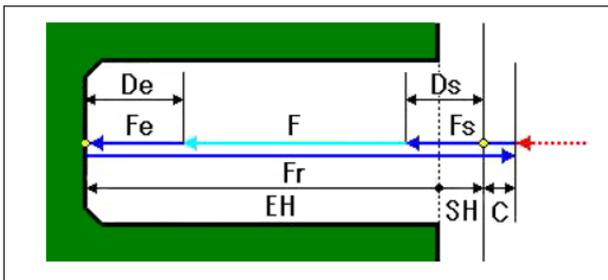
## Reaming G1003

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item	Meaning	
W	Machining type	<ul style="list-style-type: none"> <li>• [CUT]: The tool is retracted with feed speed from the bottom of the drilling.</li> <li>• [RAPID]: The tool is retracted in rapid traverse from the bottom of the drilling.</li> <li>• [DWELL]: After dwelling at the bottom of the drilling, the tool is retracted with feed speed.</li> </ul>
I	Ref.point mode	<ul style="list-style-type: none"> <li>• [INIT-1]: When traversing between borings, the return is made to reference point R. At the end, the return is made to point I. (Initial value).</li> <li>• [INIT-2]: All movements between borings, including the last return, are made to point I.</li> <li>• [REF]: All movements between borings, including the last return, are made to point R.</li> </ul>
J	I Point	I point coordinates.
L	Cut depth	Drilling depth (radius value, negative value)
C	Clearance	Distance between workpiece surface and R-position (Radius value, positive value)
F	Feed rate	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the drilling. If this is missing, 0 is assumed. (In seconds, positive value). Only in machining type [DWELL].
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>

Detail		
Data item		Meaning
A*	Start cut depth	The start feed S applies for the start cut depth A
S*	Start feed rate	Feed speed at start
D*	End cut depth	The end feed E applies for the end cut depth D
E*	End feed rate	Feed speed at end



tool track

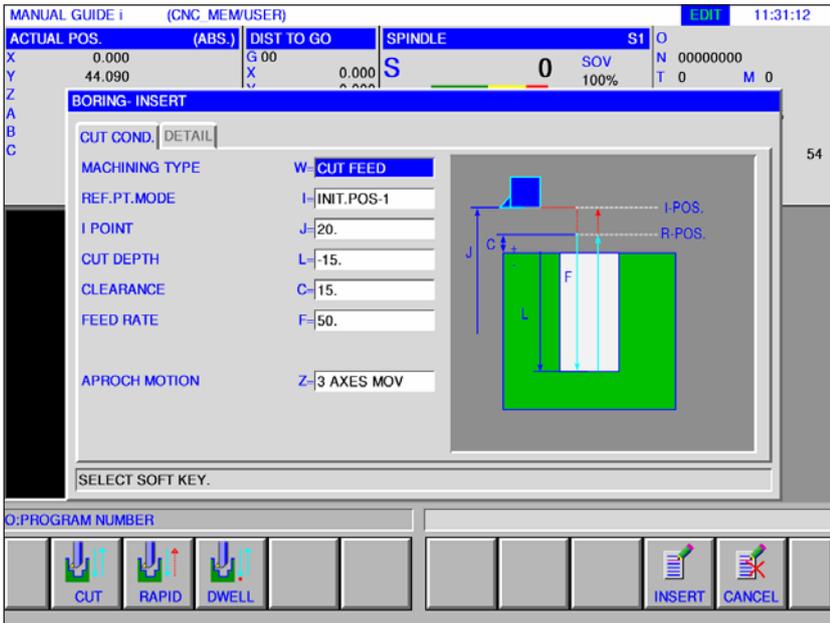
**Cycle description:**

- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the machining end position.
- 3 Stop spindle.
- 4 The spindle is turned in the opposite direction.
- 5 The tool is moved with feed speed (F) into the position "Machining start position + clearance (C)".
- 6 The spindle changes to the normal direction of rotation.



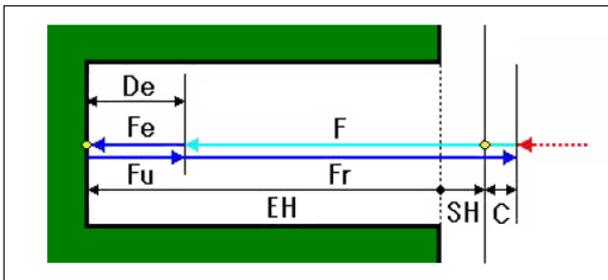
## Boring G1004

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item	Meaning	
W	Machining type	<ul style="list-style-type: none"> <li>• [CUT]: The tool is retracted with feed speed from the bottom of the drilling.</li> <li>• [RAPID]: The tool is retracted in rapid traverse from the bottom of the drilling.</li> <li>• [DWELL]: After dwelling at the bottom of the drilling, the tool is retracted with feed speed.</li> </ul>
I	Ref.point mode	<ul style="list-style-type: none"> <li>• [INIT-1]: When traversing between borings, the return is made to reference point R. At the end, the return is made to point I. (Initial value).</li> <li>• [INIT-2]: All movements between borings, including the last return, are made to point I.</li> <li>• [REF]: All movements between borings, including the last return, are made to point R.</li> </ul>
J	I Point	I point coordinates.
L	Cut depth	Drilling depth (radius value, negative value)
C	Clearance	Distance between workpiece surface and R-position (Radius value, positive value)
F	Feed rate	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the drilling. If this is missing, 0 is assumed. (In seconds, positive value). Only in machining type [DWELL].
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>

Detail		
Data item		Meaning
A*	Start cut depth	The start feed S applies for the start cut depth A
S*	Start feed rate	Feed speed at start
D*	End cut depth	The end feed E applies for the end cut depth D
E*	End feed rate	Feed speed at end

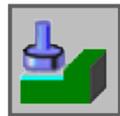


tool track

**Cycle description:**

- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the machining end position.
- 3 The tool is moved with retraction speed (Fr) into the position "Machining start position + clearance (C)".





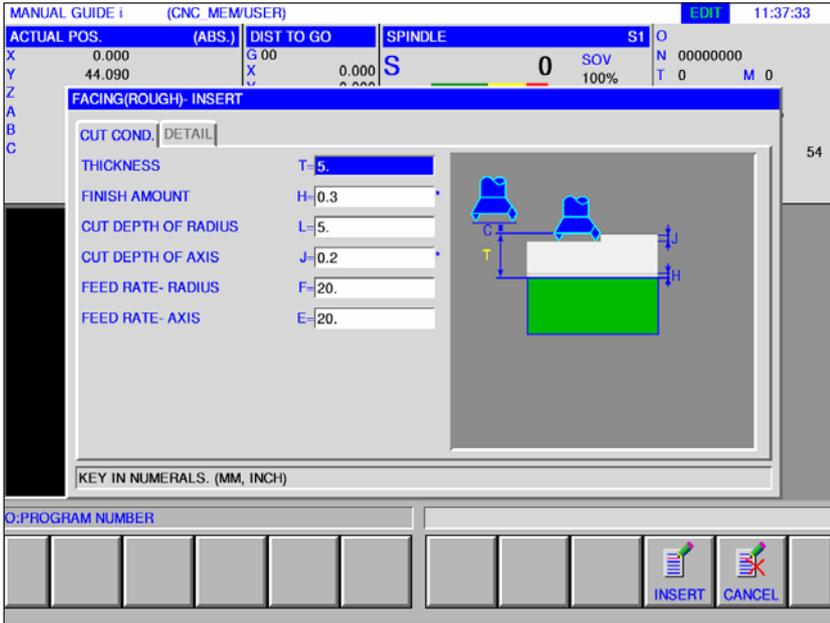
## Face machining

- Facing rough G1020
- Facing finish G1021



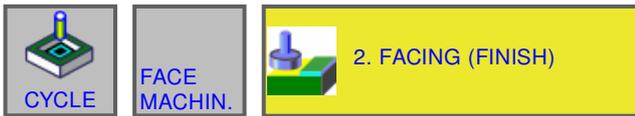
## Facing (rough) G1020

Entry fields marked with \* are optional and should not be populated.



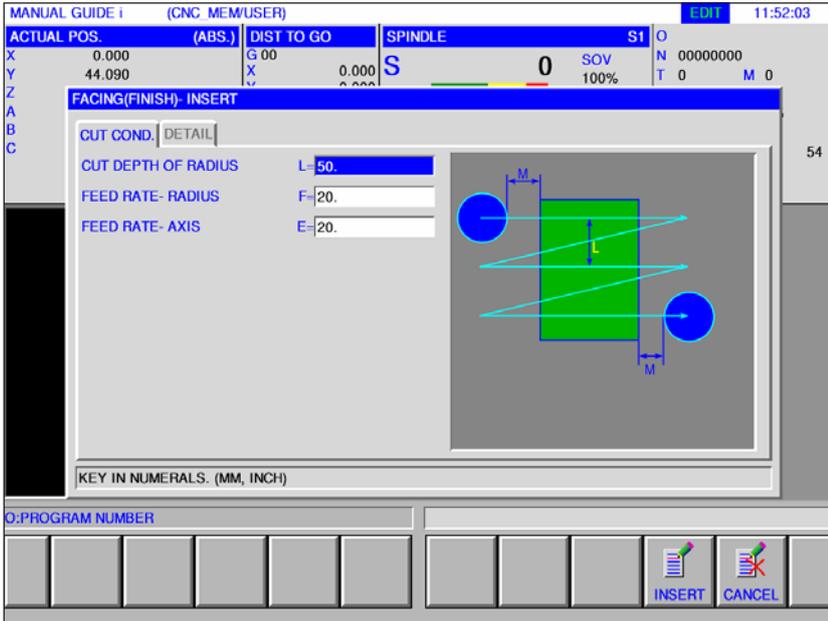
Cutting condition		
Data item		Meaning
T	Thickness	Face mill diameter
H*	Finish amount	Finished dimension in plane machining.
L	Cut depth of radius	Cut depth in tool radius direction to the next cutting track.
J*	Cut depth of axis	Cut depth in tool axis direction per cutting process.
F	Feed rate radius	Feed speed when cutting in tool radius direction.
E	Feed rate axis	Feed speed when cutting in the tool axis direction.

Detail		
Data item		Meaning
I	1 <sup>st</sup> feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cutting method	<ul style="list-style-type: none"> <li>• [SINGLE]: Cutting in tool radius direction is always done in the same direction.</li> <li>• [ZIGZAG]: Cutting in tool radius direction is done forwards and backwards.</li> </ul>
P	Path move method	<ul style="list-style-type: none"> <li>• [PULL]: Return to point R before approaching the start point for the next cutting track (in tool axis direction).</li> <li>• [KEEP]: Direct approach to the start point of the next cutting track without returning to point R.</li> </ul> Only if machining method [ZIGZAG]
V	Path move feed rate	Traverse speed with which the tool moves to the start point for the next cutting track. If the feed speed is set to 0, the tool is traversed in rapid traverse. Only if machining method [ZIGZAG]
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
M	Clearance of radius	Distance between the end of the workpiece and the end of the tool in the retraction position (radius value).
A	Cutting direction	<ul style="list-style-type: none"> <li>• [RIGHT]: Cutting is done to the right as shown in the sketch. If both directions are selected, cutting is done in the first cutting track to the right.</li> <li>• [LEFT]: Cutting is done to the left as shown in the sketch. If both directions are selected, cutting is done in the first cutting track to the left.</li> <li>• [UP]: Cutting is done upwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track upwards.</li> <li>• [DOWN]: Cutting is done downwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track downwards.</li> </ul> The actual cutting direction is determined by the coordinates axis, which is shown in the sketch.
B	Cut shift direction	<ul style="list-style-type: none"> <li>• [RIGHT]: Cutting is done to the right during the change between two cutting tracks, as shown in the sketch.</li> <li>• [LEFT]: Cutting is done to the left during the change between two cutting tracks, as shown in the sketch.</li> <li>• [UP]: Cutting is done upwards during the change between two cutting tracks, as shown in the sketch.</li> <li>• [DOWN]: Cutting is done downwards during the change between two cutting tracks, as shown in the sketch.</li> </ul> The actual cutting direction is determined by the coordinates axis, which is shown in the sketch.
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Facing (finish) G1021

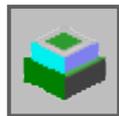
Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
L	Cut depth of radius	Cut depth in tool radius direction to the next cutting track.
F	Feed rate radius	Feed speed when cutting in tool radius direction.
E	Feed rate axis	Feed speed when cutting in the tool axis direction.

Detail		
Data item		Meaning
W	Cutting method	<ul style="list-style-type: none"> <li>• [SINGLE]: Cutting in tool radius direction is always done in the same direction.</li> <li>• [ZIGZAG]: Cutting in tool radius direction is done forwards and backwards.</li> </ul>
P	Path move method	<ul style="list-style-type: none"> <li>• [PULL]: Return to point R before approaching the start point for the next cutting track (in tool axis direction).</li> <li>• [KEEP]: Direct approach to the start point of the next cutting track without returning to point R. Only if machining method [ZIGZAG]</li> </ul>
V	Path move feed rate	<p>Traverse speed with which the tool moves to the start point for the next cutting track. If the feed speed is set to 0, the tool is traversed in rapid traverse.</p> <p>Only if machining method [ZIGZAG]</p>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
M	Clearance of radius	Distance between the end of the workpiece and the end of the tool in the retraction position (radius value).
A	Cutting direction	<ul style="list-style-type: none"> <li>• [RIGHT]: Cutting is done to the right as shown in the sketch. If both directions are selected, cutting is done in the first cutting track to the right.</li> <li>• [LEFT]: Cutting is done to the left as shown in the sketch. If both directions are selected, cutting is done in the first cutting track to the left.</li> <li>• [UP]: Cutting is done upwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track upwards.</li> <li>• [DOWN]: Cutting is done downwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track downwards.</li> </ul> <p>The actual cutting direction is determined by the coordinates axis, which is shown in the sketch.</p>
B	Cut shift direction	<ul style="list-style-type: none"> <li>• [RIGHT]: Cutting is done to the right during the change between two cutting tracks, as shown in the sketch.</li> <li>• [LEFT]: Cutting is done to the left during the change between two cutting tracks, as shown in the sketch.</li> <li>• [UP]: Cutting is done upwards during the change between two cutting tracks, as shown in the sketch.</li> <li>• [DOWN]: Cutting is done downwards during the change between two cutting tracks, as shown in the sketch.</li> </ul> <p>The actual cutting direction is determined by the coordinates axis, which is shown in the sketch.</p>
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>





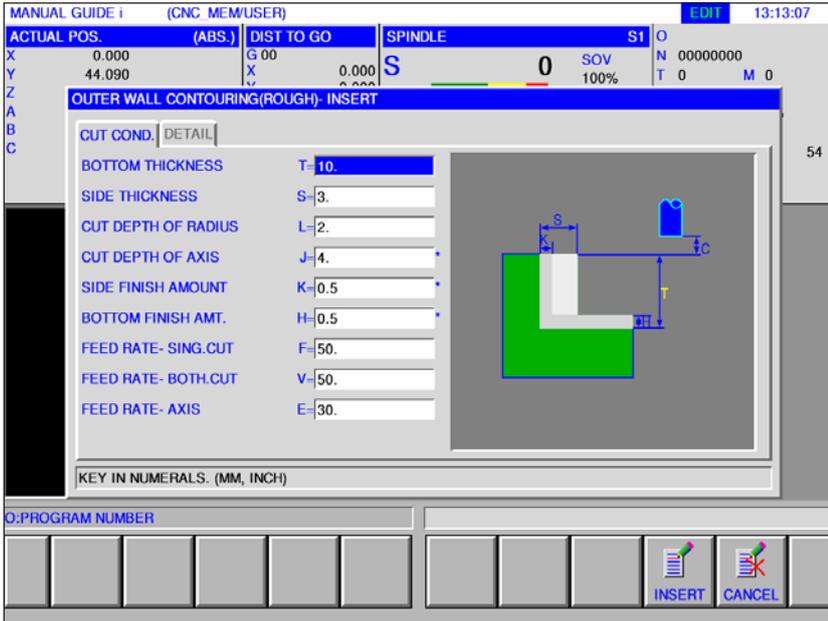
## Contouring

- Outer wall countering rough G1060
- Outer wall countering bottom finish G1061
- Outer wall countering side finish G1062
- Outer wall countering chamfer G1063
- Inner wall countering rough G1064
- Inner wall countering bottom finish G1065
- Inner wall countering side finish G1066
- Inner wall countering chamfer G1067
- Partial contouring rough G1068
- Partial contouring bottom finish G1069
- Partial contouring side finish G1070
- Partial contouring chamfer G1071



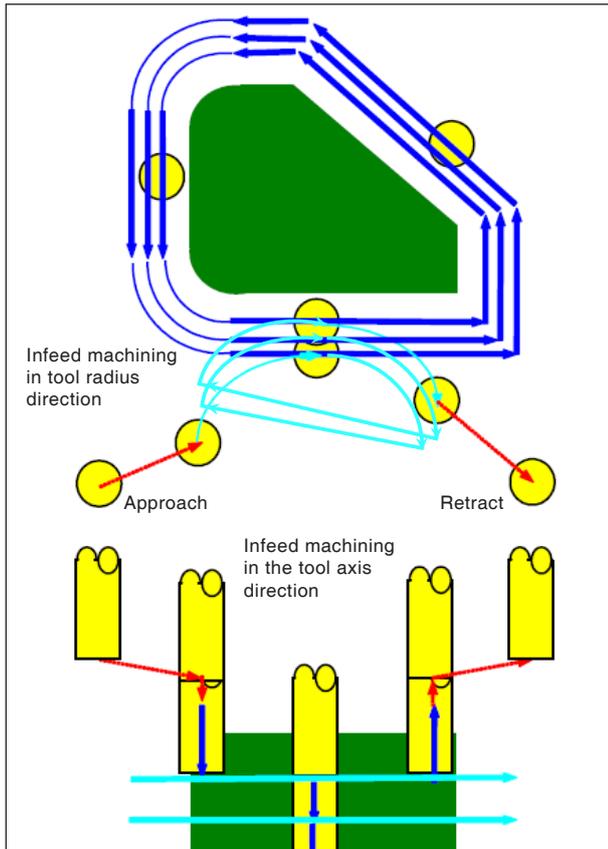
## Outer wall (rough) G1060

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item	Meaning	
T	Bottom thickness	Finishing dimension at the bottom in side surface machining (Radius value, positive value)
S	Side thickness	Side surface machining allowance (radius value, positive value).
L	Cut depth of radius	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
J*	Cut depth of axis	Cut depth in tool axis direction per cutting process (radius value, positive value). Standard is (Machining dimension at bottom - Finishing dimension at bottom).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
M	1 <sup>st</sup> feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIK]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



Tool track G1060, G1064, G1068

### Cycle description for G1060, G1064 and G1068:

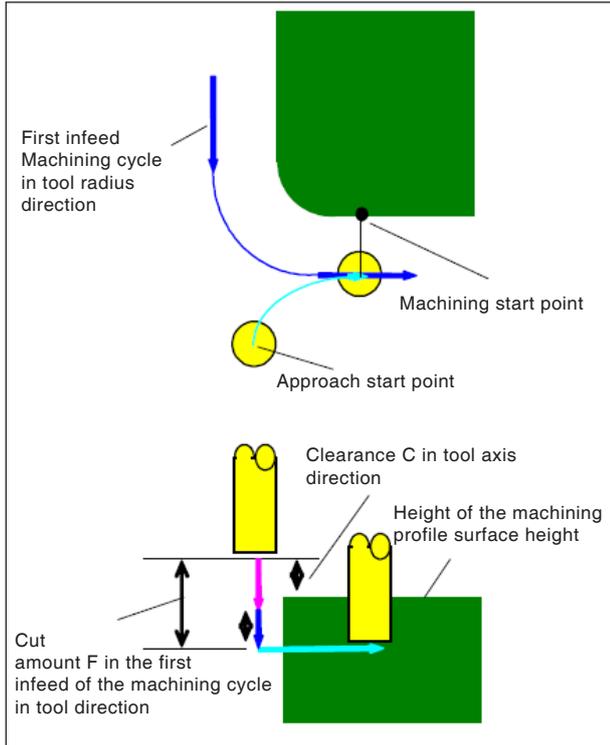
The side surface contour of the machining profile is exhausted. The following tool track is generated.

- 1 The tool is moved to the above approach start point.
- 2 The tool is moved to the height of the machining surface.
- 3 The tool cuts along the side surface contour of the machining profile.

The tool cuts by infeed machining in the tool radius direction until the machining allowance in the tool radius direction is exhausted.

- 4 Steps <2> and <3> are repeated until the machining allowance in the tool axis direction is exhausted.

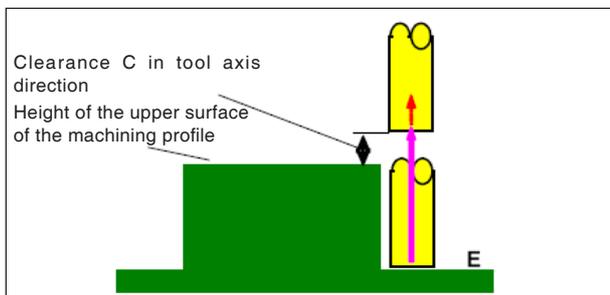
- 5 The tool retracts.



Approach G1060, G1064, G1068

**Approach cycle description:**

- 1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".
- 2 The tool traverses into the position "Amount to be cut in the first infeed machining cycle in the tool axis direction - clearance (C) in the tool axis direction" with the feed speed (E) specified for movement in the tool axis direction.
- 3 The tool approaches in the tool radius direction the start point for the first infeed machining in the tool radius direction.

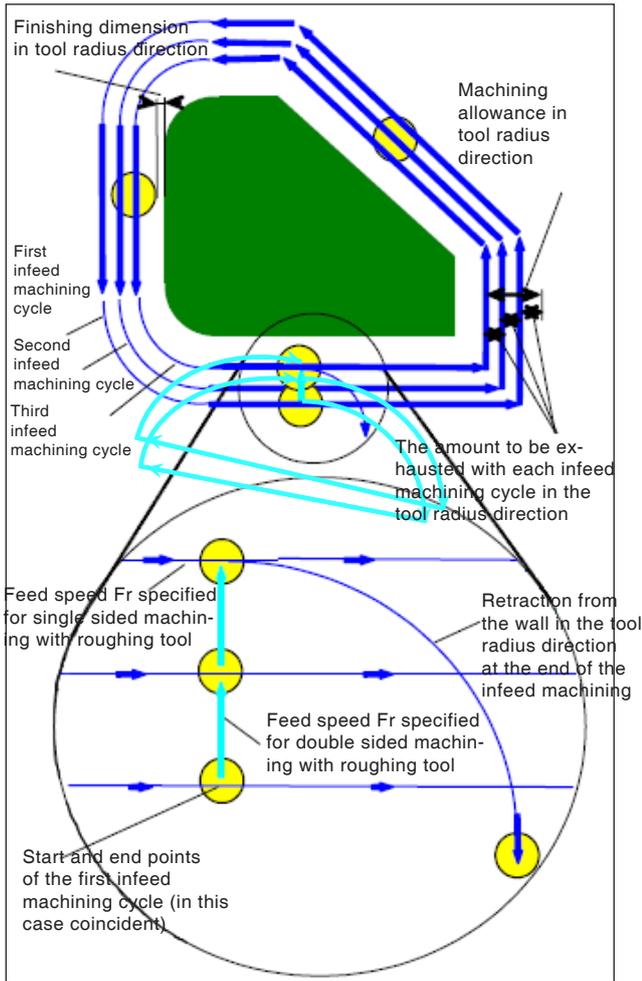


Retraction G1060, G1064, G1068

**Retraction cycle description:**

- 1 The tool is moved from the approach end point to the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction" with the feed speed (E) specified for movement in the tool axis direction.





*Infeed machining in the tool radius direction  
G1060, G1064, G1068*

**Tool radius alignment cycle description:**

**1** The tool is moved for cutting along the contour from the first infeed machining cycle start point to the end point with the feed speed (F) specified for single side machining with roughing tool.

**2** The tool approaches as follows.

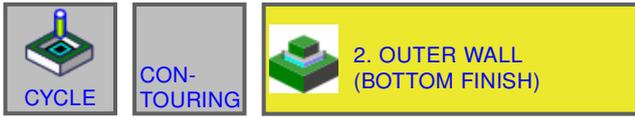
If the infeed machining start point coincides with the infeed machining end point: The tool approaches the next infeed machining start point in the normal direction with the feed speed (F) specified for double-sided machining with the roughing tool.

If the infeed machining start point does not coincide with the infeed machining end point: The tool approaches the second infeed machining start point.

**3** The tool is moved for cutting along the machining profile contour with the feed speed (F) specified for single side machining with roughing tool.

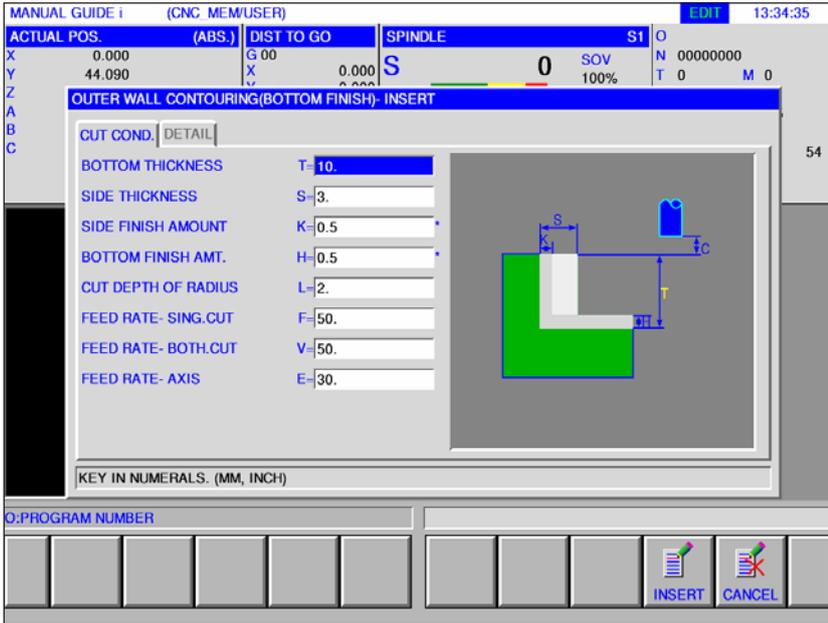
**4** Steps <2> and <3> are repeated until the machining allowance (machining allowance in the tool axis direction) is exhausted.

**5** The tool retracts.



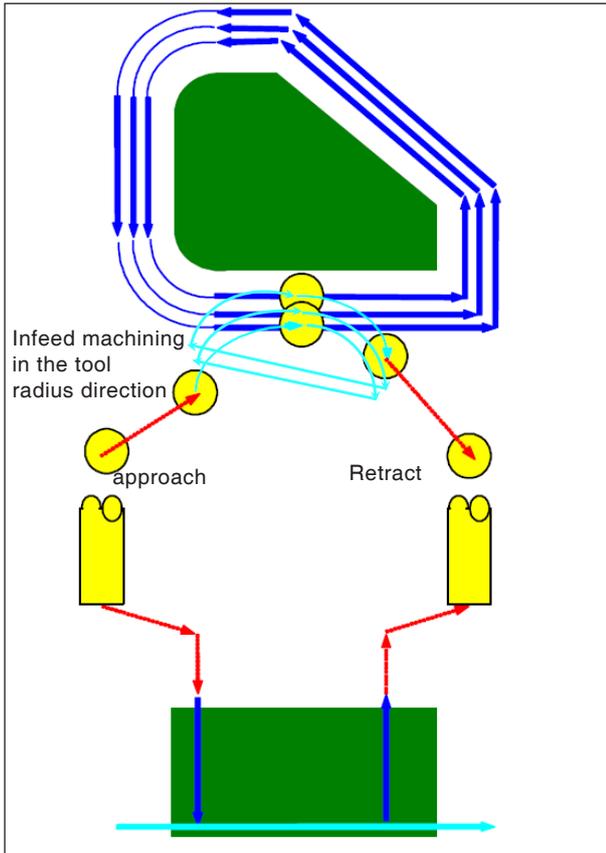
## Outer wall (bottom finish) G1061

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
T	Bottom thickness	Finishing dimension at the bottom in side surface machining (Radius value, positive value)
S	Side thickness	Side surface machining allowance (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
L	Cut depth of radius	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UPCUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWNCUT] Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



Tool track G1061, G1065, G1069

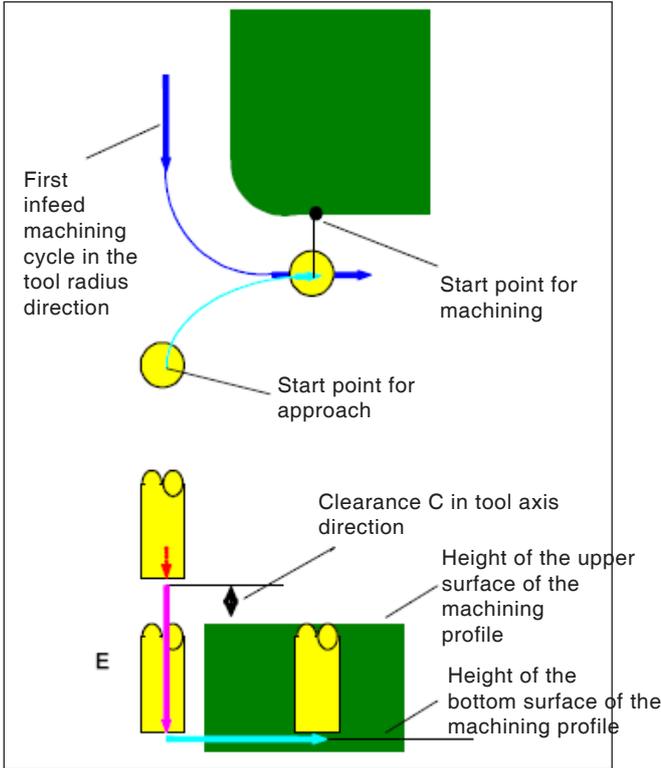
#### Cycle description for G1061, G1065 and G1069:

The bottom surface of the side surface contour of the machining profile is machined. The following tool track is generated.

- 1 The tool is moved to the machining profile approach start point.
- 2 The tool is moved to the height of the machining surface of the machining profile.
- 3 The tool is moved to cut along the side surface contour of the machining profile.

The tool cuts by infeed machining in the tool radius direction until the machining allowance in the tool radius direction is exhausted.

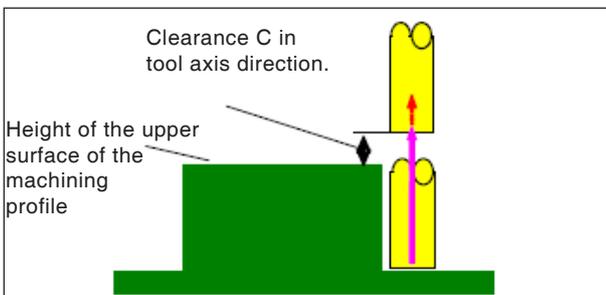
- 4 The tool retracts.



Approach G1061, G1065, G1069

**Approach cycle description:**

- 1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".
- 2 The tool traverses into the position "Machining profile bottom surface + machining allowance (Vt) in the tool axis direction" with the feed speed (E) specified for movement in the tool axis direction.
- 3 The tool approaches in the tool radius direction the start point for the infeed machining in the tool radius direction.



Retraction track G1061, G1065, G1069

**Retraction cycle description:**

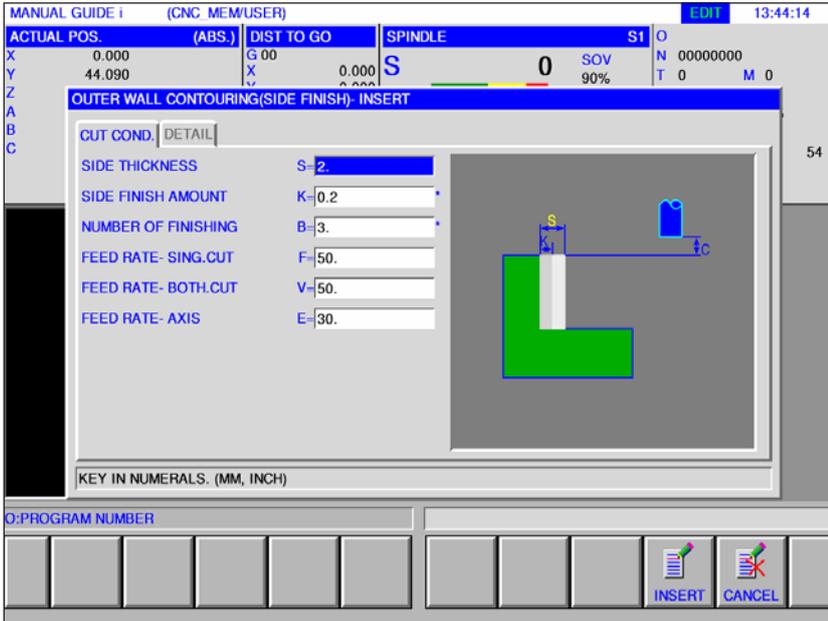
- 1 The tool is moved from the approach end point with rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".

Infeed machining in the tool radius direction. This movement is the same as for contour machining (roughing). Details of contour machining (roughing) can be found in the relevant descriptions.



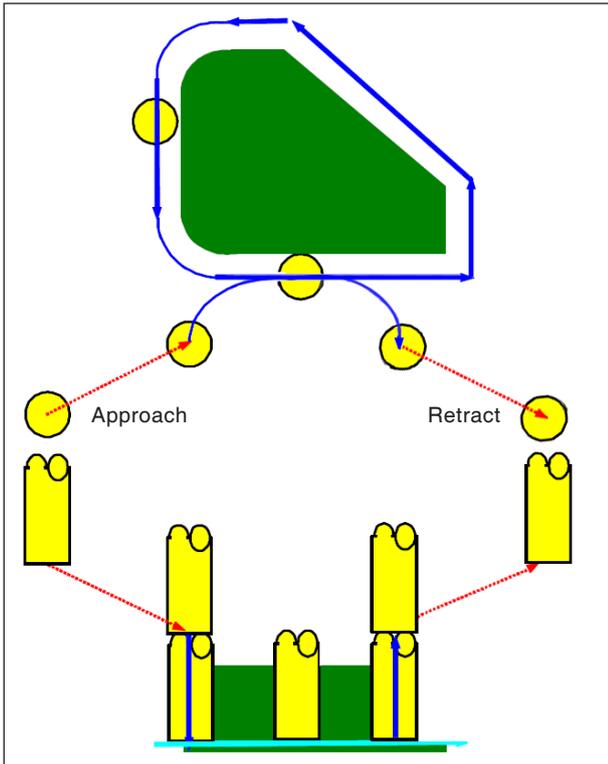
## Outer wall (side finish) G1062

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
S	Side thickness	Side surface machining allowance (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
B*	Number of finishing	Number of cuts during finishing (positive value) Depth per cut = (superfluous side thickness)/(number of finishing cuts)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UPCUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWNCUT] Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



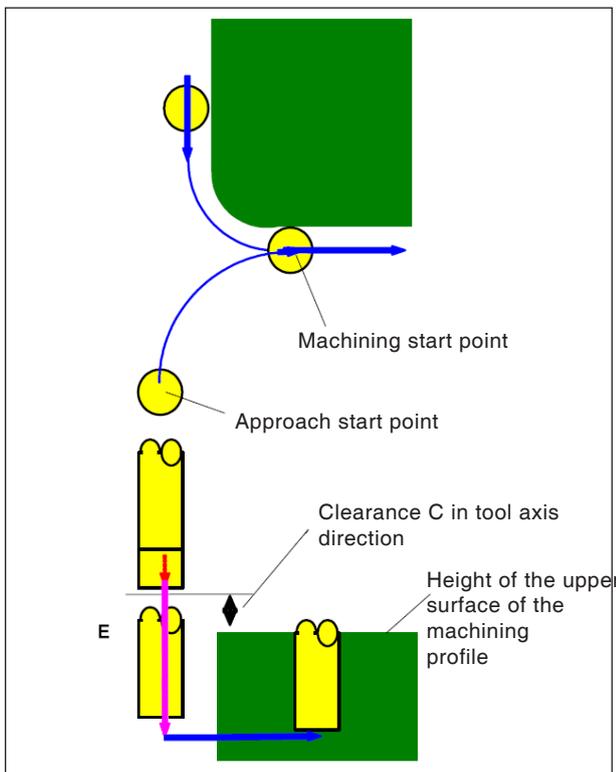
Tool track G1062, G1066, G1070

**Cycle description for G1062, G1066 and G1070:**

- 1 The tool is moved to the above approach start point.
- 2 The tool is moved to the height of the bottom surface of the machining profile.
- 3 The tool is moved to cut along the side surface contour of the machining profile.

The machining allowance ( $V_t$ ) in the tool radius direction is used up in a specified number of finishing machining cycles by infeed machining.

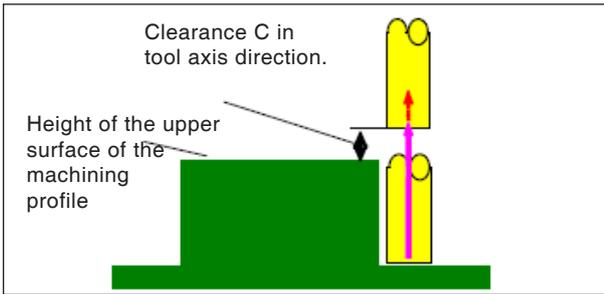
- 4 The tool retracts.



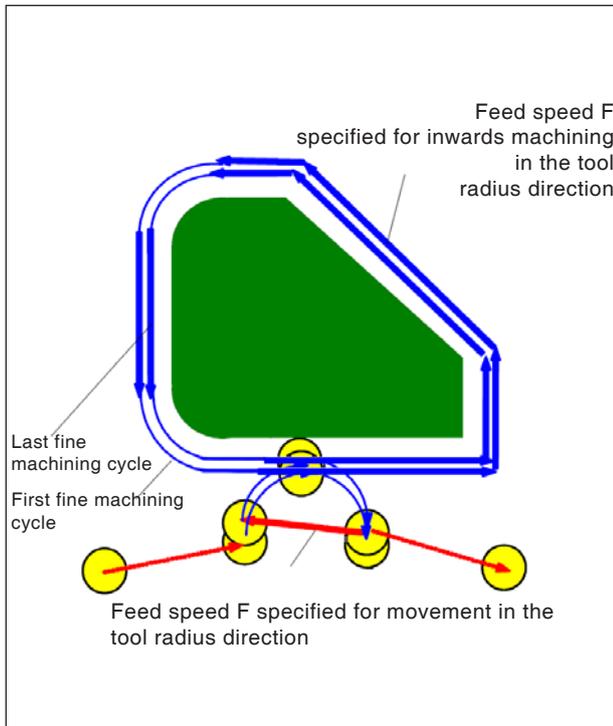
Approach G1062, G1066, G1070

**Approach cycle description:**

- 1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".
- 2 The tool traverses into the position "Machining profile bottom surface + finishing dimension (S) in the tool axis direction + clearance (C) in the tool axis direction" with the feed speed (E) specified for movement in the tool axis direction.
- 3 The tool traverses into the position "Machining profile bottom surface + finishing dimension (Tt) in the tool axis direction" with the feed speed (Ft) specified for cutting the machining profile.
- 4 The tool approaches in the tool radius direction the start point for the infeed machining cycle in the tool radius direction.



Retraction G1062, G1066, G1070



Retraction G1062, G1066, G1070

**Cycle description process:**

- 1 The tool is moved from the approach end point with rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".

**Infeed machining in tool radius direction:**

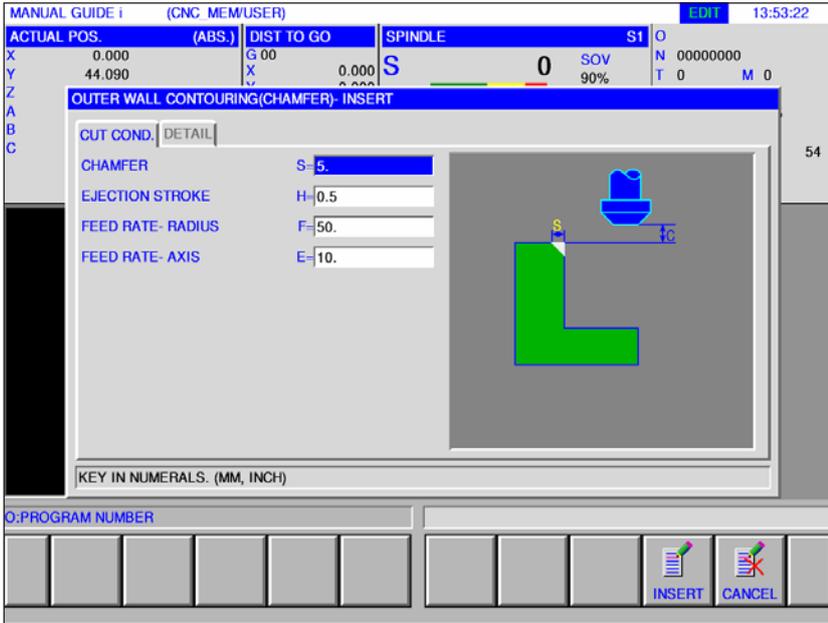
The tool moves to the cut start point in the tool radius direction with the feed speed (F) specified for cutting in the tool radius direction.

- 2 The tool is moved along the contour from the first infeed machining cycle start point to the first infeed machining end point with the feed speed (F) specified for cutting in the tool radius direction.
- 3 The tool retracts from the start point for machining in the tool radius direction with the feed speed (F) specified for cutting in the tool radius direction.
- 4 The tool moves to the next infeed machining cycle start point in accordance with the type of movement specified for infeed machining.
- 5 Steps <2> to <4> are repeated as often as the number of finishing machining cycles specified.



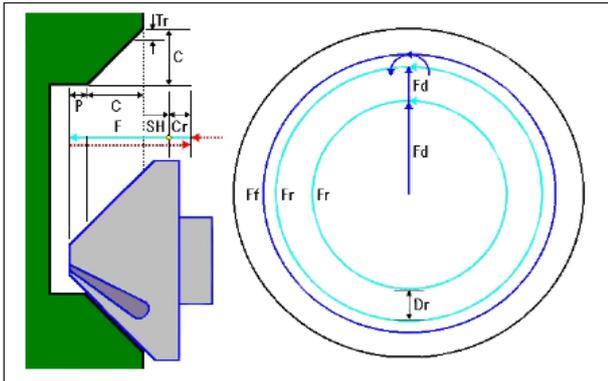
## Outer wall (chamfer) G1063

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
S	Side thickness	Chamfer length (radius value, positive value)
H	Ejection stroke	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value)
F	Feed rate radius	Feed speed when cutting in tool radius direction
E	Feed rate axis	Feed speed when cutting in the tool axis direction

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UPCUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWNCUT] Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



Tool track G1063, G1067, G1071

### Cycle description for G1063, G1067 and G1071:

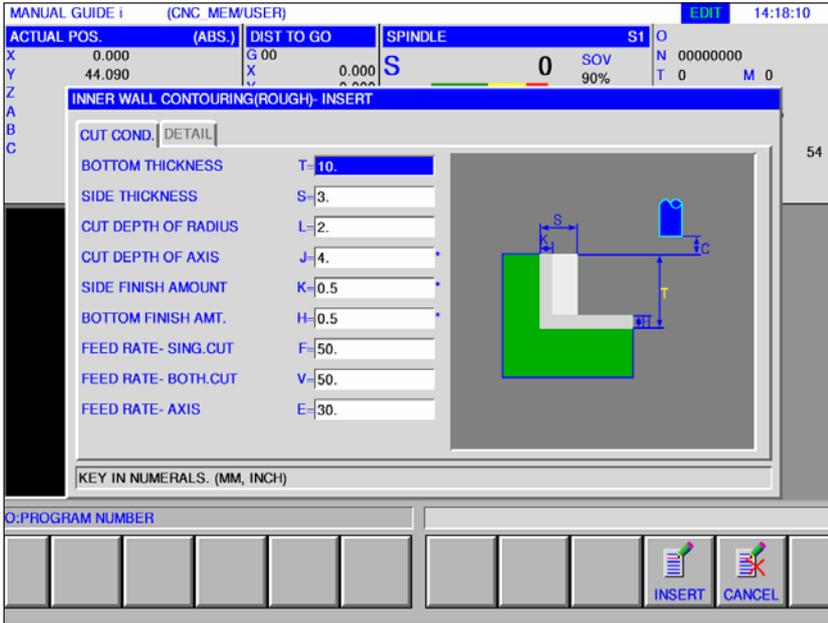
- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (Cr)".
- 2 The tool is moved into the position "Chamfering amount (C) + ejection path (P)" with the feed speed (F) specified for cutting.
- 3 The tool cuts into the section to be chamfered to the cut depth (Dr) in the tool radius direction with feed speed (F).
- 4 The tool performs the finishing machining with the feed speed (F) specified for finishing.
- 5 The tool is moved in rapid traverse into the position "Cutting start point + clearance (Cr)".





## Inner wall (rough) G1064

Entry fields marked with \* are optional and should not be populated.



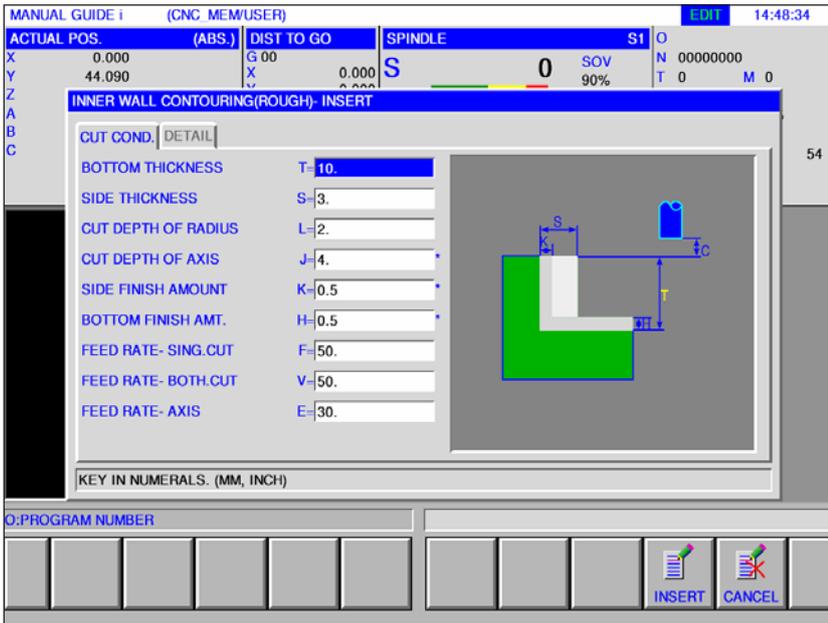
Cutting condition		
Data item	Meaning	
T	Bottom thickness	Finishing dimension at the bottom in side surface machining (Radius value, positive value)
S	Side thickness	Side surface machining allowance (radius value, positive value).
L	Cut depth of radius	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
J*	Cut depth of axis	Cut depth in tool axis direction per cutting process (radius value, positive value). Standard is (Machining dimension at bottom - Finishing dimension at bottom).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
M	1 <sup>st</sup> feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Inner wall (bottom finish) G1065

Entry fields marked with \* are optional and should not be populated.



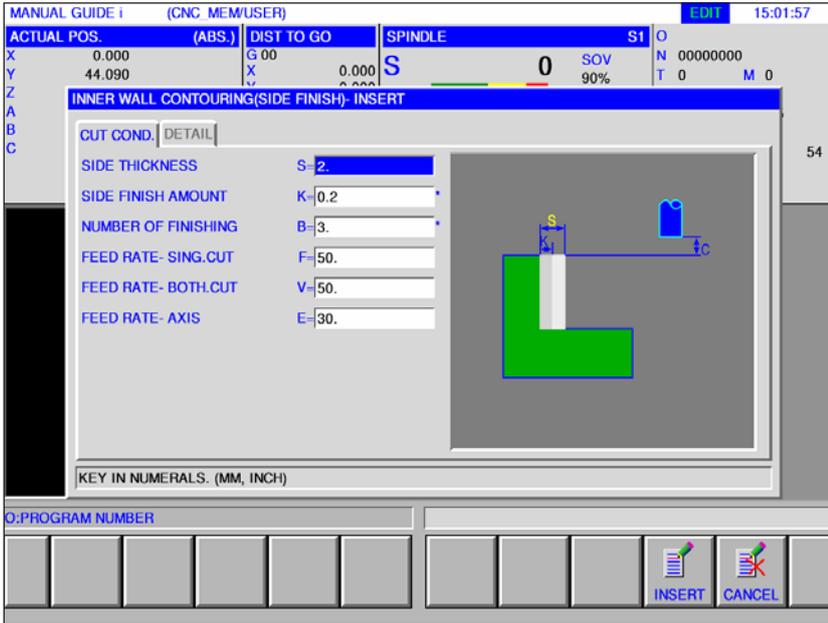
Cutting condition		
Data item		Meaning
T	Bottom thickness	Finishing dimension at the bottom in side surface machining (Radius value, positive value)
S	Side thickness	Side surface machining allowance (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
L	Cut depth of radius	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Inner wall (side finish) G1066

Entry fields marked with \* are optional and should not be populated.



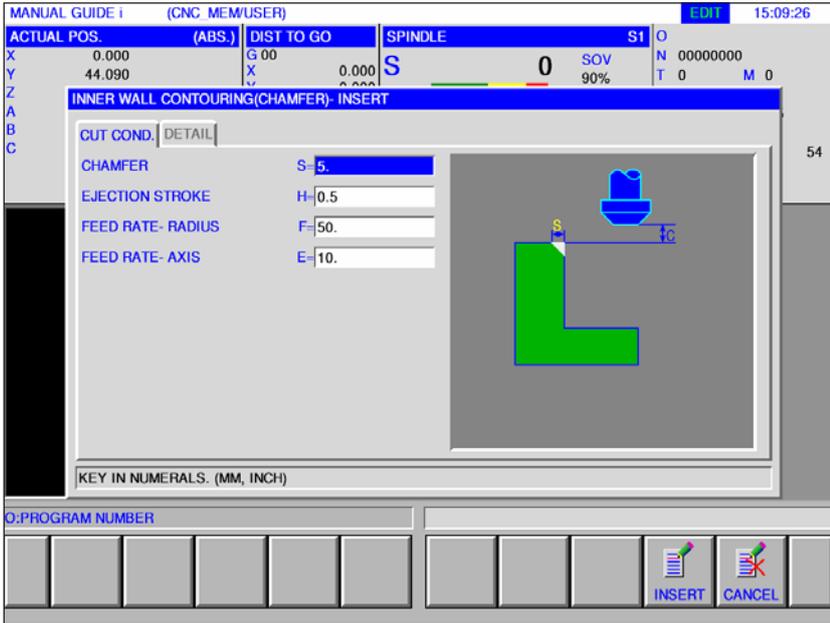
Cutting condition		
Data item		Meaning
S	Side thickness	Side surface machining allowance (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
B*	Number of finishing	Number of cuts during finishing (positive value) Depth per cut = (superfluous side thickness)/(number of finishing cuts)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Inner wall (chamfer) G1067

Entry fields marked with \* are optional and should not be populated.



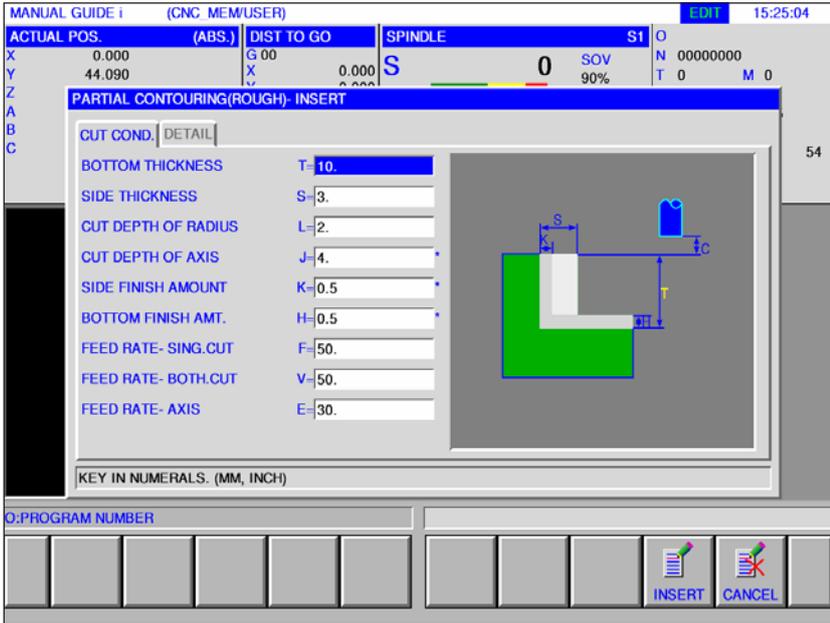
Cutting condition		
Data item		Meaning
S	Chamfer	Chamfer length (radius value, positive value)
H	Ejection stroke	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value)
F	Feed rate radius	Feed speed when cutting in tool radius direction
E	Feed rate axis	Feed speed when cutting in the tool axis direction

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Partial (rough) G1068

Entry fields marked with \* are optional and should not be populated.



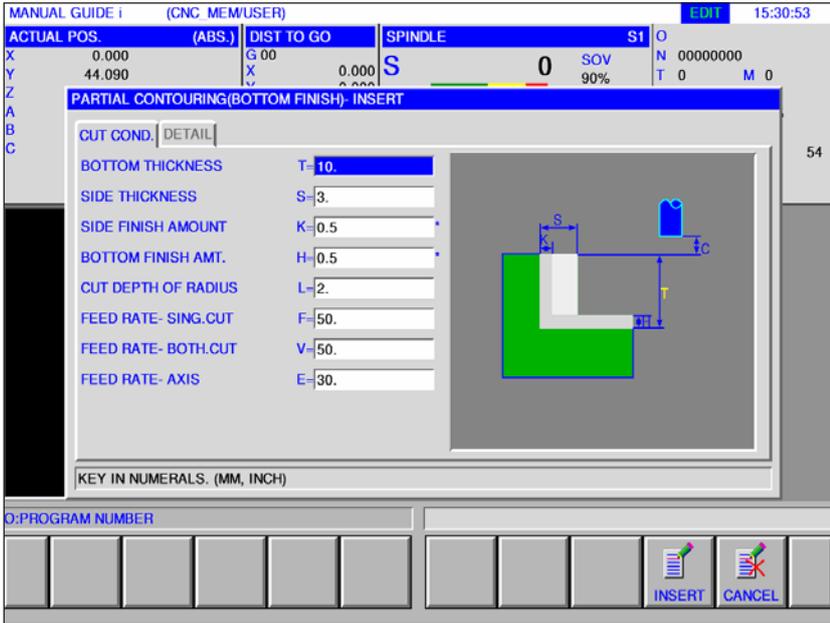
Cutting condition		
Data item	Meaning	
T	Bottom thickness	Finishing dimension at the bottom in side surface machining (Radius value, positive value)
S	Side thickness	Side surface machining allowance (radius value, positive value).
L	Cut depth of radius	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
J*	Cut depth of axis	Cut depth in tool axis direction per cutting process (radius value, positive value). Standard is (Machining dimension at bottom - Finishing dimension at bottom).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
M	1 <sup>st</sup> feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Partial (bottom finish) G1069

Entry fields marked with \* are optional and should not be populated.



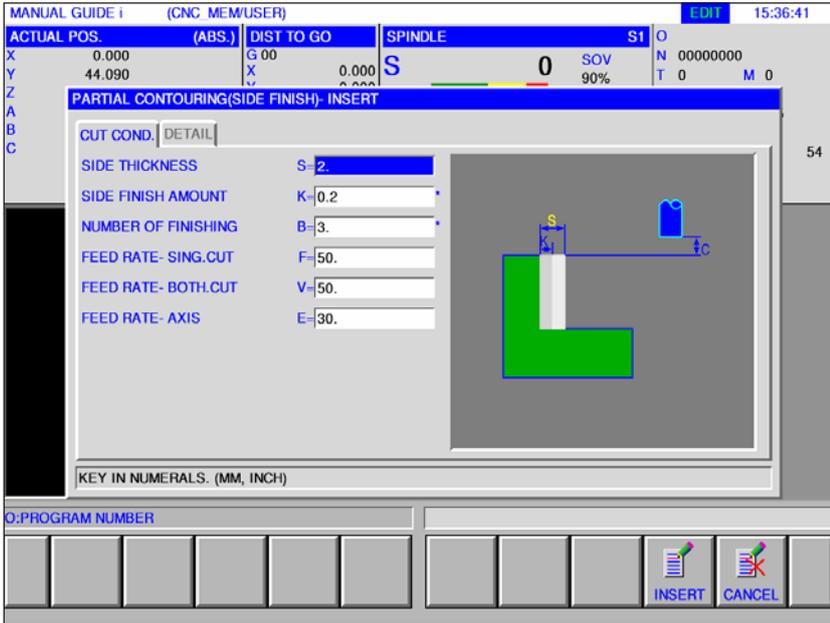
Cutting condition		
Data item	Meaning	
T	Bottom thickness	Finishing dimension at the bottom in side surface machining (Radius value, positive value)
S	Side thickness	Side surface machining allowance (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
L	Cut depth of radius	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Partial (side finish) G1070

Entry fields marked with \* are optional and should not be populated.



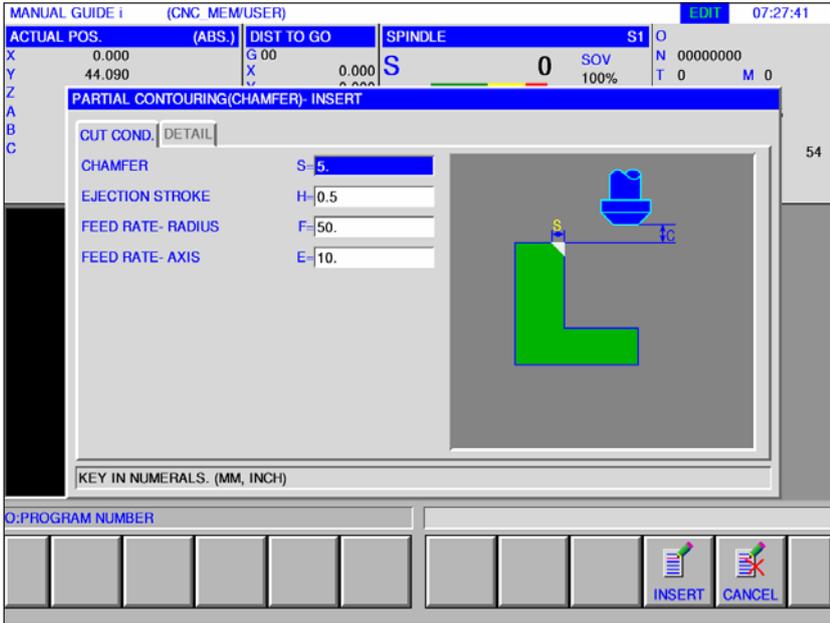
Cutting condition		
Data item		Meaning
S	Side thickness	Side surface machining allowance (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
B*	Number of finishing	Number of cuts during finishing (positive value) Depth per cut = (superfluous side thickness)/(number of finishing cuts)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



## Partial (chamfer) G1071

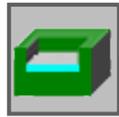
Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
S	Chamfer	Chamfer length (radius value, positive value)
H	Ejection stroke	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value)
F	Feed rate radius	Feed speed when cutting in tool radius direction
E	Feed rate axis	Feed speed when cutting in the tool axis direction

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of the unmachined part for machining and the machining start point (point R) in the direction of the tool axis (radius value)
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> <li>• [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: The approach to the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
R	Approach radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> <li>• [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining.</li> <li>• [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.</li> </ul>
X	Escape radius / -distance	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (Radius value, positive value)
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>





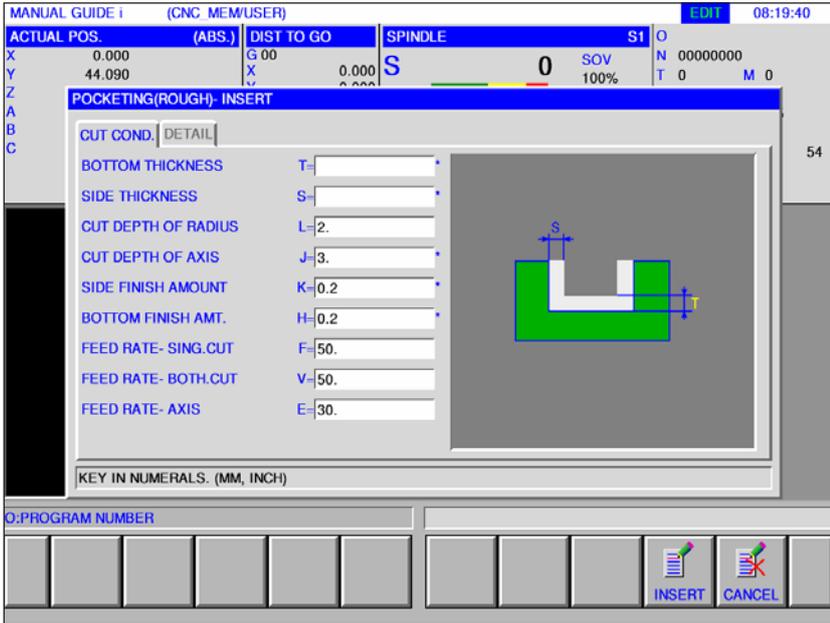
## Pocketing

- Pocketing rough G1040
- Pocketing bottom finish G1041
- Pocketing side finish G1042
- Pocketing chamfer G1043



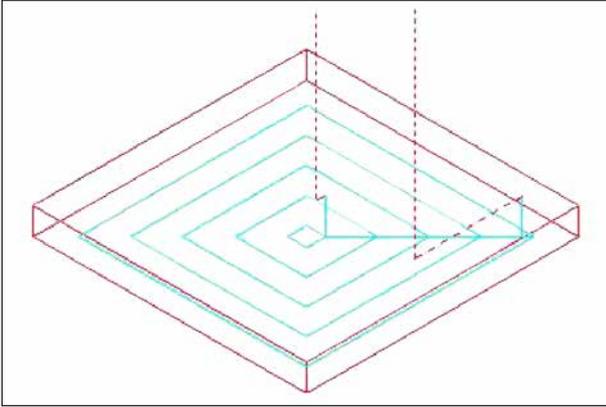
## Pocketing (rough) G1040

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
T*	Bottom thickness	Machining allowance at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
S*	Side thickness	Machining allowance at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value) Remarks: If side finishing dimension and bottom finishing dimension are ignored, a complete pocket is machined.
L	Cut depth of radius	Cut depth at the side surface (in tool radius direction) per cutting process (radius value, positive value).
J*	Cut depth of axis	Cut depth in tool axis direction per cutting process (radius value, positive value).
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
B	Clearance of radius	Distance between the pocket wall and the tool retraction position in tool radius direction (radius value, positive value).
C	Clearance of axis	Distance between the surface of an unmachined part to be machined and the machining start point (point R) in the tool axis direction (radius value, positive value).
Z	Approach motion	• [3 AXIS]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.
X	Cut depth method	<ul style="list-style-type: none"> <li>• [STRAIT]: Plunging in the form of a straight line.</li> <li>• [HELICL]: Plunging on spiral path (helix).</li> </ul>
A*	Cut angle	Angle under which the tool cuts obliquely or in a spiral into the pocket. The unmachined part is viewed as 0 degrees. (In 1-degree increments, positive value).



*tool track*

### **Cycle description**

The interior of a pocket machining profile is cut out spirally.

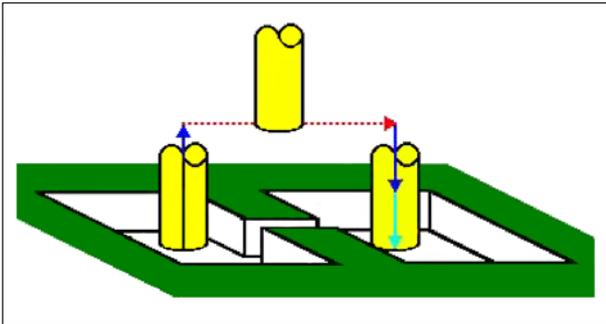
The following tool track is generated.

For a pocket machining profile, several island machining profiles and several cavity machining profiles can be defined. Island machining profiles remain unmachined. Cavity machining profiles are bypassed so that they are not machined.

The tool track is produced in a manner such that a collision to be expected with pocket machining profiles or island machining profiles is avoided.

The tool track generated is effective, as retraction in the direction of the tool axis is avoided as far as possible. Infeed machining in the tool axis direction is possible with this tool track.

Only given machine allowances can be used. Machining can be designed either for upcut or for downcut machining. The machining direction around the islands is controlled automatically.



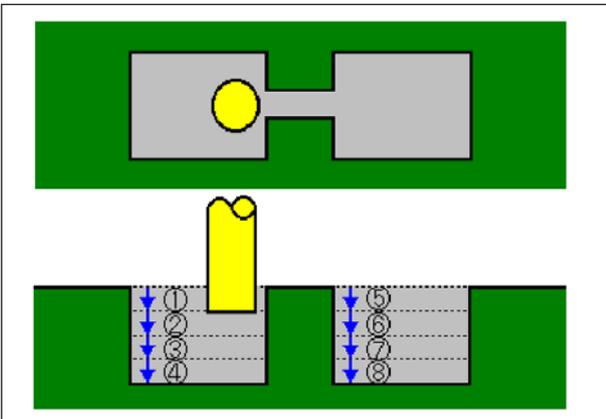
Tool track

It is possible to cut into the machining profile from the inside and from the outside.

If a residue that has not been cut away remains at a corner, it is possible to determine this automatically and cut it away.

It can be cut at any angle in the tool axis direction. This also determines the machining start point automatically.

The movement method for tool movement can be selected. However, a cutting depth on the top surface of island machining profiles can also be determined automatically.



Tool track

In the following pocket machining profile, where the tool can move through the pocket, the tool is lifted off automatically, so that only the pocket area that has to be processed is machined.

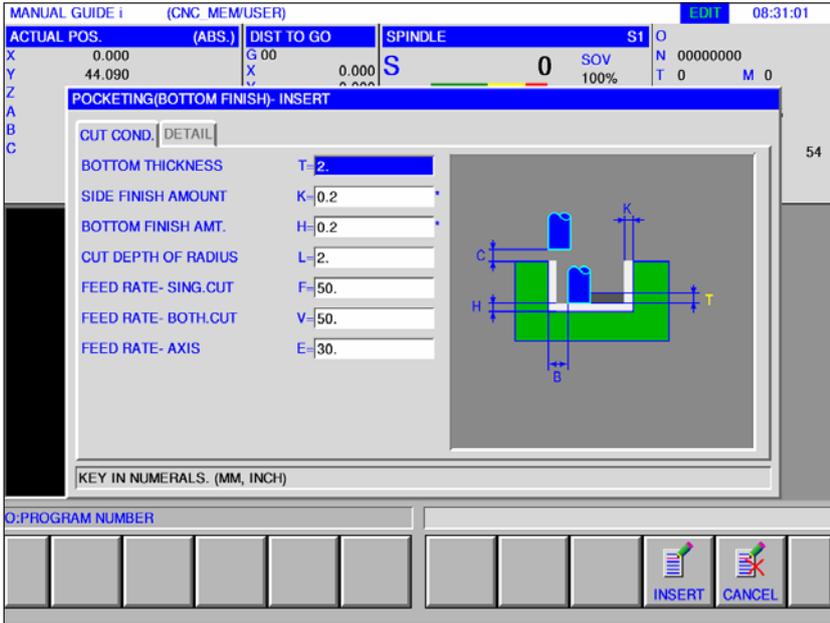
If a number of cuts have to be made in the tool axis direction, each area is machined completely before another is started.

If a number of cuts have to be made in the tool axis direction, each area is machined completely before another is started.



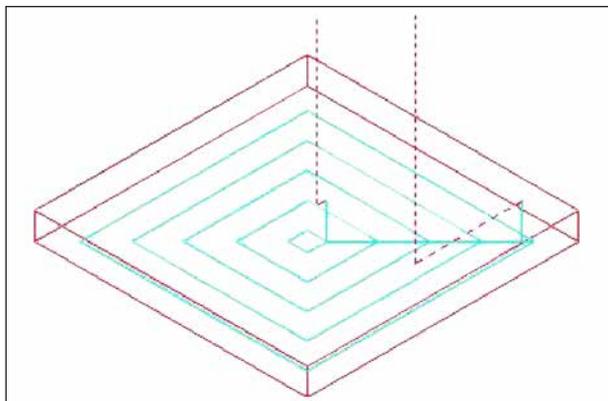
## Pocketing (bottom finish) G1041

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
T	Bottom thickness	Machining allowance at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
L	Cut depth of radius	Cut depth at the side surface (in tool radius direction) per cutting process (radius value, positive value).
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
B	Clearance of radius	Distance between the pocket wall and the tool retraction position in tool radius direction (radius value, positive value).
C	Clearance of axis	Distance between the surface of an unmachined part to be machined and the machining start point (point R) in the tool axis direction (radius value, positive value).
Z	Approach motion	• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.
X	Cut depth method	<ul style="list-style-type: none"> <li>• [STRAIT]: Plunging in the form of a straight line.</li> <li>• [HELICL]: Plunging on spiral path (helix).</li> </ul>
A*	Cut angle	Angle under which the tool cuts obliquely or in a spiral into the pocket. The unmachined part is viewed as 0 degrees. (In 1-degree increments, positive value).



Tool track

**Cycle description**

The bottom surface of a pocket machining profile is finished spirally. The tool track for this is the same as with pocket milling (roughing).

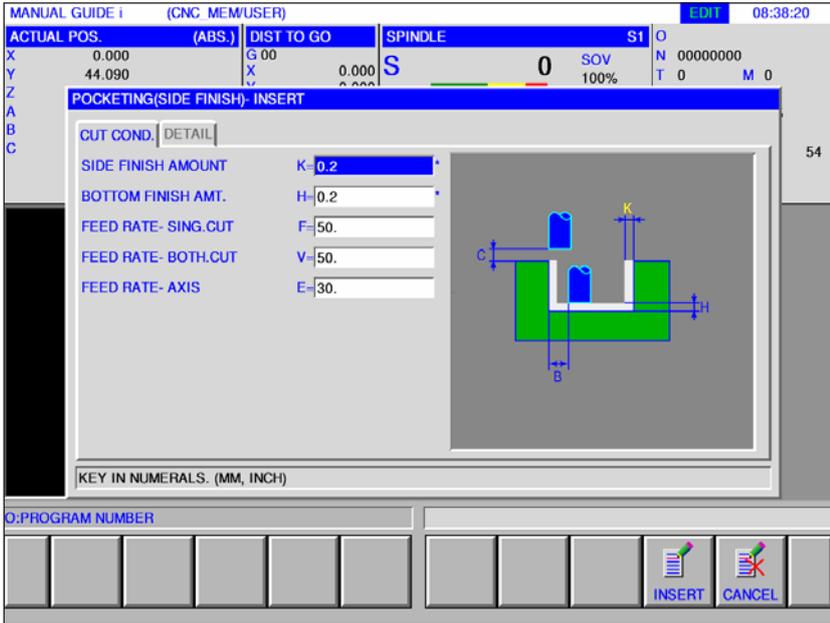
However, there is no infeed machining in the tool axis direction.

Neither is the top surface of island machining profiles machined.



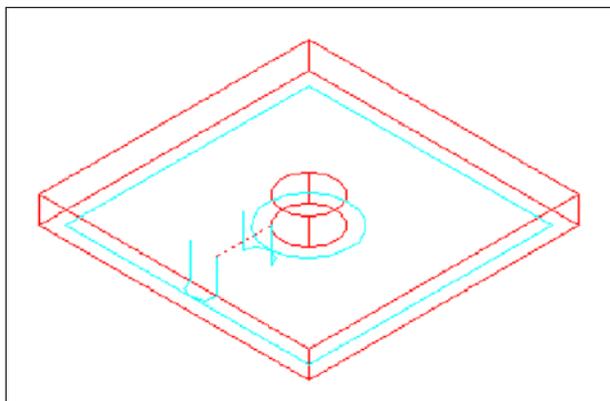
### Pocketing (side finish) G1042

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
K*	Side finish amount	Finishing dimension at the side surface. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
H*	Bottom finish amount	Finishing dimension at the bottom in side surface machining The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Feed rate single cut	Feed speed when cutting with only one side of the cut of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the face, except during on-cutting.
V	Feed rate both cut	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for on-cutting.
E	Feed rate axis	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of an unmachined part to be machined and the machining start point (point R) in the tool axis direction (radius value, positive value).
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> </ul> The setting is specified as fixed and cannot be changed.
R	Approach radius / -distance	Radius, if [ARC] is specified.
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> </ul> The setting is specified as fixed and cannot be changed.
X	Escape radius / -distance	Radius, only if [ARC] is specified.
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>



Tool track

**Cycle description**

The side contour of the pocket and of the island machining profile are finished. The tool track for this is the same as for contour machining (Finishing the side surface).

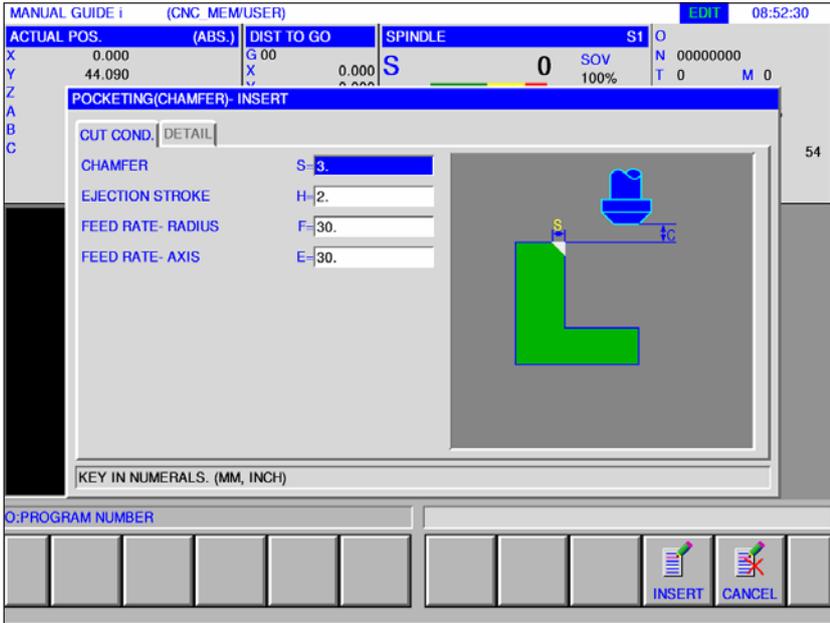
However, the specifications for the following points differ somewhat. There is no infeed machining in the tool radius direction or in the tool axis direction.

Even if you should assume that the tool might collide with a pocket or island machining profile during finishing, no tool track is produced that can prevent this collision.



### Pocketing (chamfer) G1043

Entry fields marked with \* are optional and should not be populated.



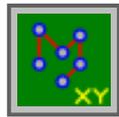
Cutting condition		
Data item		Meaning
S	Chamfer	Chamfer length (radius value, positive value).
H	Ejection stroke	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value).
F	Feed rate radius	Feed speed when cutting in tool radius direction
E	Feed rate axis	Feed speed when cutting in the tool axis direction

Detail		
Data item		Meaning
W	Upcut / downcut	<ul style="list-style-type: none"> <li>• [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise.</li> <li>• [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.</li> </ul>
C	Clearance of axis	Distance between the surface of an unmachined part to be machined and the machining start point (point R) in the tool axis direction (radius value, positive value).
P	Approach type	<ul style="list-style-type: none"> <li>• [ARC]: The approach to the side surface takes the form of an arc.</li> </ul> The setting is specified as fixed and cannot be changed.
R	Approach radius / -distance	Radius, if [ARC] is specified.
A*	Approach angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value)
Q	Escape type	<ul style="list-style-type: none"> <li>• [ARC]: Retraction from the side surface takes the form of an arc.</li> </ul> The setting is specified as fixed and cannot be changed.
X	Escape radius / -distance	Radius, only if [ARC] is specified.
Y*	Escape angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approach motion	<ul style="list-style-type: none"> <li>• [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.</li> </ul>

**Cycle description**

The top surface of the pocket wall is chamfered. The tool track for this is the same as with pocket milling (roughing).





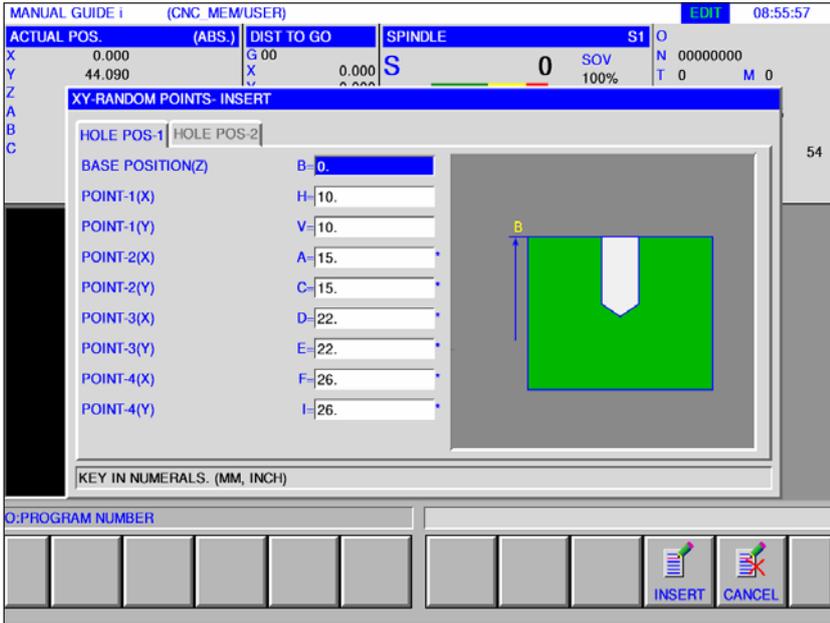
## Figure: Hole position

- Random points G1210
- Linear points G1211
- Grid points G1213
- Rectangle points G1214
- Circle points G1215
- Arc points G1216
- A-axis hole in cylinder (arc) G1772
- A-axis hole in cylinder (free) G1773



## Random Points G1210

Entry fields marked with \* are optional and should not be populated.



Cutting condition		
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Point 1 X	X-coordinates of the first drilling.
V	Point 1 Y	Y-coordinates of the first drilling.
A*	Point 2 X	X-coordinates of the second drilling.
C*	Point 2 Y	Y-coordinates of the second drilling.
D*	Point 3 X	X-coordinates of the third drilling.
E*	Point 3 Y	Y-coordinates of the third drilling.
F*	Point 4 X	X-coordinates of the fourth drilling.
I*	Point 4 Y	Y-coordinates of the fourth drilling.
J*	Point 5 X	X-coordinates of the fifth drilling.
K*	Point 5 Y	Y-coordinates of the fifth drilling.
M*	Point 6 X	X-coordinates of the sixth drilling.
P*	Point 6 Y	Y-coordinates of the sixth drilling.
Q*	Point 7 X	X-coordinates of the seventh drilling.
R*	Point 7 Y	Y-coordinates of the seventh drilling.
S*	Point 8 X	X-coordinates of the eighth drilling.
T*	Point 8 Y	Y-coordinates of the eighth drilling.

**Instructions:**

Values should not be entered for all drilling positions. However, if an entry is made, both X and Y coordinates should be entered as a pair for a drilling position.

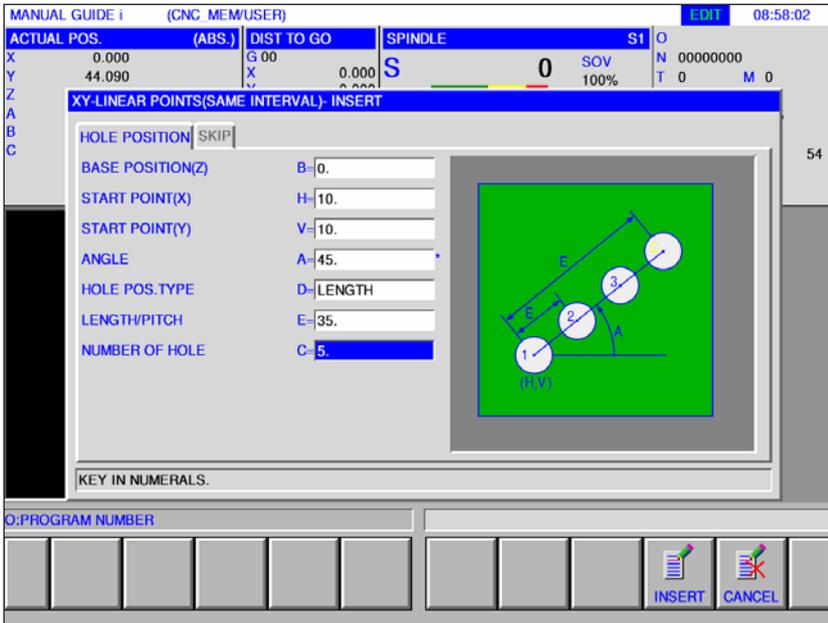






## Linear Points (same interval) G1211

Entry fields marked with \* are optional and should not be populated.



Hole position		
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Start point X	X-coordinates of the start point (first drilling) on a straight line.
V	Start point Y	Y-coordinates of the start point (first drilling) on a straight line.
A*	Angle	Angle of a straight line to the X-axis (initial value = 0).
D	Hole position type	<ul style="list-style-type: none"> <li>• [LENGTH]: Gives the distance between the first and last drilling and the number of drillings.</li> <li>• [PITCH]: Gives the distance between two adjacent drillings and the number of drillings.</li> </ul>
E	Length / Pitch	<ul style="list-style-type: none"> <li>• [LENGTH]: Distance between the first and last drillings (if [LENGTH] is selected for point D).</li> <li>• [PITCH]: Distance between two adjacent drillings (if [PITCH] is selected for point D).</li> </ul>
C	Number of hole	Number of drillings

Skip		
Data item		Meaning
F*	Omitting point 1	Point at which no hole is drilled (1)
I*	Omitting point 2	Point at which no hole is drilled (2)
J*	Omitting point 3	Point at which no hole is drilled (3)
K*	Omitting point 4	Point at which no hole is drilled (4)

**Instructions:**

It is unnecessary to enter a value for a drainage position.  
This applies also for cycles G1213 to G1216.

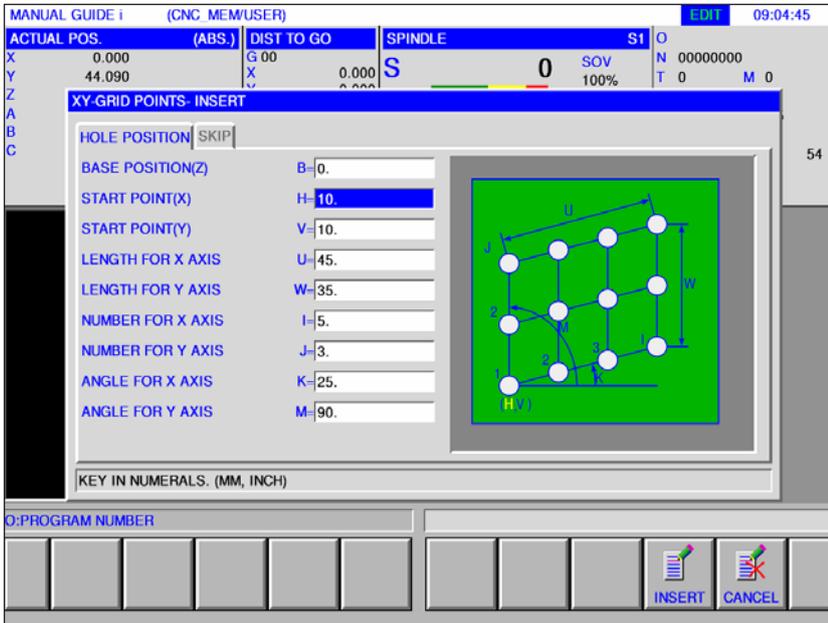




3. XY GRID POINTS

### XY Grid points G1213

Entry fields marked with \* are optional and should not be populated.



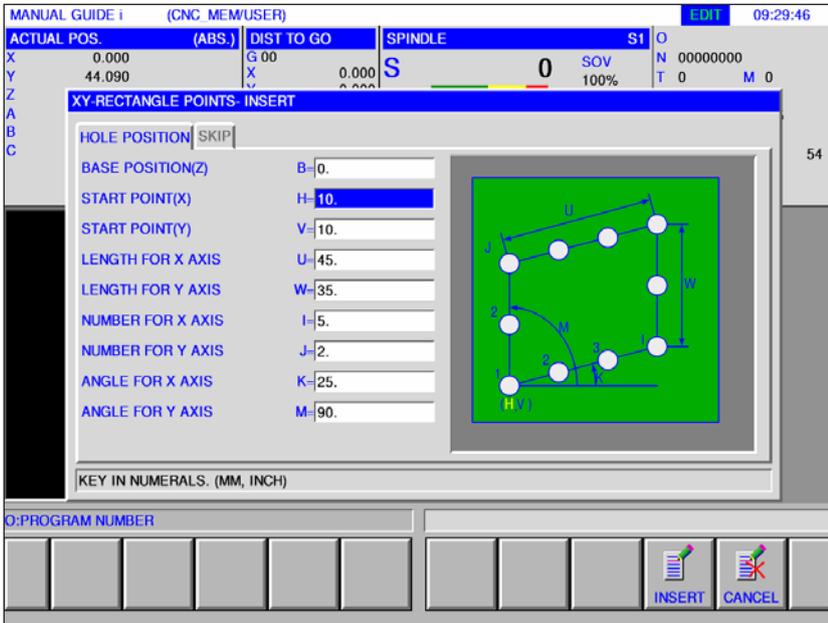
Hole position		
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Start point X	X-coordinates of the start point (first drilling) on a straight line.
V	Start point Y	Y-coordinates of the start point (first drilling) on a straight line.
U	Length for X-axis	Length of the first side of the grid (positive value).
W	Length for Y-axis	Length of the second side of the grid (positive value).
I	Number for X-axis	Number of drillings on the first side of the grid (positive value).
J	Number for Y-axis	Number of drillings on the second side of the grid (positive value).
K	Angle for X-axis	Angle of the first side of the grid (positive value). (Initial value = 0).
M	Angle for Y-axis	Angle of the second side of the grid (positive value). (Initial value = 90).

Skip		
Data item		Meaning
A*	Omitting point 1	Point at which no hole is drilled (1)
C*	Omitting point 2	Point at which no hole is drilled (2)
D*	Omitting point 3	Point at which no hole is drilled (3)
E*	Omitting point 4	Point at which no hole is drilled (4)



## XY Rectangle points G1214

Entry fields marked with \* are optional and should not be populated.



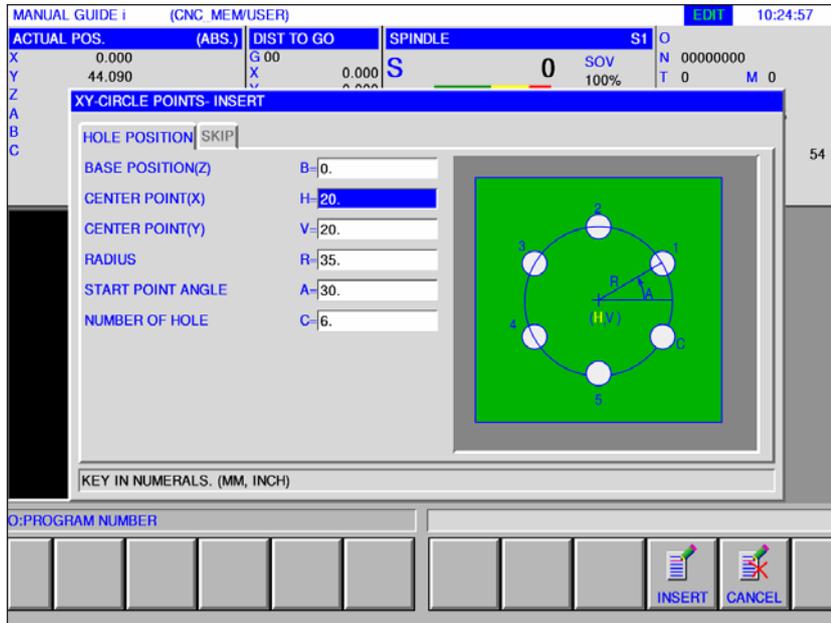
Hole position		
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Start point X	X-coordinates of the start point (first drilling) on a straight line.
V	Start point Y	Y-coordinates of the start point (first drilling) on a straight line.
U	Length for X-axis	Length of the first side of the grid (positive value).
W	Length for Y-axis	Length of the second side of the grid (positive value).
I	Number for X-axis	Number of drillings on the first side of the grid (positive value).
J	Number for Y-axis	Number of drillings on the second side of the grid (positive value).
K	Angle for X-axis	Angle of the first side of the grid (positive value). (Initial value = 0).
M	Angle for Y-axis	Angle of the second side of the grid (positive value). (Initial value = 90).

Skip		
Data item		Meaning
A*	Omitting point 1	Point at which no hole is drilled (1)
C*	Omitting point 2	Point at which no hole is drilled (2)
D*	Omitting point 3	Point at which no hole is drilled (3)
E*	Omitting point 4	Point at which no hole is drilled (4)



## XY Circle points G1215

Entry fields marked with \* are optional and should not be populated.



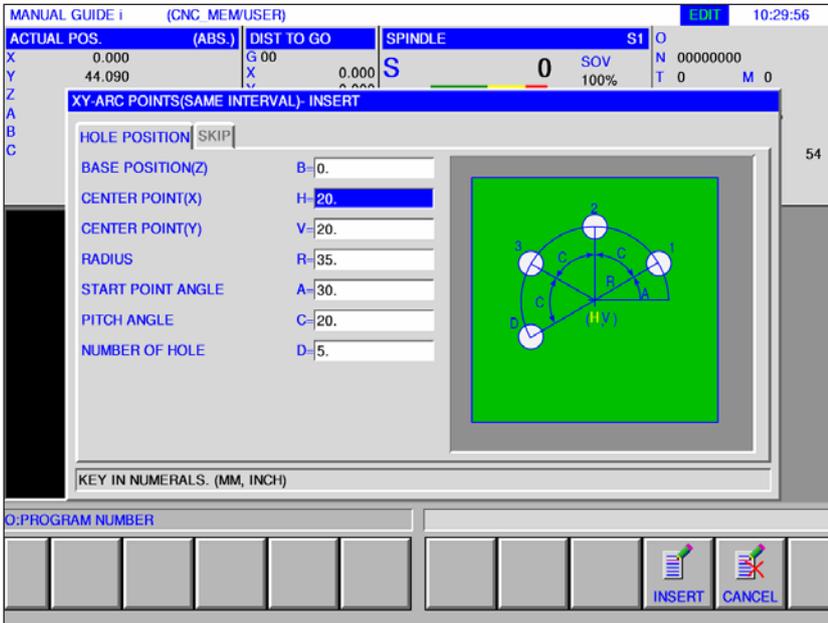
Hole position		
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Centre point X	X-coordinates of the centre of the circle
V	Centre point Y	Y-coordinates of the centre of the circle
R	Radius	Radius of the circle (positive value).
A	Start point angle	Centre angle to the X-axis of the first drilling (positive or negative value) (initial value = 0).
C	Number of hole	Number of drillings (positive value).

Skip		
Data item		Meaning
D*	Omitting point 1	Point at which no hole is drilled (1)
E*	Omitting point 2	Point at which no hole is drilled (2)
F*	Omitting point 3	Point at which no hole is drilled (3)
I*	Omitting point 4	Point at which no hole is drilled (4)



## XY Arc points (same interval) G1216

Entry fields marked with \* are optional and should not be populated.



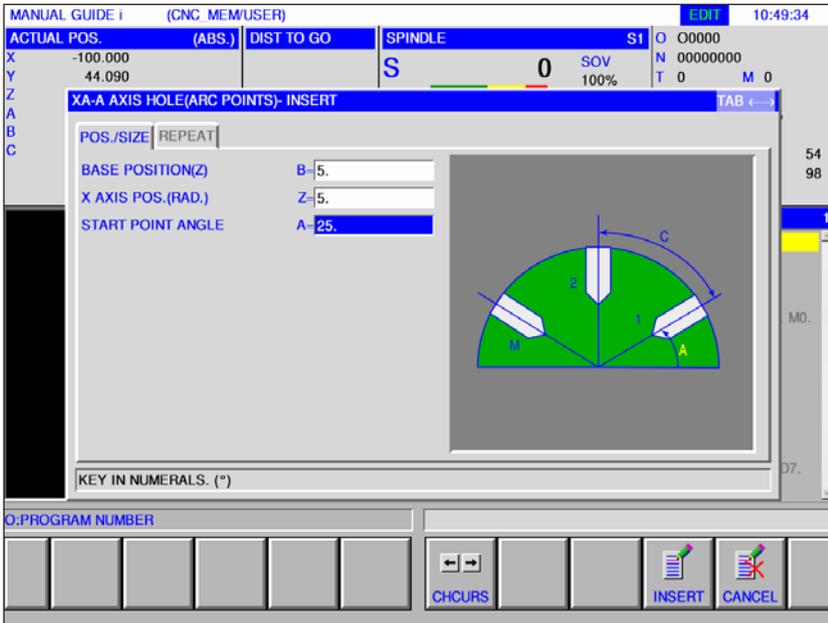
Hole position		
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Centre point X	X-coordinates of the centre of the circle
V	Centre point Y	Y-coordinates of the centre of the circle
R	Radius	Radius of the arc (positive value).
A	Start point angle	Centre angle to the X-axis of the first drilling (positive or negative value) (initial value = 0).
C	Pitch angle	Centre angle between two sequential drillings (positive or negative value).
D	Number of hole	Number of drillings (positive value).

Skip		
Data item		Meaning
E*	Omitting point 1	Point at which no hole is drilled (1)
F*	Omitting point 2	Point at which no hole is drilled (2)
I*	Omitting point 3	Point at which no hole is drilled (3)
J*	Omitting point 4	Point at which no hole is drilled (4)



## XA A-axis Hole Arc G1772

Entry fields marked with \* are optional and are not required to be populated.



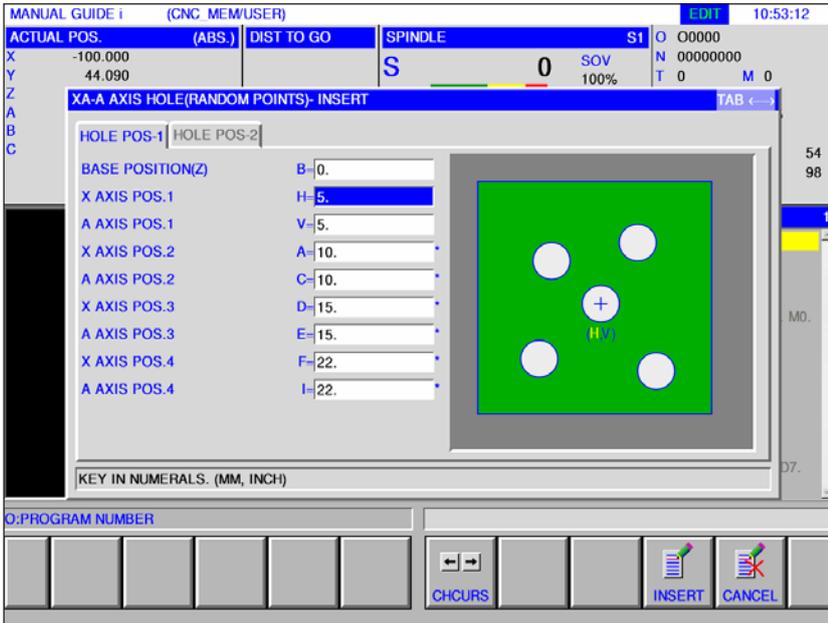
Machining Hole Position		
Data element		Meaning
B	Basis position Z	Z-coordinates of the work piece surface.
Z	X-axis position radius	X-coordinate of the hole position (positive value).
A	Start point angle	Centre angle to the X-axis of the first hole (positive or negative value) (initial value = 0).

Machining Outlet Position		
Data element		Meaning
C	Pitch angle	Centre angle between two sequential drillings (positive or negative value).
M	Number of holes	Number of holes (positive value).



## XA A-axis Hole Free G1773

Entry fields marked with \* are optional and are not required to be populated.

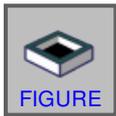


Machining Hole Position		
	Data element	Meaning
B	Basis position Z	Z-coordinates of the work piece surface.
H	X-axis position 1	X-coordinates of the first hole.
V	A-axis position 1	A-coordinate of the first hole.
A*	X-axis position 2	X-coordinate of the second hole.
C*	A-axis position 2	A-coordinate of the second hole.
D*	X-axis position 3	X-coordinates of the third hole.
E*	A-axis position 3	A-coordinate of the third hole.
F*	X-axis position 4	X-coordinate of the fourth hole.
I*	A-axis position 4	A-coordinate of the fourth hole.
J*	X-axis position 5	X-coordinates of the fifth hole.
K*	A-axis position 5	A-coordinate of the fifth hole.
M*	X-axis position 6	X-coordinates of the sixth hole.
P*	A-axis position 6	A-coordinate of the sixth hole.
Q*	X-axis position 7	X-coordinates of the seventh hole.
R*	A-axis position 7	A-coordinate of the seventh hole.
S*	X-axis position 8	X-coordinates of the eighth hole.
T*	A-axis position 8	A-coordinate of the eights hole.

## Figure: Front Surface Contour

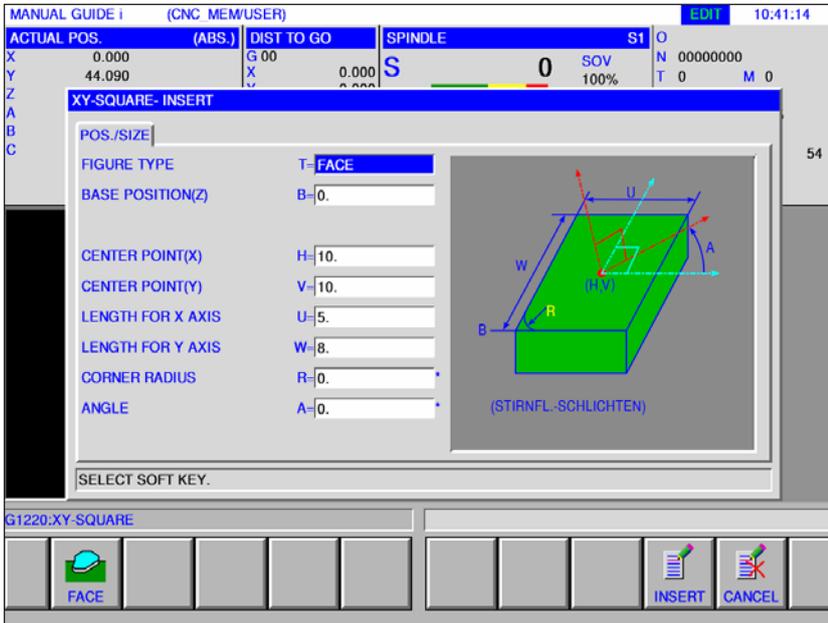
- Square G1220





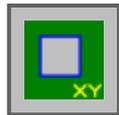
## XY Square facing G1220

Entry fields marked with \* are optional and should not be populated.



Position / Size		
Data item		Meaning
T	Figure type	• [FACE]: used as contour in plane machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
U	Length for X-axis	Side length in X-axis direction (Radius value, positive value)
W	Length for Y axis	Centre angle to the X-axis of the first drilling (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Angle	Gradient of a rectangular contour to the X-axis (positive or negative value).





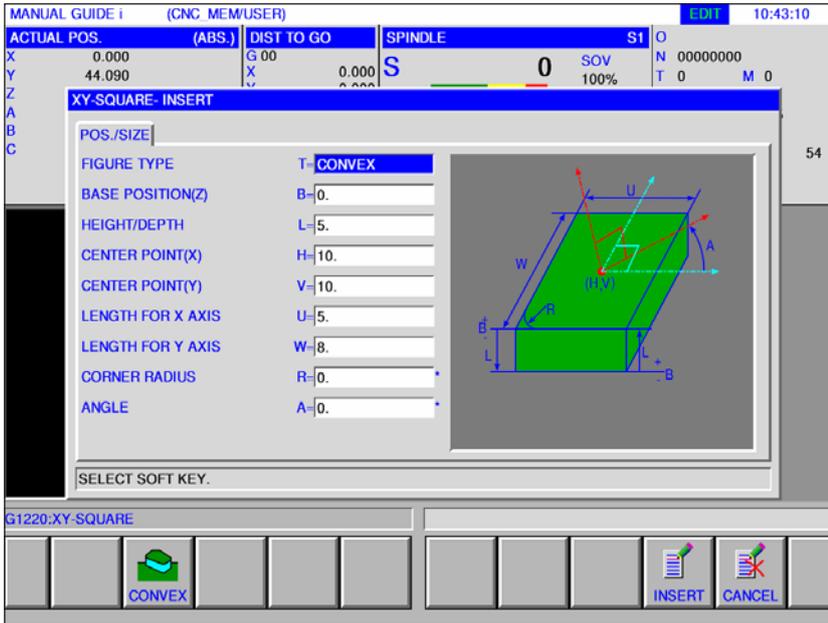
## Figure: Contouring figure

- Square G1220 convex
- Circle G1221 convex
- Track G1222 convex
- Polygon G1225 convex
- Free figure
- XA plane free convex figure for cylinder G1700
- Square G1220 concave
- Circle G1221 concave
- Track G1222 concave
- Polygon G1225 concave
- Free concave figure
- XA plane free concave figure for cylinder G1700
- Free open figure
- XA plane free open figure for cylinder G1700



## XY Square convex G1220

Entry fields marked with \* are optional and should not be populated.



Position / Size		
Data item	Meaning	
T	Figure type	• [CONVEX]: used as outside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
U	Length for X-axis	Side length in X-axis direction (Radius value, positive value)
W	Length for Y-axis	Centre angle to the X-axis of the first drilling (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

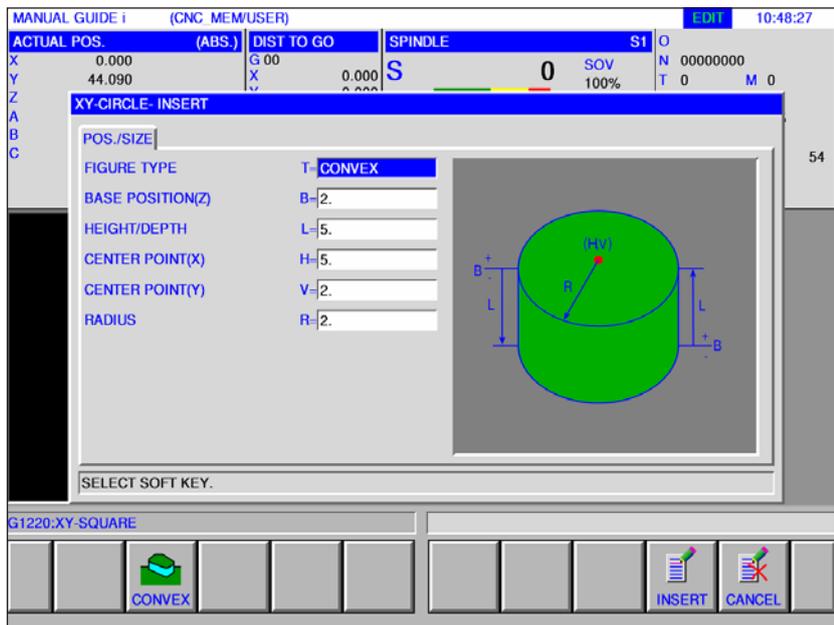


CONT. FIGURE



## XY Circle convex G1221

Entry fields marked with \* are optional and should not be populated.

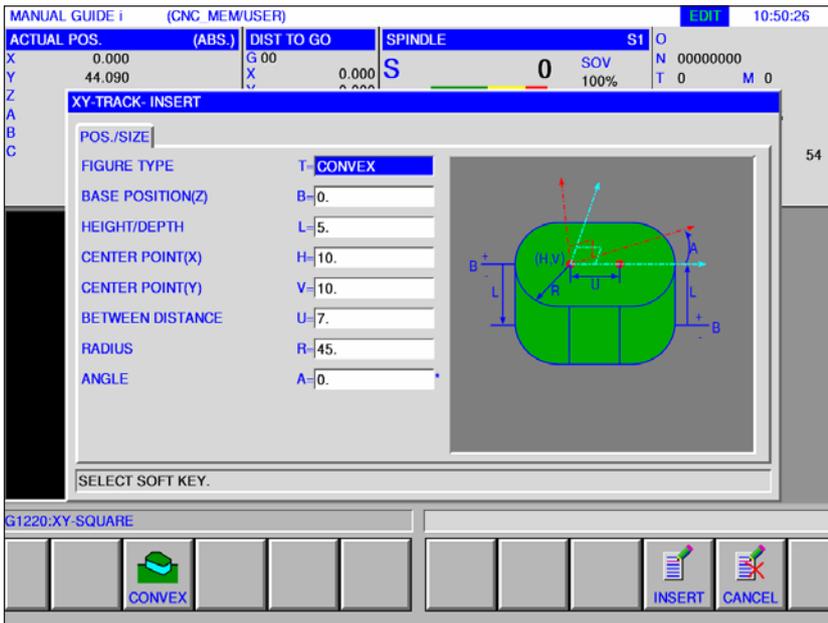


Position / Size		
Data item		Meaning
T	Figure type	• [CONVEX]: used as outside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a circular contour.
V	Centre point Y	Y-coordinates of the centre point of a circular contour.
R	Radius	Radius of a circular contour (radius value, positive value)

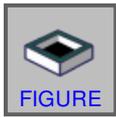


## XY Track convex G1222

Entry fields marked with \* are optional and should not be populated.



Position / Size		
Data item	Meaning	
T	Figure type	• [CONVEX]: used as outside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	Y-coordinates of the centre point of the left semicircle.
V	Centre point Y	Y-coordinates of the centre point of the left semicircle.
U	Between distance	Distance between the centre points of the right and left semicircles (radius value, positive value).
R	Radius	Radius of the left and right semicircles (Radius value, positive value)
A*	Angle	Gradient of an oval contour to the X-axis (positive or negative value)

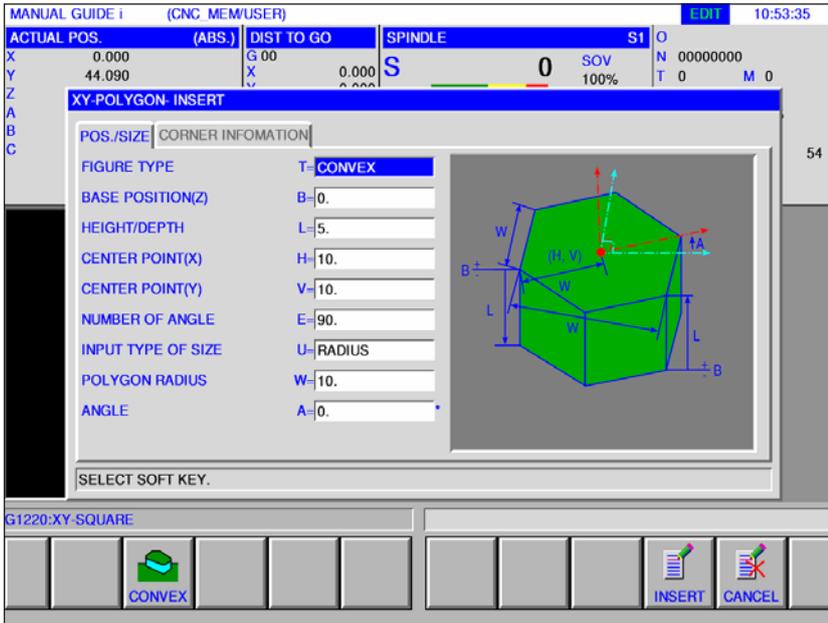


CONT. FIGURE

4. XY POLYGON CONVEX

## XY Polygon convex G1225

Entry fields marked with \* are optional and should not be populated.

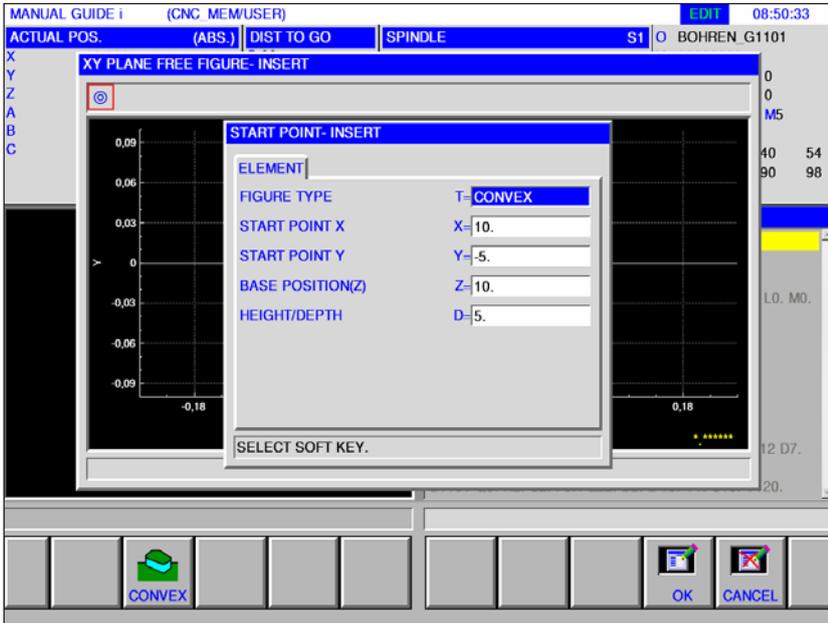


Position / Size		
Data item		Meaning
T	Figure type	<ul style="list-style-type: none"> <li>[CONVEX]: used as outside contour for contour machining.</li> </ul>
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
E	Number of angle	Number of edges, positive integer between 3 and 99.
U	Input type of size	<ul style="list-style-type: none"> <li>[RADIUS]: Radius</li> <li>[LENGTH]: Edge length</li> <li>[WIDTH]: Key width</li> </ul>
W	Polygon radius, U1	Dependent on U
A*	Angle	Gradient of a straight line which connects an apex and the centre point relative to the 1 <sup>st</sup> axis.
Corner information		
Data item		Meaning
C	Corner type	<ul style="list-style-type: none"> <li>[NOTHIN]:</li> <li>[CHAMFR]:</li> <li>[ARC]:</li> </ul>
R*	Corner size	Chamfer radius Only for chamfer or arc



## XY Free figure convex

Entry fields marked with \* are optional and should not be populated.

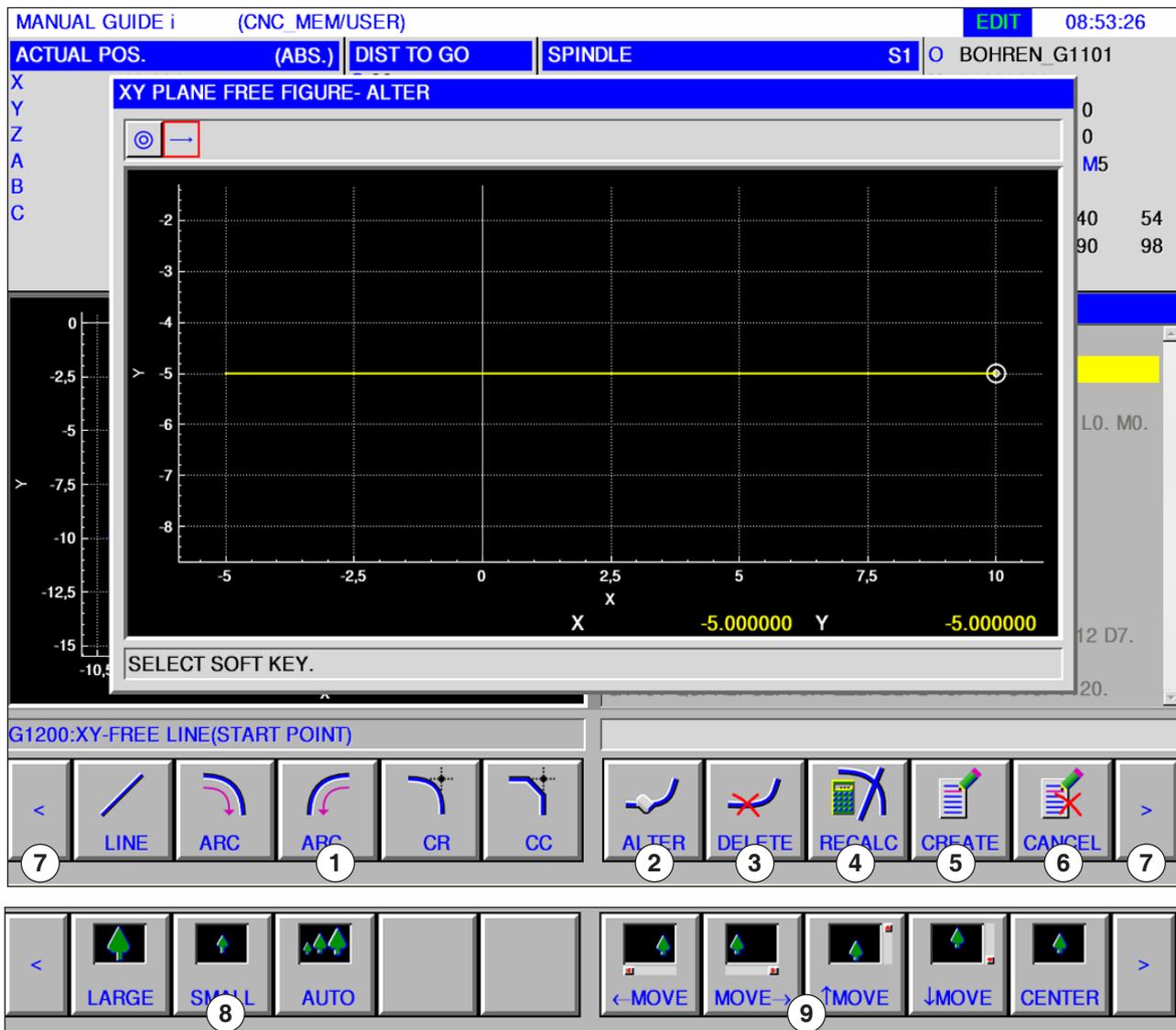


Startpoint insert		
Data item		Meaning
T	Figure type	• [CONVEX]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position (Z)	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface

**Instructions:**  
 INPUT DATA are elements that are displayed when changing or editing in the data input window.  
 This applies for cycles G1201 to G1206.

**Instructions:**  
 OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can call these up only for program display purposes.  
 This applies also for cycles G1201 to G1206.

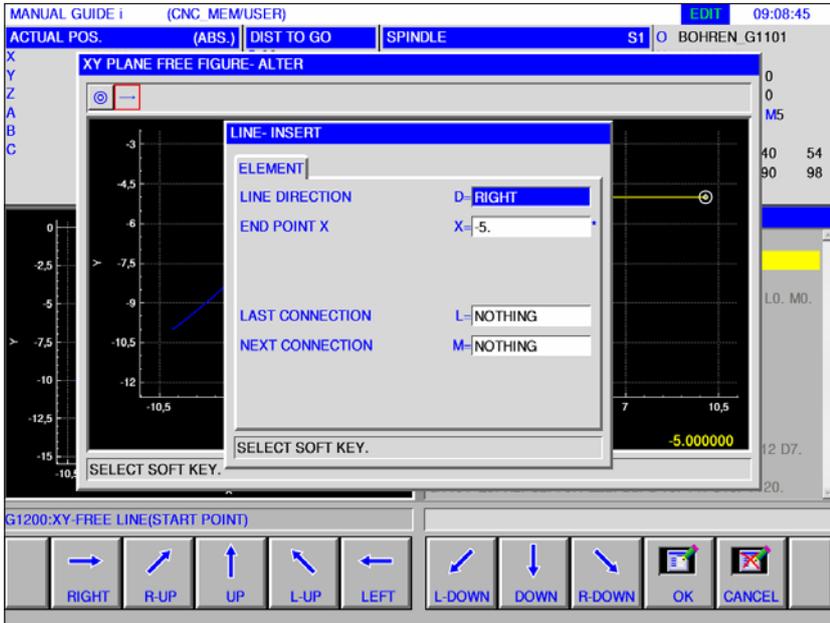
## Overview of the input elements for free contour programming



- |  |                                   |
|--|-----------------------------------|
| 1 Input elements: Line, arc, radius, chamfer | 6 Cancel input                    |
| 2 Change contour element                     | 7 Expansion buttons               |
| 3 Delete contour element                     | 8 Zoom graphic display in and out |
| 4 Recalculate contour data                   | 9 Move graphic display            |
| 5 Generate contour                           |                                   |

## Input items of line (XY plane) G1201

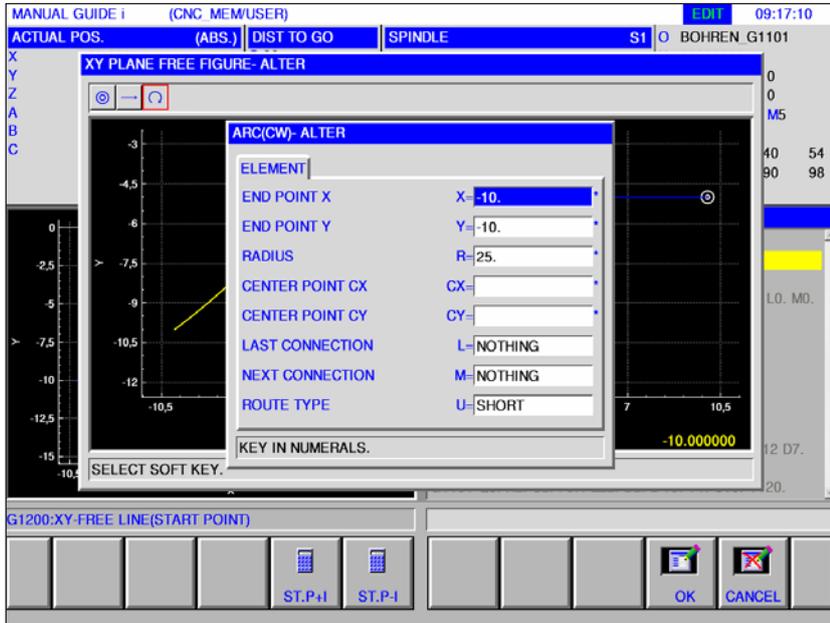
Entry fields marked with \* are optional and should not be populated.



Element data input		
Data item		Meaning
D	Line direction	The line direction is selected by means of a softkey in the toolbar. <ul style="list-style-type: none"> <li>• [RIGHT]:</li> <li>• [R-UP]:</li> <li>• [UP]:</li> <li>• [L-UP]:</li> <li>• [LEFT]:</li> <li>• [L-DOWN]:</li> <li>• [DOWN]:</li> <li>• [R-DOWN]:</li> </ul>
X*	End point X	X-coordinates of the line end point.
Y*	End point Y	Y-coordinates of the line end point.
A*	Angle	Angle of the straight line
L	Last connection	<ul style="list-style-type: none"> <li>• [TANGNT]: Contact with the immediately preceding contour.</li> <li>• [NO SET]: No contact with the immediately preceding contour (initial value).</li> </ul>
M	Next connection	<ul style="list-style-type: none"> <li>• [NO SET]: No contact with the immediately preceding contour (initial value).</li> </ul>

## Input items for arc (XY plane) G1202, 1203

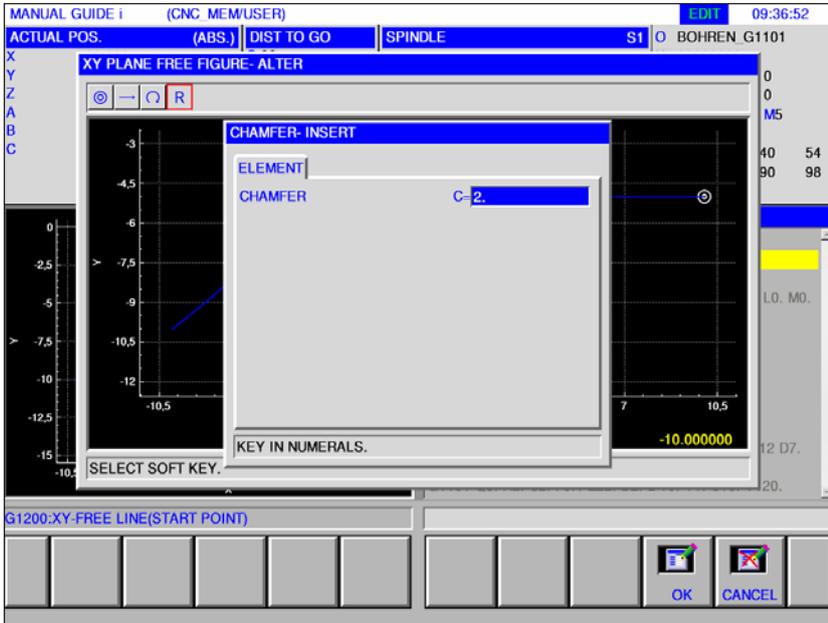
Entry fields marked with \* are optional and should not be populated.



Element data input		
Data item		Meaning
X*	End point X	X-coordinates of an arc endpoint Incremental dimension programming is possible.
Y*	End point Y	Y-coordinates of an arc endpoint Incremental dimension programming is possible.
R*	Radius	Radius of curvature
CX*	Centre point CX	X-coordinates of the centre of the arc
CY*	Centre point CY	Y-coordinates of the centre of the arc
L	Last connection	<ul style="list-style-type: none"> <li>[TANGNT]: Contact with the immediately preceding contour.</li> <li>[NO SET]: No contact with the immediately preceding contour.</li> </ul>
M	Next connection	[NO SET]: No contact with the immediately preceding contour (initial value).
U	Route type	<ul style="list-style-type: none"> <li>[SHORT]: An arc with a long route is generated.</li> <li>[LONG]: An arc with a short route is generated.</li> </ul>

### Input items of chamfer (XY plane) G1204

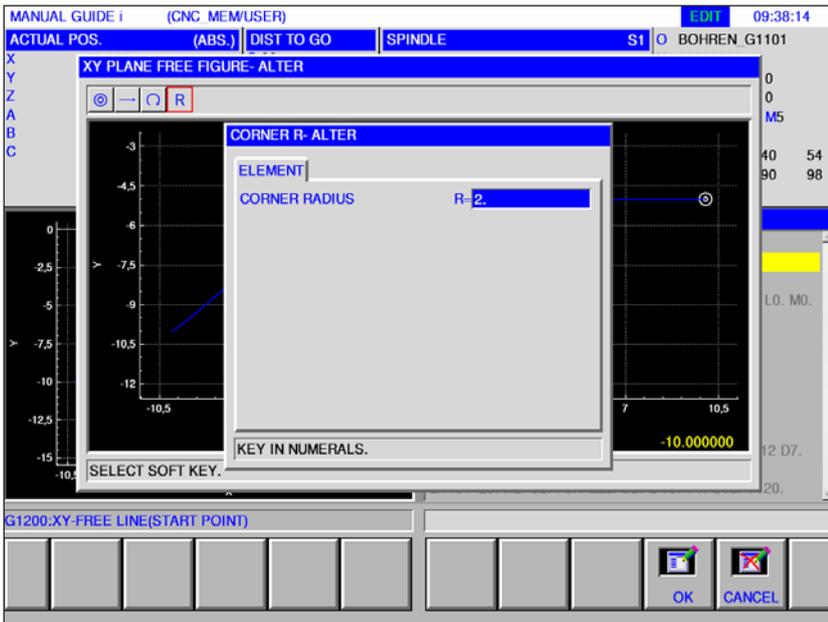
Entry fields marked with \* are optional and should not be populated.



Element data input		
Data item		Meaning
C	Chamfer	Chamfer, positive value

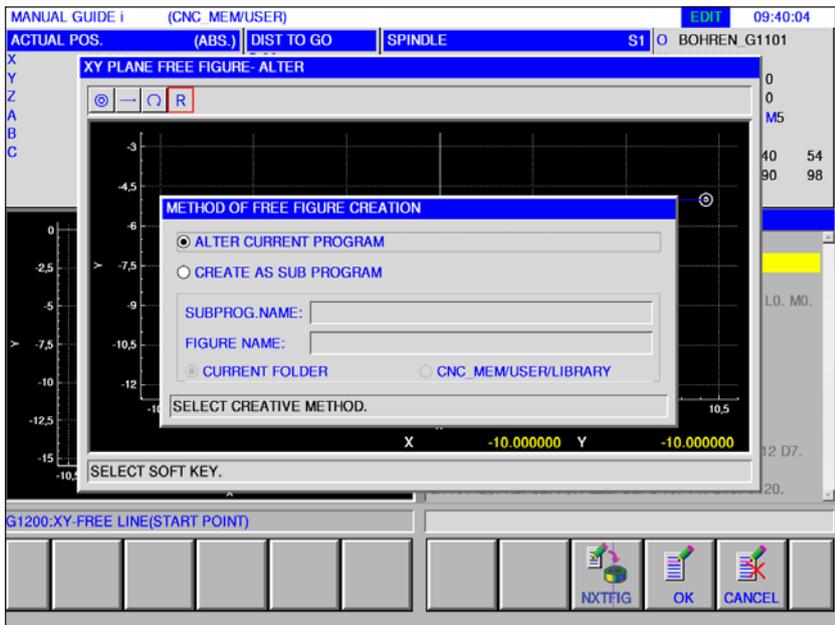
### Input items of corner (XY plane) G1205

Entry fields marked with \* are optional and should not be populated.



Element data input		
Data item		Meaning
R	Corner radius	Radius value, positive value

## Input items of end of arbitrary figures G1206



### Close contour

- Press softkey.

A dialogue with 2 selections pops up:

- Insert into the current program
- Save as a separate subroutine  
With this option, you can also select the folder in which the subroutine is to be saved.

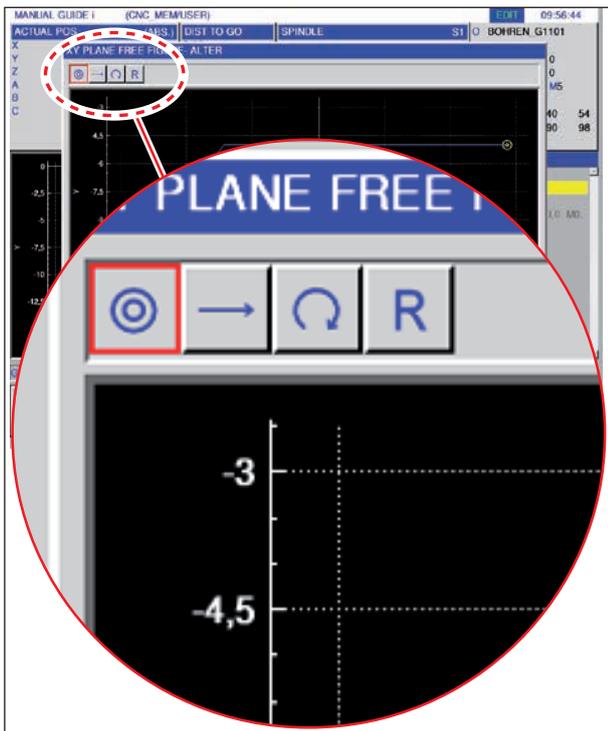
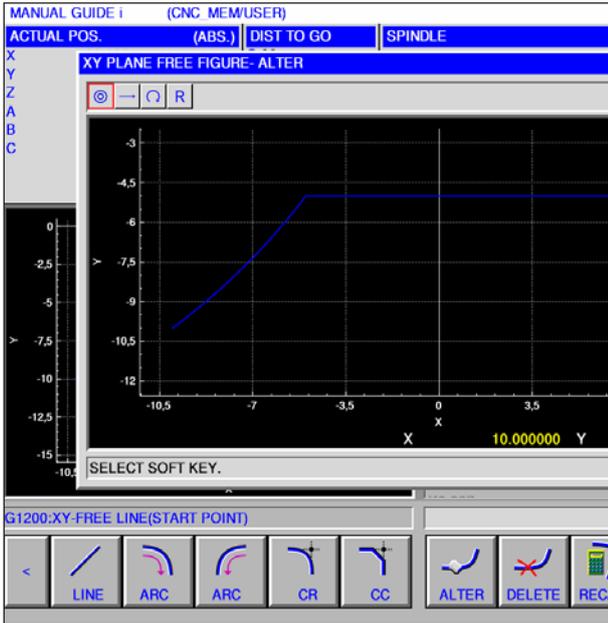


- Enter the next figure  
This softkey opens the contour editor input mask. Other free contours can be created.



- "OK" closes the input.

## Symbol representation of the contour elements



Contour element	Symbol	Meaning
Start point		Start point of the figure
Straight up Straight down		Straight line in 90° grid
Straight left Straight right		Straight line in 90° grid
Straight line anywhere		Straight line with arbitrary pitch
Curve right Curve left		Arc
Radius		
Chamfer		

**Instructions:**

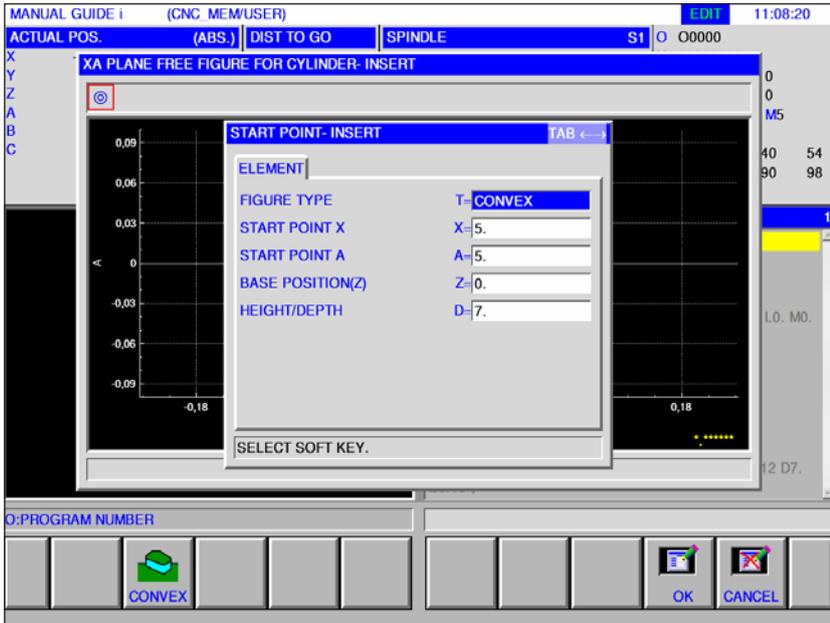
The end of a contour is not a contour element, therefore there is no symbol for it.





## XA plane free convex figure for cylinder G1700

Entry fields marked with \* are optional and should not be populated.



Start point		
Element		Meaning
T	Figure type	• [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position Z	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface

**Note:**

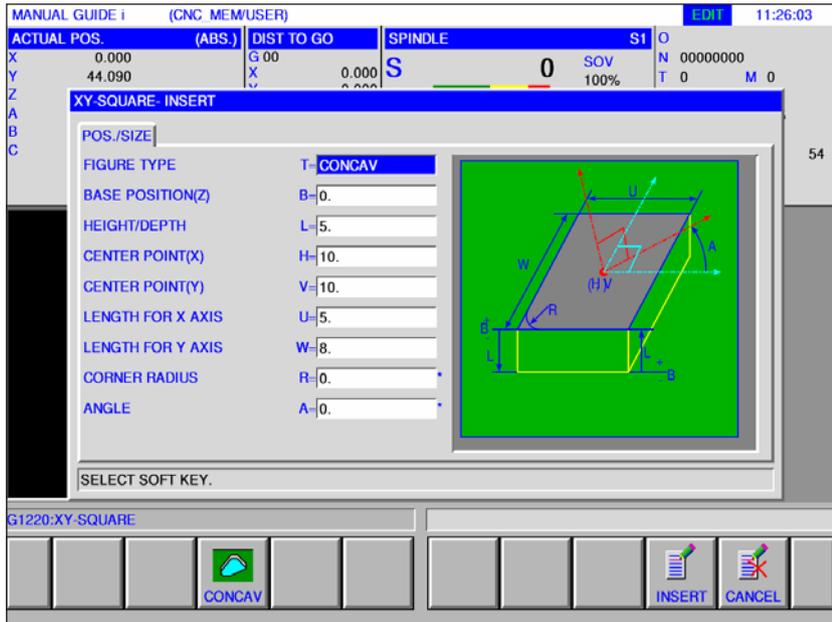
For more instructions how to create a free contour, see also "XY square concave G1220".





## XY Square concave G1220

Entry fields marked with \* are optional and should not be populated.

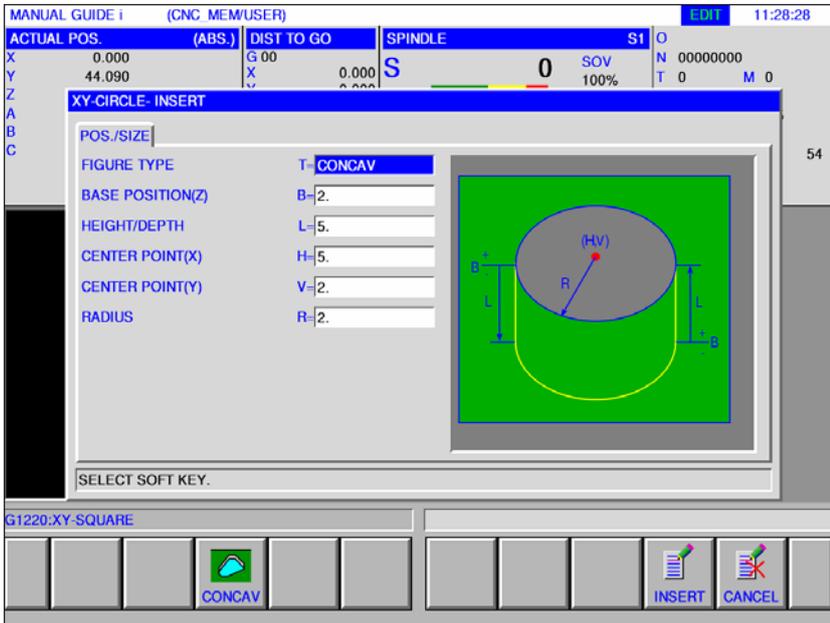


Position / Size		
Data item	Meaning	
T	Figure type	• [CONCAV]: used as inside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
U	Length for X-axis	Side length in X-axis direction (Radius value, positive value)
W	Length for Y-axis	Centre angle to the X-axis of the first drilling (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Angle	Gradient of a rectangular contour to the X-axis (positive or negative value).



## XY Circle concave G1221

Entry fields marked with \* are optional and should not be populated.

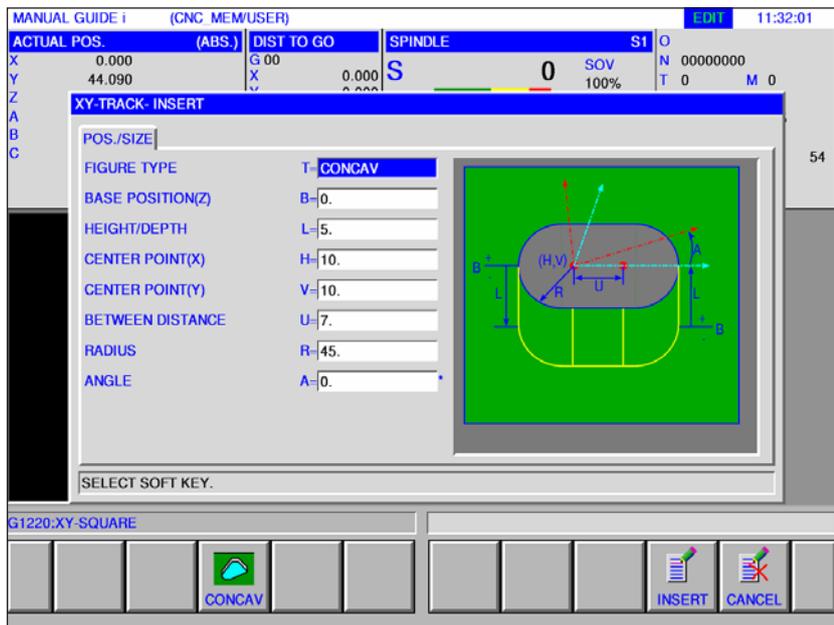


Position / Size		
Data item	Meaning	
T	Figure type	• [CONCAV]: used as inside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a circular contour.
V	Centre point Y	Y-coordinates of the centre point of a circular contour.
R	Radius	Radius of a circular contour (radius value, positive value)



## XY Track concave G1222

Entry fields marked with \* are optional and should not be populated.

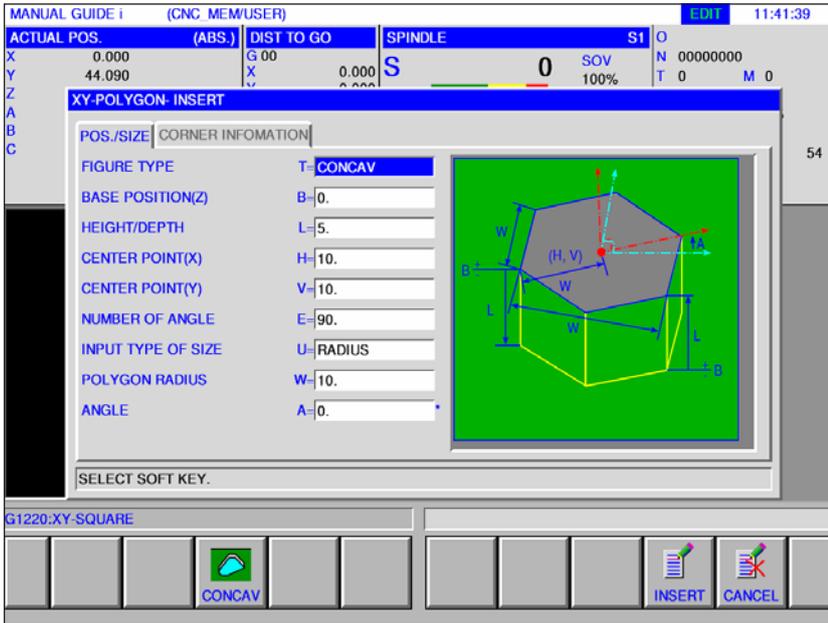


Position / Size		
Data item	Meaning	
T	Figure type	• [CONCAV]: used as inside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	Y-coordinates of the centre point of the left semicircle.
V	Centre point Y	Y-coordinates of the centre point of the left semicircle.
U	Between distance	Distance between the centre points of the right and left semicircles (radius value, positive value).
R	Radius	Radius of the left and right semicircles (Radius value, positive value)
A*	Angle	Gradient of an oval contour to the X-axis (positive or negative value)



## XY Polygon concave G1225

Entry fields marked with \* are optional and should not be populated.

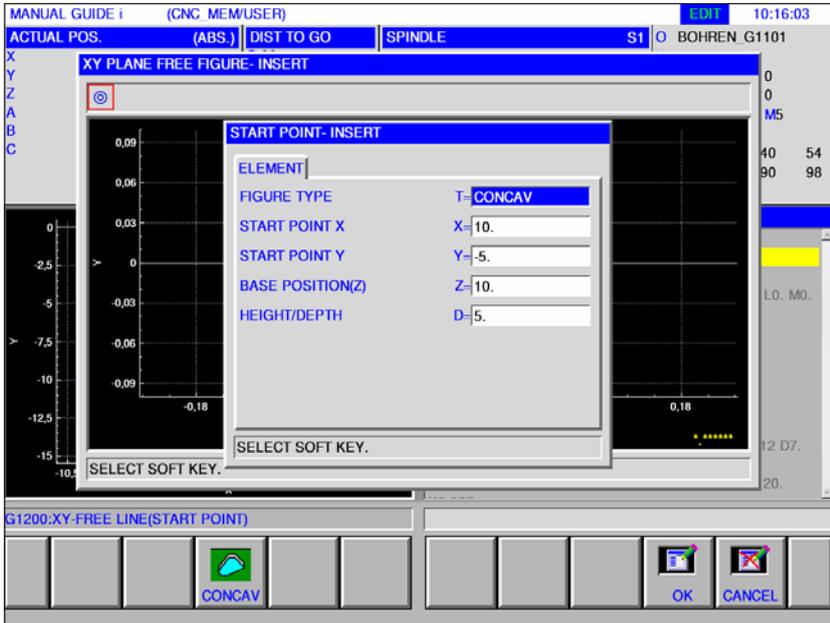


Position / Size		
Data item		Meaning
T	Figure type	<ul style="list-style-type: none"> <li>[CONCAV]: used as inside contour for contour machining.</li> </ul>
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
E	Number of angle	Number of edges, positive integer between 3 and 99.
U	Input type of size	<ul style="list-style-type: none"> <li>[RADIUS]: Radius</li> <li>[LENGTH]: Edge length</li> <li>[WIDTH]: Key width</li> </ul>
W	Polygon radius, U1	Dependent on U
A*	Angle	Gradient of a straight line which connects an apex and the centre point relative to the 1 <sup>st</sup> axis.
Corner information		
Data item		Meaning
C	Corner type	<ul style="list-style-type: none"> <li>[NOTHIN]:</li> <li>[CHAMFR]:</li> <li>[ARC]:</li> </ul>
R*	Corner size	Chamfer radius Only for chamfer or arc.



## XY Free concave figure

Entry fields marked with \* are optional and should not be populated.

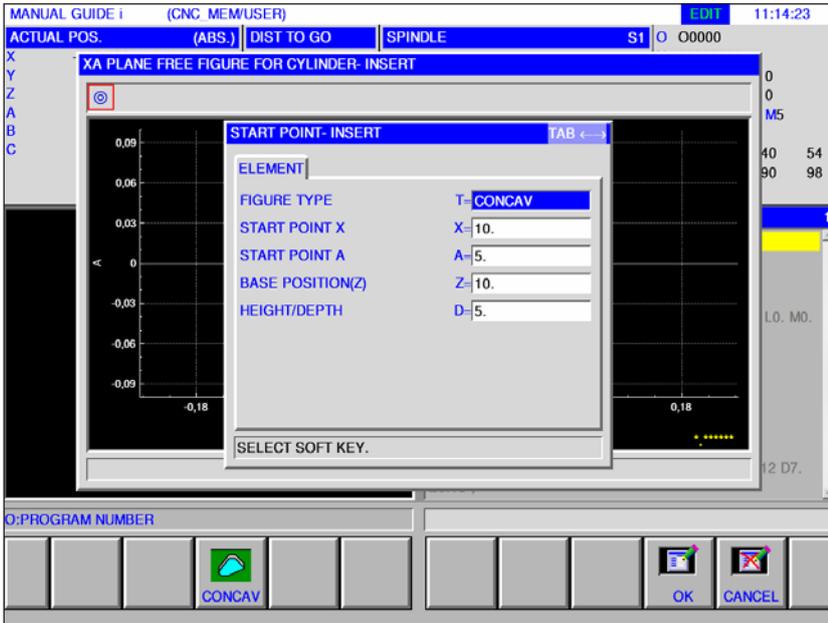


Start point		
Element		Meaning
T	Figure type	• [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position Z	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface



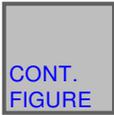
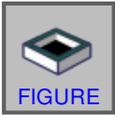
## XA plane free concave figure for cylinder G1700

Entry fields marked with \* are optional and should not be populated.

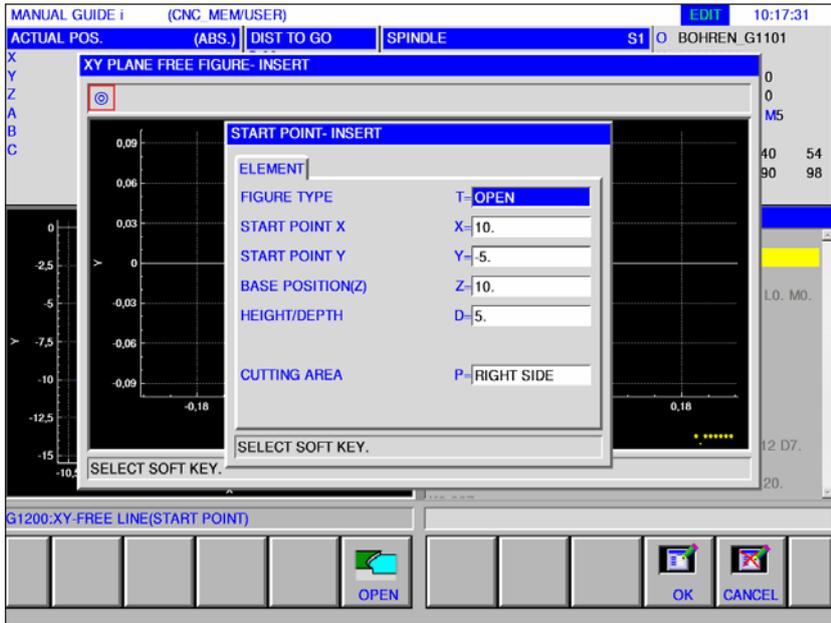


Start point		
Element		Meaning
T	Figure type	• [OPEN]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position Z	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface

**Note:** Description for the input elements see "Free figure convex".



## XY Free open figure



Entry fields marked with \* are optional and should not be populated.

Start point		
Element		Meaning
T	Figure type	• [OPEN]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position Z	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface
P	Cutting area	• [RIGHT]: right side • [LEFT]: left side

**Note:**

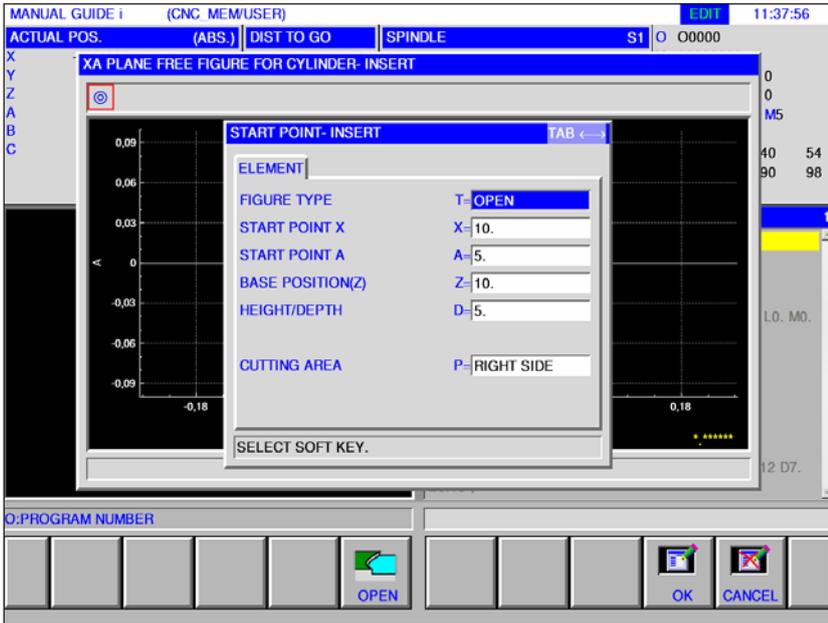
Description for the input elements see "Free figure convex".





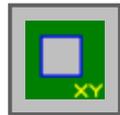
## XA plane free open figure for cylinder G1700

Entry fields marked with \* are optional and should not be populated.



Insert startpoint		
Data element		Meaning
T	Figure type	• [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position (Z)	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface
P	Cutting area	• [RIGHT]: right side • [LEFT]: left side

**Note:**  
Description for the input elements see "Free figure convex".



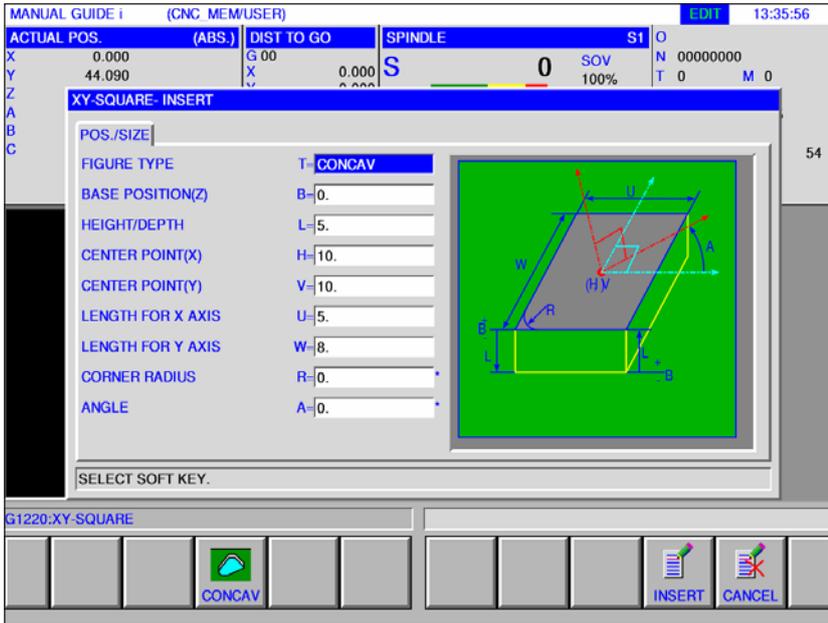
## Figure: Pocket figure

- Square concave G1220
- Circle concave G1221
- Track concave G1222
- Polygon concave G1225
- Free concave figure
- XA plane free open figure for cylinder G1700



## XY Square concave G1220

Entry fields marked with \* are optional and should not be populated.

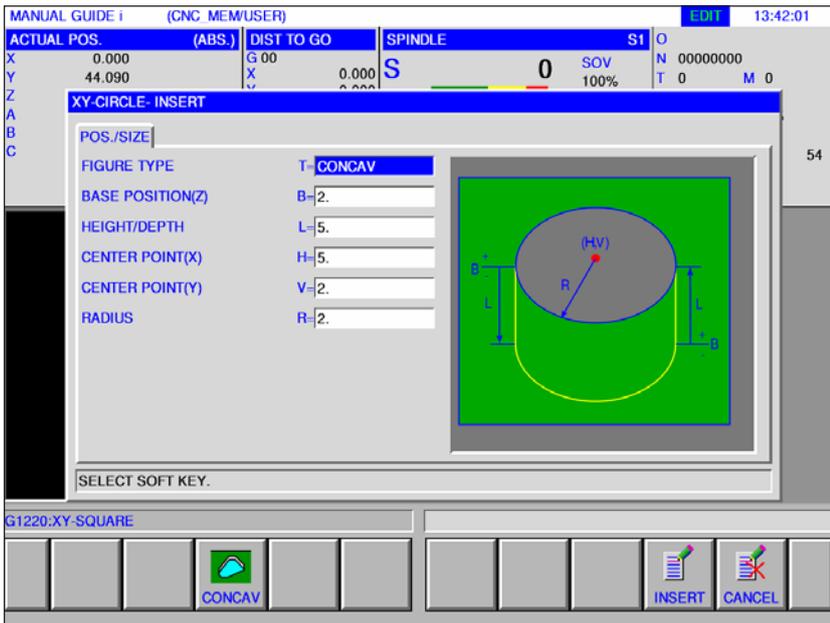


Position / Size		
Element		Meaning
T	Figure type	• [CONCAV]: used as inside contour for contour machining or as contour for pocket milling.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
U	Length for X-axis	Side length in X-axis direction (Radius value, positive value)
W	Length for Y-axis	Centre angle to the X-axis of the first drilling (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Angle	Gradient of a rectangular contour to the X-axis (positive or negative value).



## XY Circle concave G1221

Entry fields marked with \* are optional and should not be populated.

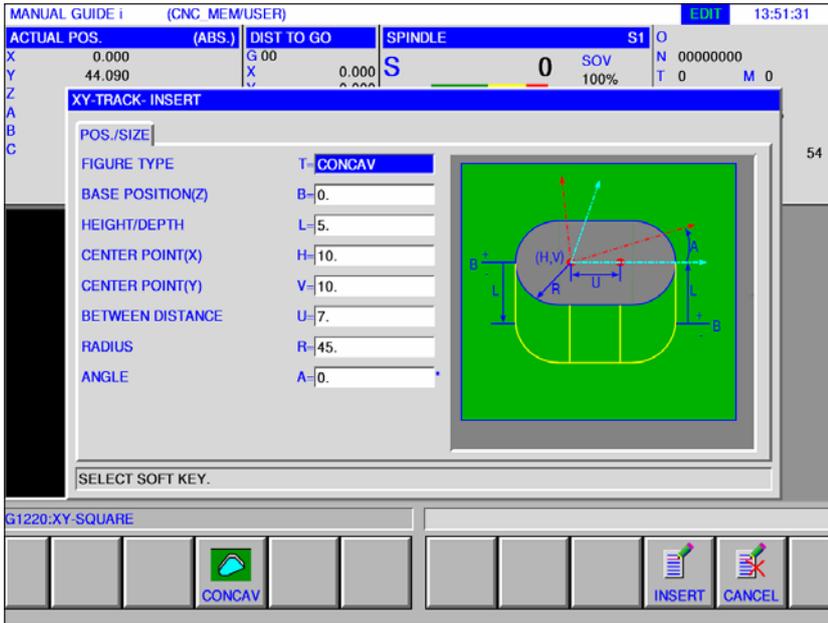


Position / Size		
Element		Meaning
T	Figure type	• [CONCAV]: used as inside contour for contour machining or as contour for pocket milling.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a circular contour.
V	Centre point Y	Y-coordinates of the centre point of a circular contour.
R	Radius	Radius of a circular contour (radius value, positive value)



## XY Track concave G1222

Entry fields marked with \* are optional and should not be populated.

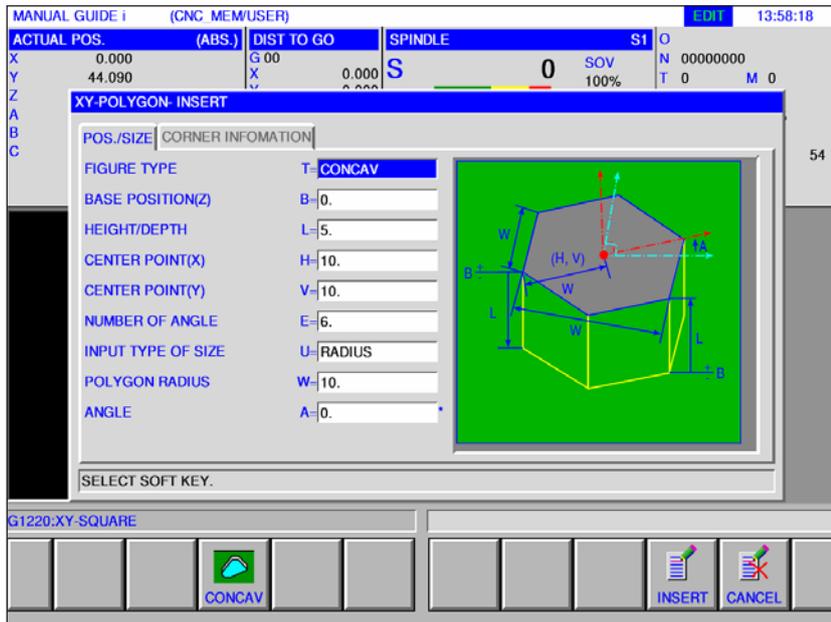


Position / Size		
Element		Meaning
T	Figure type	• [CONCAV]: used as inside contour for contour machining or as contour for pocket milling.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of the left semicircle.
V	Centre point Y	Y-coordinates of the centre point of the left semicircle.
U	Between distance	Distance between the centre points of the right and left semicircles (radius value, positive value).
R	Radius	Radius of the left and right semicircles (Radius value, positive value)
A*	Angle	Gradient of an oval contour to the X-axis (positive or negative value)



## XY Polygon concave G1225

Entry fields marked with \* are optional and should not be populated.

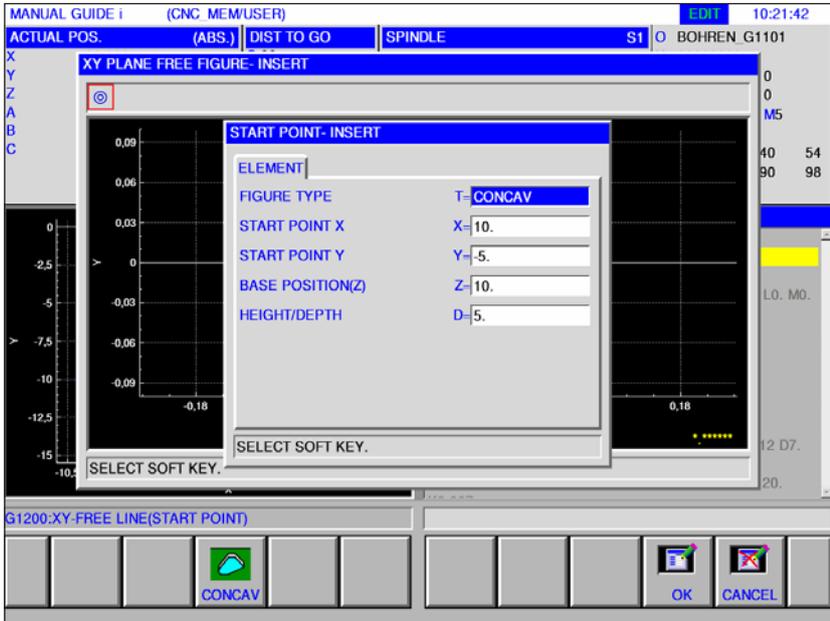


Position / Size		
Element	Meaning	
T	Figure type	<ul style="list-style-type: none"> <li>[CONCAV]: used as inside contour for contour machining or as contour for pocket milling.</li> </ul>
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom surface of the side is given as a negative value (radius value). → Depth If the bottom surface of a workpiece is selected as BASISPOSITION, the distance to the top surface of the workpiece is given as a positive value (radius value). → Height
H	Centre point X	X-coordinates of the centre point of a rectangular contour.
V	Centre point Y	Y-coordinates of the centre point of a rectangular contour.
E	Number of angle	Number of edges, positive integer between 3 and 99.
U	Input type of size	<ul style="list-style-type: none"> <li>[RADIUS]: Radius</li> <li>[LENGTH]: Edge length</li> <li>[WIDTH]: Key width</li> </ul>
W	Polygon radius, U1	Dependent on U
A*	Angle	Gradient of a straight line which connects an apex and the centre point relative to the 1 <sup>st</sup> axis.
Corner information		
Element	Meaning	
C	Corner type	<ul style="list-style-type: none"> <li>[NOTHIN]:</li> <li>[CHAMFR]:</li> <li>[ARC]:</li> </ul>
R*	Corner size	Chamfer radius



## XY Free concave figure

Entry fields marked with \* are optional and should not be populated.



Insert startpoint		
Data element		Meaning
T	Figure type	• [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position (Z)	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface

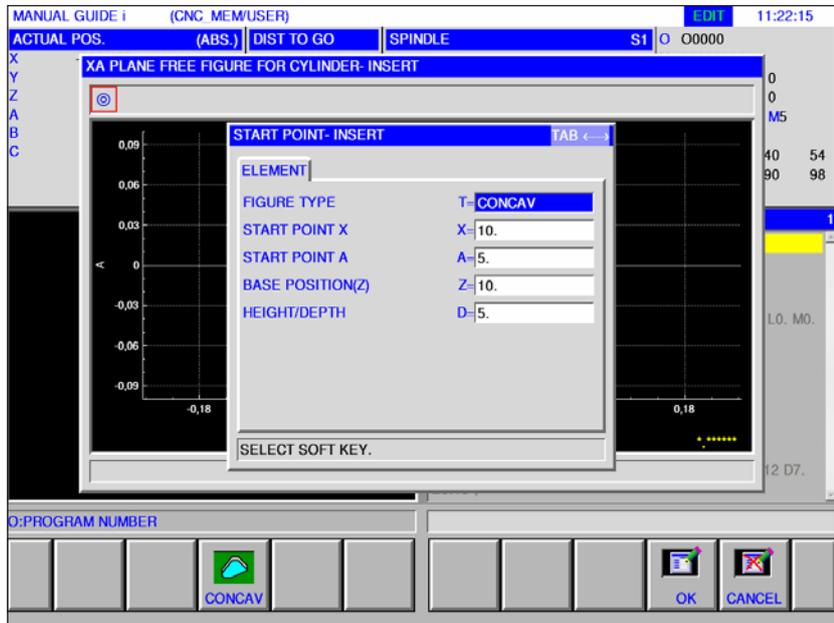
**Note:** Description for the input elements see "Free figure convex".



**6. XA PLANE FREE CONCAVE FIGURE FOR CYLINDER**

## XA plane free concave figure for cylinder G1700

Entry fields marked with \* are optional and should not be populated.



Insert startpoint		
Data element		Meaning
T	Figure type	• [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position (Z)	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface

**Note:**  
Description for the input elements see "Free figure convex".







**Subroutines**



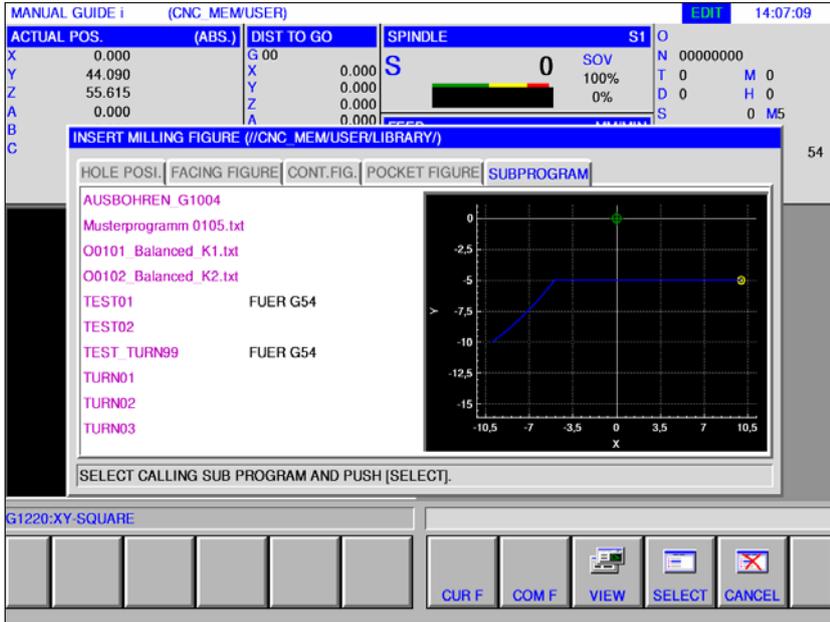
**Fixforms**



**M-code menu**



# Sub-routines



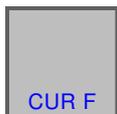
1 Select "Edit" mode.



2 Press softkey and register card



3 Set the cursor on the subroutine to be selected and open it for machining with the "SELECT" softkey.



4 The register card "SUBROUTINE" lists all current subroutines that are stored in the directory for the currently open program.



5 A list of programs is shown in a communal folder:  
CNC\_MEM/USER/LIBRARY



6 The softkey "VIEW" shows a preview of the subprogram. Press the softkey again to turn off the preview.

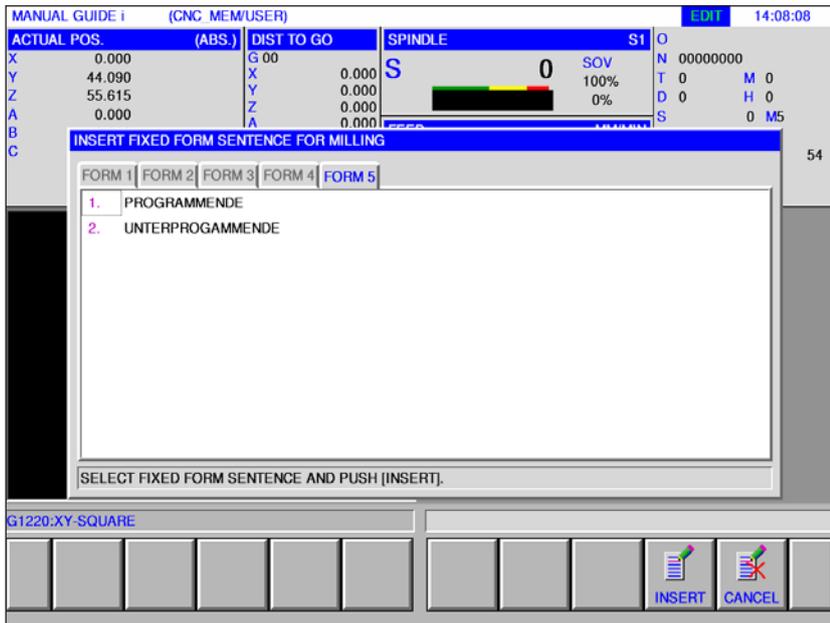
## Programming

- M98 Call subroutine
- M99 Jump back to the calling program

## Example

M98 P1234

## Fixed forms



Frequently recurring machining processes can be saved as fixed form and inserted in the NC program.

This method saves the user repeated entry of the same machining processes.



1 Select "Edit" mode.



2 Press softkey.

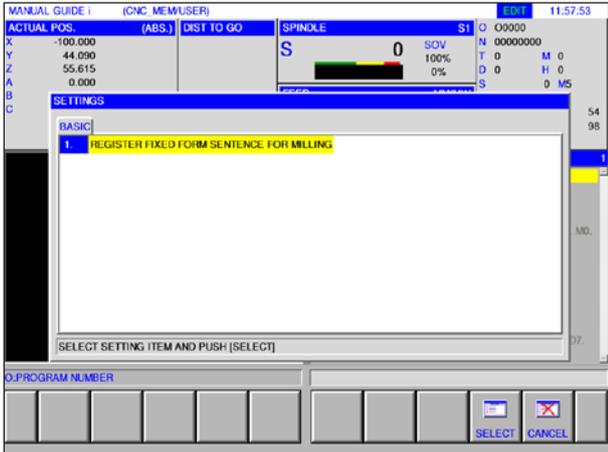
The predefined fixed forms can be selected by means of the register cards FORM1 to FORM5.



3 Select the desired Fixform with "INSERT" and insert it into the program.

## Creating fixed forms

1 Press softkey "SETTING" to open editor for fixed forms.

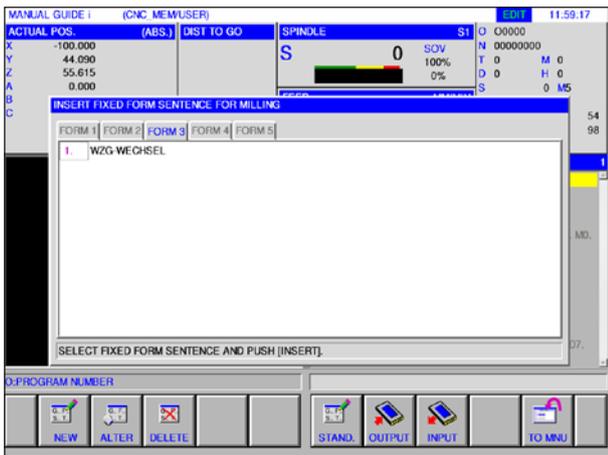


**Note:**  
Fixed forms can be created in all operating modes.

2 Press "SELECT" to open a pre-defined block for turning or milling.



3 Press "CANCEL" to leave the settings menu.



**Note:**

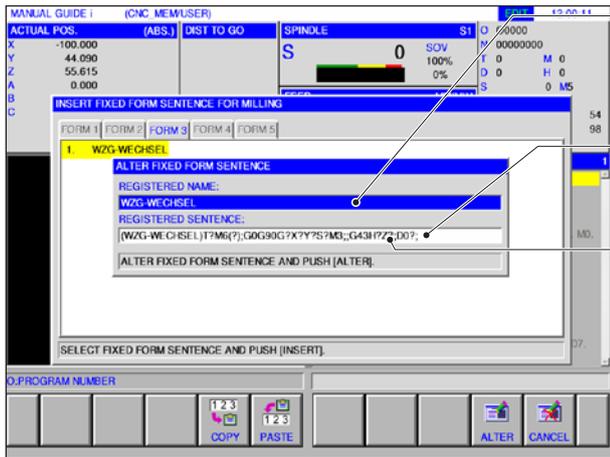
- The fixed form set menu displayed under the tab "FORM 1" has the same content as the menu displayed under "START".
- The fixed form set menu displayed under the tab "FORM 5" has the same content as the menu displayed under "END".

4 Press the softkey to create a new fixed form or to process an existing fixed form.



5 Press softkey to delete a fixed form.

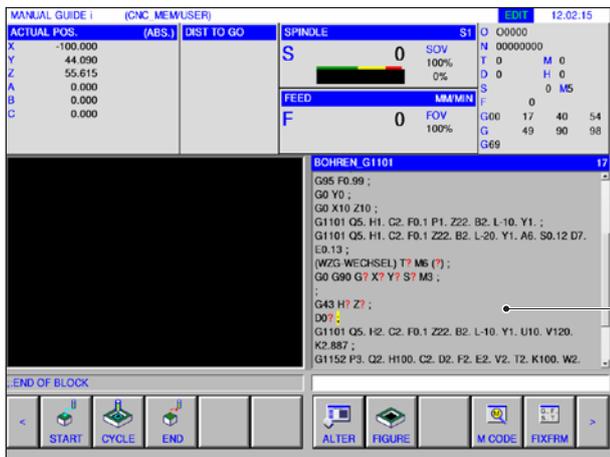




6 Enter or change registration name.

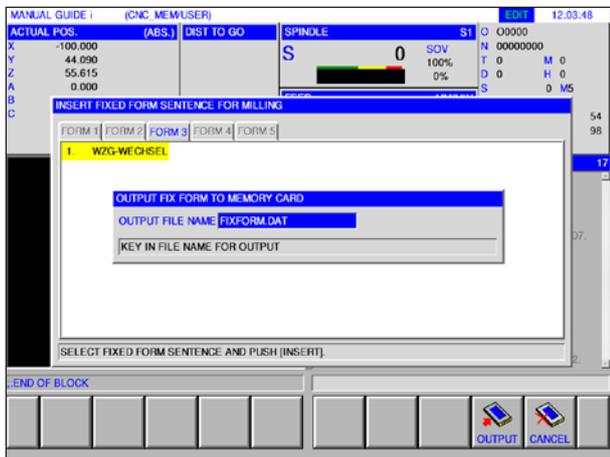
7 Enter or change registration set.

When the special character "?" appears in the parts program, the operator is requested to enter defined data.



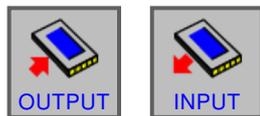
**Example:** Tool change for milling

(TOOL CHANGE) T???? M6 (?);  
G0 G90 G? X? Y? S? M3;



8 Save pre-defined block on external memory card or upload it.

9 Press softkey to open dialogue for saving or uploading.



10 The softkey "STAND." performs a reset of the fixed forms to the condition after the software installation (corresponds to the condition pre-set by the factory).

Fixed form sets that have been entered or altered will be deleted or reset.



11 "TO MNU" Return to the selection of a pre-defined block.

## M-code menu



1 Select "Edit" mode.

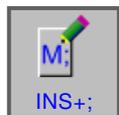


2 Open M-code menu.

Select the desired M-code with the cursor keys.



3 Pressing the "INSERT" softkey inserts the selected M-code into the program directly after the cursor position.



4 Pressing the "INS+;" softkey inserts an End of Block (EOB) directly after the M-code. If several sequential M-codes are inserted in a block, press the "INS +;" softkey to close the entry.

# E: G code programming



## Instructions:

This programming manual describes all functions that can be executed with WinNC. Depending on the machine you are operating with WinNC, not all functions may be available.

## Example:

The Concept MILL 55 milling machine has no position-controlled main spindle, so it is also not possible to program spindle positions.

## Overview

### M-commands

M00	Programmed stop	M25	Open clamping device
M01	Optional stop	M26	Close clamping device
M02	End of program	M27	Swivel divider
M03	Spindle ON clockwise	M29	Thread tapping without compensation chuck
M04	Spindle ON counterclockwise	M30	End of main program
M05	Spindle stop	M51	C-axis mode activation
M06	Perform tool change	M52	C-axis mode deactivation
M07	Minimal lubrication ON	M71	Exhaust ON
M08	Coolant ON	M72	Exhaust OFF
M09	Coolant OFF, minimal lubrication OFF	M98	Call subroutine
M10	Divider clamp on	M99	Jump back to calling program
M11	Slacken divider clamping		



## Calculation operators for NC program

Command	Meaning
[, ], *, /, +, -, =	Calculation functions
SIN()	Sine function
COS()	Cosine function
TAN()	Tangent function
ASIN()	Arc sine function
ACOS()	Arc cosine function
ATAN()	Arc tangent function (value)
ATAN2(,)	Arcurod function (X-section, Y-section)
SQRT()	Square root function
EXP()	Exponential function (base e)
LN()	Natural logarithmic function
ABS()	Absolute function
RND()	Rounding function
MOD()	Modulo function
FIX()	Detach
ROUND	Round down

## Overview of machine G-commands

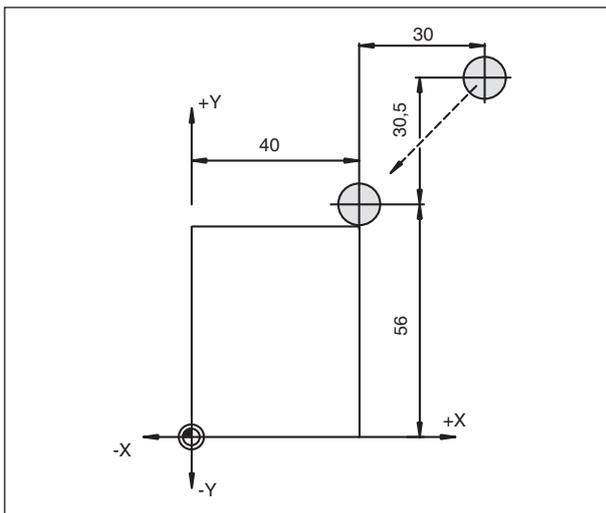
G-code	Group	Meaning
G00	01	Rapid traverse
G01		Straight interpolation
G02		Circular interpolation/clockwise
G03		Circular interpolation/counterclockwise
G04	00	Dwell time
G09		Precision stop (block-by-block)
G10		Data setting
G15	17	End polar coordinates command
G16		Polar coordinates command
G17	02	XY plane selection
G18		ZX plane selection
G19		YZ plane selection
G20	06	Switch to imperial (inches) input
G21		Switch to metric input
G40	07	End mill radius compensation
G41		Left mill radius compensation
G42		Right mill radius compensation
G43	08	Tool length offset plus
G44		Tool length offset minus
G49		Tool length offset end
G50	11	End scaling
G51		Scaling
G50.1	22	End programmable axis mirroring
G51.1		Programmable axis mirroring
G52	0	Additive zero point offset programming
G53		Zero point offset suppression
G54	14	Workpiece coordinates system 1 selection
G55		Workpiece coordinates system 2 selection
G56		Workpiece coordinates system 3 selection
G57		Workpiece coordinates system 4 selection

Code Grp. A	Group	Meaning
G58	14	Workpiece coordinates system 5 selection
G59		Workpiece coordinates system 6 selection
G61	15	Precision stop (effective modally)
G64		Path control operation
G65	00	Macro call
G66	12	Modal macro call
G67		End modal macro call
G68	16	Rotate coordinates system
G73	09	Deep hole boring with chip breaking
G74		Left-hand thread cutting cycle
G76		Fine boring cycle
G80		End fixed cycle
G81		Boring cycle (modal)
G82		Boring cycle with dwell time
G83		Deep hole boring with chip removal
G84		Right-hand thread cutting cycle
G85		Boring cycle with withdrawal (in feed)
G89		Boring cycle with dwell time and withdrawal
G90		
G91		Incremental dimension programming
G94	10	Feed in mm/min
G95		Feed in mm/revolution



## Brief description of G-commands

This description is an extract from the programming instructions for the WinNC for Fanuc 31i controller and is considered in first line as a programming aid.



*Absolute and incremental dimensional data*

### G00 Rapid traverse

#### Format

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

The carriages are traversed at maximum speed to the programmed objective (tool change position, start point for the next machining process).

#### Notes

- A programmed carriage feed  $F$  is suppressed during G00.
- The rapid traverse speed is set to fixed.
- The feed offset switch is limited to 100%.

#### Example

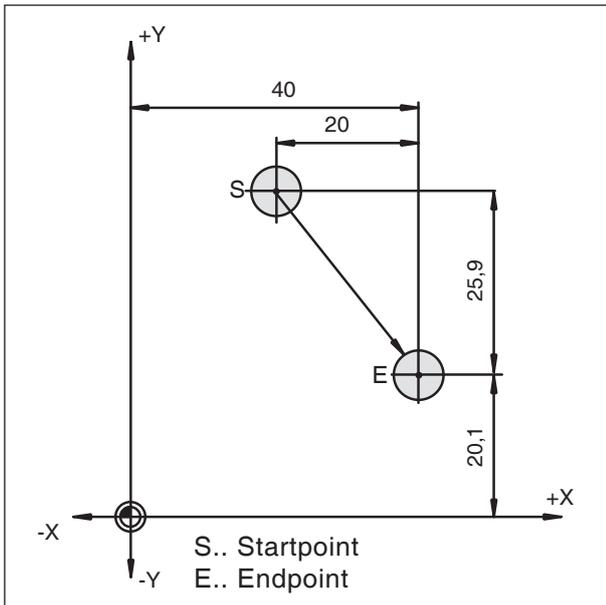
##### absolute G90

```
N40 G00 X70 Y86,5
N50 G00 X40 Y56
```

##### incremental G91

```
N40 G00 X70 Y86.5
N50 G00 G91 X-30 Y-30.5
```

## G01 Straight interpolation



Absolute and incremental values for G01

### Format

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

Straight line movement with programmed feed speed.

### Example

#### absolute G90

N.. G94

....

N10 G00 X20 Y46

N20 G01 X40 Y20.1 F200

#### incremental G91

N.. G94 F200

....

N10 G00 X20 Y46

N20 G01 G91 X20 Y-25.9

## Entering chamfers and radii

### Format

....

N... G01 X... Y... ,C/,R

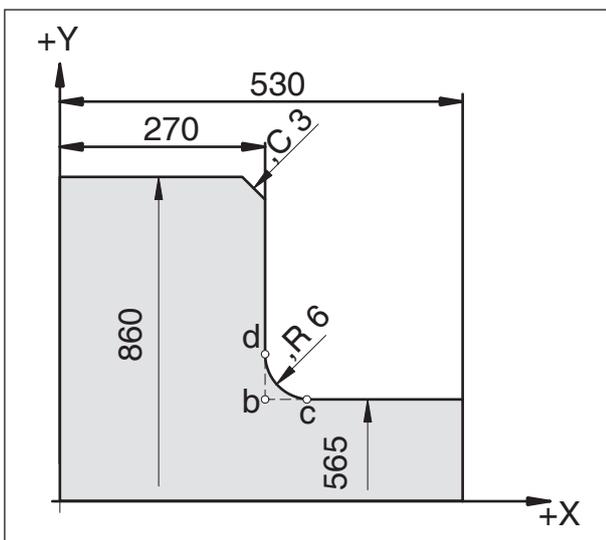
N... G01 X... Y...

### Notes

- Programming chamfers and radii is possible only for the active plane in each case. What follows defines the programming for the XY plane (G17).
- The movement which is programmed in the second block must begin with point b in the diagram. With incremental programming, the distance from point b must be programmed.
- In individual block mode, the tool stops first at point c and then at point d.

The following situations cause an error message:

- If the travel in one of the two blocks G00/G01 is so small that no intersection point was generated when the chamfer or radius was entered, error message no. 55 is generated.
- If no G00/G01 command is programmed in the second block, error message no. 51 or 52 is generated



Entering chamfers and radii

## Direct drawing dimension input

	Commands	Tool movements
1	$X_2...$ ( $Y_2...$ ) , $A...$	

### Notes

- Missing intersection points must not be calculated.  
Angles ( $A$ ), chamfers ( $C$ ) and radii ( $R$ ) can be programmed directly into the program.  
The block after a block with  $C$  or  $R$ , must be a block with  $G01$ .  
Programming a chamfer is only possible with the comma symbol " $C$ ", otherwise an error message appears because of unauthorized use of the  $C$ -axis.
- Input of angles ( $A$ ) is possible only with the comfort programming option.
- The following G-commands should not be used for blocks with chamfer or radius:
  - G-commands in Group 00:  
 $G7.1$ ,  $G10$ ,  $G11$ ,  
 $G52$ ,  $G53$ ,  
 $G73$ ,  $G74$ ,  $G76$ ,  $G77$ ,  $G78$
  - G-commands in Group 01:  
 $G02$ ,  $G03$ ,
  - G-commands in Group 06:  
 $G20$ ,  $G21$
- These must not be used between blocks with chamfer or radius which define the sequence numbers.



## G02 Circular interpolation, clockwise

## G03 Circular interpolation, counterclockwise

### Format

N... G02 X... Y... Z... I... J... K... F...

or

N... G02 X... Y... Z... R... F...

X,Y, Z,..... End point of the arc

I,J, K,..... Incremental circular parameter

(Distance from start point to the centre of the circle, I is assigned to the X-axis, J to the Y-axis, K to the Z-axis)

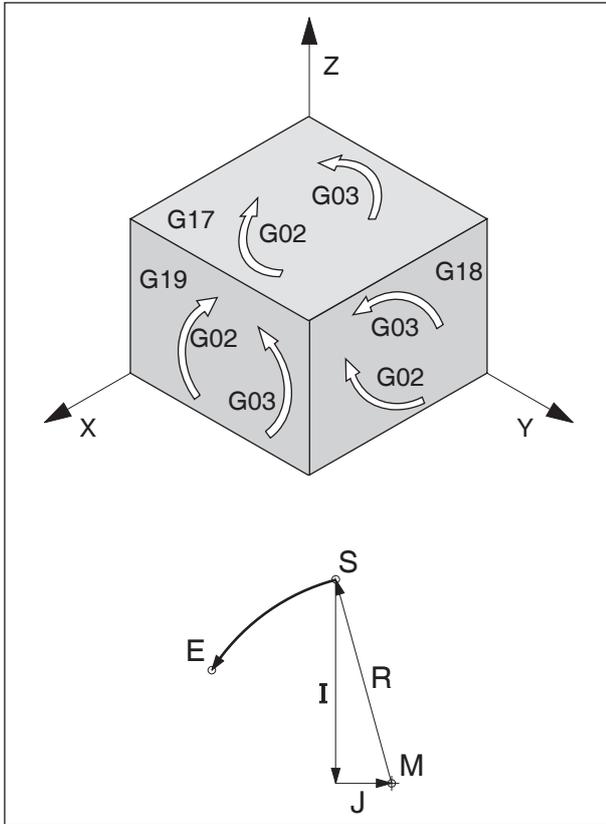
R..... Radius of the arc

Circle < semicircle for +R, > semicircle for -R, can be given instead of the parameters I, J, K.

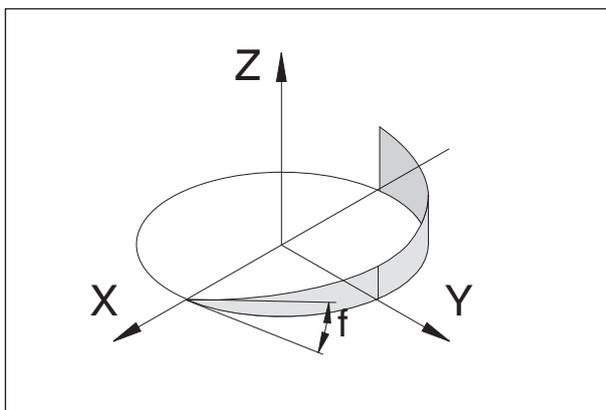
The tool is moved along the defined arc to the objective with the feed programmed under F.

### Notes

- You can only run a circular interpolation in the active plane.
- If I, J or K are equal to 0, the relevant parameters must not be given. The turning direction for G02, G03 is always considered vertically on the active plane.



G02 and G03 turning directions



Helical curve

## Helical interpolation

Normally, only two axes are given for a circle and these also determine the planes in which the circle is.

If a third, vertical axis is given, the movements of the axis carriages are coupled in such a way that a helical line results.

The programmed feed speed is not observed on the actual path, but on the circular (projected) path. The third axis, moved in a straight line, is also controlled so that it reaches the end point at the same time as the axes moved in a circle.

## G04 Dwell time

### Format

N G04 X [sec]

or

N G04 P [msec]

The tool is held for the time defined under X or P (in the last position reached) - sharp edges - clean transfers, mill base, precise stop.

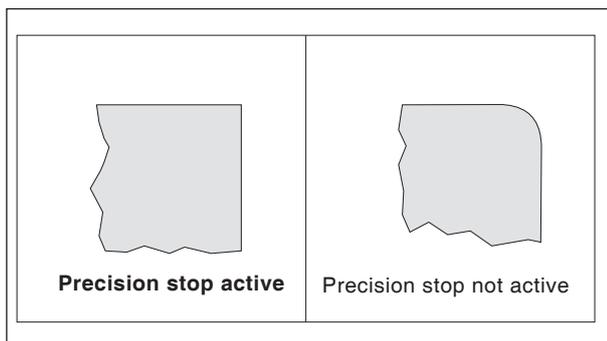
### Notes

- A decimal point cannot be used with the address P.
- The dwell time begins after the feed speed has reached that of the preceding block "NULL".

### Examples

N75 G04 X2.5 (dwell time = 2.5 sec)

N95 G04 P1000 (dwell time = 1 sec = 1000 msec)



## G09 Precision stop (block-by-block)

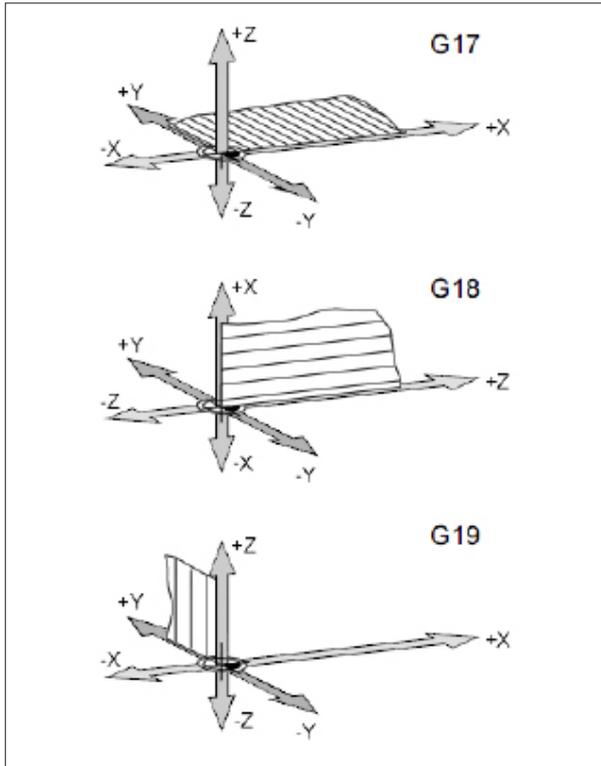
### Format

N G09

A block will only be processed if the carriages are braked to a stop.

In this way the corners will not be rounded and precise transitions achieved.

G09 is effective block-by-block



Planes in the workspace

## G17-G19 Plane selection

### Format

N.. G17/G18/G19

G17 to G19 determine the planes in which circular interpolation and polar coordinate interpolation can be executed and in which the mill radius offset can be calculated.

The tool offset is executed in the vertical axis on the active plane.

G17 XY plane

G18 ZX plane

G19 YZ plane

## G20 Dimensional data in inches

### Format

N.. G20

The G20 programming converts the following data into the imperial measurement system:

- Feed F [mm/min, inches/min, mm/rev, inches/rev]
- Offset values (NPV, geometry and wear) [mm, inches]
- Traverses [mm, inches]
- Current position indication [mm, inches]
- Cutting speed [m/min, feet/min]

## G21 Dimensional data in millimetres

### Format

N.. G21

Comments and instructions similar to those for G20.

## G28 Return to reference point

### Format

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

X, Y, Z, Absolute intermediate coordinates

The G28 command is used to approach the reference point via an intermediate position (X, Y, Z). First the movement to X, Y and Z, then movement to the reference point.

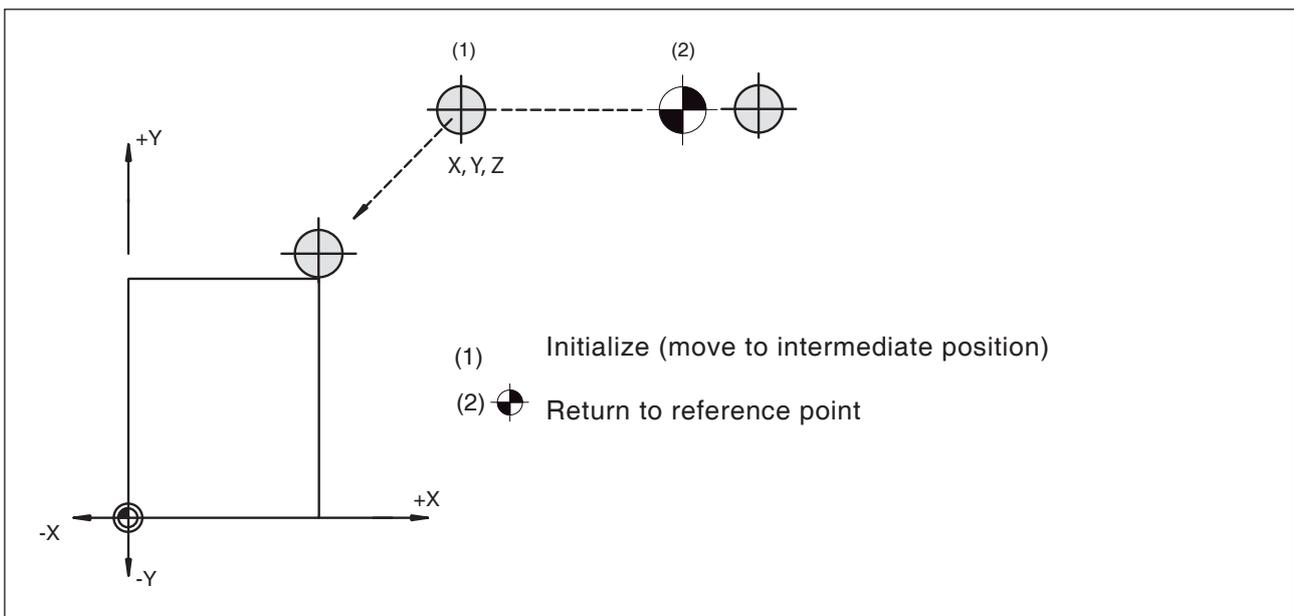
Both movements with G0.

The movement to the intermediate position can be programmed incrementally:

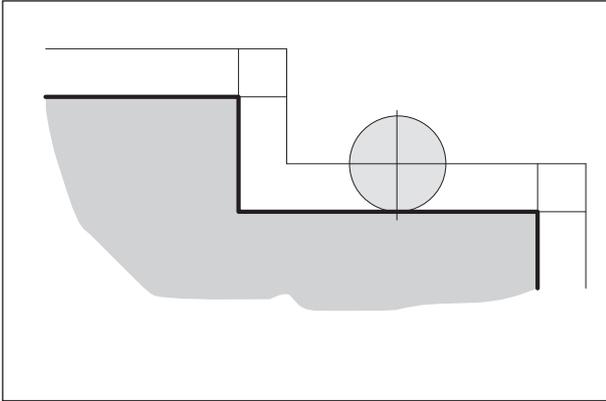
### G-Code group B/C

G91 G28 X10 Y10 Z10

G90



*Return to reference point*



Radius offset tool path

## Mill radius offset

If mill radius offset is used, a parallel contour path will be calculated by the controller automatically and the mill radius compensated in this way.

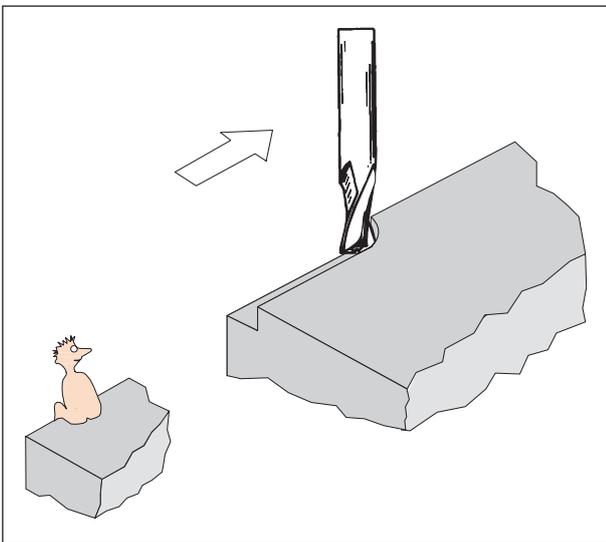
## G40 Deselection mill radius offset

G40 deselects mill radius offset.

Deselection is possible only in connection with a straight-line traverse movement (G00, G01).

G40 can be programmed in the same block as G00 or G01 or in the preceding block.

G40 is usually defined in the withdrawal to the tool change point.



Definition of G41 Mill radius correction left

## G41 Mill radius offset left

If the tool (viewed in the feed direction) is to the **left** of the contour to be machined, G41 must be programmed.

So that a radius can be calculated, when selecting the mill radius offset, you must state a D parameter from the table:

Tool correction => TOOL RADIUS OFFSET gaps

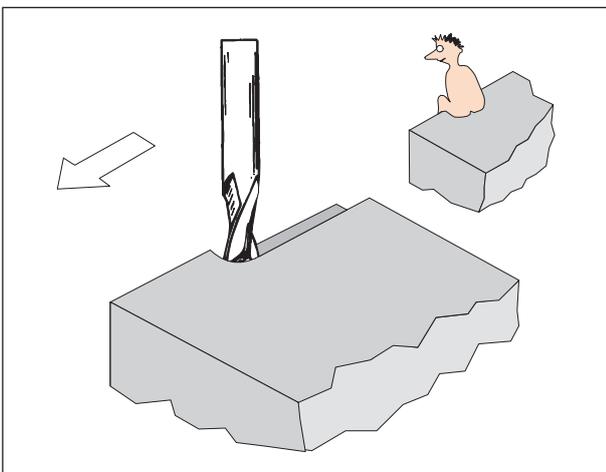
=> GEOMETRY AND WEAR

which equals the mill radius, e.g.

N.. G41 D..

### Notes

- A direct switch between G41 and G42 is not allowed - prior deselection with G40.
- Approach in connection with G00 or G01 required.
- The mill radius data is essential, the H parameter is effective only until it is deselected with H0 or a different H parameter is programmed.



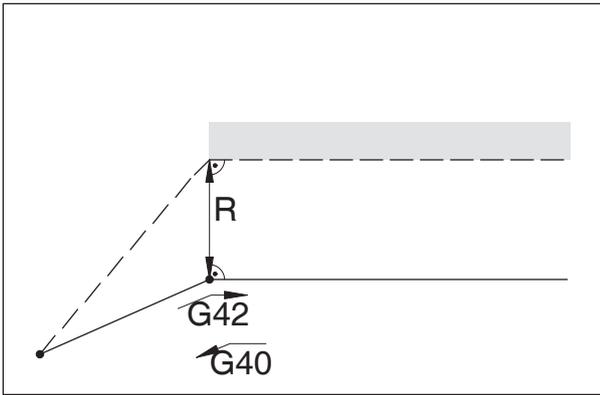
Definition of G42 Mill radius correction right

## G42 Mill radius offset right

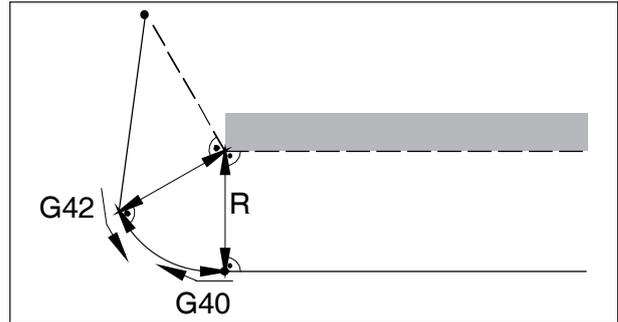
If the tool (viewed in the feed direction) is to the **right** of the contour to be machined, G42 must be programmed.

For instructions see G41!

**Tool paths when selecting/deselecting mill radius offset**

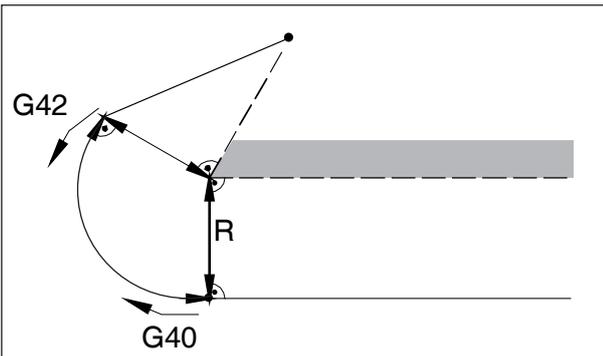


*Approach or move away from a corner point from the front*



*Approach or move way from the side to the rear*

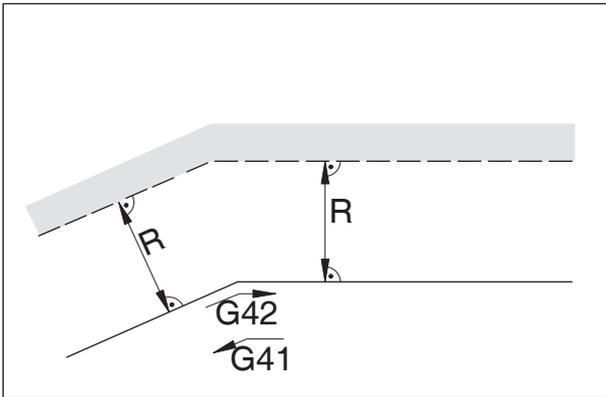
--- Programmed tool path  
 ——— Actual tool path



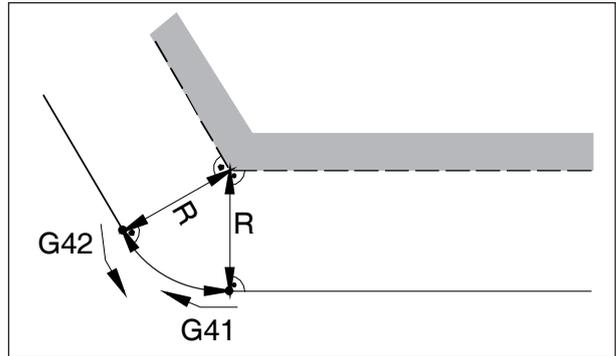
*Approach or move away from a corner point from behind*

For arcs, you always approach on the tangents in the circle start/end point.  
 The approach path to the contour and the path away from the contour must be greater than the mill radius R, otherwise the program will stop with an alarm.  
 If contour elements are less than the mill radius R, contour infringements can occur.

**Tool paths in the program sequence with mill radius offset**

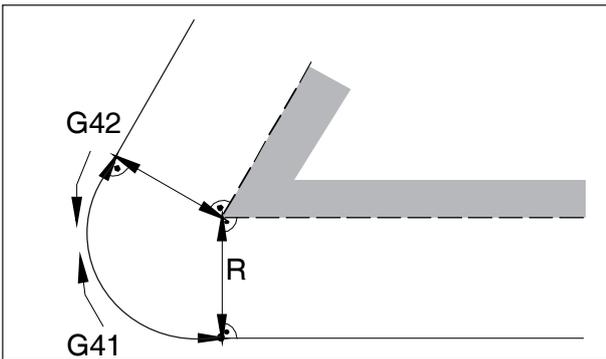


*Tool path at an inside corner*



*Tool path at an outside corner > 90°*

--- Programmed tool path  
 — Actual tool path



*Tool path at an outside corner < 90°*

For arcs, you always approach on the tangents in the circle start/end point.

If contour elements are less than the mill radius R, contour infringements can occur.

**G43 Tool length offset positive****G44 Tool length offset negative****Format**

N.. G43/G44 H..

G43 and G44 can call up a value from the offset register (GEOMT) and add or subtract it as a tool length. This value is added or subtracted for all subsequent Z movements (with active XY plane - G17) in the program.

**Example**

N.. G43 H05

The value in line 5 of the table:  
Tool correction gaps TOOL LENGTH CORRECTION GEOMETRY AND WEAR  
is added as tool length to all subsequent Z movements.

**G49 Deselection tool length offset**

The positive (G43) or negative (G44) offset is cancelled.

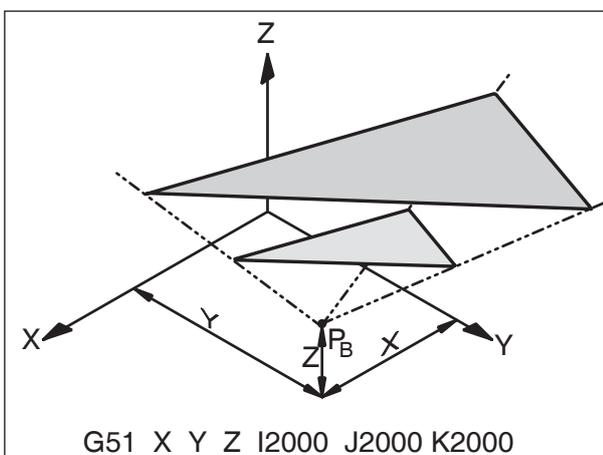
**G50 Deselection scaling factor****G51 Scaling factor****Format**

N.. G50

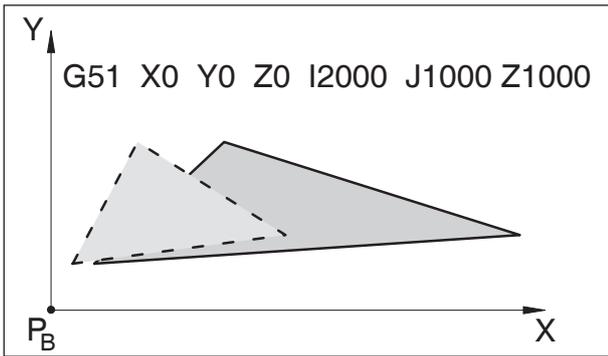
N.. G51 X.. Y.. Z.. I.. J.. K..

G51 computes all position data to scale, until the scale is deselected with G50. X, Y and Z determines a reference point  $P_B$ , from which all the dimensions are calculated.

I, J and K determine an appropriate scaling factor (in 1/1000) for each axis.



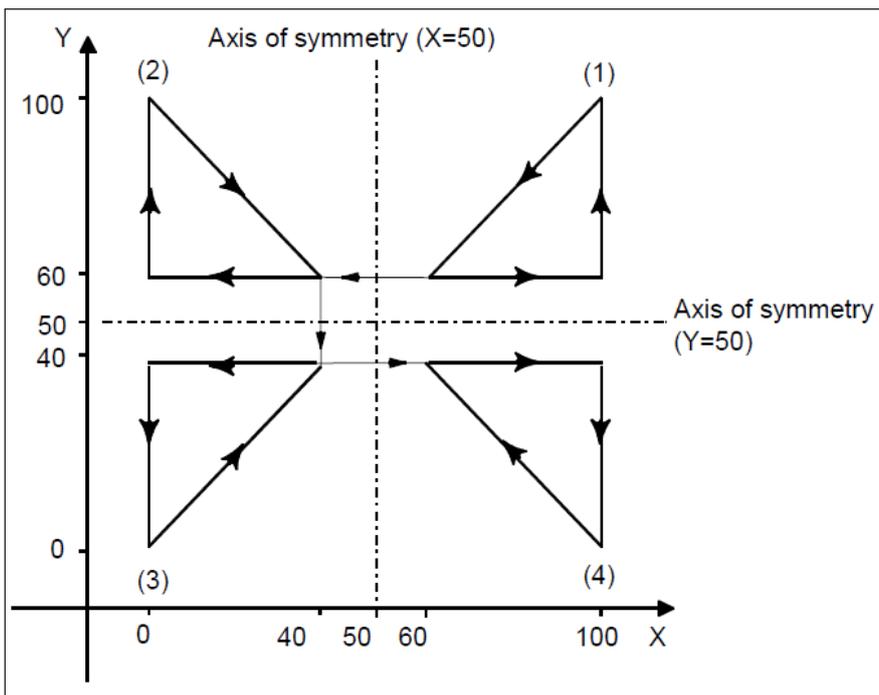
*Enlarging a contour*



Distorting a contour: X 1:2, Y,Z 1:1

If different scaling factors are given for the individual axes, the contours are distorted. Circular movements must not be distorted or an alarm will sound.

### G51.1 Mirroring a contour G50.1 Deselecting mirroring



Programmable mirror image

- (1) Original image of a programmed command
- (2) Image symmetrical about a line parallel to the Y-axis and crossing the X-axis at 50
- (3) Image symmetrical about point (50,50)
- (4) Image symmetrical about a line parallel to the X-axis and crossing the Y-axis at 50

## G52 Local coordinates system

### Format

N.. G52 X.. Y.. Z..

G52 offsets the instantaneous coordinate source around the X, Y and Z values. Use this to create a sub-coordinate system to the existing coordinate system.

The programmed offset is maintained until a different offset is called.

## G53 Machine coordinates system

### Format

N.. G53

The machine zero point is set by its manufacturer (EMCO milling machines: at the left front edge of the machine table).

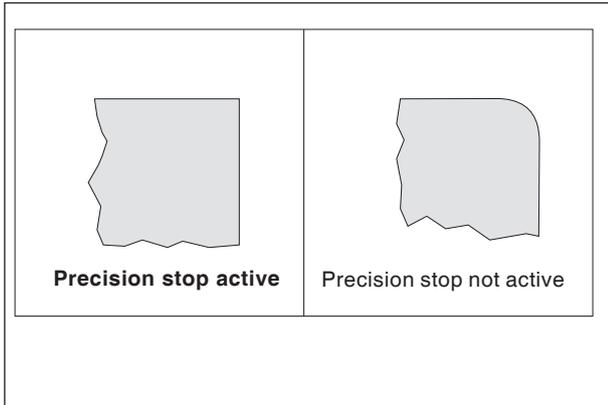
Certain work steps (tool change, measurement position, etc.) are always executed at the same position in the workspace.

G53 disables the zero point offset for a program block and the coordinate data is related to the machine zero point.

## G54-G59 Zero point offsets 1-6

Six positions in the workspace can be preset as zero points (e.g. points on fixed mounted clamping devices). These zero point offsets are called with G54 - G59.

See chapter A Entering zero point offset fundamentals.



## G61 Precision stop (modally effective)

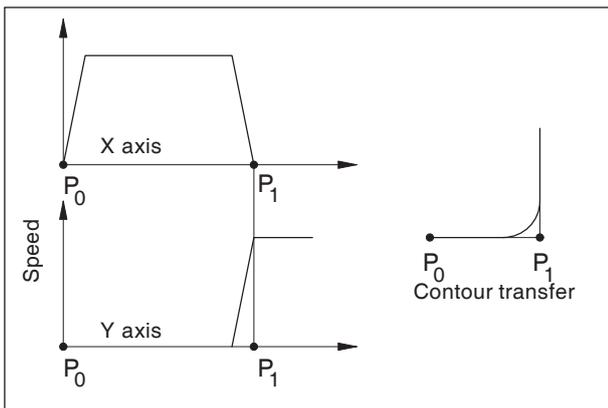
### Format

N.. G61

A block will only be processed if the carriages are braked to a stop.

In this way the corners will not be rounded and precise transitions achieved.

G61 effective until it is deselected with G64.



Speed characteristic of the carriage with G64

## G64 Cutting mode

### Format

N.. G64

The Y-axis is accelerated before reaching the target point in the X-direction. This achieves an even movement during contour transfers. The contour transfer is not an exactly sharp angle (parabola, hyperbola).

The contour transfer size is normally within the tolerance range in the drawings.

## G65 Macro call

G65 calls up a macro with user-defined values. A macro is a subroutine which runs a given operation with values assigned to variable parameters (boring image, contours).

### Format

G65 Pxxxx Lrrrr Arguments

or

G65 Pxxxx Lrrrr Arguments

Xxxx is the macro number (e.g. O0123)

- rrrr the repetition value
- "program.CNC" is the name of the macro file
- Arguments is a list of variable designators and values.

Arguments for macro calls are given by using the letters A-Z, excluding G, L, N, O and P.

Macros are written like normal programs. However, macro programs can access their arguments with numbers: #1 for A, #2 for B etc. (exceptions: # 4-6 for I-K, # 7-11 for D-H).

A macro can use the negative of an argument with a minus sign in front of the '#'. Other computing operations are not supported.

Macros can call other macros (up to 4 levels in depth), macro M-functions and sub-routines. Macro M-functions and sub-routines can call macros.

### Example for a main program

G65 <TEST.CNC> A5 B3 X4

Macro TEST.CNC:

G1 X#X Y#A Z-#B

This call will produce

G1 X4 Y5 Z-3

### G66 Macro call (modal)

**Format**

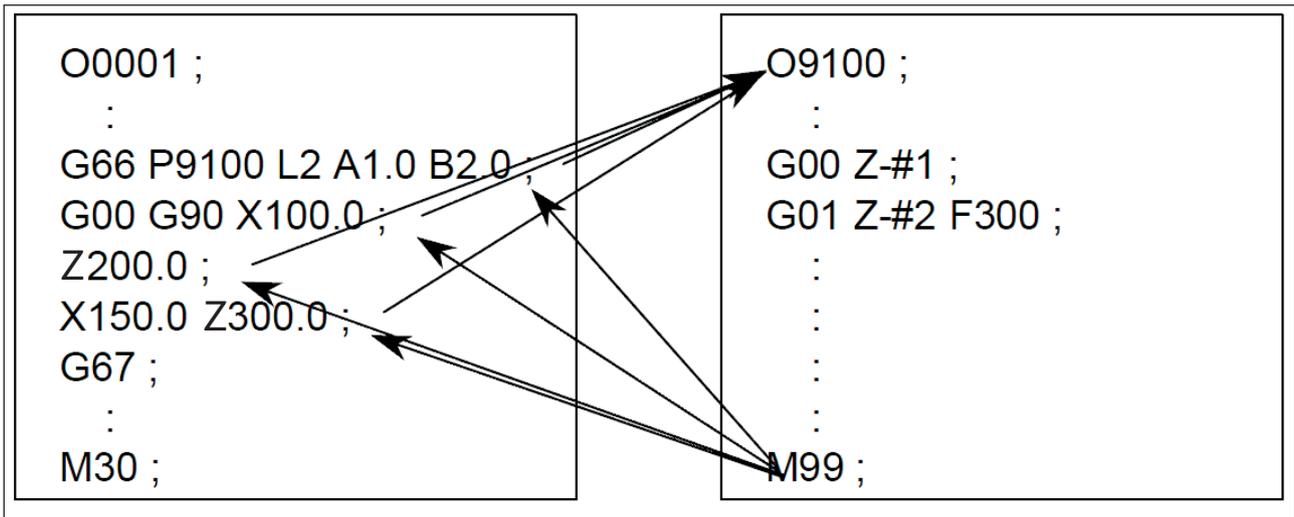
N.. G66 P.. L..Argument

..

P ..... Program number

L..... Number of repetitions (default is 1)

Argument... A list of variable designators and values which are transferred to the macro.



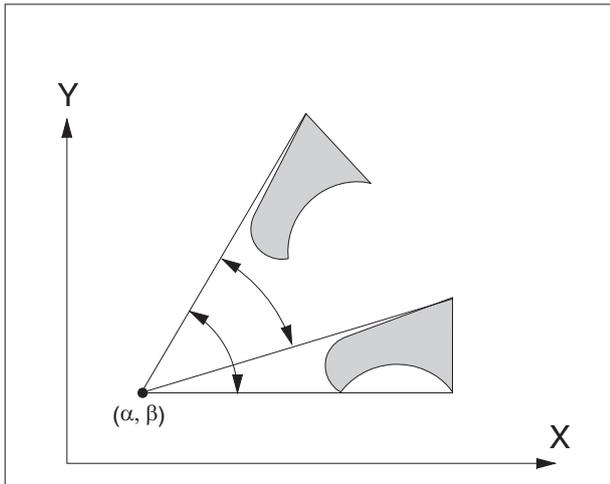
Example of macro call programming

### G67 Macro call (modal) End

**Format**

N.. G67

This function ends the macro call.



G68/G69 coordinates system rotation

## G68 Coordinate system rotation

### Format

N.. G68 X.. Y.. R..

..

N.. G69

G68 ..... Coordinates system rotation ON

G69 ..... Coordinates system rotation OFF

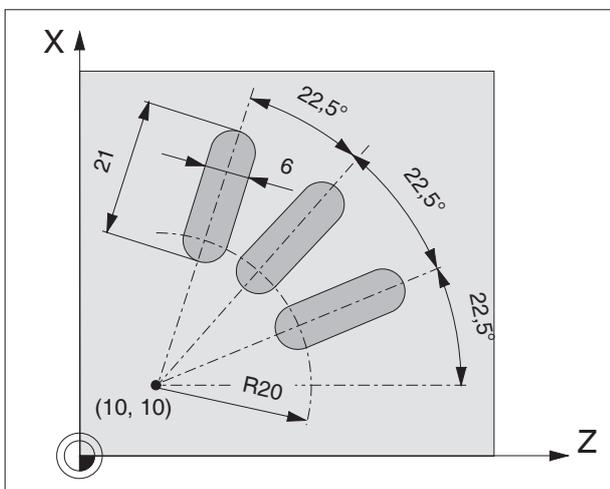
X / Y ..... Designates the coordinates of the rotation centre point in the relevant plane.

R ..... Gives the angle of rotation

This function can, for example, change programs by using a rotation command.

### Instructions:

Rotation takes place in the applicable plane (G17, G18 or G19) in each case.



Example/Coordinates system rotation

### Example

N5 G54

N10 G43 T10 H10 M6

N15 S2000 M3 F300

N20 M98 P030100 ; sub-routine call

N25 G0 Z50

N30 M30

00100 (sub-routine 0100)

N10 G91 G68 X10 Y10 R22.5

N15 G90 X30 Y10 Z5

N20 G1 Z-2

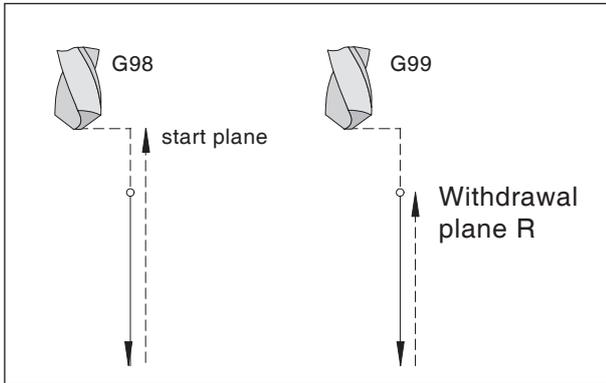
N25 X45

N30 G0 Z5

N35 M99

## Drilling Cycles G73 - G89

### Systematic G98/G99



Movements with G98 and G99

G98.....After reaching the drilling depth the tool retracts to the start plane

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

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

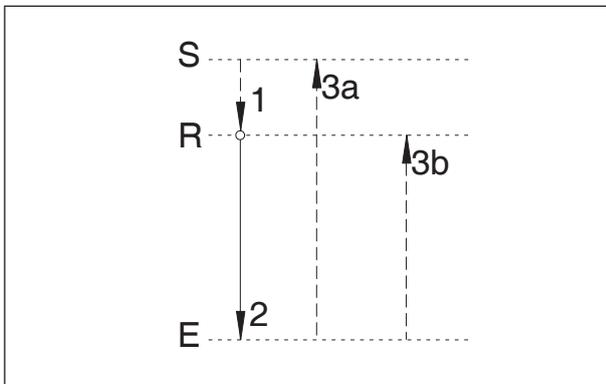
The computation of the R parameter is different with incremental and absolute programming:

**Absolute programming (G90):**

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

**Incremental programming (G91):**

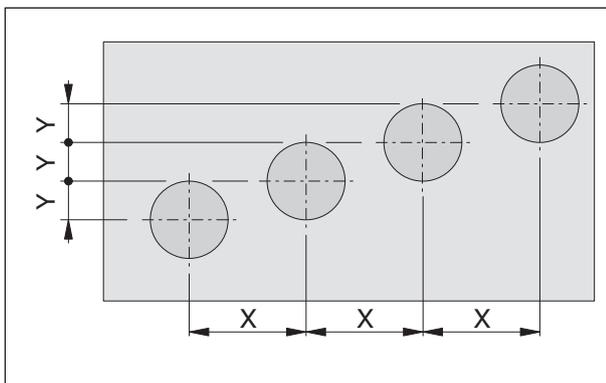
R defines the height of the withdrawal plane related to the last Z position (start position of the drilling cycle). With a negative value for R the withdrawal plane will be below the start position, with a positive value the withdrawal plane will be over the start position.



Sequence of movements G98, G99

### Sequence of movements

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



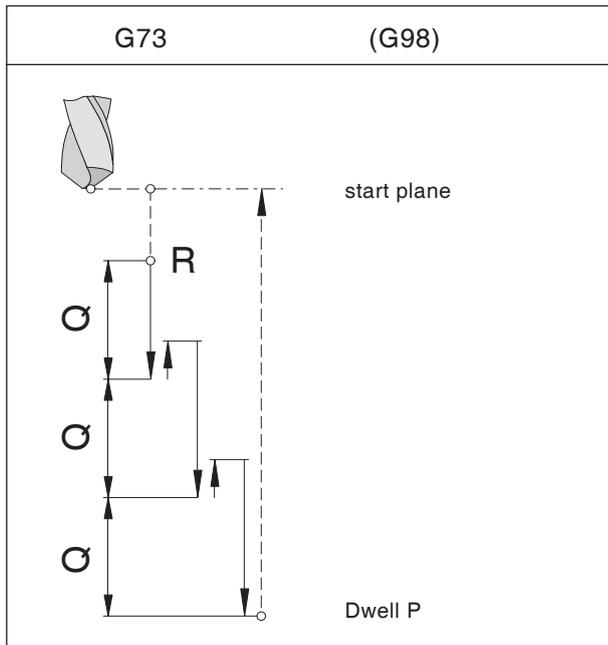
Cycle repetition for a row of holes

### Number of repetitions

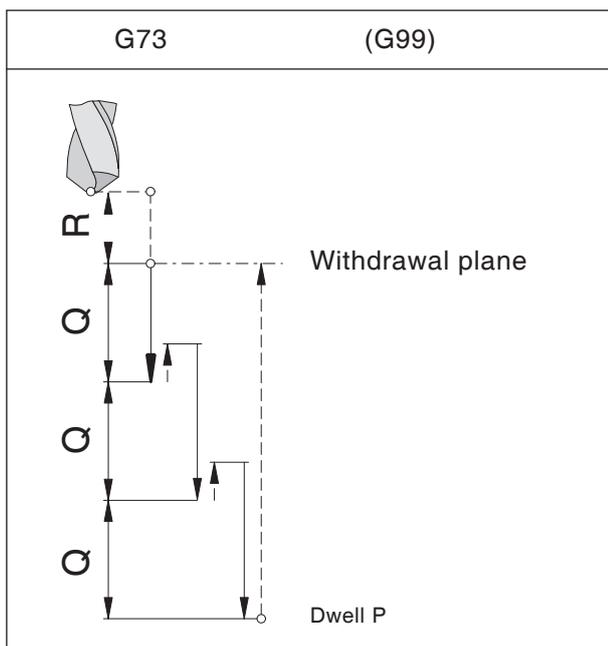
The K parameter defines the number of repetitions of the cycle.

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

Make sure that G98 is active!



Chip break drilling with retraction to the startplane



Chip break drilling with retraction to the withdrawal plane

## G73 Chip Break Drilling Cycle

### Format

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

The tool dips into the work piece for the infeed Q, drives back 0,5 mm to break the chips, dips in again etc. until end depth is reached and retracts with rapid feed.

### Applications

deep borings, material with bad cutting property

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

X, Y ..... Hole position

Z ..... Absolute (incremental) drilling depth

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

P [msec]..... Dwell at the hole bottom  
P1000 = 1 sec

F ..... Feed rate

Q [mm]..... Cutting division - infeed per cut

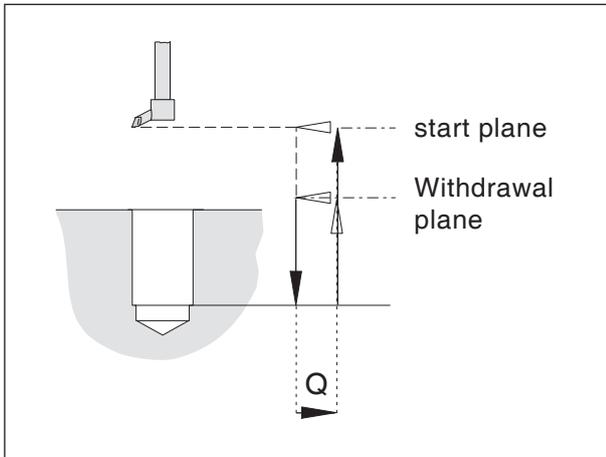
K ..... Number of repetitions

## G74 Left Tapping Cycle

With this cycle left threads can be produced. The cycle G74 works like G84 but with reversed turning directions.

Tapping chuck without length compensation is activated with M29.

See Tapping Cycle G84.



Fine drilling cycle

## G76 Fine Drilling Cycle

Only for machines with oriented spindle stop.

### Format

N...G98(G99) G76 X... Y... Z... (R)... F... Q...  
K...

This cycle is for enlarging borings with boring and facing heads.

The tool traverses with rapid feed to the withdrawal plane, with the programmed feed to the end depth, the milling spindle will be stopped oriented, the tool traverses with rapid speed horizontally (Q) off the surface in direction of the positive Y-axis and traverses with rapid speed to the withdrawal plane (G99) or start plane (G98) and traverses back for the value Q to the original position.

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

X, Y ..... Hole position

Z ..... Absolute (incremental) drilling depth

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

F ..... Feed

Q ..... Horizontal traverse-off value

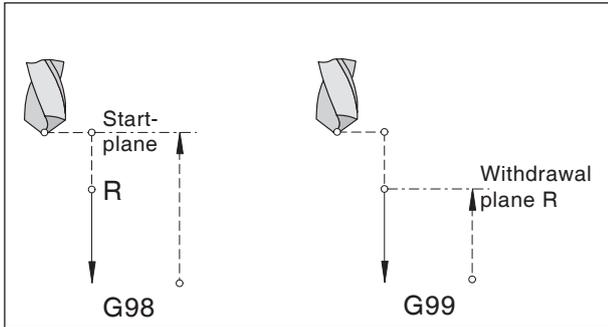
K ..... Number of repetitions

## G80 Cancel Drilling Cycles

### Format

N... G80

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



Drilling cycle

## G81 Drilling Cycle

### Format

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

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

### Application:

Short drillings, material with good cutting properties

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

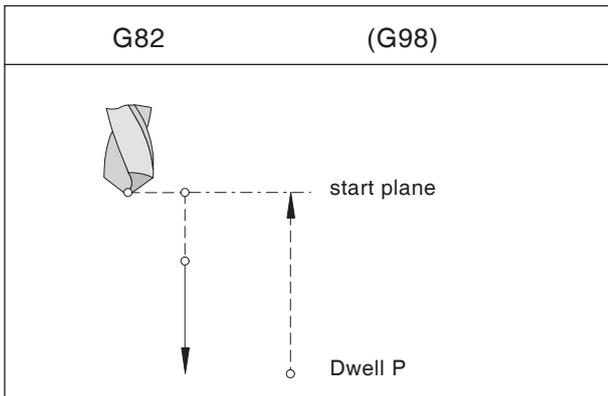
X, Y ..... Hole position

Z ..... Absolute (incremental) drilling depth

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

F ..... Feed

K ..... Number of repetitions



Drilling cycle with dwell and retraction to the start plane

## G82 Drilling Cycle with Dwell

### Format

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

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

### Applications

Short borings, material with good cutting property

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

X, Y ..... Hole position

Z ..... Absolute (incremental) drilling depth

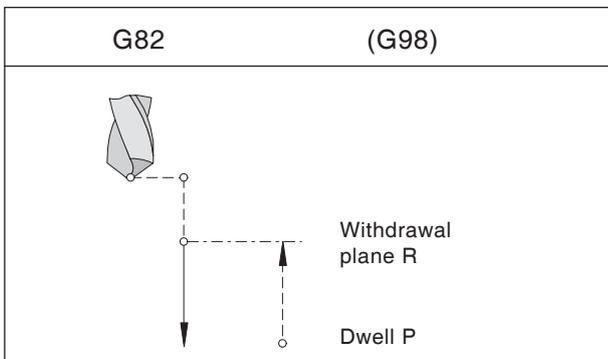
R [mm]..... Absolute (with G91 incremental) value of the withdrawal plane

P [msec]..... Dwell at the hole bottom

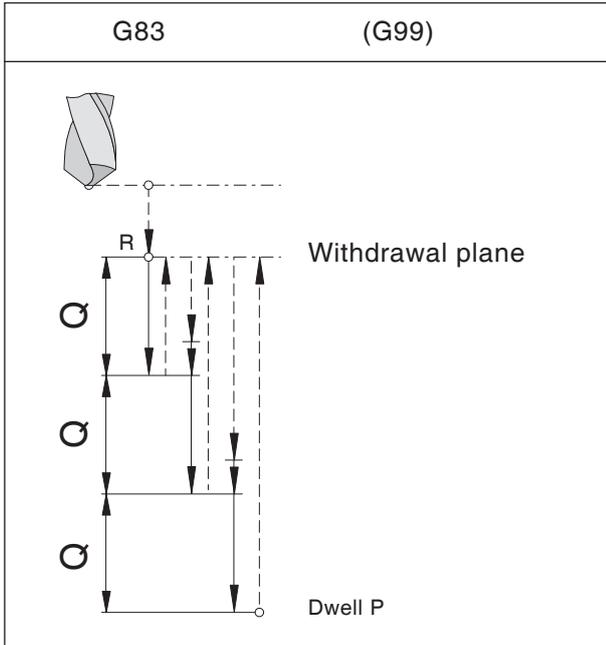
P1000 = 1 sec

F ..... Feed rate

K ..... Number of repetitions



Drilling cycle with dwell and retraction to the withdrawal plane



Deep hole drilling with retraction to the withdrawal plane

## G83 Withdrawal Drilling Cycle

### Format

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

The tool dips into the work piece for the infeed Q, drives back to the withdrawal plane, to break the chips and remove it from the hole, traverses with rapid speed until 0,5 mm over the previous drilling depth, dips in again for the infeed Q etc. until end depth is reached and retracts with rapid feed.

### Applications

deep borings, (soft) material with long chips

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

X, Y ..... Hole position

Z ..... Absolute (incremental) drilling depth

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

P [msec]..... Dwell at the hole bottom  
P1000 = 1 sec

F ..... Feed rate

Q [mm]..... Cutting division - infeed per cut

K ..... Number of repetitions

## G84 Tapping Cycle without length compensation

### Format

N.. M29 S... G98(G99) G84 X... Y... Z... (R)... F... P... K...

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

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

X, Y ..... Hole position

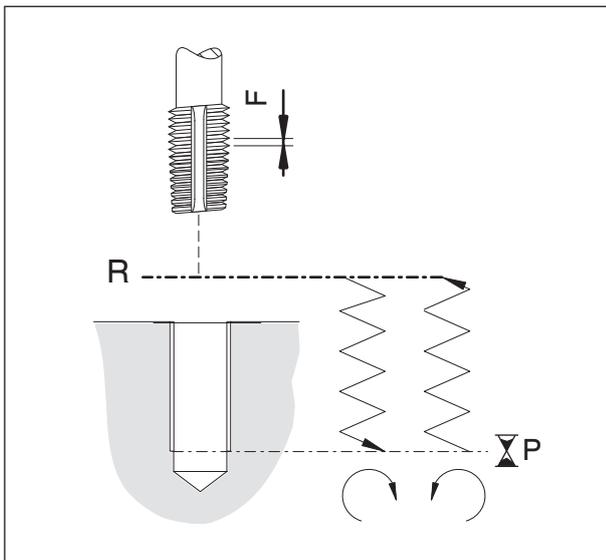
Z ..... Absolute (incremental) tapping depth

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

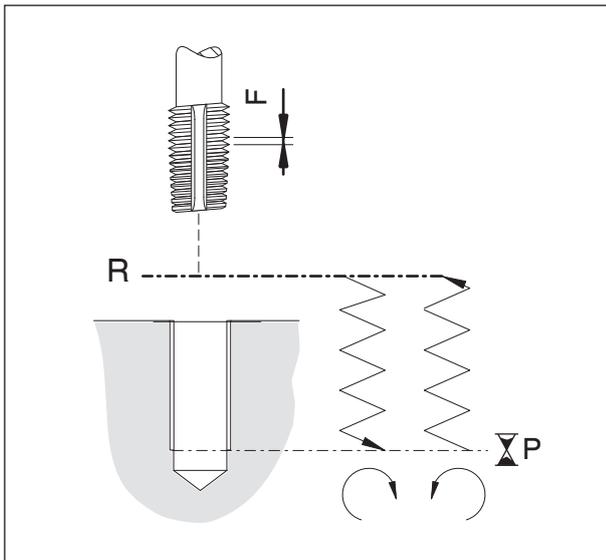
F ..... Feed rate with G94  
Thread pitch with G95

P [msec]..... Dwell at thread ground  
P 1000 = 1 sec

K ..... Number of repetitions



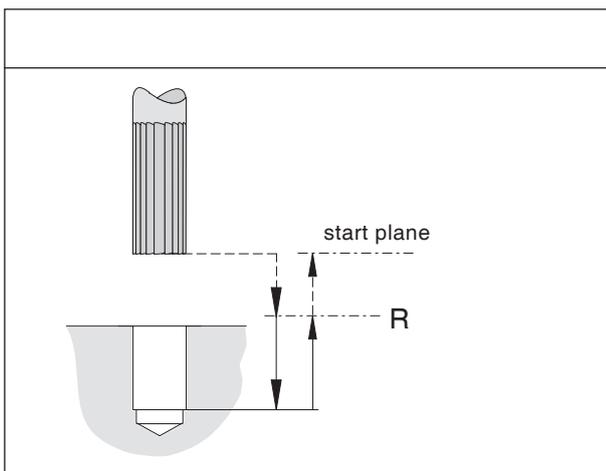
Tapping cycle (with G99)



Tapping cycle (with G99)

G85

(G98)



Reaming cycle with withdrawal to the start plane

## G84 Tapping Cycle with length compensation

### Format

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

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

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

X, Y ..... Hole position

Z ..... Absolute (incremental) tapping depth

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

F ..... Feed rate with G94  
Thread pitch with G95

P [msec] ..... Dwell at thread ground  
P 1000 = 1 sec

K ..... Number of repetitions

## G85 Reaming Cycle

### Format

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

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

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

X, Y ..... Hole position

Z ..... Absolute (incremental) drilling depth

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

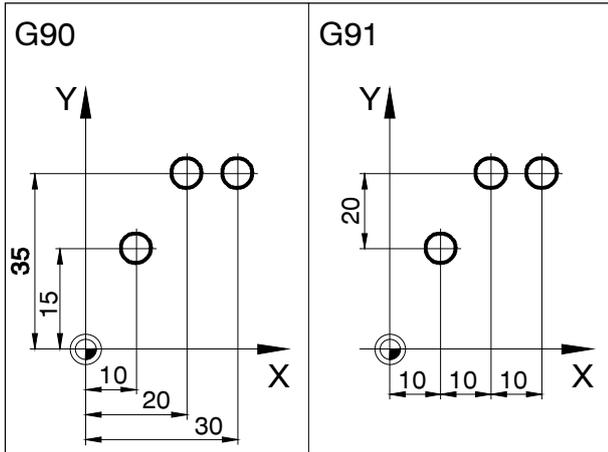
F ..... Feed rate

K ..... Number of repetitions

## G89 Reaming cycle with dwell time

See G85

The tool moves at feed speed to the final depth and stops for the dwell time P. The retraction to the retraction plane takes place with feed, dependent on G98, the start plane is reached with rapid traverse.



*Absolute value and increment value programming*

## G90 Absolute value programming

### Format

N.. G90

There are two ways to program travels of the tool: the absolute programming, and the incremental programming.

In the absolute programming, coordinate value of the end position is programmed.

The incremental programming is used to program the amount of a tool movement.

### Notes

- A direct change between G90 and G91 is also allowed block by block.
- G90 (G91) may also be programmed in conjunction with other G-functions:  
(N.. G90 G00 X... Y... Z...)

## G91 Incremental value programming

### Format

N.. G91

Instructions similar to those for G90.

## G94 Feed per minute

G94 means all values defined under F (feed) as mm/min.

### Format

N.. G94 F...

## G95 Feed per revolution

G95 means all values defined under F (feed) as mm/rev.

### Format

N.. G95 F...



# F: Tool management

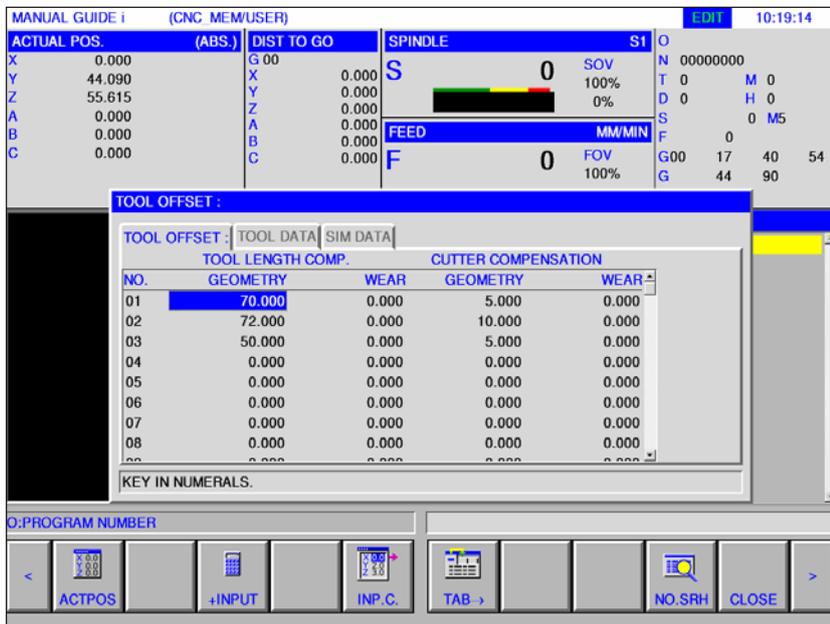
## Tool settings



- Press softkey.
- "T-OFS" displays the tool offset data window.

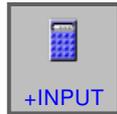
3 register cards are available for selection:

- Tool offset (tool lengths and tool radius compensation)
- Tool data
- Simulation data



Parameter	Meaning
NR	Tool number. A maximum of 99 tool offsets is available
<b>Tool length offset</b>	
Geometry	Entry of the measured geometry data (with default setting device)
Wear	Tool wear entry
<b>Tool radius compensation</b>	
Geometry	Entry of the measured geometry data (with default setting device)
SCHN-wear	Tool wear entry

## Tool length offset entry

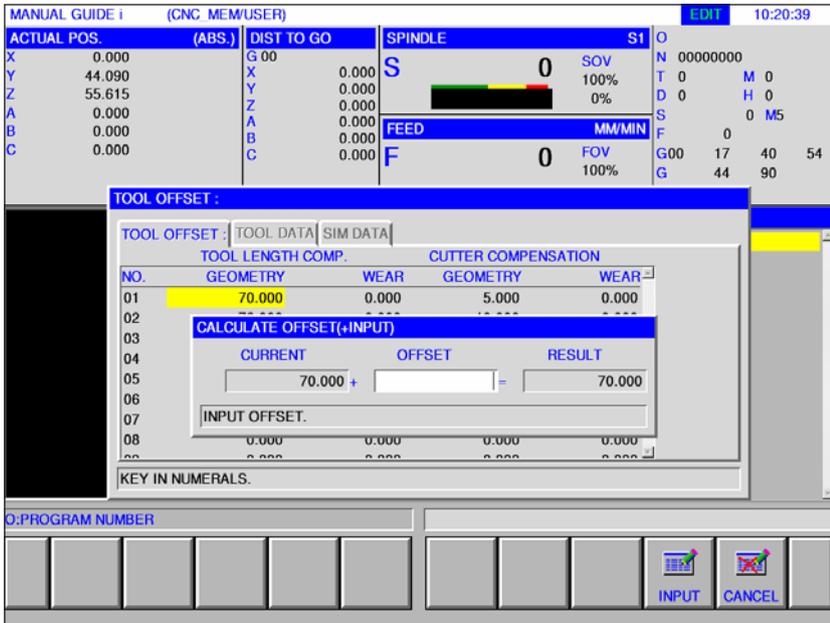


- Press softkey.

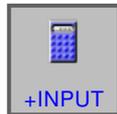
"+INPUT" opens the entry mask.

Current value + workpiece geometry offset is displayed.

Enter the value calculated previously for the geometry offset.  
"INPUT" closes the data entry.



## Tool radius offset entry

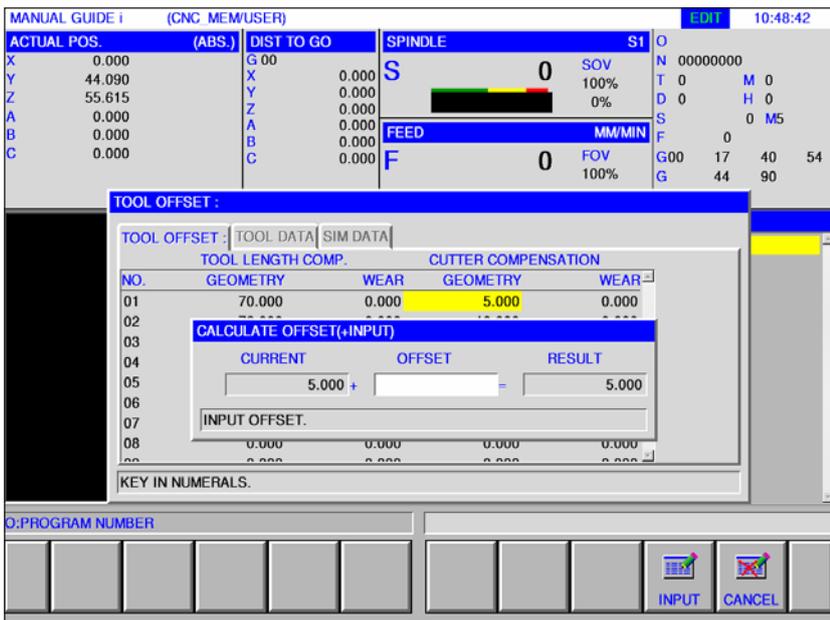


- Press softkey.

"+INPUT" opens the entry mask.

Current value + workpiece geometry offset compensation is displayed.

Enter the value calculated previously for the geometry offset.  
"INPUT" closes the data entry.



## Tool wear offset

Tool wear offset entries should be made in the same way as those for tool length and tool radius offsets.

MANUAL GUIDE i (CNC MEM/USER) EDIT 10:50:01

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	O
X	0.000	G 00	S	0	N 00000000
Y	44.090	X 0.000	SOV	100%	T 0 M 0
Z	55.615	Y 0.000		0%	D 0 H 0
A	0.000	Z 0.000	FEED	MM/MIN	S 0 0 M5
B	0.000	A 0.000	F	0	F 0
C	0.000	B 0.000	FOV	100%	G00 17 40 54
		C 0.000			G 44 90

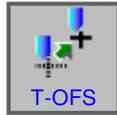
TOOL OFFSET :				
TOOL LENGTH COMP.		CUTTER COMPENSATION		
NO.	GEOMETRY	WEAR	GEOMETRY	WEAR
01	70.000	0.000	5.000	0.000
02	72.000	0.000	10.000	0.000
03	50.000	0.000	5.000	0.000
04	0.000	0.000	0.000	0.000
05	0.000	0.000	0.000	0.000
06	0.000	0.000	0.000	0.000
07	0.000	0.000	0.000	0.000
08	0.000	0.000	0.000	0.000
09	0.000	0.000	0.000	0.000

KEY IN NUMERALS.

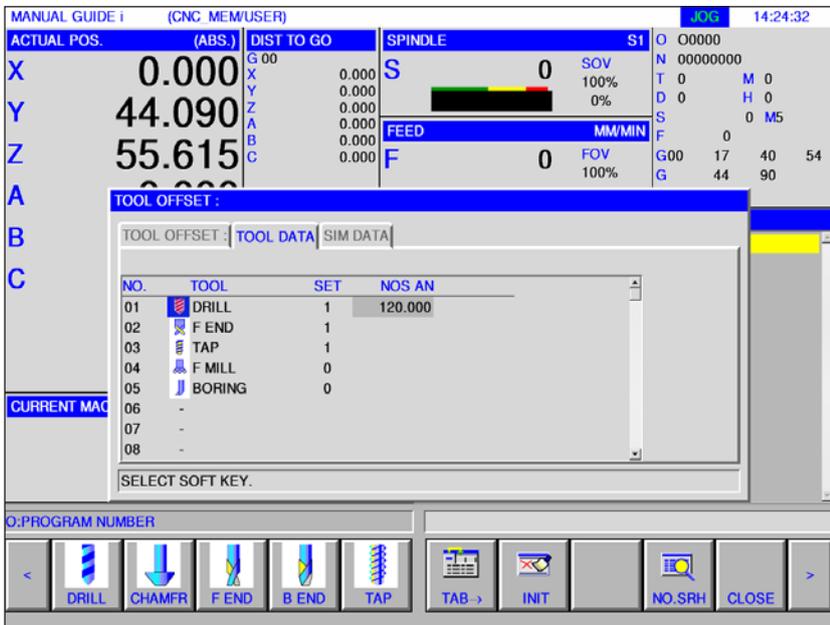
O:PROGRAM NUMBER

ACTPOS +INPUT INP.C. TAB-> NO.SRH CLOSE

## Tool data



- Press softkey.
- "T-OFS" displays the tool offset data window.



Parameter	Description
NR	Tool number
	Use the softkey to select the tool
TOOL	Tool name
SET	Setting 1 to 4. The tool tip is defined by the tool mounting position.
NOS ANGLE	Cutting angle

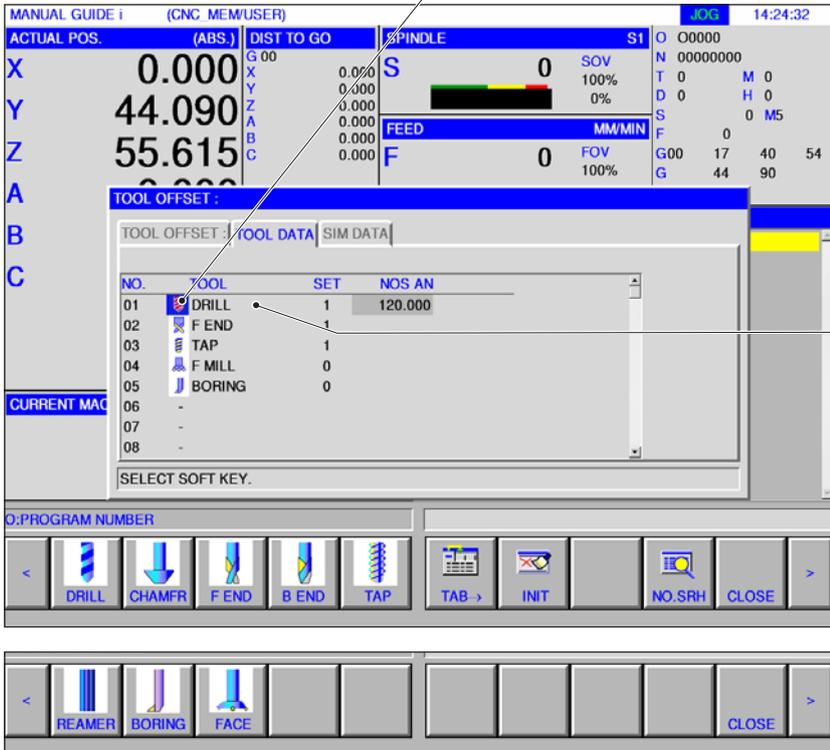
### Instructions:

The available parameters can differ, according to the tool selected.



### Select tool

- Click on the tool symbol or move the cursor on to it with the arrow keys and select the wanted tool.  
The following tools are available for lathes.



Enter tool name

Softkey bar with the available tools. Scroll the pages forwards and backwards with the expansion button.

**Instructions:**

The available parameters can differ, according to the tool selected.



**Tool setting number**

- Mark a tool setting number with the cursor for an instruction window to pop up in the right hand portion of the screen.  
For a boring tool, this is:  
Enter the tool setting number (1 to 4) to make the setting.

MANUAL GUIDE I (CNC MEM/USER) EDIT 10:53:03

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	O
X	0.000	G00	S	0	N 00000000
Y	44.090	X	SOV	100%	T 0 M 0
Z	55.615	Y	0%		D 0 H 0
A	0.000	Z	FEED	MM/MIN	S 0 M5
B	0.000	A	F	0	F 0
C	0.000	B	FOV	100%	G00 17 40 54
		C			G 44 90

**TOOL OFFSET :**

TOOL OFFSET: TOOL DATA SIM DATA

NO.	TOOL	SET	NOS AN
01	DRILL	1	120.000
02	F END	1	
03	TAP	1	
04	F MILL	0	
05	BORING	0	
06	-		
07	-		
08	-		

KEY IN NUMERALS.

O:PROGRAM NUMBER

ACTPOS TAB → INIT NO.SRH CLOSE

Settings for a boring tool

MANUAL GUIDE I (CNC MEM/USER) EDIT 10:53:47

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	O
X	0.000	G00	S	0	N 00000000
Y	44.090	X	SOV	100%	T 0 M 0
Z	55.615	Y	0%		D 0 H 0
A	0.000	Z	FEED	MM/MIN	S 0 M5
B	0.000	A	F	0	F 0
C	0.000	B	FOV	100%	G00 17 40 54
		C			G 44 90

**TOOL OFFSET :**

TOOL OFFSET: TOOL DATA SIM DATA

NO.	TOOL	SET	NOS AN
01	DRILL	1	120.000
02	F END	1	
03	TAP	1	
04	F MILL	0	
05	BORING	0	
06	-		
07	-		
08	-		

KEY IN NUMERALS.

O:PROGRAM NUMBER

ACTPOS TAB → INIT NO.SRH CLOSE

Settings for a milling tool

### Setting angle, corner angle

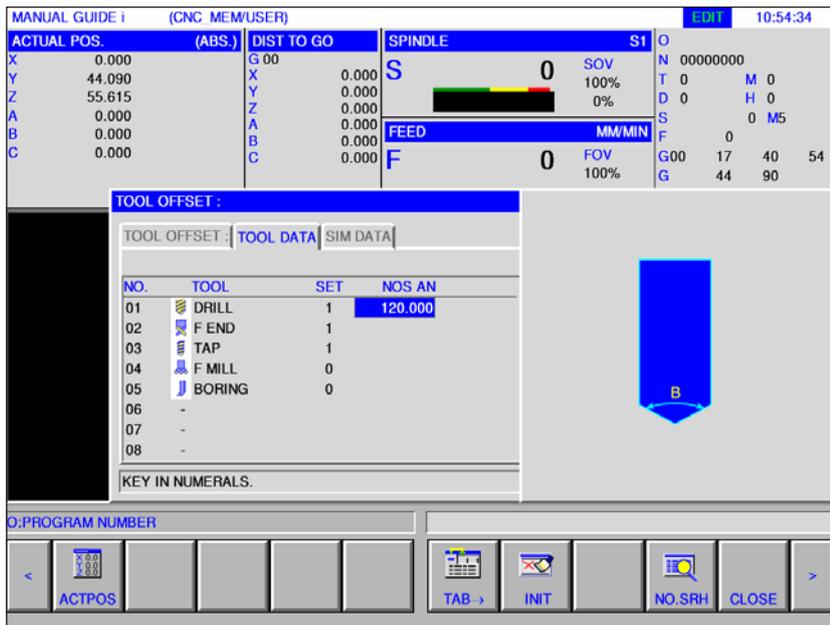
- Mark the tool data with the cursor and an instruction window pops up. Enter the tool data here.

When entering in degrees, three integers and one decimal place are allowed.

If you are entering in inches or millimetres, an eight-digit number is allowed. If numbers with several decimal places are entered, they are rounded to 1 decimal place.

The descriptions and numbers of the elements are dependent on the tool type. Details are in the table below. Tools that do not need tool data settings are not included. No tool data is required to execute milling cycles.

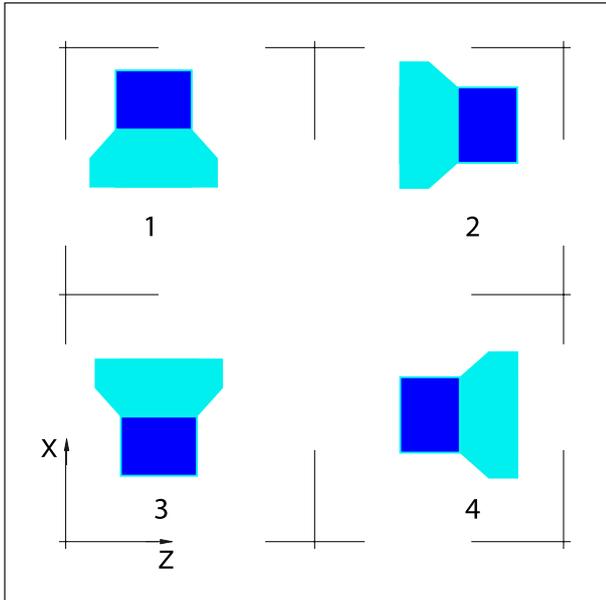
Milling cycles can therefore be executed, although no tool data was set.



### Tool form data for milling

Tool type	Borer	Countersink	
Data 1	Tip angle <sup>1)</sup>	Mill diameter	

<sup>1)</sup> the cycle can be machined without a setting.



*Cutting position of drilling and milling tools*

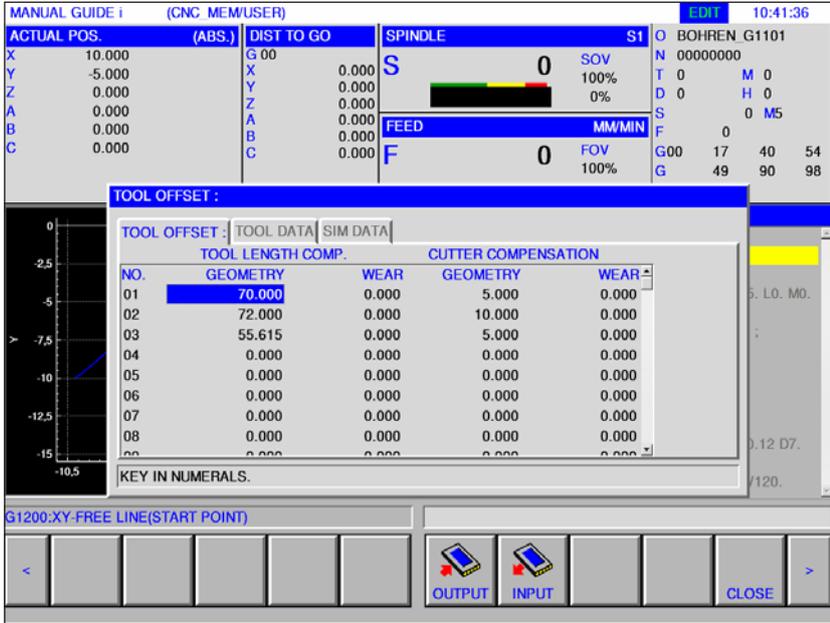
**Cutting position of milling and drilling tools**

**Milling tools**

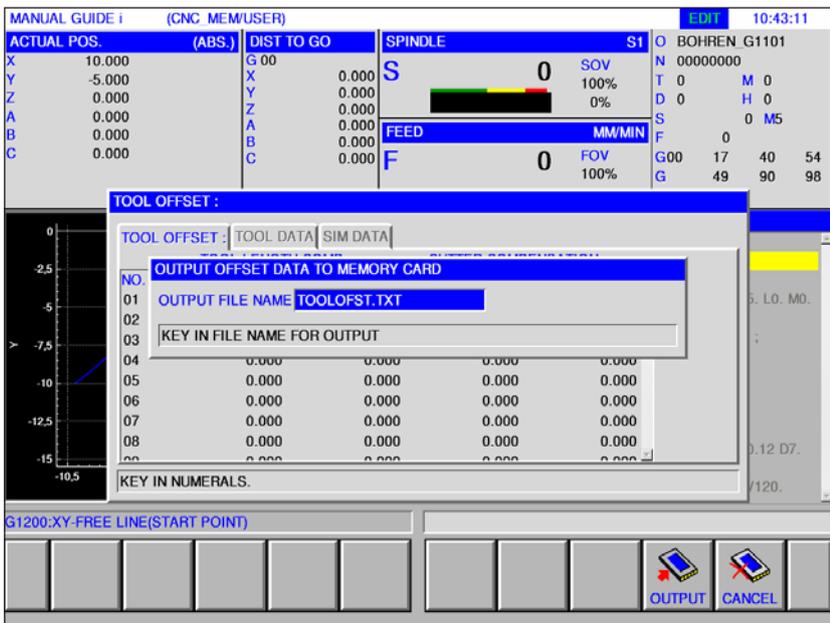
- Countersink
- Face milling cutter
- High speed face milling cutter
- Tap
- Surface milling cutter

## Tool correction and tool data input and output

The path to save and read the data is set in EMConfig under the "Directory exchange" heading.



- Press the data output softkey

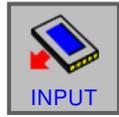


The default save for tool correction data is in the file TOOLOFST.TXT.

You can change the file name, subject to obeying the following restrictions:

A maximum of 8 characters and no special characters should be used.

The tool data is filed in the file TOOLDB.DAT.



- Press the data input softkey

MANUAL GUIDE I (CNC MEM/USER) EDIT 10:45:20

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	O
X	10.000	G 00	S	0	BOHREN G1101
Y	-5.000	X 0.000		SOV	N 00000000
Z	0.000	Y 0.000		100%	T 0 M 0
A	0.000	Z 0.000		0%	D 0 H 0
B	0.000	A 0.000			S 0 M5
C	0.000	B 0.000			F 0
		C 0.000			G00 17 40 54
					G 49 90 98

FEED MM/MIN F 0 FOV 100%

TOOL OFFSET :

INPUT OFFSET DATA FROM MEMORY CARD

NO.	FILE NAME	SIZE	DATE
0302	TOOLDB.DAT	349	2015.05.12
0303	TOOLOFS1.TXT	6833	2015.05.12
0304	TOOLOFS9.TXT	6833	2015.05.12
0305	TOOLOFST.TXT	6833	2015.05.12

KEY SELECT FILE NUMBER.AND SELECT SOFT KEY.

G1200:XY-FREE LINE(START POINT)

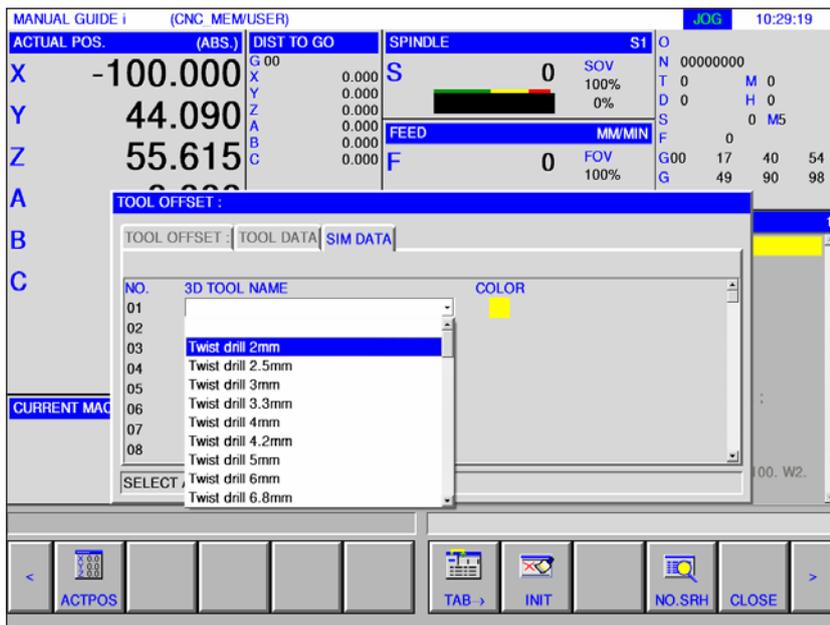
INPUT CLOSE

Select the desired file and open with "INPUT".

## Simulation data

### 3D tool

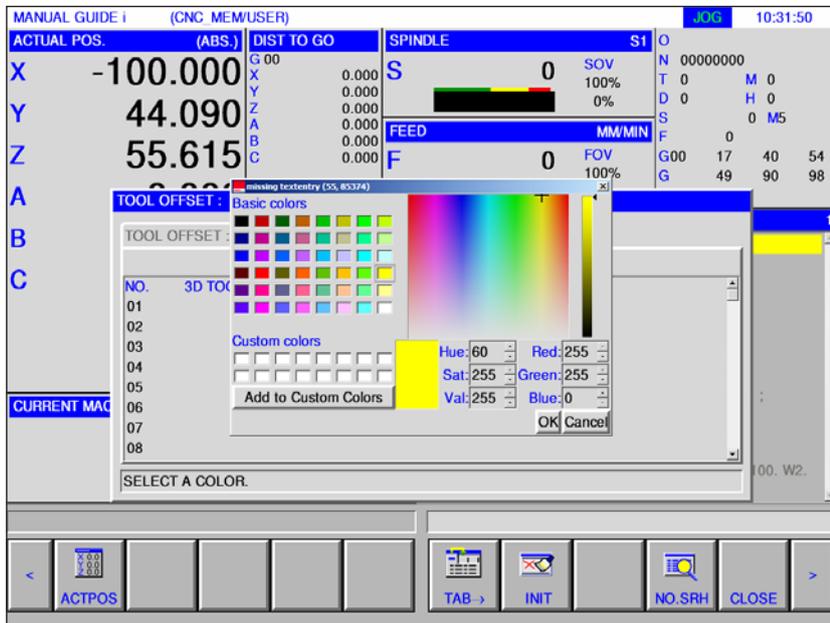
3D tools from the tool manager can be imported into the tool list. An independent color assignment for the individual tools can be made.



- 1 Press softkey.
- 2 Select simulation data.
- 3 Double-click the 3D tool to activate the tool selection (drop down menu). Press the space bar to scroll in the tool selection.
- 4 To deactivate a tool, the blank line (at the very top) must be selected in the drop down menu.

## Color select

Colors can be assigned to tools in order to display them better in the simulation.



- 1 Double-click the color selection (alternatively use the space bar) to open the color selection window.
- 2 Predefined colors are stored as basic colors, user defined colors as custom colors
  - Create Custom colors:  
Move the mouse in the colored area to the required color and click to select. Alternatively the values for R,G,B can be entered manually.
  - By clicking "Add to Custom Colors" a new color is added.
- 3 To deselect a color, select black.
- 4 Confirm with OK or quit with Cancel.

### Instructions:

If no color is selected, it is taken over from the 3D tool manager. Otherwise a selected color has priority.



## Measuring tool manually

### Scratch method

The workpiece must be clamped in the workspace such that the measuring point on the workpiece with the tool housing reference point can be reached with all measuring tools.

The tool housing reference point for the EMCO Concept Mill 105 is on the reference tool, which must be clamped in position in advance.

If a mechanical meter is used instead of the reference tool, this must be mounted on the machine table.

1 Call "JOG" mode.

2 If you do not use a meter, then place a thin sheet of paper between the workpiece and the milling spindle.

Approach the stationary spindle on the workpiece with the tool housing reference point. Reduce the feed to 1%, so that the sheet of paper you inserted is still just movable.

3 If you are using a meter, approach this with the tool housing reference point.

4 Press the "ACTPOS" softkey to display the relative coordinates.

5 Press the "PRESET" softkey and enter the value 0 in the Z axis.



ACTUAL POS. (RELATIVE)	
X	645.000
Y	45.000
Z	595.000
C	0.000



PRESET RELATIVE POSI.	
X-AXIS	645.000
Y-AXIS	45.000
Z-AXIS	0.000
C-AXIS	0.000

INPUT PRESET VALUE.



6 Accept changes with the "ALTER" softkey.



7 Tilt the workpiece to be measured.

8 Scratch the tool on the workpiece or the meter.

9 Press softkey.

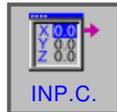


TOOL OFFSET :

TOOL OFFSET :		TOOL DATA	SIM DATA
		TOOL LENGTH COMP.	CUTTER COMPENSATION
NO.	GEOMETRY	WEAR	GEOMETRY WEAR
01	0.000	0.000	0.000 0.000
02	0.000	0.000	0.000 0.000
03	0.000	0.000	0.000 0.000
04	0.000	0.000	6.000 0.000
05	0.000	0.000	0.000 0.000
06	0.000	0.000	4.000 0.000
07	0.000	0.000	0.000 0.000
08	0.000	0.000	0.000 0.000

KEY IN NUMERALS.

10 Set the cursor to the correct tool number.



11 Press softkey.

INPUT RELATIVE COORD.

X-AXIS	645.000
Y-AXIS	45.000
Z-AXIS	43.767
C-AXIS	0.000

12 Set the cursor to Z-AXIS.



13 Press softkey. The tool is now measured in Z.

TOOL OFFSET :

TOOL OFFSET :		TOOL DATA	SIM DATA
		TOOL LENGTH COMP.	CUTTER COMPENSATION
NO.	GEOMETRY	WEAR	GEOMETRY WEAR
01	0.000	0.000	0.000 0.000
02	0.000	0.000	0.000 0.000
03	43.767	0.000	5.000 0.000
04	0.000	0.000	6.000 0.000
05	0.000	0.000	0.000 0.000
06	0.000	0.000	4.000 0.000
07	0.000	0.000	0.000 0.000
08	0.000	0.000	0.000 0.000

14 Enter the tool radius geometry under "Cutter compensation".

# G: Program sequence



**Instructions:**

During the program sequence the tool data for the tools used may not be changed.

## Preconditions

### Set zero points

The zero points used must be measured and entered.

### Tools

The tools used must be measured and entered. The tools must be in the corresponding positions (t) in the tool changer.

### Reference point

The reference point must be approached in all axes.

### Machine

The machine must be ready for operation. The workpiece must be securely clamped. Loose parts (spanners, etc.) must be removed from the work area to avoid collisions. The machine doors must be closed at the start of the program.

### Alarms

There must be no alarms pending.



## NC start

For the NC program run to be started, a Fanuc program must be open. The file name of the currently open program is shown in the centre of the simulation window.



## NC reset

Use this button to stop the NC program run and reset to the start conditions.



## NC stop

This button stops the NC program run. The simulation can be continued with this "Cycle start" button.

## Program start, program stop

- Open a program for execution.



- Select "AUTO" mode.



- Open program.

- This softkey opens the selected program.

- Press the "Cycle start" button.



- Stop program with "Cycle stop", continue with "Cycle start".



- Cancel program with "NC reset".

## Repositioning



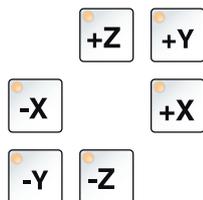
If a program interruption occurs in automatic mode, e.g. after a tool break, then the tool can be moved away from the contour in manual mode. To avoid later collisions the axes must be moved to a safe position.

The coordinates of the interruption position will be saved.

The travel difference between the axes that has been brought about in manual mode is shown in the current values window. This travel difference is called "Repos shift".

## Continue program execution

- Select REPOS mode. Thus the tool can again be brought to the contour of the workpiece.



- Select each axis to be moved one after the other and move to the cancellation position.

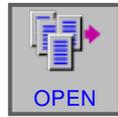


- Continue machining in automatic mode again with "Cycle start".

## Block scan



- Select "Edit" mode.



- Open program.

Select the appropriate program line from which the program is to start.



- Select "AUTO" mode.



- Press NC start button.

The following message appears:  
"Start in the middle of the Start/Ref program"

Press the NC start button again to confirm.



### Caution:

- All command that are before the selected line are ignored.
- This relates these to the chosen tools, zero point shifts, etc.

# H: Alarms and Messages

## 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 / 100 / 105 / 125 / 155 Concept MILL 55 / 105 / 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.

#### 6008:MISSING CAN SUBSCRIBER

Check fuses or EMCO customer service.  
Contact EMCO Service.

#### 6009: SAFETY CIRCUIT FAULT

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

#### 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.

#### 6019: VICE TIME EXCEED

The electric vice has not reached a stop position within 30 seconds.

The control or the clamping device board are defective, the vice is stuck. Adjust the proximity switches of the stop position.

#### 6020: VICE FAILURE

When the electric vice is closed, the signal "clamping device clamped" of the clamping device board has failed.

The control, the clamping device board or the wiring are defective.

K 2017-07

**6022: CLAMPING DEVICE BOARD DEFECTIVE**

The signal "clamping device clamped" is constantly released, although no command has been given.

Replace the board.

**6024: MACHINE DOOR OPEN**

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

**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.

**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 sticks (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 tothing 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.

**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: 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 sticks 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 sticks 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 auxiliary drives will be switched off.

Remedy: service automatic dividing device  
lock the dividing device

**6074: DIVIDING TIME EXCEEDED**

Cause: dividing device sticks mechanically  
locking switch defective  
cabling defective  
fuses defective  
insufficient compressed-air supply.

A running program will be aborted.

The auxiliary drives will be switched off.

Remedy: Check for collision, check the compressed-air supply or contact the EMCO service.

**6075: M27 AT RUNNING MAIN SPINDLE**

Cause: Programming mistake in NC program.

A running program will be aborted.

The auxiliary drives will be switched off.

Remedy: Correct NC program

**6110: 5TH AXIS NOT CONNECTED**

Cause: 4th/5th axis was selected in EMConfig, but not connected electrically.

Remedy: Connect 4th/5th axis or deselect in EM-Config.

**6111: 5TH AXIS CONNECTED**

Cause: 4th/5th axis was deselected in EMConfig, but is electrically connected.

Remedy: Remove 4th/5th axis from machine or select in EMConfig.

**6112: MOTOR PROTECTION RELEASED**

Cause: A motor protection switch has triggered. A possibly active NC program is stopped immediately.

Remedy: Check the device connected to the motor protection switch and switch on again. If recurring, contact EMCO service department.

**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.

**7001: NO M6 PROGRAMMED**

For an automatic tool change you also have to program a M6 after the T word.

**7007: FEED STOP!**

The axes have been stopped by the robotics interface (robotics entry FEEDHOLD).

**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

**7023: WAITING TIME MAIN DRIVE!**

The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly opened/ closed (under 20 seconds).

**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.

**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 HEAD NOT LOCKED!**

Either the dividing head is in an undefined position after the machine has been switched on, or the locking signal after a dividing process is missing.

Initiate the dividing process, check, respectively adjust the proximity switch for locking.

**7054: VICE OPEN**

Cause: the workpiece is not clamped

When switching on the main spindle with M3/M4 alarm 6072 (vice not ready) will be released.

Remedy: Clamp

**7055: OPEN TOOL CLAMPING SYSTEM**

A tool is clamped in the main spindle and the control does not recognize the corresponding T number.

Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

**7056: SETTING DATA INCORRECT**

An invalid tool number is stored in the setting data.

Delete the setting data in the machine directory xxxxx.pls.

**7057: TOOLHOLDER OCCUPIED**

The clamped tool cannot be positioned in the tool turret since the position is occupied.

Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

**7058: RETRACTING THE AXES**

The position of the tool turret arm cannot be clearly defined during the tool change.

Open the machine door, push the tool turret magazine backwards to the stop. Move the milling head in the JOG mode upwards to the Z reference switch and then traverse the reference point.

**7087: MOTOR PROTECTION HYDRAULIC CLAMPING RELEASED!**

Hydraulic motor is defective, stiff, circuit breaker is set incorrectly.

Replace motor or check circuit breaker and replace if necessary.

**7090: ELECTRICAL CABINET OVERRIDE SWITCH ACTIVE**

The cabinet door can only be opened when the key switch is switched on without raising an alarm.

Switch off key switch.

**7107: MOTOR PROTECTION RELEASED**

A motor protection switch has triggered. A possibly running NC program will be completed. A new NC start will be prevented.

Check the device connected to the motor protection switch and switch on again. If recurring, contact EMCO service department.

**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 50 / 55 / 105 / 120 / 125 / 155**  
**Concept TURN 55 / 60 / 105 / 155 /**  
**250 / 260 / 460**  
**Concept MILL 250**  
**EMCOMAT E160**  
**EMCOMAT E200**  
**EMCOMILL C40**  
**EMCOMAT FB-450 / FB-600**

**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.

**6005: K2 OR K3 NOT DE-ENERGIZED**

Turn machine on/off. Defective security board.

**6006 EMERGENCY-OFF RELAY K1 NOT DE-ENERGIZED**

Turn machine on/off. Defective security board.

**6007 SAFETY CIRCUIT FAULT****6008: MISSING CAN SUBSCRIBER**

The PLC-CAN board is not identified by the control.  
 Check the interface cable and the power supply of the CAN board.

**6009: SAFETY CIRCUIT FAULT****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.

**6011: DRIVE Z-AXIS NOT READY**

see 6010.

**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.

**6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING****6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING**

In the tool turret that can be coupled, the position of the coupling and uncoupling magnet is monitored by means of two proximity switches. It has to be made sure that the coupling is in the rear stop position so that the tool turret can get to the next tool position. Equally, during operation with driven tools the coupling has to be safe in the front stop position.

Check and adjust the cables, the magnet and the stop position proximity switches.

**6018: AS SIGNALS, K4 OR K5 NOT DE-ENERGIZED**

Turn machine on/off. Defective security board.

**6019: POWER SUPPLY MODULE NOT READY**

Turn machine on/off. Power supply module, defective axis controller 6020 AWZ drive failure turn machine on/off, defective axis controller.

**6020: DRIVEN TOOL DRIVE MALFUNCTION**

The driven tool power supply defective or the driven tool drive is too hot, fuse defective, mains overvoltage or undervoltage. A running CNC program will be aborted, the auxiliary drives will be stopped. Check the fuses or contact EMCO service.

**6021: COLLET TIME OUT**

During closing of the clamping device the pressure switch has not reacted within one second.

**6022: CLAMPING DEVICE BOARD DEFECTIVE**

The signal "clamping device clamped" is constantly released, even though no command has been given. Replace the board.

**6023: COLLET PRESSURE MONITORING**

The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

**6024: MACHINE DOOR OPEN**

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.

**6026: MOTOR PROTECTION COOLANT PUMP RELEASED**

**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.

**6029: TAILSTOCK QUILL TIME EXCEED**

The tailstock quill does not reach a final position within 10 seconds.

Adjust the control and the stop position proximity switches, or the tailstock quill is stuck.

**6030: NO PART CLAMPED**

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

**6031: QUILL FAILURE**

**6032: TOOL CHANGE TIMEOUT**

see alarm 6041.

**6033: TOOL TURRET SYNC ERROR**

Hardware defective. Contact EMCO service.

**6037: CHUCK TIMEOUT**

The pressure switch does not react within one second when the clamping device is closed.

**6039: CHUCK PRESSURE FAILURE**

The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

**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 sticks (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.

**6044: BRAKING RESISTANCE - MAIN DRIVE OVERLOADED**

Reduce number of speed changes in the program.

**6045: TOOL TURRET SYNC MISSING**

Hardware defective.  
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.

**6067: NO COMPRESSED AIR**

Turn the compressed air on, check the setting of the pressure switch.

**6068: MAINDRIVE OVERTEMPERATURE****6070: LIMIT SWITCH TAILSTOCK SLEEVE ACTIVE**

Cause: The axis arrived in the tailstock sleeve.  
Remedy: Drive the travel off the tailstock sleeve.

**6071: LIMIT SWITCH X AXIS ACTIVE**

Cause: The axis arrived to the end switch.  
Remedy: Drive the axis off the end switch again.

**6072: LIMIT SWITCH Z AXIS ACTIVE**

see 6071

**6073: CHUCK GUARD OPEN**

Cause: The chuck guard is open.  
Remedy: Close the chuck guard.

**6074: NO FEEDBACK FROM USB-PLC**

Turn machine on/off. Check cabling, defective USB board.

**6075: AXIS LIMIT SWITCH TRIGGERED**

see 6071

**6076: DRIVE Y-AXIS NOT READY**

see 6010

**6077 VICE NOT READY**

Cause: Loss of pressure in clamping system.

Remedy: Check pressurised air and air ducts.

**6078 MOTOR PROTECTION TOOL MAGAZINE RELEASED**

Cause: Swing intervals are too short.

Remedy: Raise swing intervals.

**6079 MOTOR PROTECTION TOOL CHANGER RELEASED**

see 6068

**6080 PRESSURE SWITCH FOR TANI MISSING**

Cause: The pressure switch fails to active when the clamping closes. No pressurised air or mechanical problem.

Remedy: Check pressurised air.

**6081 CLAMPING FOR TANI NOT OPEN**

see 6080

**6082 FAULT AS/SIGNAL**

Cause: Active Safety-Signal X/Y-controller is faulty.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

**6083 FAULT AS/SIGNAL**

Cause: Active Safety-Signal main spindle/Z-controller is faulty.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

**6084 FAULT AS/SIGNAL UE-MODUL**

Cause: Active Safety-Signal Uncontrolled power supply module is faulty.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

**6085 N=0 RELAY NOT DE-ENERGIZED**

Cause: Rotation zero relay did not drop.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO (replace relay).

**6086 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC**

Cause: ACC-PLC and USBSPS receive different door status reports.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**6087 DRIVE A-AXIS NOT READY**

see 6010

**6088 PROTECT SWITCH DOOR CONTROL UNIT RELEASED**

Cause: Door drive overload.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace motor, drive).

**6089 DRIVE B-AXIS NOT READY**

see 6010

**6090 CHIP CONVEYOR CONTACTOR NOT DE-ENERGIZED**

Cause: Chip conveyor guard not down.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace guard).

**6091 AUTOMATIC DOOR CONTACTOR NOT DE-ENERGIZED**

Cause: Automatic door guard not down.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace guard).

**6092 EMERGENCY-OFF EXTERNAL**

**6093 FAULT AS/SIGNAL A-AXIS**

Cause: Active Safety-Signal A control element faulty.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO.

**6095 OVERHEATING IN THE SWITCHGEAR CABINET**

Cause: Temperature monitoring responded.

Remedy: Check switchgear cabinet filter and fan, raise triggering temperature, switch machine on and off.

**6096 SWITCHGEAR CABINET DOOR OPEN**

Cause: Switchgear cabinet door opened without key switch release.

Remedy: Close switchgear cabinet door, switch machine off and on.

**6097 EMERGENCY-OFF TEST REQUIRED**

Cause: Functional test of Emergency off.  
 Remedy: Press EMERGENCY-OFF button on the control panel and unlock again. Press Rest-button to acknowledge the emergency stop state.

**6098 FLOAT SWITCH HYDRAULIC MISSING**

Effect: Auxiliary units are switched off.  
 Meaning: The hydraulic float switch has released.  
 Remedy: Refill hydraulic oil.

**6099 PROX. SWITCH SPINDLE-BRAKE 1 MISSING**

Effect: Feed stop, Read in lock  
 Meaning: M10 Spindle brake on → Inductive proximity switch stays 0.  
 M11 Spindle brake off → Inductive proximity switch stays 1.  
 Remedy: Check inductive proximity switch, check magnetiv valve spindle brake.

**6100 LOW PRESSURE QUILL**

Effect: Auxiliary units are switched off.  
 Meaning: At the moment of a spindle start command the tailstock pressure was not built up or the pressure dropped while spindle run.  
 Remedy: Check clamping device pressure and pressure switch.  
 Check program.

**6101 QUILL -B3 OR -B4 MISSING**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for the movement of the quill was activated, but the switches -B3 and -B4 does not alter its status.  
 Remedy: Check switches, magnetic valves.

**6102 QUILL POSITION ALARM (PART MOVED?)**

Effect: Feed stop, Read in lock  
 Meaning: The tailstock target position was overtravelled in AUTOMATIC mode.  
 Remedy: Check tailstock target position, check technology (clamping device pressure higher, tailstock pressure lower)

**6103 QUILL NO BACKPOSITION**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for the tailstock was activated, but the switch for "tailstock back" stays 0.  
 Remedy: Check magnetic valve, check switch.

**6104 LOW PRESSURE CLAMPING EQUIPMENT 1**

Effect: Auxiliary units are switched off.  
 Meaning: At the moment of a spindle start command the clamping pressure was not built up or the clamping pressure dropped while spindle run.  
 Remedy: Check clamping device pressure and pressure switch.  
 Check program.

**6105 CLAMPING EQUIPMENT 1 NOT OPEN**

Effect: Feed stop, Read in lock  
 Meaning: The analogous proximity switch for clamping device 1 was not actuated.  
 Remedy: Adjust inductive proximity switch (see "Machine Description - chapter "C Programming and operation").

**6106 CLAMPING EQUIPMENT 1 NOT CLOSED**

Effect: Feed stop, Read in lock  
 Meaning: The pressure switch for "clamping device closed" does not switch.  
 Remedy: Check pressure switch

**6107 LIMIT SWITCH CLAMPING EQUIPMENT 1**

Effect: Auxiliary units are switched off.  
 Remedy: Adjust clamping device (do not clamp in end position of the clamping system - see "Machine Description, chapter C Programming and Operation").

**6108 PARTS CATCHER FORWARD MISSING**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for "collection tray forward/backward" was activated, but the switch for "collection tray forward/backward" does not alter its status.  
 Remedy: Check switches, magnetic valves.

**6109 PARTS CATCHER ROTATE IN MISSING**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for "collection tray in/out" was activated, but the switch for "collection tray in/out" does not alter its status.  
 Remedy: Check switch, magnetic valve.

**6900 USBPLC not available**

Cause: USB communication with the safety board could not be established.  
 Remedy: Switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6901 Error emergency-off relay USBPLC**

Cause: USBPLC EMERGENCY-OFF relay error.  
Remedy: Switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6902 Error standstill monitoring X**

Cause: Unauthorized movement of the X axis in the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6903 Error standstill monitoring Z**

Cause: Unauthorized movement of the Z axis in the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6904 Error alive circuit PLC**

Cause: Error in the connection (Watchdog) of the safety board with the PLC.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6906 Error overspeed spindle**

Cause: The main spindle speed exceeds the maximum permissible value for the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6907 Error enable pulses ER-module**

Cause: ACC-PLC did not shutdown the input/negative feeder-module.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6908 Error standstill monitoring main drive**

Cause: Unexpected warm up of the main spindle in the operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6909 Error main drive enable without spindle start**

Cause: The release of the control unit of the main spindle was given by the ACC-PLC without the spindle-start key being pressed.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6910 Error standstill monitoring Y**

Cause: Unauthorized movement of the Y axis in the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6911 Error standstill axes**

Cause: Unauthorized movement of the axis in the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6912 Error overspeed axis**

Cause: The feed of the axes exceeds the maximum permissible value for the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6913 Error overspeed X**

Cause: The feed of the X axis exceeds the maximum permissible value for the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6914 Error overspeed Y**

Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.  
Remedy: Delete the alarm with the RESET button and switch the machine off and on again.  
Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6915 Error overspeed Z**

Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.

Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

**6916 ERROR: X-INDUCTIVE PROXIMITY SWITCH DEFECT**

Cause: No signal is delivered by X axis Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**6917 ERROR: Y-INDUCTIVE PROXIMITY SWITCH DEFECT**

Cause: No signal is delivered by Y axis Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**6918 ERROR: Z-INDUCTIVE PROXIMITY SWITCH DEFECT**

Cause: No signal is delivered by Z axis Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**6919 ERROR: SPINDLE-INDUCTIVE PROXIMITY SWITCH DEFECT**

Cause: No signal is delivered by main spindle Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**6920 INVERSION OF X-DIRECTION TOO LONG "1"**

Cause: The change in direction of X axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

**6921 INVERSION OF Y-DIRECTION TOO LONG "1"**

Cause: The change in direction of Y axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

**6922 INVERSION OF Z-DIRECTION TOO LONG "1"**

Cause: The change in direction of Z axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

**6923 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC**

Cause: ACC-PLC and USBSPS receive different door status reports.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**6924 ERROR ENABLE PULSES MAIN DRIVE**

Cause: The pulse release on the main spindle control element was interrupted by the USBSPS, as the PLC did not shut it down in a timely fashion.

Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

**6925 GRID PROTECTION ERROR!**

Cause: Grid protection does not drop out in current operating state, or does not engage.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

**6926 MOTOR PROTECTION ERROR!**

Cause: Motor protection drops out in current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

**6927 EMERGENCY OFF ACTIVE ERROR!**

Cause: Emergency off button was pressed.

Remedy: Restart the machine.

**6928 TOOL CHANGER SHUTDOWN MONITORING ERROR**

Cause: Unauthorised tool changer movement in the current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

**6929 MACHINE DOOR CLOSING/LOCKING ERROR**

Cause: State of the door lock not plausible or door closure unserviceable.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

**6930 BEROS MAIN SPINDLE PLAUSIBILITY ERROR**

Cause: Beros main spindle signal different.  
 Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

**6931 MAIN DRIVE QUICK STOP FUNCTION PLAUSIBILITY ERROR**

Cause: Main drive actuator does not confirm the quick stop function in the current operating state.  
 Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

**6999 USB-EXTENSION FOR ROBOTIK NOT AVAILABLE**

Cause: The USB extension for robotics cannot be addressed by ACC.  
 Remedy: Contact EMCO.

**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.

**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.

**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.

**7022: COLLECTION DEVICE MONITORING**

Time exceed of the swivelling movement. Check the pneumatics, respectively whether the mechanical system is jammed (possibly a workpiece is jammed).

**7023: ADJUST PRESSURE SWITCH !**

During opening and closing of the clamping device the pressure switch has to turn off and on once.  
 Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

**7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !**

When the clamping device is open and the position stop control is active, the respective proximity switch has to feed back that the clamping device is "Open".  
 Check and adjust the clamping device proximity switch, check the cables.

**7025 WAITING TIME MAIN DRIVE !**

The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly opened/ closed (under 20 seconds).

**7026 PROTECTION MAIN MOTOR FAN RELEASED!**

**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.

**7041: GEARBOX COVER OPEN**

The main spindle cannot be switched on and NC start cannot be activated. Close the gearbox cover in order to start a CNC 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.

**7060 RETRACT SLEEVE LIMIT SWITCH !**

The axis arrived in the tailstock sleeve. Drive the travel off the tailstock sleeve.

**7061 RETRACT X AXIS LIMIT SWITCH !**

The axis arrived to the end switch. Drive the axis off the end switch again.

**7062 RETRACT Z AXIS LIMIT SWITCH !**

see 7061

**7063 OIL LEVEL CENTRAL LUBRICATION !**

Low oil level in central lubrication. Refill oil as per maintenance instructions to the machine.

**7064 CHUCK GUARD OPEN !**

The chuck guard is open. Close the chuck guard.

**7065 MOTOR PROTECTION COOLANT PUMP RELEASED !**

Overheated coolant pump. Check the coolant pump for ease of motion and presence of dirt. Ensure sufficient amount of coolant fluid in the coolant facility.

**7066 CONFIRM TOOL !**

To confirm the tool change, press T after the change has been completed.

**7067 MANUAL OPERATING MODE**

The Special Operation key switch is in the Set position (manual).

**7068 X AXIS HANDWHEEL ACTIVE**

The safety wheel is locked for manual travel movement. The safety wheel locking is monitored by contactless switches. With the manual wheel locked, the axis feed cannot be switched on. For automatic processing of a program, the manual wheel must be released again.

**7069 Y AXIS HANDWHEEL ACTIVE**

see 7068

**7070 Z AXIS HANDWHEEL ACTIVE**

see 7068

**7071 VERTICAL TOOL CHANGE**

The sheath for manual clamping of the tool holder is monitored by a switch. The switch reports a unaccepted socket wrench or a sheath which was left open. Remove the socket wrench after clamping the tool and close the sheath.

**7072 HORIZONTAL TOOL CHANGE**

The turning knob for manual tool clamping on the horizontal spindle is monitored by a switch. The switch reports a tightened turning knob. The spindle gets locked. Release the turning knob after clamping the tool.

**7073 RETRACT Y AXIS LIMIT SWITCH !**

see 7061

**7074 CHANGE TOOL**

Clamp programmed tool.

**7076: SWIVEL UNIT VOR MILLING HEAD UNLOCKED**

The milling head is not fully swung. Fix the milling head mechanically (the end switch must be pushed).

**7077: ADJUST TOOL TURRET**

No valid machine data for tool change are available. Contact EMCO.

**7078: POCKET NOT IN HOME POSITION**

Cancel during tool change. Swing back tool recessed in setup operation.

**7079: TOOL ARM NOT IN HOME POSITION**

see 7079

**7080: INCORRECT TOOL CLAMPED !**

The tool cone lies beyond tolerance. The clamped tool is twisted by 180°. Bero tool clamping is displaced. Check the tool and clamp it again. If this problem occurs with more tools, contact EMCO.

**7082: MOTOR PROTECTION CHIP CONVEY-OR RELEASED**

The scrap belt is overloaded. Check the conveyor belt for ease of motion and remove jammed scrap.

**7083: MAGAZINE IS ACTIVE !**

A tool has been removed from the non-chaotic tool administration from the main spindle. Fill the tool drum.

**7084: VICE OPEN !**

The vice is not clamped. Clamp the vice.

**7085 ROUNDAXIS A MOVE TO 0 DEGRE !**

Cause: The MOC only shuts down if the A Round axis is at 0°.

When 4.5. is present, a round axis must be made each time before the machine is switched off.

Remedy: Move round axis to 0°.

**7088 SWITCHGEAR CABINET OVERHEATING**

Cause: Temperature monitoring responded.

Remedy: Check switchgear cabinet filter and fan, raise trigger temperature.

**7089 SWITCHGEAR CABINET DOOR OPEN**

Cause: Switchgear cabinet door open.

Remedy: Close switchgear cabinet door.

**7091 WAIT FOR USB-I2C SPS**

Cause: Communication with the USB-I2C PLC could not be established.

Remedy: If the message does not self-extinguish, switch off and on the machine. Please contact EMCO service when the message occurs permanently even after power off.

**7092 TEST STOP ACTIVE**

Cause: Safety test for monitoring the safety functions is active.

Remedy: Wait until the safety test is finished.

**7093 SET REFERENCE POINT**

Cause: The reference point acknowledge mode was activated by the operator.

**7094 SET X-REFERENCE POINT**

Cause: The reference value of the X axis was adopted to the acc.esd file.

**7095 SET Y-REFERENCE POINT**

Cause: The reference value of the Y axis was adopted to the acc.esd file.

**7096 SET Z-REFERENCE POINT**

Cause: The reference value of the Z axis was adopted to the acc.esd file.

**7097 FEED OVERRIDE 0**

Cause: The override switch (feed control) was set by the operator on 0%

**7098 SPINDLE-BRAKE 1 ACTIVE**

Effect: spindle stop.

**7099QUILL DRIVES FORWARD**

Effect: Feed stop, Read in lock

Meaning: M21 was programmed → pressure switch "tailstock in front position" is not yet 1.

Remedy: Disappears with pressure switch front position.

**7100 QUILL DRIVES BACKWARD**

Effect: Feed stop, Read in lock

Meaning: M20 was programmed → limit switch "tailstock back" is not yet 1

Remedy: Disappears with limit switch "backposition".

**7101 REFERENCE POINT TOOL TURRET MISSING**

Effect: Feed stop, Read in lock

Meaning: At NC start the tool turret is not referenced.

Remedy: Reference tool turret in JOG mode with tool turret key.

**7102 TOOL TURRET IN MOTION**

Effect: Tool turret swivels to the programmed position.

**7103 LIMIT-SWITCH CLAMPING EQUIPMENT 1**

Effect: Locking of NC start and main drive start, spindle stop S1

Meaning: The pick-up for the analouge value detects the clamping position as end position.

Remedy: Change clamping range of the clamping device (see "Machine Description - chapter C Programming and Operation").

**7104 QUILL IN INTERMEDIATE POSITION**

Effect: Feed hold / read in stop.

**7105 SET AWZ REFERENCE POINT**

Effect: The reference value of the AWZ motor was transferred to the acc.msds file.

**7900 INITIALIZE EMERGENCY OFF!**

Cause: The emergency off button must be initialized.

Remedy: Press and then release emergency off button.

**7901 INITIALIZE MACHINE DOORS!**

Cause: The machine doors must be initialized.

Remedy: Open the machine doors and close them again.

**7906 SET A-REFERENCE POINT**

Cause: The reference value of the A axis was adopted to the acc.esd file.

## Inputunit alarms 1700 - 1899

These alarms and messages are raised by the control keyboard.

### **1701 Error in RS232**

Cause: Serial port settings are invalid or the connection to the serial keyboard were interrupted.

Remedy: Check the settings of the serial interface and/or turn keyboard off/on and check the control cable connection.

### **1703 Ext. keyboard not available**

Cause: Connection with the external keyboard can not be made.

Remedy: Check the settings of the external keyboard and/or check the cable connection.

### **1704 Ext. keyboard: checksum error**

Cause: Error in the transmission.

Remedy: The connection to the keyboard is automatically restored. If this fails, turn off or on the keyboard.

### **1705 Ext. keyboard: general error**

Cause: The attached keyboard reported an error.

Remedy: Plug the keyboard off and on again. Contact EMCO Customer Service if the error occurs on several occasions.

### **1706 General USB error**

Cause: Error in the USB communication.

Remedy: Plug the keyboard off and on again. Contact EMCO Customer Service if the error occurs on several occasions

### **1707 Ext. Keyboard: no LEDs**

Cause: Fehlerhaftes LED-Kommando wurde an die Tastatur gesandt.

Remedy: EMCO-Service kontaktieren.

### **1708 Ext. Keyboard: unknown command**

Cause: Unknown command was sent to the keyboard.

Remedy: Contact EMCO Customer Service

### **1710 Installation of Easy2control is damaged!**

Cause: Incorrect installation of Easy2control

Remedy: Reinstall software and/or contact EMCO Customer Service

### **1711 Initialization of Easy2Control failed!**

Cause: Configuration file onscreen.ini for Easy2control is missing.

Remedy: Reinstall software and/or contact EMCO Customer Service.

### **1712 USB-Dongle for Easy2control could not be found!**

Cause: USB-Dongle for Easy2control is not connected. Easy2control is displayed but can not be operated.

Remedy: Connect USB-Dongle for Easy2control.

### **1801 Keytable not found!**

Cause: The file with the keytable couldn't be found.

Remedy: Reinstall software and/or contact EMCO Customer Service.

### **1802 Connection to keyboard lost**

Cause: Connection to the serial keyboard was interrupted.

Remedy: Turn keyboard off/on and check the cable connection.



## Axis Controller Alarms

### 8000 - 9000, 22000 - 23000, 200000 - 300000

#### 8000 Fatal Error AC

##### 8100 Fatal init error AC

Cause: Internal error

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

##### 8101 Fatal init error AC

see 8101.

##### 8102 Fatal init error AC

see 8101.

##### 8103 Fatal init error AC

see 8101.

##### 8104 Fatal system error AC

see 8101.

##### 8105 Fatal init error AC

see 8101.

##### 8106 No PC-COM card found

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

Remedy: Mount board, adjust other address with jumper

##### 8107 PC-COM card not working

see 8106.

##### 8108 Fatal error on PC-COM card

see 8106.

##### 8109 Fatal error on PC-COM card

see 8106.

##### 8110 PC-COM init message missing

Cause: Internal error

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

##### 8111 Wrong configuration of PC-COM

see 8110.

##### 8113 Invalid data (pccom.hex)

see 8110.

##### 8114 Programming error on PC-COM

see 8110.

##### 8115 PC-COM packet acknowledge missing

see 8110.

##### 8116 PC-COM startup error

see 8110.

##### 8117 Fatal init data error (pccom.hex)

see 8110.

##### 8118 Fatal init error AC

see 8110, ev. insufficient RAM memory

##### 8119 PC interrupt no. not valid

Cause: The PC interrupt number can not be used.

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

##### 8120 PC interrupt no. unmaskable

see 8119

##### 8121 Invalid command to PC-COM

Cause: Internal error or defective cable

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

##### 8122 Internal AC mailbox overrun

Cause: Internal error

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

##### 8123 Open error on record file

Cause: Internal error

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

##### 8124 Write error on record file

Cause: Internal error

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

##### 8125 Invalid memory for record buffer

Cause: Insufficient RAM, record time exceeding.

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

##### 8126 AC Interpolation overrun

Cause: Ev. insufficient computer performance.

Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

##### 8127 Insufficient memory

Cause: Insufficient RAM

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

##### 8128 Invalid message to AC

Cause: Internal error

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

##### 8129 Invalid MSD data - axisconfig.

see 8128.

##### 8130 Internal init error AC

see 8128.

##### 8130 Internal init error AC

see 8128.

**8132 Axis accessed by multiple channels**

see 8128.

**8133 Insufficient NC block memory AC**

see 8128.

**8134 Too much center points programmed**

see 8128.

**8135 No centerpoint programmed**

see 8128.

**8136 Circle radius too small**

see 8128.

**8137 Invalid for Helix specified**

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

Remedy: Program correction.

**8140 Maschine (ACIF) not responding**

Cause: Machine off or not connected.

Remedy: Switch on machine or connect.

**8141 Internal PC-COM error**

Cause: Internal error

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

**8142 ACIF Program error**

Cause: Internal error

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

**8143 ACIF packet acknowledge missing**

see 8142.

**8144 ACIF startup error**

see 8142.

**8145 Fatal init data error (acif.hex)**

see 8142.

**8146 Multiple request for axis**

see 8142.

**8147 Invalid PC-COM state (DPRAM)**

see 8142.

**8148 Invalid PC-COM command (CNo)**

see 8142.

**8149 Invalid PC-COM command (Len)**

see 8142.

**8150 Fatal ACIF error**

see 8142.

**8151 AC Init Error (missing RPG file)**

see 8142.

**8152 AC Init Error (RPG file format)**

see 8142.

**8153 FPGA program timeout on ACIF**

see 8142.

**8154 Invalid Command to PC-COM**

see 8142.

**8155 Invalid FPGA packet acknowledge**

see 8142 or hardware error on ACIF board (contact EMCO Service).

**8156 Sync within 1.5 revol. not found**

see 8142 or Bero hardware error (contact EMCO Service).

**8157 Data record done**

see 8142.

**8158 Bero width too large (referencing)**

see 8142 or Bero hardware error (contact EMCO Service).

**8159 Function not implemented**

Bedeutung: In normal operation this function can not be executed

**8160 Axis synchronization lost axis 3..7**

Cause: Axis spins or slide is locked, axis synchronisation was lost

Remedy: Approach reference point

**8161 X-Axis synchronization lost**

Step loss of the step motor. Causes:

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

**8162 Y-Axis synchronization lost**

see 8161

**8163 Z-Axis synchronization lost**

see 8161

**8164 Software limit switch max axis 3..7**

Cause: Axis is at traverse area end

Remedy: Retract axis

**8168 Software limit overtravel axis 3..7**

Cause: Axis is at traverse area end

Remedy: Retract axis

**8172 Communication error to machine**

Cause: Internal error

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

**8173 INC while NC program is running**

Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

**8174 INC not allowed**

Cause: At the moment the axis is in motion.

Remedy: Wait until the axis stops and then traverse the axis.

**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.

**8183 Gear too high**

Cause: The selected gear step is not allowed at the machine.

**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**

see 8175.

**8191 Invalid jog feed unit**

Cause: The machine does not support the rotation feed in the JOG operating mode.

Remedy: Order a software update from EMCO.

**8192 Invalid axis in command**

see 8175.

**8193 Fatal PLC error**

see 8175.

**8194 Thread without length**

Cause: The programmed target coordinates are identical to the starting coordinates.

Remedy: Correct the target coordinates.

**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.

**8212 Rotation axis not allowed**

see 8199.

**8213 Circle and rotation axis can't be interpolated****8214 Thread and rotation axis cant't be interpolated****8215 Invalid state**

see 8199.

**8216 No rotation axis for rotation axis switch**

see 8199.

**8217 Axis type not valid!**

Cause: Switching during the rotary axis operating mode when the spindle is running.

Remedy: Stop the spindle and switch over to the rotary axis operating mode.

**8218 Referencing round axis without selected round axis!**

see 8199.

**8219 Thread not allowed without spindle encoder!**

Cause: Thread cutting, respectively tapping is only possible with spindles with encoders.

**8220 Buffer length exceeded in PC send message!**

see 8199.

**8221 Spindle release although axis is no spindle!**

see 8199.

**8222 New master spindle is not valid**

Cause: The indicated master spindle is not valid when switching over to the master spindle.

Remedy: Correct the spindle number.

**8224 Invalid stop mode**

see 8199.

**8225 Invalid parameter for BC\_MOVE\_TO\_IO!**

Cause: The machine is not configured for touch probes. A traversing movement with rotary axis is not allowed during touch probe operating mode.

Remedy: Remove the rotary axis movement from the traversing movement.

**8226 Rotary axis switch not valid (MSD data)!**

Cause: The indicated spindle does not have a rotary axis.

**8228 Rotary axis switch not allowed while axis move!**

Cause: The rotary axis has moved during switching over to the spindle operating mode.

Remedy: Stop the rotary axis before switching.

**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!**

Cause: Transmit is not possible at this machine.

**8232 Axis configuration (MSD) for TRACYL not valid!**

Cause: Tracyl is not possible at this machine.

**8233 Axis not available while TRANSMIT/TRACYL is active!**

Cause: Programming of the rotary axis is not allowed during Transmit/ Tracyl.

**8234 Axis control grant removed by PLC while axis interpolates!**

Cause: Internal error

Remedy: Delete error with reset and inform EMCO.

**8235 Interpolation invalid while axis control grant is off by PLC!**

see 8234.

**8236 TRANSMIT/TRACYL activated while axis or spindle moves!**

see 8234.

**8237 Motion through pole in TRANSMIT!**

Cause: It is not allowed to move through the coordinates X0 Y0 in Transmit.

Remedy: Alter the traversing movement.

**8238 Speed limit in TRANSMIT exceeded!**

Cause: The traversing movement gets too close to the coordinates X0 Y0. In order to observe the programmed feed rate, the maximum speed of the rotary axis would have to be exceeded.

Remedy: Reduce the feed rate. Set the value of the C-axis feed limitation in WinConfig, machine data settings / general machine data/ to 0.2. Thus, the feed rate will be automatically reduced near the coordinates X0 Y0.

The distance to the center is calculated with the following formula:

for CT155/CT325/CT450:

$$F[\text{mm/min}] * 0.0016 = \text{distance} [\text{mm}]$$

for CT250:

$$F[\text{mm/min}] * 0.00016 = \text{distance} [\text{mm}]$$

This applies for rapid traverse in transmit:

CT155/250/325: 4200 mm/min

CT450: 3,500 mm/min

**8239 DAU exceeded 10V limit!**

Cause: Internal error

Remedy: Start the software again or install it anew. Report the error to EMCO.

**8240 Function not valid during active transformation (TRANSMIT/TRACYL)!**

Cause: The Jog and INC operating mode are not possible during Transmit in X/C and during Tracyl in the rotary axis.

**8241 TRANSMIT not enabled (MSD)!**

Cause: Transmit is not possible at this machine.

**8242 TRACYL not enabled (MSD)!**

Cause: Tracyl is not possible at this machine.

**8243 Round axis invalid during active transformation!**

Cause: It is not allowed to program the rotary axis during Transmit/Tracyl.

**8245 TRACYL radius = 0!**

Cause: When selecting Tracyl, a radius of 0 was used.

Remedy: Correct the radius.

**8246 Offset alignment not valid for this state!**

see 8239.

**8247 Offset alignment: MSD file write protected!****8248 Cyclic supervision failed!**

Cause: The communication with the machine keyboard is interrupted.

Remedy: Start the software again or install it anew. Report the error to EMCO.

**8249 Axis motion check alarm!**

see 8239

**8250 Spindle must be rotation axis !**

see 8239

**8251 Lead for G331/G332 missing !**

Cause: The threading pitch is missing or the starting coordinates are identical to the target coordinates.

Remedy: Program the threading pitch.  
Correct the target coordinates.

**8252 Multiple or no linear axis programmed for G331/G332 !**

Remedy: Program exactly one linear axis.

**8253 Speed value for G331/G332 and G96 missing !**

Cause: No cutting speed has been programmed.

Remedy: Program the cutting speed.

**8254 Value for thread starting point offset not valid!**

Cause: The thread starting point offset is not within the range of 0 to 360°.

Remedy: Correct the thread starting point offset.

**8255 Reference point not in valid software limits!**

Cause: The reference point has been defined outside the software limit switches.

Remedy: Correct the reference points in WinConfig.

**8256 Spindle speed too low while executing G331/G332!**

Cause: During tapping the spindle speed has decreased. Perhaps the incorrect threading pitch was used or the core drilling is not correct.

Remedy: Correct the threading pitch. Adapt the diameter to the core drilling.

**8257 Real Time Module not active or PCI card not found!**

Cause: ACC could not be started correctly or the PCI card in the ACC was not recognized.

Remedy: Report the error to EMCO.

**8258 Error allocating Linux data!**

see 8257.

**8259 Current thread in sequence not valid!**

Cause: One block of a thread in sequence has been programmed without thread G33.

Remedy: Correct the program.

**8260 Change of leading axis within the thread sequence !**

Cause: Occurs if the thread run-out of the longitudinal thread is set so that it's not possible to reach the target point. with the required stopping distance

Remedy: The thread run-out should be at least as large as the pitch. this error also occurs. if the pitch of a thread chain is too large when changing the leading axis.

**8261 Missing thread in sequence !**

Cause: A successive thread has not been programmed for a thread in sequence, the number has to be in accordance with the SETTHREADCOUNT () that has been defined before.

Remedy: Correct the number of threads in the thread in sequence and add a thread.

**8262 Reference marks are not close enough !**

Cause: The settings of the linear scale have been changed or the linear scale is defective.

Remedy: Correct the settings. Contact EMCO.

**8263 Reference marks are too close together!**

see 8262.

**8265 No or wrong axis in axis switch command!**

Cause: Internal error.

Remedy: Please contact the EMCO after-sales service.

**8266 Invalid tool**

Cause: Programmed tool is not set in magazine.

Remedy: Correct tool number and/or load tool in magazine.

**8267 Speed difference to high**

Cause: Die Soll- und Istgeschwindigkeit der Achse weichen zu stark voneinander ab.

Remedy: Run the program again with reduced feed. If this does not remedy the problem, contact EMCO.

**8269 USBSPS and ACC speed values or override are different**

Cause: USBSPS and ACC have different rotations saved.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

**8270 Reference switch defective**

Cause: The reference switch did not switch within the specified range.

Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

**8271 Tool load in locked place not possible**

Cause: There was an attempt to swing a tool into a locked place in the magazine.

Remedy: Choose a free, unlocked place in the magazine and then swing the tool into the magazine.

**8272 Old PLC version, update necessary**

Cause: The PLC version is too old to fully support randomised tool management.

Remedy: Update the PLC.

**8273 Spindle overload**

Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).

Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

**8274 Define tool before loading**

Cause: The tool must be defined in the tool list before it is possible to transfer the tool into the spindle.

Remedy: Create the tool in the tool list, then load.

**8275 Could not read position from absolute encoder**

Cause: The position of an absolute encoder could not be read.

Remedy: Turn off and on the machine. Contact EMCO service if the error occurs multiple times

**8276 Physical axis left the valid travel range**

Cause: An axis with absolute encoder is outside the valid range for travel.

Remedy: Turn off and on the machine. Contact EMCO service if the error occurs multiple times.

**8277 Sinamics drive package error active**

Cause: Error in Sinamics drive.

Remedy: Turn off and on the machine. Contact EMCO service if the error occurs multiple times.

**8278 Control not compatible with ACpn**

Cause: The WinNC control is incompatible with the ACpn machine.

Remedy: Install a WinNC control which is compatible with the ACpn.

**8279 Connection to drive got lost**

Cause: the connection between ACpn and CU320 was lost.

Remedy: Turn off and on the machine. Contact EMCO service if the error occurs multiple times

**8280 Reference point in setting data not equivalent with machine data, please check!**

Cause: The reference point saved in the AC setting data does not match the reference point in the machine data (ACC\_MSD).

Remedy: Measure the reference point of all axes anew and enter into EMConfig.

**8704 Feed override absent, REPOS is not executed**

Cause: The REPOS command is not executed because the feed override is set to 0%.

Remedy: Change the feed override and restart REPOS.

**8705 Tool sorting active**

Cause: The tools will be re-sorted with random tool management to facilitate non-random operation (tool 1 at place 1, tool 2 at place 2, etc.).

Remedy: Wait until sorting has finished. The controller will delete the report independently.

**8706 Check new controller - tool table**

Cause: The controller was changed with random tool management active.

Remedy: Check the tool or place table to clear the alarm.

**8707 Ending with auxiliary drives switched on not possible**

Cause: An attempt was made to end the controller, although the auxiliary drives are still switched on.

Remedy: Switch off the auxiliary drives and then end the controller.

**8710 Initiating communication with drives**

Cause: ACpn connects to the Sinamics drives

Remedy: Wait until the connection was established.

**8712 JOG in X and C deactivated during Transmit**

Cause: JOG in X and C is not possible at active Transmit.

**22000 Gear change not allowed**

Cause: Gear step change when the spindle is active.

Remedy: Stop the spindle and carry out a gear step change.

**22270 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

**200000 to 300000 are specific to the drives and show up only in combination with the alarm # "8277 Sinamics error".****201699 - „(F) SI P1 (CU): Test of the shutdown path is necessary“**

Cause: A test of the shutdown path is necessary. The machine remains operational.

Remedy: The test is performed automatically during the next restart of the WinNC.

**235014 TM54F: Teststop necessary**

Cause: A teststop is necessary.

Remedy: Shutdown and restart the WinNC. The test is performed automatically during restart.



## Axis Controller Messages

### 8700 Execute REPOS in all axes before program start

Cause: After the program was stopped, the axes were manipulated with the hand wheel or with the jog keys and then a restart of the program was attempted.

Remedy: Before starting the program again, one should reposition the axes along the contour by executing "REPOS".

### 8701 No NCStop during offset align

Cause: The machine is currently executing an automatic offset adjustment. NC stop is not possible at this time.

Remedy: Wait until the offset adjustment is finished and then stop the program with NC stop.

### 8702 No NCStop during positioning after block search

Cause: The machine is currently finishing the block search operation and then it starts to go back to the last programmed position. No NC stop is possible in the meantime.

Remedy: Wait until positioning is finished and then stop the program with NC stop.

### 8703 Data record done

The recording of data is finished and the file record.acp has been copied to the installation folder.

### 8705 Feed-override missing, REPOS will not be executed

Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).

Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

### 8706 Tool sorting active

Cause: The tools were resorted during randomised tool management in order to enable non-randomised operation (tool 1 to place 1, tool 2 to place 2, etc.).

Remedy: Wait till the sorting is finished. The message will be deleted automatically by the control system.

### 8707 New control - please check tool table

Cause: The control system was changed when randomised tool management was active.

Remedy: Check the tool or place table to cancel the alarm.

### 8708 Switch off auxiliary drives for shutdown

Cause: There was an attempt to shut down the control system although the auxiliary drives are still switched on.

Remedy: Switch off the auxiliary drives and then shut down the control system.

### 8709 Insert tool in spindle for loading

Cause: During loading a tool must be physically available in the spindle.

Remedy: Clamp tool in the spindle. The message disappears.

## Control alarms 2000 - 5999

The alarms are released by the software.

**Fagor 8055 TC/MC**  
**Heidenhain TNC 426**  
**CAMConcept**  
**EASY CYCLE**  
**Sinumerik OPERATE**  
**Fanuc 31i**  
**Heidenhain TNC 640**

### 2200 Syntax error in line %s, column %s

Cause: Syntax error in the program code.

### 2001 Circle end point invalid

Cause: The start-middle point and middle-end point distances differ by more than 3 µm.

Remedy: Correct circular arc point.

### 2300 tracyl without corresponding round-axis invalid

Cause: Maybe the machine has no rotary axis.

### 3000 Traverse feed axis manually to position %s

Remedy: Move the axis manually to the required position.

### 3001 Tool T.. change!

Cause: A new tool was programmed in the NC program.

Remedy: Clamp the required tool in the machine.

### 4001 slot width too small

Cause: The tool radius is too large for the slot to be milled.

### 4002 slot length to small

Cause: The slot length is too small for the slot to be milled.

### 4003 length equal zero

Cause: Pocket length, pocket width, stud length, stud width are zero.

### 4004 slot width too big

Cause: The programmed slot width is larger than the slot length.

### 4005 depth equal zero

Cause: No machining takes place since no effective cutting depth has been defined.

### 4006 corner radius too big

Cause: The corner radius is too large for the size of the pocket.

### 4007 diameter too big

Cause: The remaining material (nominal diameter - diameter of the prebore) /2 is larger than the tool diameter.

### 4008 diameter too small

Cause: The tool diameter is too large for the intended bore.

Remedy: Enlarge the nominal diameter and use a smaller milling cutter.

### 4009 length too small

Cause: Width and length must be larger than the double tool radius.

### 4010 diameter less equal zero

Cause: The pocket diameter, the stud diameter, etc. must not be zero.

### 4011 blank diameter too big

Cause: The diameter of the machined pocket must be larger than the diameter of the premachined pocket.

### 4012 blank diameter too small

Cause: The diameter of the machined stud must be smaller than the diameter of the premachined stud.

**4013 start angle equal to end angle**

Cause: Start angle and end angle for hole pattern are identical.

**4014 tool radius 0 not permitted**

Cause: Tool radius zero is not permitted.

Remedy: Select a valid tool.

**4015 no outer contour defined**

Cause: The contour file indicated in the cycle was not found.

**4017 tool radius too big**

Cause: For the programmed machining, a tool being too large was selected. Therefore, machining is not possible.

**4018 allowance must not be 0**

Cause: There were programmed finishing operations without finishing offset.

**4019 too many iterations**

Cause: The contour definitions are too complex for the roughing-out cycle.

Remedy: Simplify the contour.

**4020 illegal radian correction**

Cause: An error has occurred during the programming of the radius compensation.

Remedy: Check the cycle parameters.

**4021 can't calculate parallel contour**

Cause: The control was not able to calculate the tool radius compensation.

Remedy: Check the programmed contour for plausibility. Maybe contact EMCO.

**4022 illegal contour definition**

Cause: The programmed contour is not suited to the selected machining.

Remedy: Check the programmed contour.

**4024 no contour definition**

Cause: The contour file being defined in the cycle has not been found.

**4025 internal calculation error**

Cause: An unexpected error has occurred during calculation of the cycle movements.

Remedy: Please inform the EMCO after-sales service.

**4026 allowance too big**

Cause: A part of the finishing offset (for several finishing passes) is larger than the total finishing offset.

Remedy: Correct the finishing offsets.

**4028 pitch 0 not permitted**

Cause: The thread was programmed with pitch zero.

**4029 undefined working mode**

Cause: Internal error (invalid machining type for the thread).

**4030 function not yet supported**

Cause: Roughing out with pockets is not implemented yet.

Remedy: Please inform the EMCO after-sales service.

**4031 value not permitted**

Cause: An invalid retracting direction was programmed during inside turning.

**4032 plunging must be defined**

Cause: For the programmed cycle no cutting depth has been programmed.

**4033 radius/chamfer too big**

Cause: The radius, respectively the chamfer, cannot be inserted in the programmed contour.

Remedy: Reduce the radius, respectively the chamfer.

**4034 diameter too big**

Cause: The programmed starting point and the machining diameter are contradictory.

**4035 diameter too small**

Cause: The programmed starting point and the machining diameter are contradictory.

**4036 unknown working direction**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4037 unknown working type**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4038 unknown sub cycle**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4039 rounding not possible**

Cause: The programmed radius contradicts the rest of the cycle parameters.

**4042 illegal tool width**

Cause: The tool width for the cutting-off cycle must be defined.

**4043 groove width too small**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4044 distance not defined**

Cause: The distance for the multiple grooving cycle must not be zero.

**4045 illegal allowance type**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4046 invalid speed**

Cause: The spindle speed must be nonzero.

**4047 invalid end point**

Cause: The programmed end point contradicts the rest of the cycle definition.

**4048 tool cut width too small**

Cause: The cutting edge is too small for the programmed cutting depth.

**4050 invalid distance**

Cause: The hole patterns do not tally with the selected distance.

**4052 working pattern not possible**

Cause: Error in the definition of the hole pattern. The number of bores is contradictory.

**4053 invalid start point**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4055 illegal working direction**

Cause: The machining direction is contradictory to the rest of the cycle definition.

**4057 plunging angle less equal zero**

Cause: The plunging angle must be between 0 and 90 degree.

**4058 chamfer too large**

Cause: The programmed chamfer is too large for the pocket cycle.

**4062 radius/chamfer too small**

Cause: The radius, respectively the chamfer, cannot be machined with the current tool radius.

**4066 invalid mill step**

Cause: The mill step must be greater than zero.

**4069 invalid angle**

Cause: An angle of zero degree is not permitted.

**4072 plunging too small**

Cause: For the cycle, a cutting depth has been selected that leads to extra-long machining time.

**4073 invalid clearance angle**

Cause: The clearance angle indicated for the tool cannot be machined.

Remedy: Correct the clearance angle for the tool.

**4074 contour-file not found**

Cause: The contour file indicated in the cycle has not been found.

Remedy: Please select the contour file for the cycle.

**4075 not machinable with selected tool**

Cause: The tool is too wide for the programmed groove.

**4076 reciprocating plunge cut impossible (initial move too short)**

Cause: The first movement of the contour is shorter than the double tool radius and cannot be therefore used for the swinging delivery.

Remedy: Extend the first movement of the contour.

**4077 wrong tool type in grooving or cut-off cycle**

Cause: The wrong tool type was used in the cutting cycle.

Remedy: Use only grooving and punch tools in the cutting cycles.

**4078 radius of helix too small**

Cause: The pitch of the helix is  $\leq 0$ .

Remedy: Program the radius  $> 0$ .

**4079 pitch of helix too small**

Cause: The radius of the helix is  $\leq 0$ .

Remedy: Program the pitch  $> 0$ .

**4080 radius of helix or tool too large**

Cause: The helical approach cannot be executed with the selected data for the helix and the current tool radius without a contour breach.

Remedy: Use a tool with a smaller radius or reduce the radius of the helix.

**4200 leaving movement is missing**

Cause: No movement after the tool radius compensation was deactivated in the current plane.

Remedy: Insert the departing movement in the current plane after having deactivated the tool radius compensation.

**4201 TPC off missing**

Cause: The tool radius compensation has not been deactivated.

Remedy: Deactivate the tool radius compensation.

**4202 TPC requires at least three movements**

Cause: The tool radius compensation requires at least 3 movements in the current plane in order to calculate the tool radius compensation.

**4203 approaching movement not possible**

Cause: It was not possible to calculate the approaching movement.

**4205 leaving movement not possible**

Cause: It was not possible to calculate the departing movement.

**4208 TPC curve could not be calculated**

Cause: It was not possible to calculate the tool radius compensation for the programmed contour.

**4209 switching the plane is not allowed when TPC is switched on**

Cause: The programmed plane must not be changed during the tool radius compensation.

Remedy: Remove the change of planes during the tool radius compensation.

**4210 tool path compensation already activated**

Cause: G41 is active and G42 was programmed or G42 is active and G41 was programmed.

Remedy: Switch tool radius compensation off with G40 before programming the radius compensation again.

**4211 Bottleneck detected**

Cause: In the radius correction calculation some parts of the contour were omitted, as too large a milling cutter tool was used.

Remedy: Use a smaller milling cutting tool to process the contour completely.

**4212 Infeed has been programmed twice during approach**

Cause: After the approach movement a second infeed has been programmed, without previously moving to the work plane.

Remedy: First program a movement to the work plane before programming a second infeed.

**5000 drill manually now****5001 contour has been adjusted to the programmed clearance angle**

Cause: The programmed contour was adapted to the programmed clearance angle. Maybe there will remain rest material that cannot be machined with this tool.

**5500 3D simulation: Internal error**

Cause: Internal error inside the 3D simulation.

Remedy: Restart the software or, if necessary, report the error to EMCO customer service.

**5502 3D simulation: Tool place invalid**

Cause: Tool place unavailable on the machine used.

Remedy: Correct tool call-up.

**5503 3D simulation: Chuck invalid owing to the unmachined part definition**

Cause: The distance from the front of the unmachined part is > the unmachined part length.

Remedy: Change the distance.

**5505 3D simulation: Unmachined part definition invalid**

Cause: Implausibility in the unmachined part geometry (e.g. expansion in one axis  $\leq 0$ , inside diameter > outside diameter, unmachined part contour not closed, etc.).

Remedy: Correct unmachined part geometry.

**5506 3D simulation: STL chuck file has auto-overcuts**

Cause: Error in the chuck description.

Remedy: Correct file.

**5507 3D simulation: Pole transit on TRANS-MIT!**

Cause: Travel comes too close to the X0 Y0 coordinates.

Remedy: Change travel.



# I: Fanuc 31i controller alarms

## Controller alarms 0001 - 88000

The controller triggers these alarms. These are the same alarms as would have occurred on the Fanuc 31i controller.

### 0006 ILLEGAL USE OF MINUS SIGN

**Declaration:** Illegal minus sign (-) in a NC command word or a system variable.

### 0010 IMPROPER G-CODE

**Declaration:** Improper G-Code commanded. The parameter for piercing with continuous circular motion is not effective. The activation signal for piercing with continuous circular motion is "0".

### 0030 ILLEGAL OFFSET NUMBER

**Declaration:** Illegal offset number specified. Alarm also comes on if, in tool offset memory B, the number of tool form offsets exceeds the maximum number of tool offset sets.

### 0051 MISSING MOVE AFTER CNR/CHF

**Declaration:** Improper move or travel in the set after chamfering or rounding. Correct program.

### 0055 MISSING MOVE VALUE IN CHF/CNR

**Explanation:** In the set for chamfering/rounding, the path is less than the amount of the chamfer or rounding. Correct program.

### 0077 TOO MANY SUB,MACRO NESTING

**Declaration:** In total, more subroutines and macro calls have been commanded than allowed. Further subroutine calls during subroutine call from external memory

### 0114 ILLEGAL EXPRESSION FORMAT

**Declaration:** Format error in a printout of a client macro instruction. The punched tape format parameter is incorrect.

### 0115 VARIABLE NO. OUT OF RANGE

**Declaration:** A local, global or system variable in a client macro contains an out of range number. A non-existent client macro variable number has

been given in the "EGB axis fadeout" function (G31.8). Or the number of client macro variables for saving jump positions is insufficient.

High speed cycle machinings are incorrect. The alarm is triggered in the following cases:

- 1) The program head corresponding to the called machining cycle number is missing.
- 2) The value of the cycle connection information is outside the permitted range (0 to 999).
- 3) The number of data elements in the program head is outside the permitted range (1 to 65535).
- 4) The memory start data variable number of the executable data is outside the permitted range (#20000 to #85535/#200000 to #986431/#2000000 to #3999999).
- 5) The memory end data variable number of the executable data is outside the permitted range (#85535/#986431/#3999999).
- 6) The memory start data variable number of the executable data is the same variable number as that used by the program head.

### 0116 WRITE PROTECTED VARIABLE

**Declaration:** In a client macro, a variable should be used on the left page of a printout that may only appear on the right page.

### 0128 ILLEGAL MACRO SEQUENCE NUMBER

**Explanation:** The figure program includes The given set number was not found in the set number search.

The set number given in GOTO-- and M99P-- as the jump destination was not found.

### 0175 ILLEGAL G07.1 AXIS

**Declaration:** Axis with which cylindrical interpolation is impossible. Several axes in a G07.1 set. Cylindrical interpolation is ended for an axis that is not in this mode.

Set the axis for cylindrical interpretation in parameter 1022 not on 0, but on 5, 6 or 7 (parallel axis), in order to describe the arc with the rotation axis (ROT parameter 1006#1 on 1, the parameter 1260 is configured).

### 0310 FILE NOT FOUND

**Declaration:** The file was not found when calling up the subroutine or macro.

A 2014-12

**0312 ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING**

**Declaration:** Illegal command format for direct drawing dimensions programming. An illegal G-Code was used for direct drawing dimensions programming.

There are two or more sets without a positioning command between two commands for direct drawing dimensions programming. Decimal point in direct drawing dimensions programming, although decimal points must not be used (bit 4 of parameter no. 3405 = 1).

**1330 ILLEGAL OFFSET NUMBER**

**Declaration:** Spindle number higher than the number of controlled spindles while loading parameters or lead error offset data from punched tape or with G10.

**1960 ACCESS ERROR (MEMORY CARD)**

**Declaration:** Illegal memory card access. This alarm is also triggered while reading if the end of the file is reached and no EOR code '%' is found.

**3506 WRONG CUTTING AREA**

**Declaration:** The cutting area is illegal.  
**Remedy:** Change the cutting program so that a correct cutting area is given, so that part contour and raw part contour match with the cutting profile.

**3507 WRONG CUTTING CONDITION**

**Declaration:** The cutting conditions are illegal.  
**Remedy:** Change the cutting program so that the cutting conditions, such as feed speed, are normal.

**3514 WRONG FIGURE DATA**

**Declaration:** The figure data is illegal.  
**Remedy:** Change the cutting program so that the figure data is correct.

**3516 NO NECESSARY ADDRESS**

**Declaration:** The necessary addresses for a cycle cutting command or other 4-digit G-commands were not entered.  
**Remedy:** Change the cutting program, e.g. by adding the necessary addresses.

**3530 WRONG MACHINING TYPE**

**Explanation:** The machining type specification is illegal.  
**Remedy:** Change the machining program so that the machining type is appropriate.

**3531 WRONG RETURN MODE**

**Explanation:** The return mode is illegal.  
**Remedy:** Change the machining program so that the return mode is appropriate.

**3535 WRONG THREADING LEAD**

**Declaration:** The threading lead setting is illegal.  
**Remedy:** If, say, a negative value was entered as the threading lead, change the machining program so that the threading lead is correct.

**3538 WRONG CUTTING DIRECTION**

**Explanation:** The cutting direction setting is illegal.  
**Remedy:** A value was entered that must not be given for turning or other machining directions. Change the machining program so that the machining direction is permitted.

**3539 WRONG CUT DEPTH DIRECTION**

**Declaration:** The cut depth direction setting is illegal.  
**Remedy:** A value was entered that must not be given for turning or other cut depth directions. Change the machining program so that the cut depth direction is permitted.

**3541 WRONG CHAMFERING AMOUNT**

**Explanation:** The chamfering amount setting is illegal.  
**Remedy:** A value was entered that is not permitted for chamfering or other chamfer values, e.g. a negative value. Change the machining program so that the chamfering amount is permitted.

**3542 WRONG EJECTION STROKE**

**Declaration:** The ejection stroke setting is illegal.  
**Remedy:** A value was entered that is not permitted for chamfering or other plane area ejection stroke amounts, e.g. a negative value. Change the machining program so that the ejection stroke amount is permitted.

**3543 WRONG THICKNESS**

**Explanation:** The given machining thickness is illegal.  
**Remedy:** A value was entered that is not permitted for pocket milling or other machining thicknesses, e.g. a negative value. Change the machining program so that the machining thickness is permitted.

**3547 WRONG CUTTING DEPTH ANGLE**

**Declaration:** The cutting depth angle is illegal.

**Remedy:** A value was entered that is not permitted for pocket milling or other cutting depth angles. Change the machining program so that the cutting depth angle is permitted.

**3548 WRONG CLEARANCE**

**Declaration:** - The clearance is illegal.

**Remedy:** A clearance value was entered that is illegal, e.g. a negative value. Change the machining program so that the clearance is permitted.

**3551 WRONG FINISHING NUMBER**

**Declaration:** - The number of finished machining processes is illegal.

**Remedy:** An illegal value for the number of thread cutting processes or for other finished machining process was entered, e.g. 0. Change the machining program so that the number of finished machining processes is permitted.

**3552 WRONG APPROACH SETTING**

**Declaration:** The approach setting is illegal.

**Remedy:** A value was entered that is not permitted for contour cutting or other approaches. Change the machining program so that the approach is permitted.

**3553 WRONG ESCAPE SETTING**

**Declaration:** The escape setting is illegal.

**Remedy:** A value was entered that is not permitted for contour cutting or other escapes. Change the machining program so that the escape is permitted.

**3575 WRONG FIGURE TYPE**

**Explanation:** The figure type is illegal.

**Remedy:** A figure type that is not permitted was selected for a figure set. Change the machining program so that the figure type is permitted.

**3579 WRONG CORNER SETTING**

**Declaration:** The corner rounding setting is illegal.

**Remedy:** A value for rounding the corner of a figure type that is not permitted was entered. Change the machining program so that the entered value is permitted.

**3582 WRONG FIGURE RADIUS**

**Declaration:** The figure radius setting is illegal.

**Remedy:** A value for rounding the corner of a figure type that is not permitted was entered.

Change the machining program so that the entered value is permitted.

**3584 WRONG PITCH SETTING**

**Declaration:** The pitch setting is illegal.

**Remedy:** A value for a pitch for a figure set that is not permitted was entered. Change the machining program so that the set value is permitted.

**3585 WRONG HOLE/GROOVE NUMBER**

**Declaration:** The number of holes/grooves is illegal.

**Remedy:** A value for the number of holes or grooves in a figure set that is not permitted was entered. Change the machining program so that the set value is permitted.

**3586 WRONG COORDINATE POSITION**

**Declaration:** The coordinate positions are illegal.

**Remedy:** A value for the coordinates of a figure set that is not permitted was entered. Change the machining program so that the set value is permitted.

**3592 WRONG OMITTED POINT**

**Declaration:** A random figure is not closed.

**Remedy:** A random figure that was entered for plane cutting, pocket milling or turning was not closed. Change the machining program so that there is a closed figure that has identical start and end points.

**5010 END OF RECORD**

**Declaration:** The EOR (end of record) code was in the middle of a set. An alarm is also generated if the percent sign is read at the end of the program.

**5044 G68 FORMAT ERROR**

**Declaration:** Error in the three-dimensional coordinate conversion command:

- (1) No I, J or K in the three-dimensional coordinate conversion command string (without the "Coordinate rotation" option).
- (2) I, J or K are all 0 in the three-dimensional coordinate conversion command string.
- (3) No rotating angle R in the three-dimensional coordinate conversion command string.



# W: Accessory Functions

## Activating accessory functions

According to the machine (turn/mill) the following accessories can be taken into operation:

- automatic tailstock
- automatic vice/clamping device
- Air purge system
- Dividing attachment
- Robot interface
- Automatic doors
- Win3D view simulation software
- DNC interface

The accessories are activated with EMConfig.

## Robotic Interface

The robotic interface is used to connect the concept machines to an FMS/CIM system.

The most important functions of a concept machine can be automated via the inputs and outputs of an optional hardware module.

The following functions can be controlled via the robotic interface:

- Program START / STOP
- Door open / closed
- Quill clamp / back
- Clamping device open / closed
- Feed STOP

### Note:

The functionality of the Win3D-View is depending on the control software that is used.



## Automatic doors

Preconditions for activation:

- The auxiliary drives must be switched on.
- The main spindle must be still (M05 or M00) - this also means that the run-out phase of the main spindle must be ended (program dwell time if required).
- The feed axes must be still.
- The tool changer must be still.

Behavior when automatic doors active:

Opening door

The door can be opened manually, via the robot interface or DNC interface.

In addition, the door opens when the following commands are executed in the CNC program:

- M00
- M01
- M02
- M30

Closing door:

The door can be closed by manually pressing the button via the robot interface. It is not possible to close the door via the DNC interface.

## Win3D View

Win3D View is a 3D simulation for turning and milling, which is offered as an additional option for the WinNC product. Graphical simulations of CNC controls are primarily designed for industrial practice. The Win3D View screen representation goes beyond the industrial standard. Tools, raw parts, clamping devices and the processing steps are represented extremely realistically. The programmed movement paths of the tool are checked by the system for a collision with clamping device and raw part. A warning message is issued when there is danger. This makes possible to have understanding and control of the manufacturing process already on the screen.

Win3D View is used to visualize and prevent costly collisions.

Win3D View offers the following advantages:

- Extremely realistic representation of workpiece
- Tool and clamping device collision control
- Cut representation
- Zoom functions and turning of views
- Representation as solid or wireframe model

## DNC interface

The DNC interface (Distributed Numerical Control) enables the control system (WinNC) to be controlled remotely via a software protocol.

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

During the installation of the control software, the DNC interface is enabled and configured, and can be reconfigured with EMConfig later on.

The DNC interface creates a connection between a higher-level computer (production control computer, FMS computer, DNS host computer, etc.) and the control computer of an NC machine. After activation of the DNC drive the DNC computer (Master) takes over control of the NC machine (Client). The DNC computer takes over complete control of the manufacturing. The automation fittings such as door, chuck (collet), quill, coolant, etc. can be controlled from the DNC computer. The current status of the NC machine is visible on the DNC computer.

The following data can be transferred or loaded via the DNC interface:

- NC Start
- NC Stop
- NC programs \*)
- Zero point shifts \*)
- Tool data \*)
- RESET
- Approach reference point
- Periphery control
- Override data

The DNC interface can be operated with the following CNC control types:

- SINUMERIK Operate T and M
- FANUC 31i T and M

Further details of the functions and the DNC protocol can be found in the accompanying product documentation.

If the DNC interface is operated with TCP/IP, it will wait for incoming connections on port 5557.

\*) not for SINUMERIK Operate and FANUC 31i

# X: EMConfig

**Note:**

The settings which are available in EMConfig are depending on the machine and the control that is used.



## General

EMConfig is a configuration software for WinNC. EMConfig helps you to alter the settings of WinNC.

The most important settings are:

- Control language
- System of measurement mm - inch
- Activate accessories
- Selection of interface for control keyboard

Using EMConfig you can also activate diagnostic functions in case of troubles - that way you get help immediately.



Safety-related parameters are protected by a password. They can only be activated by set-up technicians or by technical support representatives.



Icon for EMConfig

## How to start EMConfig

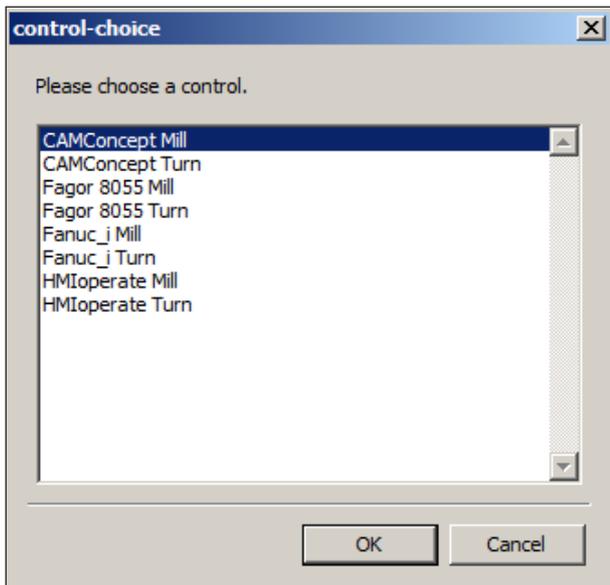
Open EMConfig.

In case several control types are installed, a selection box will appear on the screen.

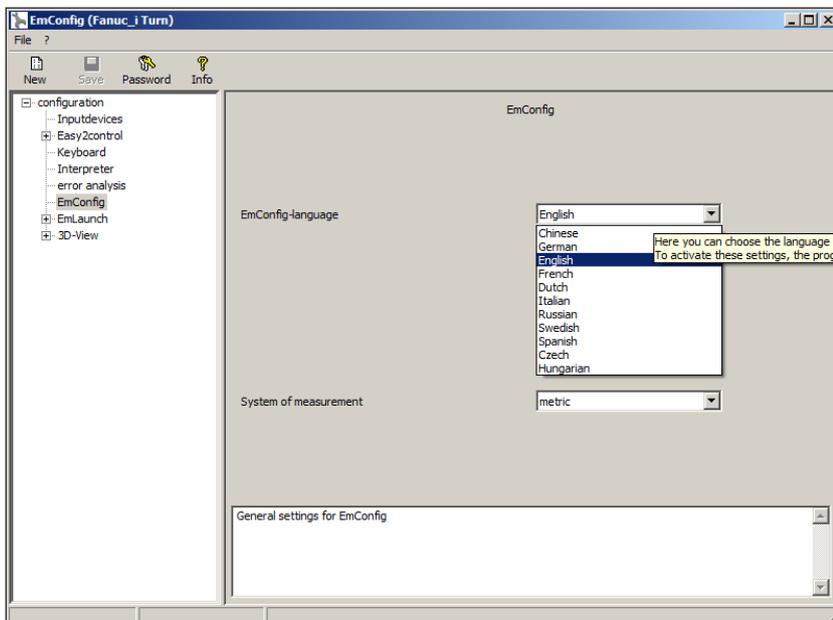
Select the required control type and click OK.

The following settings are only valid for the selected control type.

The window for EMConfig appears on the screen.



Selection box for control type



Change the language of EMConfig

Here you can change the language of EMConfig. In order to activate the settings, restart the program.

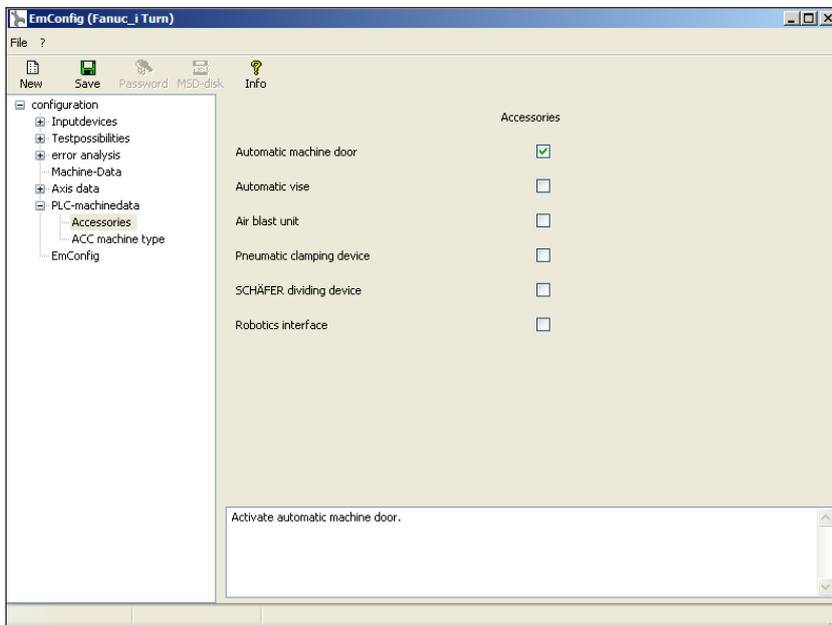
### Note:

Select the desired menu item. The appropriate function is explained in the text box.



## How to activate accessories

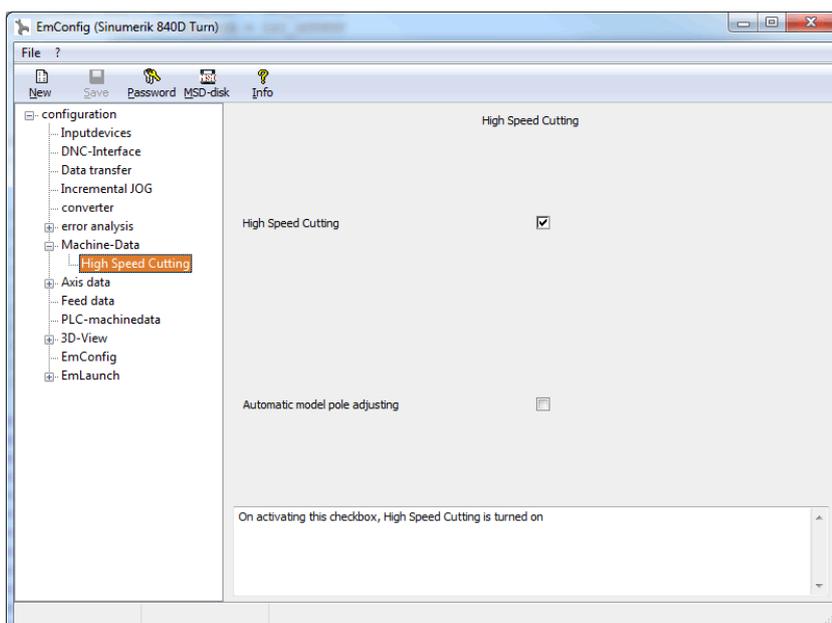
When you install accessories on your machine, you need to activate them here.



Activate accessories

## High Speed Cutting

On activating this checkbox, High Speed Cutting is turned on.



Activate High Speed Cutting

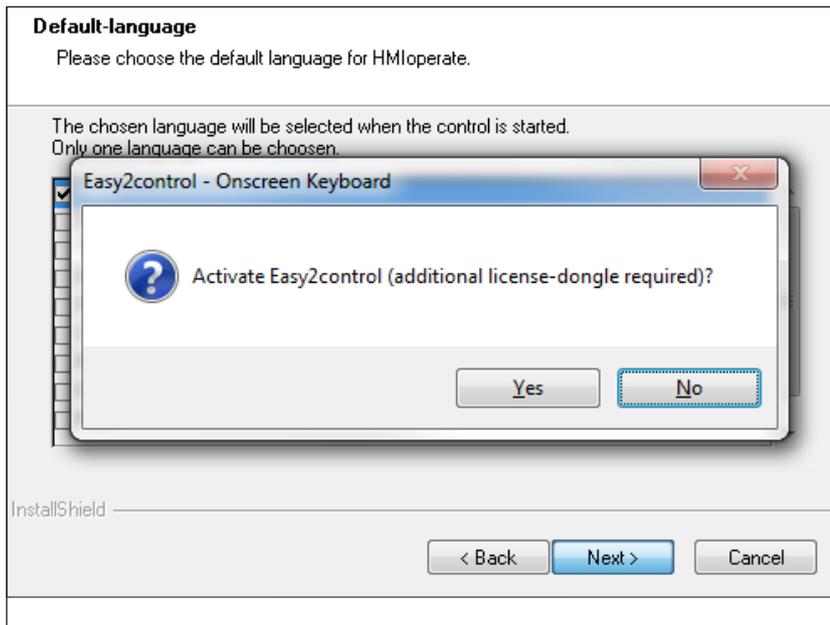
By using high speed cutting, the setting of the axis controller is adjusted. This gain is only effective until the programmed feed rate of 2500 mm/min and allows contour faithful retraction of the tool path and generating of sharp edges. If the feed is set up to higher than 2500 mm/min, it is automatically reset to the normal operating mode and sanded and rounded edges are created.

**Note:**

If Easy2control is used without the hardware-dongle, the controls are deactivated and an appropriate alarm is output by the controller. However, the virtual keyboard is displayed completely.

## Easy2control on screen operation

Installation and activation using the example of WinNC for Sinumerik Operate.

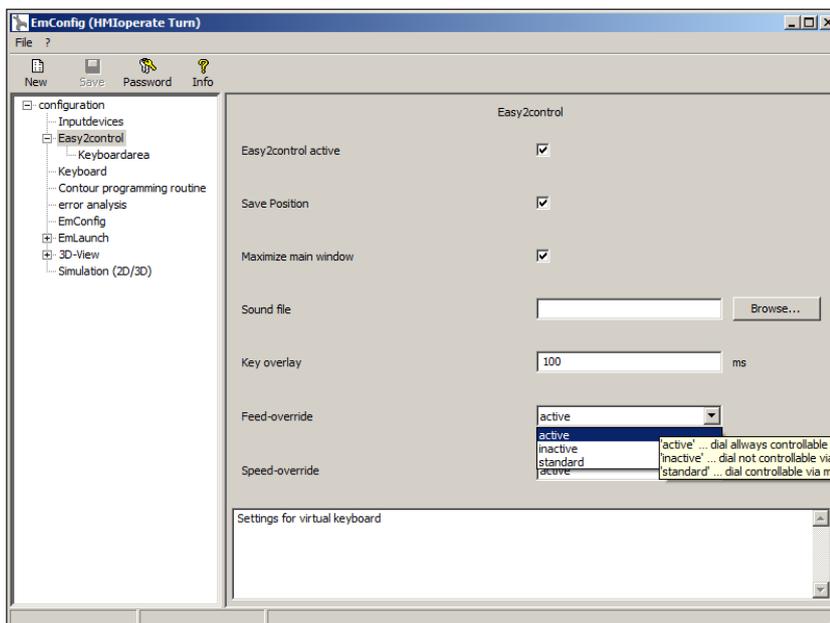


When installing the software WinNC for Sinumerik Operate you will be prompted to activate Easy2control. In order to use the Software without restriction, the supplied dongle must be connected to a free USB port.

Activating Easy2control

## Settings

This mask allows you to enable or disable Easy2control and make settings.



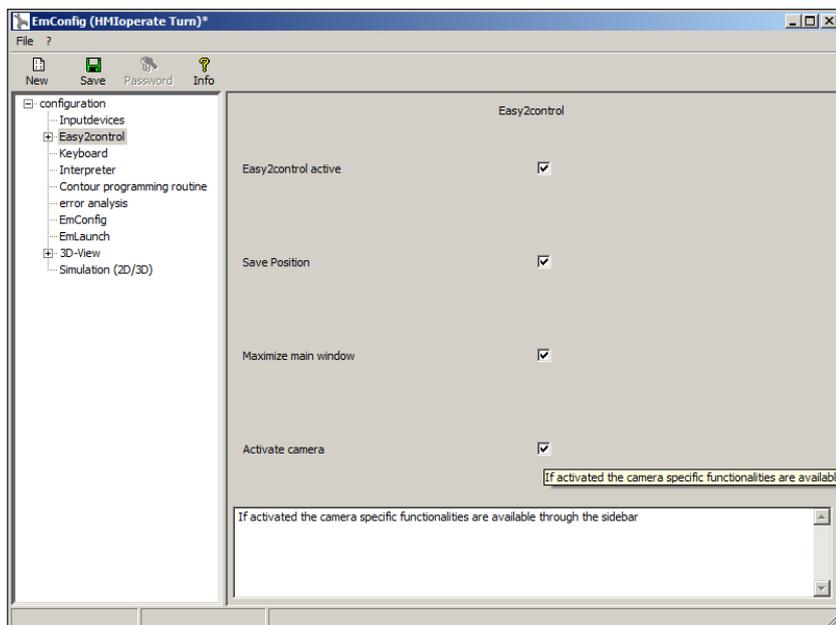
### Dial feed-override and dial speed-override:

- **Active:** dial always controllable via mouse/touchscreen (even with available hardware-version).
- **Inactive:** dial not controllable via mouse/touchscreen.
- **Standard:** dial controllable via mouse/touchscreen when no hardware-version is available.

Easy2control settings

## Machine room camera

The accessory machine room camera is available for all controls which support Easy2control.



*Activate camera*

Please refer to chapter Y “external input devices” for more information about the installation guide for the camera.



### Attention:

The camera may not be operated without the supplied waterproof case.

If the camera is operated without the waterproof case it may be damaged by coolant and chips.



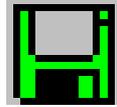
### Danger:

Make sure the machine room camera is placed properly in the work area that collisions with the tool turret and the axes are avoided.

## How to save changes

After the settings, the changes must be saved.

Select "Save" or click on the icon.



### Note:

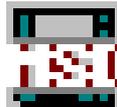
Input fields highlighted in red indicate inadmissible values. Inadmissible values are not saved in EMConfig.



After saving the changes, create a machine data floppy disk (MSD) or a machine data USB flash drive.

## How to create machine data floppy disk or machine data USB flash drive

After having changed the machine data, the machine data floppy disk or the machine data USB flash drive must be in the appropriate drive. Otherwise your changes cannot be saved and get lost.



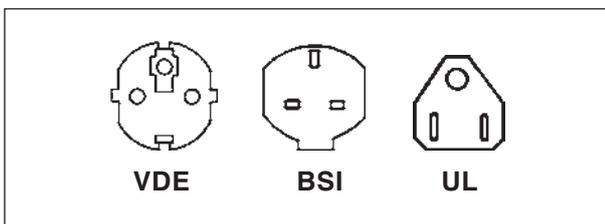
# Y: External Input Devices

## EMCO Control Keyboard USB

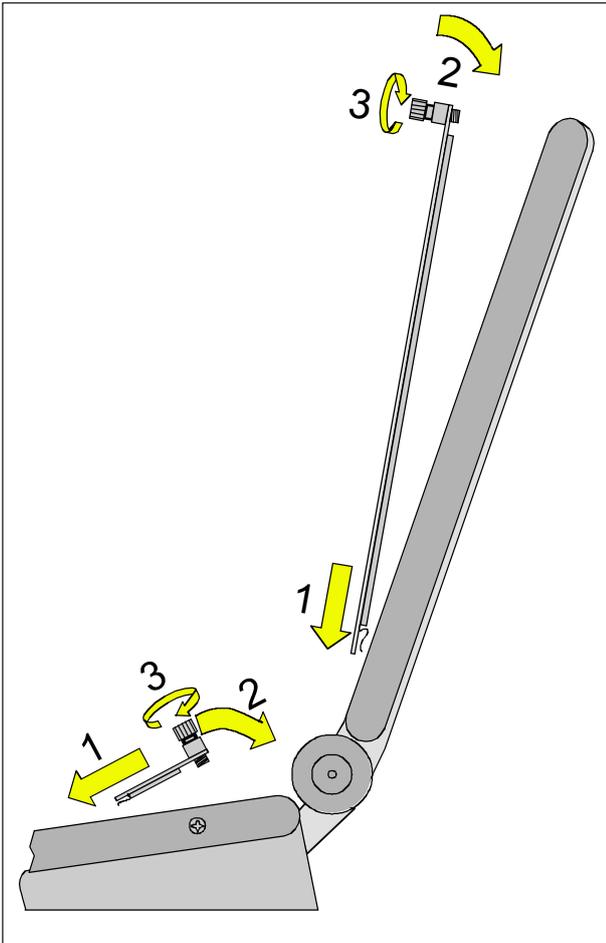
### Scope of supply

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

- Basic case
- Key module



Ref. No.	Description	
X9B 000	Basic unit with USB cable	
X9Z 600	TFT Display with screen cable and power supply unit	
A4Z 010	Mains cable VDE	
A4Z 030	Mains cable BSI	
A4Z 050	Mains cable UL	
X9Z 050N	Key module FAGOR 8055 TC 2 key sheets with keys	
X9Z 055N	Key module FAGOR 8055 MC 2 key sheets with keys	
X9Z 426N	Key module HEIDENHAIN 426/430 2 key sheets with keys 1 package exchange keys	
X9Z 060	Key module WinNC for SINUMERIK OPERATE 2 key sheets with keys	
X9Z 030	Key module WinNC for FANUC 31i 2 key sheets with keys 1 package exchange keys	
X9Z 640	Key module Emco WinNC for HEIDENHAIN TNC 640	



### Assembling

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

**Note:**

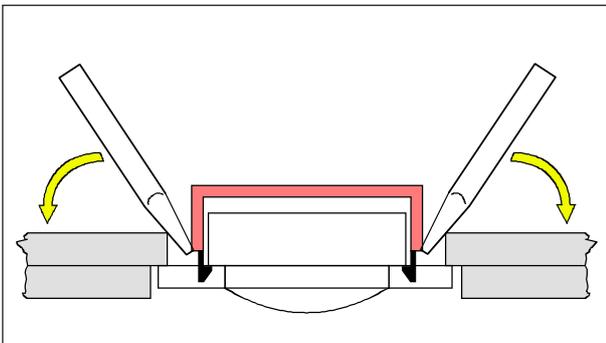
The key sheets must not be bended, otherwise the switching function can not be warranted.

### Exchange of single key caps

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

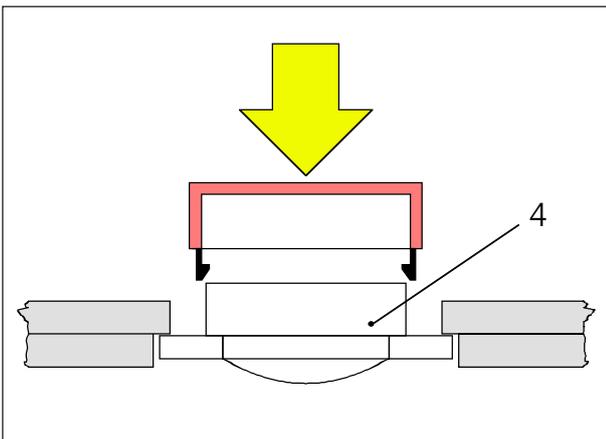
The scope of supply includes a package of exchange key caps to equip the keyboard for milling.

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



**Note:**

For the control type Emco WinNC for HEIDENHAIN TNC 640 only a milling version is available.



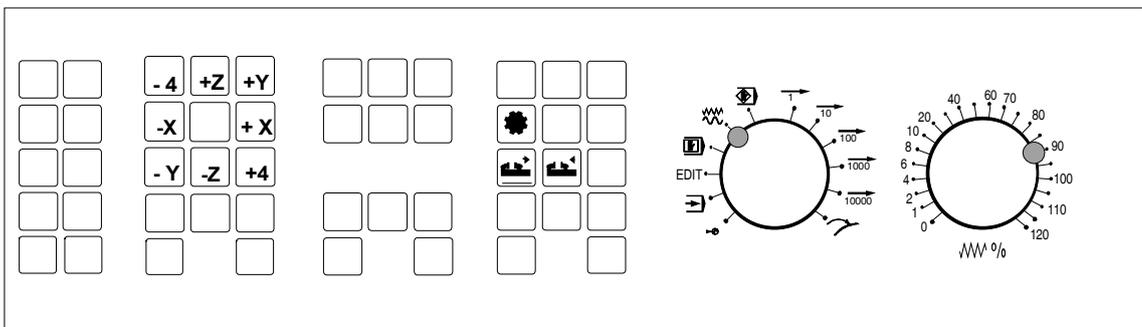
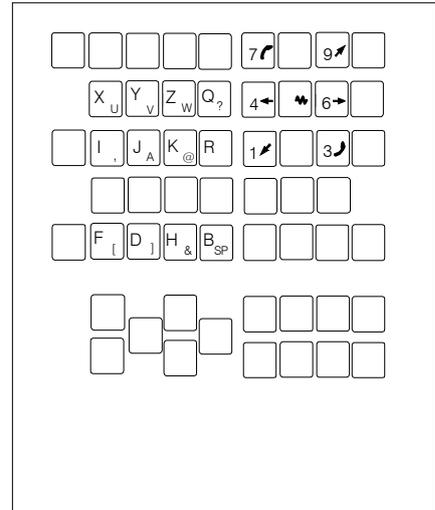
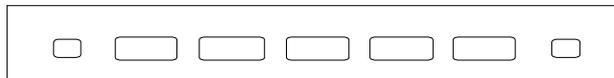
**Take off**

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

**Clip on**

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

**FANUC 31iM**  
Exchange key caps  
for milling



### Connection to the PC

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

### Settings at the PC software

#### Setting during new installation of the PC software

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

#### Setting in case of PC software already installed

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

## Easy2control On Screen operation

Easy2control adds a range of attractive applications to the successful interchangeable control system used in EMCO training machines. Suitable for use in machines and simulation workplaces alike, it displays additional control elements directly on the screen and, when used together with a touchscreen monitor, provides the ideal input interface.

### Scope of supply

The software for Easy2control is part of the control software.

The dongle for a workstation license is delivered:

**Ref. No.** X9C 111

Technical data for the screen:

16:9 Full-HD Monitor (1920x1080) at the minimum

Currently available controls (T and M):

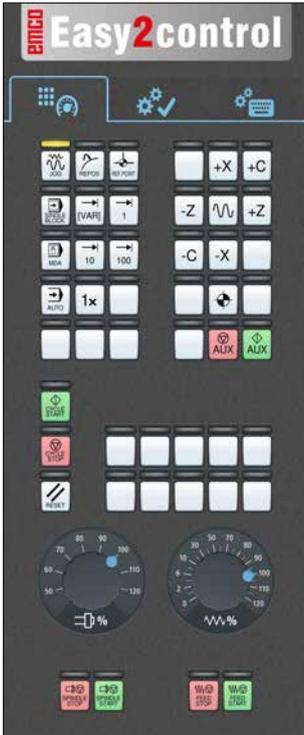
- Sinumerik Operate
- Fanuc 31i
- Emco WinNC for Heidenhain TNC 640 (M only)
- Emco WinNC for Heidenhain TNC 426 (M only)
- Fagor 8055

**Note:**

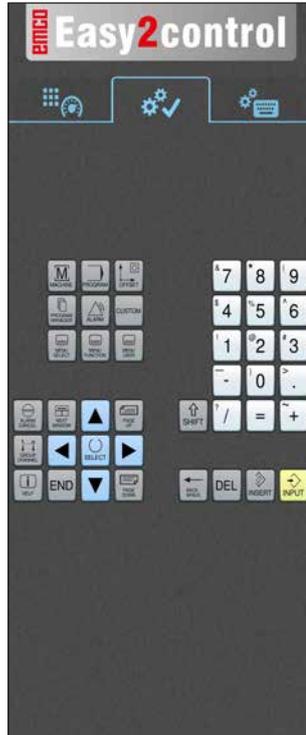
If a Full HD monitor is used without touchscreen function, the control is operated just with mouse and keyboard.

# Operating areas

## Sinumerik Operate



Machine control panel

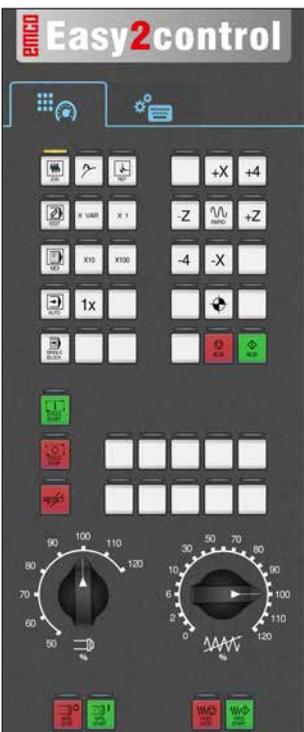


Controller-specific operation



Control operation complete

## Fanuc 31i



Machine control panel



Control operation complete

### Emco WinNC for Heidenhain TNC 640



Machine control panel

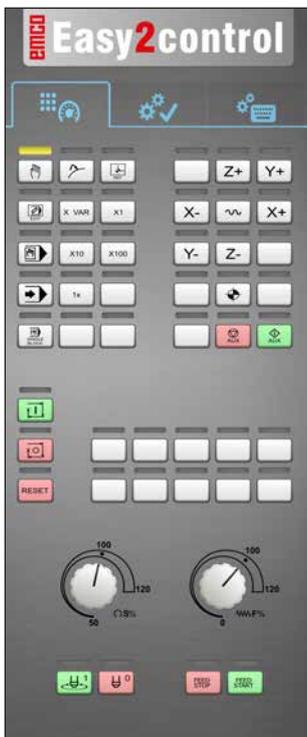


Controller-specific operation



Control operation complete

### Heidenhain TNC 426



Machine control panel

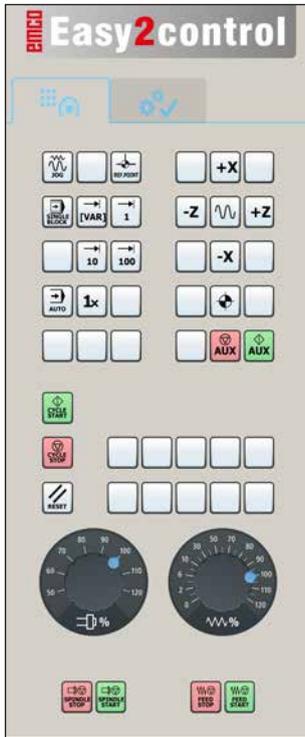


Controller-specific operation



Control operation complete

## Fagor 8055



Machine control panel



Control operation complete

Refer to the chapter “Key Description” of the respective control description for operation and key function.

**Note:**

The screen display, based on customer-specific configurations, may look different.



## Machine room camera

This accessory can be ordered using the following order number:

Ref. No.: S4Z750

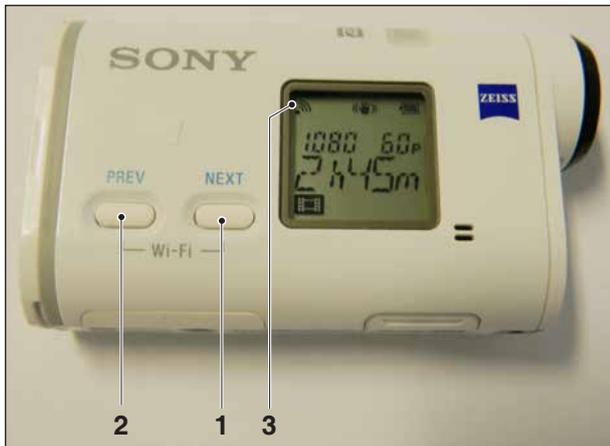
### Installing the camera

#### Requirement

USB WLAN adapter for the machine.

#### Setup WLAN

- press NEXT (1) or PREV (2) until an operating mode appears which supports WLAN, e.g. MOVIE. The symbol for WLAN (3) appears in the corner on top left.
- Open EMConfig and activate the camera.
- Connect the WLAN adapter to the USB port to the machine.
- Open windows network and sharing centre (4).
- Select the wireless network, enter the password and finish the connect setup. The name of the network (5) and the password are supplied together with the camera.
- Open the control software with activated Easy-2control.



Activating WLAN



Connect WLAN

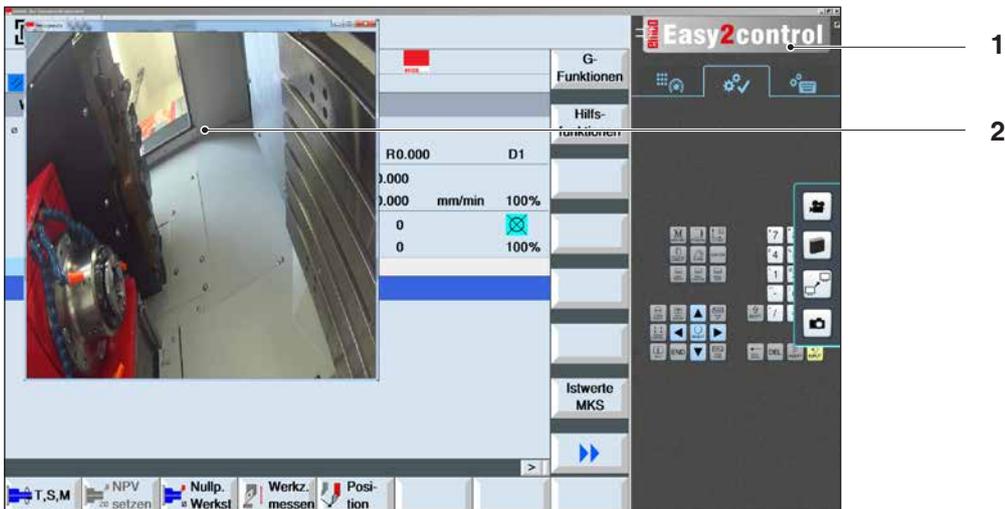
5 4

## Operating the camera

- Click the Easy2control logo to open the sidebar.

### Functions of the sidebar

- Click the camera symbol to open the preview window (2).
- Open the operating manual of the control.
- Option for the second screen:
  - duplicate screen
  - screen extension to two monitors
- Creates a screenshot of the control in \*.png format.



Operating the machine room camera

**Note:**

The option for the second screen is only available for machines CT/CM 260 and 460.



**Attention:**

The camera may not be operated without the supplied waterproof case.

If the camera is operated without the waterproof case it may be damaged by coolant and chips.





# Z: Software Installation Windows

## System prerequisites

### Machine with integrated control PC

- All Concept machines
- Machines that were converted to ACC
- MOC with Windows 7 or higher (32 / 64 Bit)

### Machines with included control PC and programming stations

- Windows 7 or higher (32 / 64 Bit)
- free hard drive space 400 MB
- Programming station: 1\*USB, machine version: 2\*USB
- TCP/IP-capable network card for machine version)

### Recommended system prerequisites

- PC Dual Core
- Working memory 4 GB RAM
- free hard drive space 2 GB

## Software installation

- Start Windows
- Start the installation application on the USB stick or your download file.
- Follow the instructions from the installation guide.

For more informations regarding software installation and / or software update please refer to the documentation "short description for WinNC update installation".

### Note:

PC TURN and PC MILL have to be equipped with the conversion kit for ACC in order to operate EMCO WinNC.



## Variants of WinNC

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

- WinNC for SINUMERIK Operate T and M
- WinNC for FANUC 31i T and M
- HEIDENHAIN TNC 426
- Emco WinNC for HEIDENHAIN TNC 640
- FAGOR 8055 TC and MC
- CAMConcept T and M

In case there are several control types installed, a menu appears when starting EM Launch from which you can select the desired type.

The following versions can be installed from the WinNC variants:

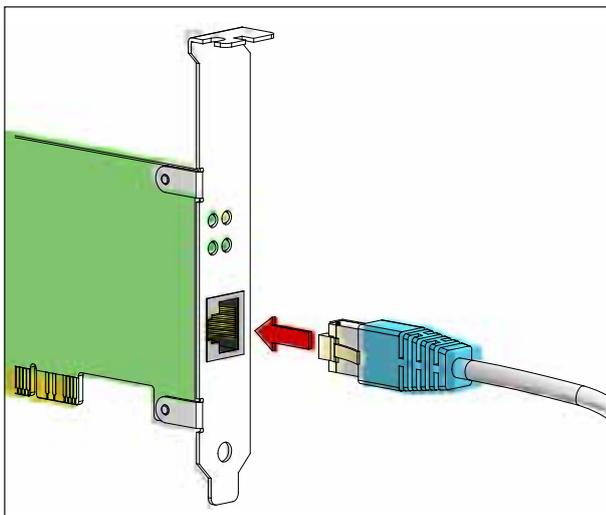
- Demo licence:  
The demo licence is valid for 30 days after the first use. 5 days before the demo licence expires, you can enter another valid licence key (see licence manager)
- Programming station:  
Programming and operation of the appropriate CNC control type is simulated by WinNC on your PC.
- Single user licence:  
Authorizes to external programming of CNC-controlled machine tools on one PC workstation.
- Multi-user licence:  
Authorizes to external programming of CNC-controlled machine tools. The multi-user licence can be installed on an unlimited number of PC workstations or in a network within the institute registered by the licensor.
- Educational licence version:  
Is a time-limited multi-licence especially for schools and educational institutes.
- Machine licence:  
This licence allows to directly operate a PC-controlled machine (PC TURN, Concept TURN, PC MILL, Concept MILL) of WinNC as if it was operated by an ordinary CNC control.

**Danger:**

Removal and installation of the network card must only be carried by skilled personnel. The computer must be disconnected from the power supply (pull the power plug).

**Note:**

During a machine installation one network-card is reserved exclusively for the control of the machine.



Connection of the machine to the PC

**Network card (ACC)**

for:

Concept Turn 55  
 Concept Mill 55  
 Concept Turn 105  
 Concept Mill 105  
 Concept Turn 60

Only for machines with ACC kit:

PC Turn 50  
 PC Mill 50  
 PC Turn 100  
 PC Mill 120

Network card type: TCP/IP compatible network card

Setting the network card for the local connection to the machine:

**IP address: 192.168.10.10**  
**Subnetmask 255.255.255.0**



In case of problems observe the instructions of your operating system (Windows help).

**Instructions:**

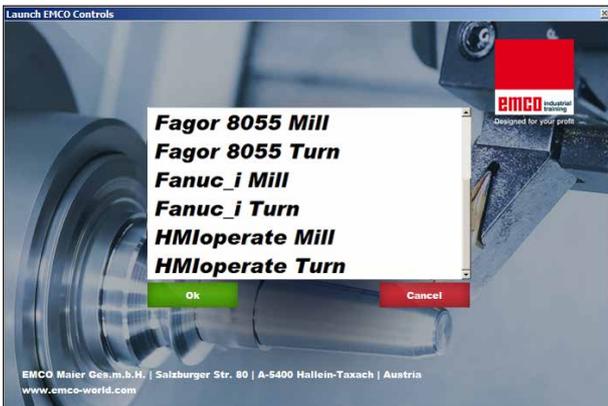
If the network connection to the machine could not be established at the start, the above adjustments are to be made.

## Starting WinNC

If you choose AUTO START YES during the installation of your machine version, WinNC starts automatically after switching on the PC.

Otherwise proceed as follows:

- 1 Switch the machine on.
- 2 Wait for 20 seconds to ensure that the machine operating system is running before the network connection to the PC is established. Otherwise it is possible that no connection can be established.
- 3 Switch the PC on and start Windows.
- 4 Click on the start symbol at the bottom.
- 5 Select program, EMCO and click on WinNC.
- 6 The start image will be shown on the screen. The licence holder is registered in the start screen.
- 7 If you have only installed one CNC control type, it starts immediately.
- 8 If you have installed several CNC control types, the selection menu appears.
- 9 Select the desired CNC control type (use cursor buttons or mouse) and press ENTER to start it.
- 10 If you use the control keyboard, you can select the desired CNC control type with the cursor buttons or mouse and start with the "NC-Start" button.



Selection menu EMLaunch

### Note:

EMLaunch displays all WinNC und CAMConcept controls that are installed in the same directory.



## Terminating WinNC

- 1 Switch off auxiliary drive with AUX OFF.  
Only for machine places, not for programming stations.
- 2 By simultaneously pressing these buttons WinNC for Sinumerik Operate will be terminated specifically.  
This corresponds to Alt+F4 on the PC keyboard.

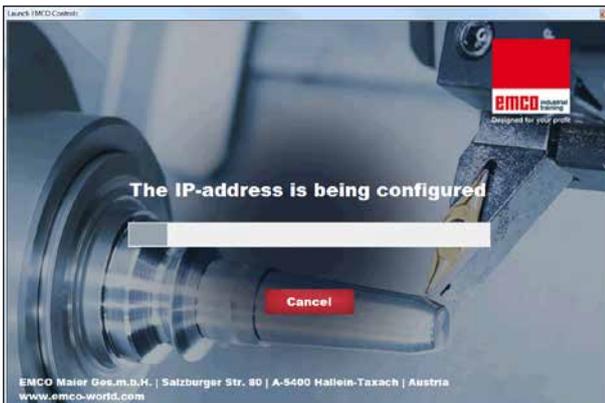
## Checks by EmLaunch

In the ACC / ACpn-machine version EmLaunch is checking if a machine is available:

During the network configuration, the IP address is not configured correctly and DHCP for automatic configuration of the IP address is disabled. Connection to the machine is not possible.



*DHCP disabled*



*IP-address configuration*



*Setup the connection to the machine*

It is attempt to configure the IP address automatically via DHCP.

The IP configuration is correct and the connection to the machine is checked. Once the machine is available, the selection of the available controls is displayed.

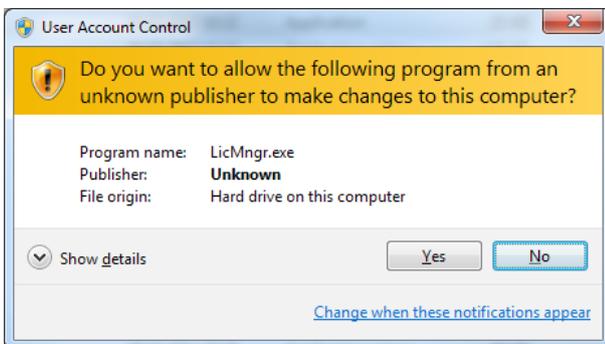


*Connection to the machine is OK*

The connection to the machine is completed and the corresponding control can be started.



Input window licence key enquiry



Run EMCO licence manager as an administrator



EMCO Licence Manager

## Licence input

After the installation of an EMCO software product, an input window appears during initial operation and asks for name, address and licence key. This input window appears for every software product that is installed. In case a demo licence is desired (see page Z1), please select "DEMO". Then the input window reappears only 5 days before the expiry date of the demo licence. A subsequent input of a licence key is also possible via the licence manager (see licence manager below).

## Licence manager

The query in the UAC dialog box must be confirmed with Yes in order to start the Licence Manager.

For the release of additional function groups of existing EMCO software products it is necessary to enter a new licence key (exception: demo licence).

The **EMCO Licence Manager** (see picture on the bottom on the left) enables the input of further new licence keys. For this purpose select the new product in the selection window and confirm the input.

The next time you start your control software an input window appears and asks you to enter name, address and licence key (see picture on the top left).

Please note that the licence key is asked for each software product individually. The picture on the left shows e.g. the input prompt for the licence key for the software product "Heidenhain TNC 426".

Input licence key:

Start the WinNC with the option "Run as Administrator" right after installing the programm or launching the licence manager.