**Software description**
WinNC for Fanuc 31i Mill**Ref. no. EN 1846**
Edition C 2016-02 **Rev 00**

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

Original operating instructions**software release effective from 01.02**

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
E-Mail: service@emco.at



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

emco

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>i</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 several 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 highlighted text to clipboard	C19
Move highlighted text to clipboard	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>i</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
Tool modelling with the 3D-Tool Generator.....	W2
DNC interface	W6

X: EMConfig

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

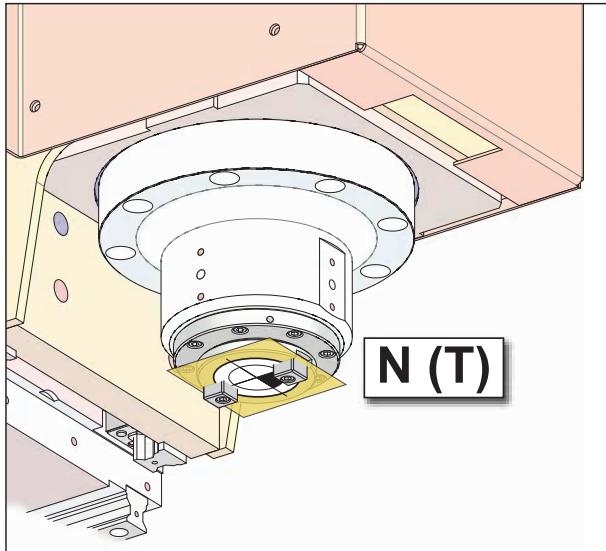
Y: External Input Devices

Easy2control On Screen operation.....	Y1
Scope of supply	Y1
Operating areas.....	Y2
Machine room camera.....	Y5
Installing the camera.....	Y5
Operating the camera.....	Y6

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

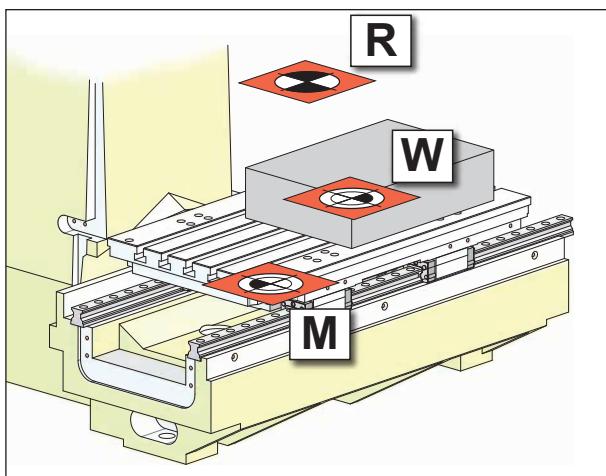
⊕ N (T) = Tool zero point

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

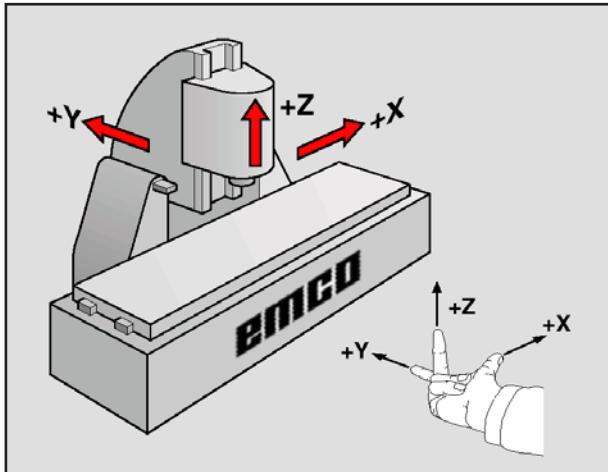
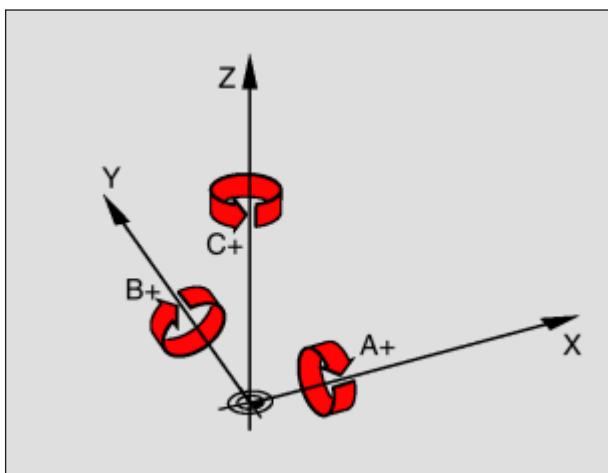
The workpiece zero point W can be freely programmed by the user. By programming a workpiece zero point, the origin of the coordinate system will be shifted from the machine zero point M to the workpiece zero point W.

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**Assignment of the rotating axes to the main axes*

Milling machine reference system

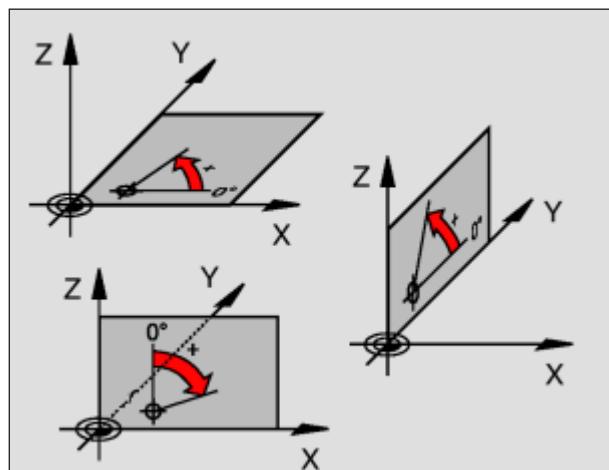
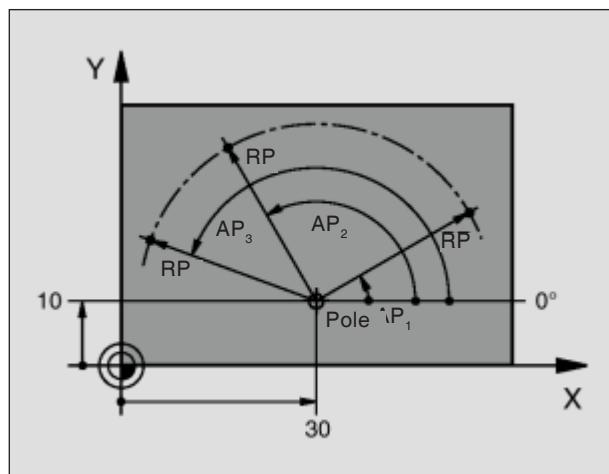
With a reference system, you define positions in a plane or in space unambiguously. Positional information always relates to a given point and is described by means of coordinates.

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.

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:

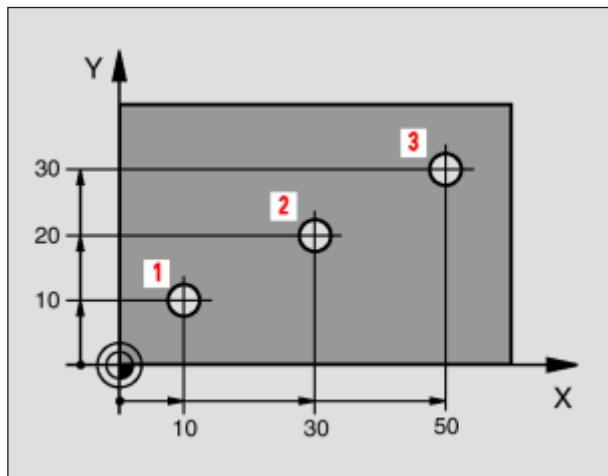
- Polar coordinates radius (RP): the distance from the pole to the position.
- Polar coordinates angle (AP): The angle between the angle reference axis and the path that connects the pole with the position.
(See picture above left)

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

Incremental workpiece positions

Incremental coordinates refer to the 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

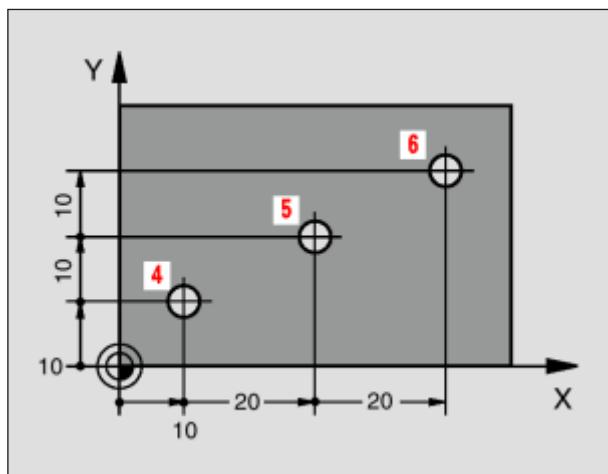
IX = 10 mm
IY = 10 mm

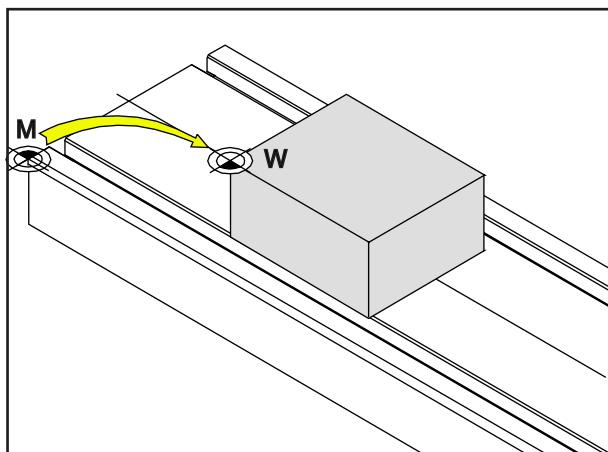
Drilling 5, related to 4

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.

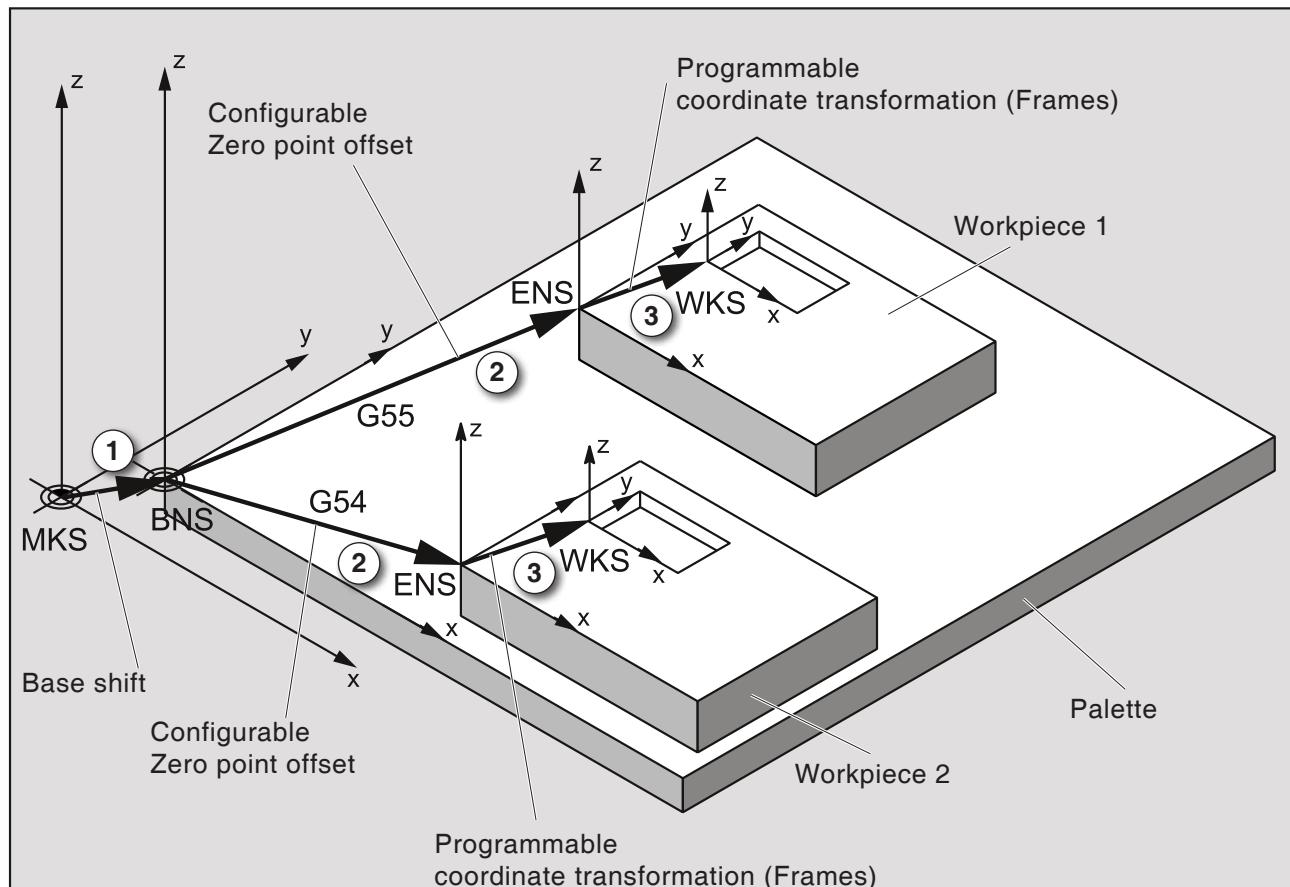
Programmable coordinate transformation (Frames)

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).

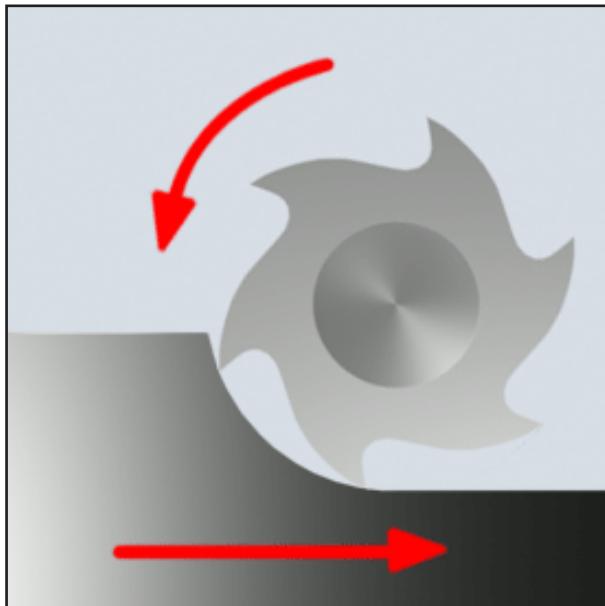
Machine zero point and workpiece zero point are usually not identical. The distance between the points is the entire zero point offset and is made up of various shifts:



- 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.

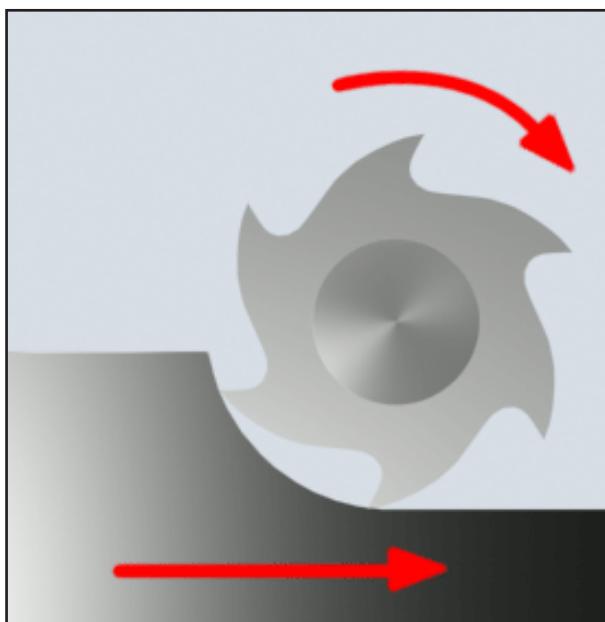
Down-cut milling is generally to be preferred if the machine permits it (backlash-free table drive in EMCO CNC machines).

Up-cut Milling

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.



Up-cut milling

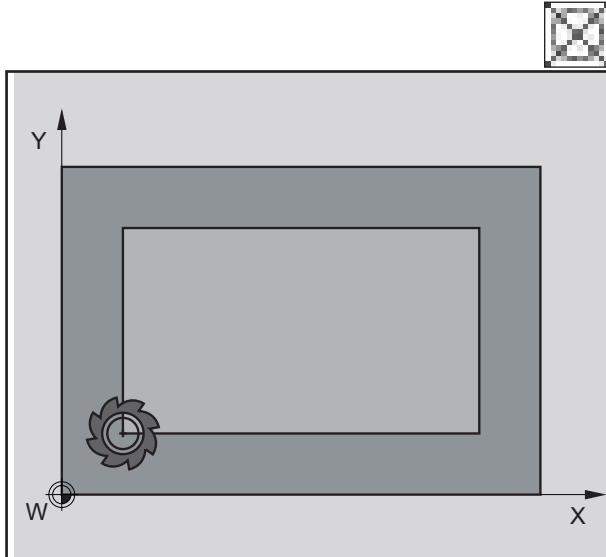
Down-cut/up-cut milling

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

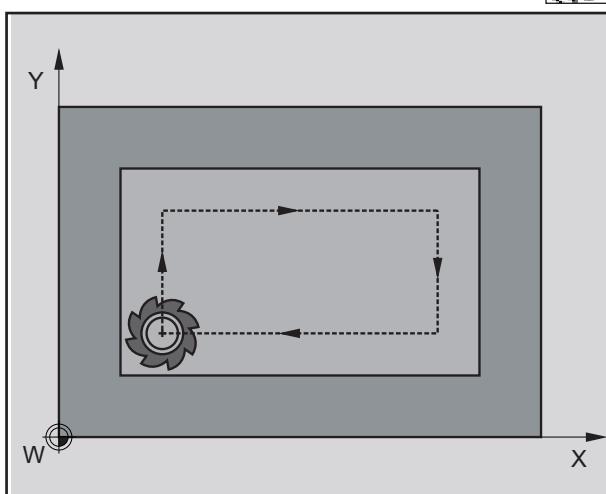
Tool radius compensation

Without tool radius compensation

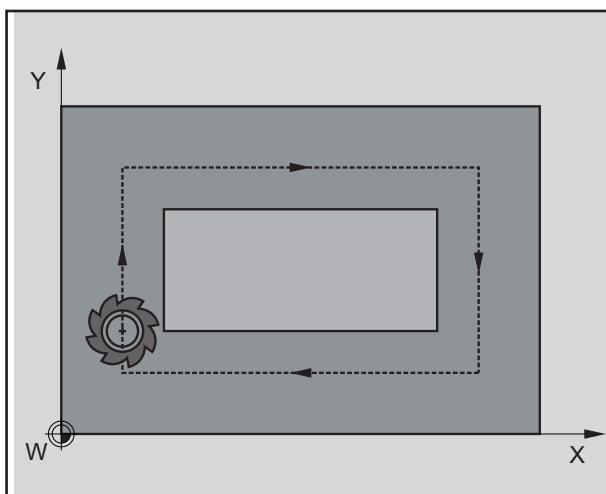
If tool radius compensation is switched off, the tool traces the contour's central track.



Without tool radius compensation



Right-hand tool radius compensation



Left-hand tool radius compensation



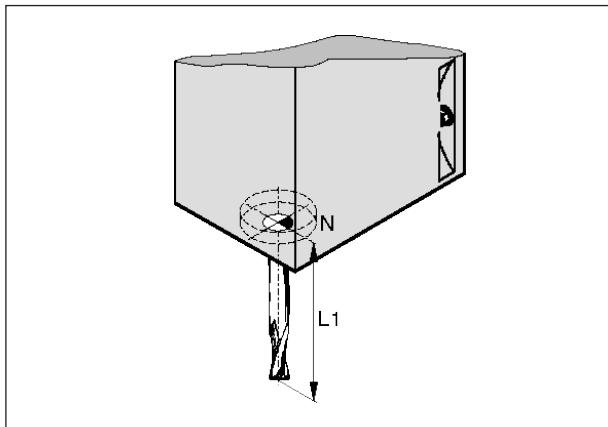
Right-hand tool radius compensation

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.

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.

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.

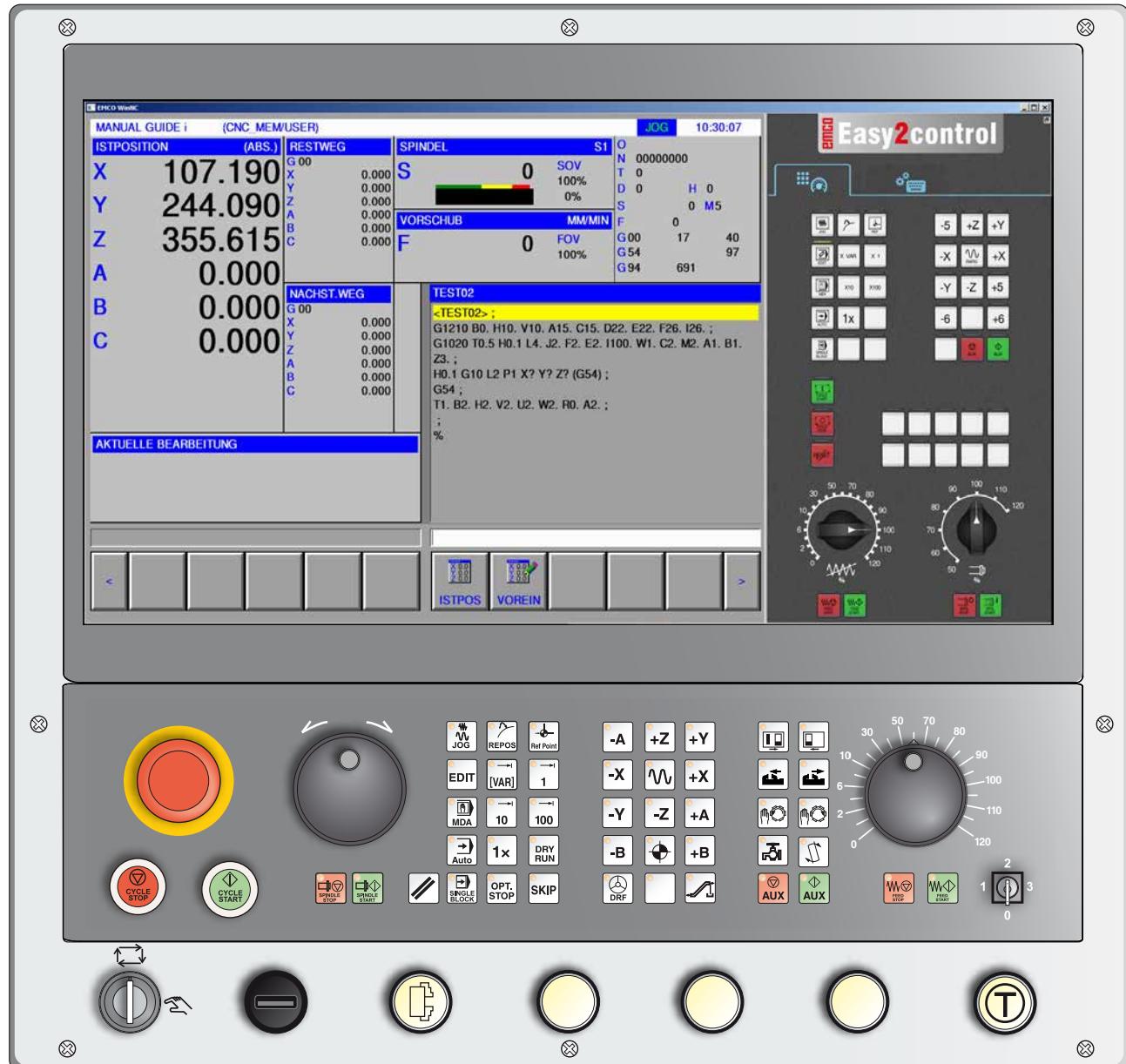
The measured lengths and mill radius can be stored in the tool list.

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:

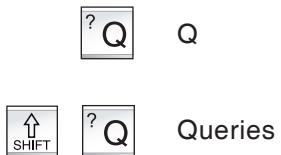
Depending on the machine you are operating with EMCO WinNC for Fanuc 31i, not all functions and machine keys may be available.

Address and number pad



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

Example:



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

A>

JOG	***	***	***	07:59:19	
1	2	3			
ABSOLUTE	RELATIVE	ALL			

Actual position

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



Release overview display

ACTUAL POSITION TEST BOHREN_G1101 N00000

		ABSOLUTE		F		0 MM/MIN	
X			0.000				
Y			44.090				
Z			55.615		VERSIONLIST		
A			0.000		name act. Version		
B			0.000		Version: SERIES 31I G41Z-07.0		

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

S 0 OV 100 LM 0 A>

JOG	***	***	***	08:03:29	
1	2	3			
ABSOLUTE	RELATIVE	ALL			
VERSION					

Release overview

Displays the current WinNC software release



Alarm and message display.

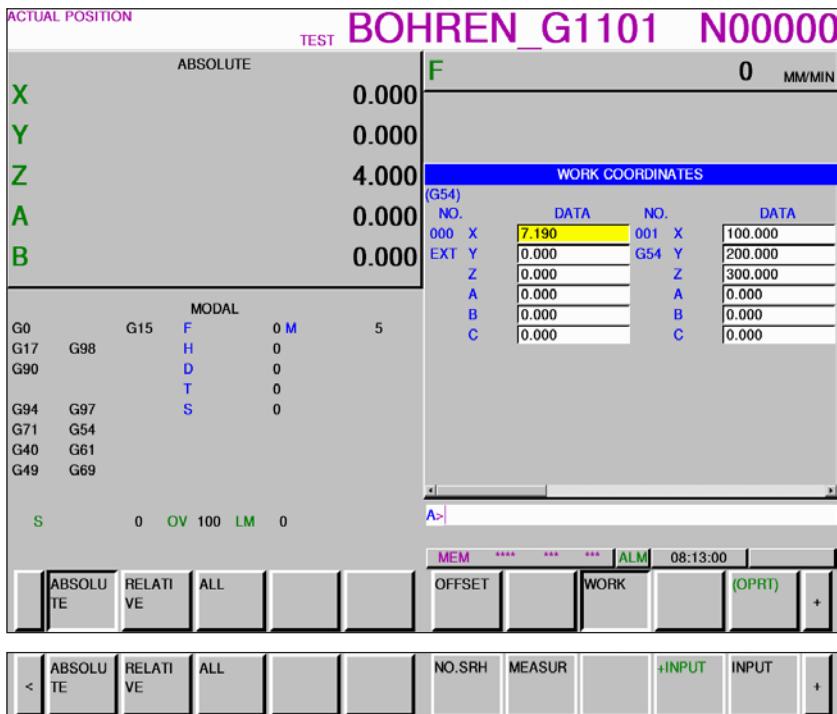


Alarm and report overview

Displays all alarms and reports

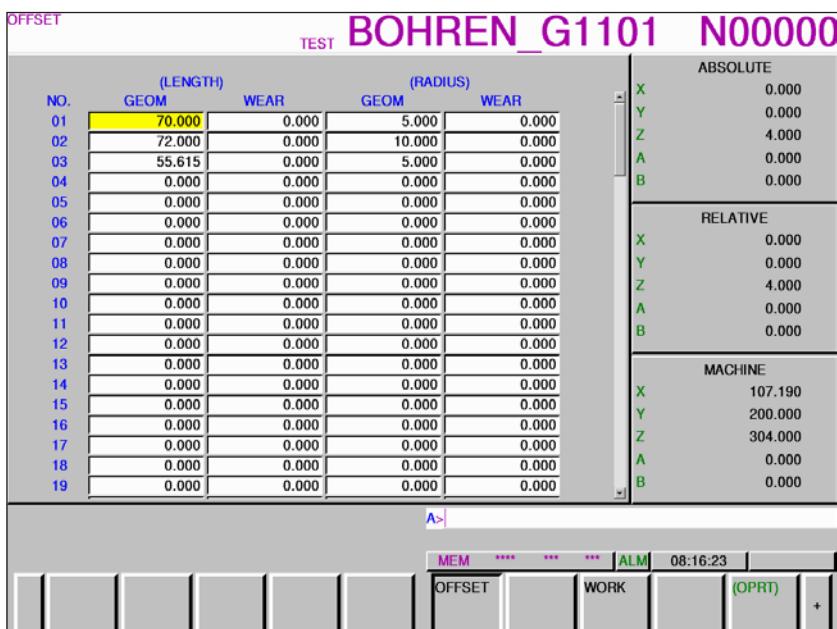


Zero point offset, tool correction and customer macro variables setting and display

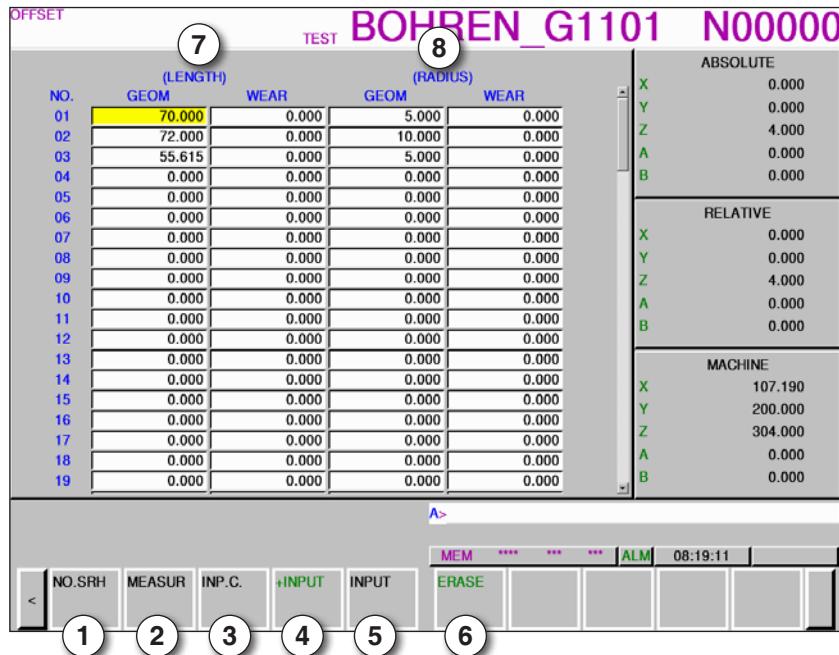


Zero point offset

- 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.

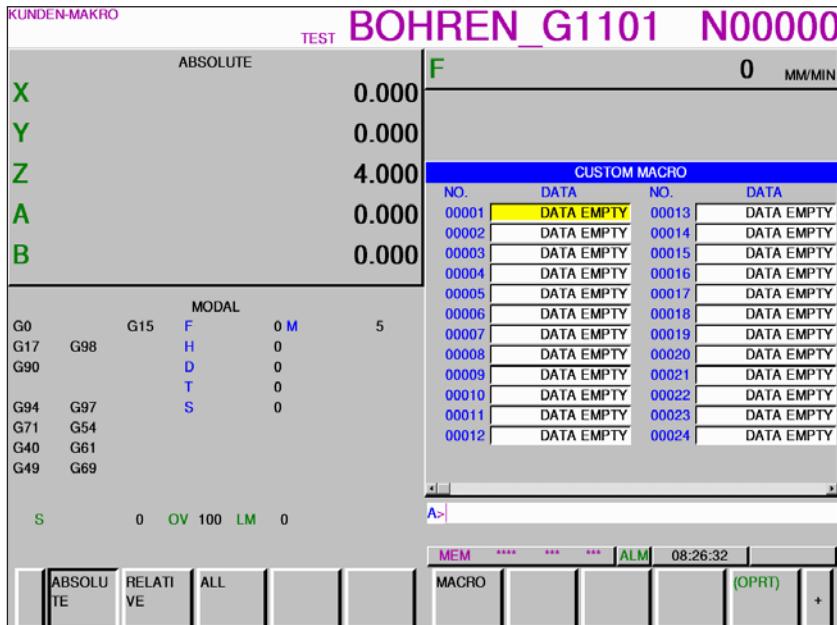


Tool correction



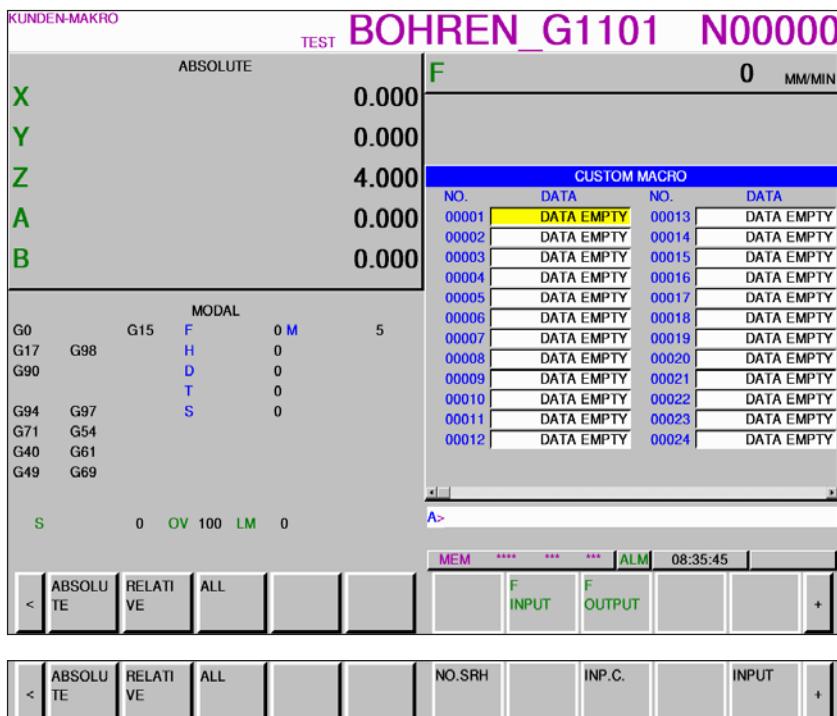
- 1 Search for tool number
- 2 Measure tool
- 3 Coordinates entry
- 4 Computes current value + entry from entry line
- 5 Accept value from entry line
- 6 Delete
- 7 Tool length data
- 8 Tool radius data

The "+" expansion button calls up the customer macro variables page

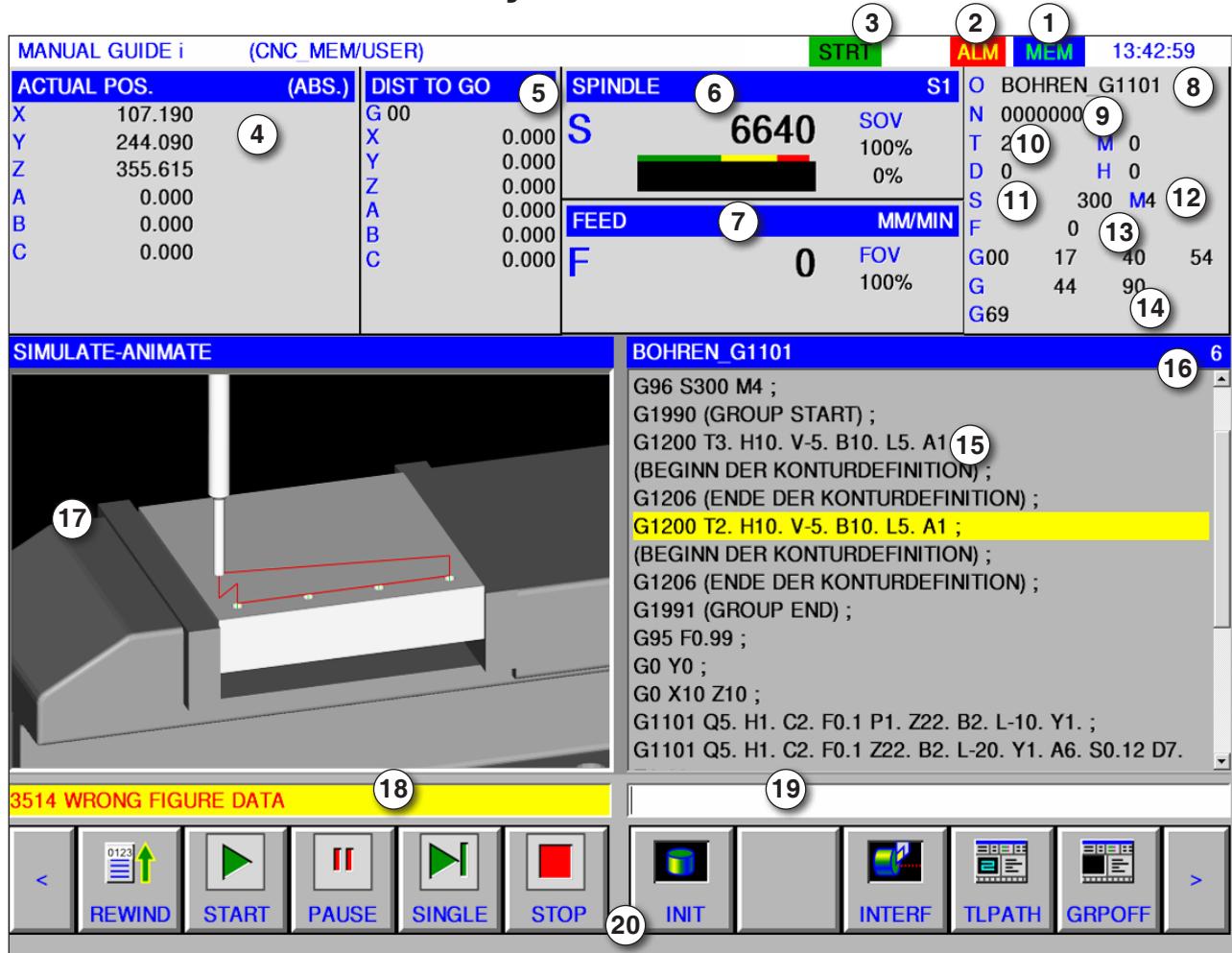


Customer macro variables

- 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 MAKRO.TXT.
- The path to save and read the data is set in EMConfig under the "Directory exchange" heading.



Screen layout Manual Guide i



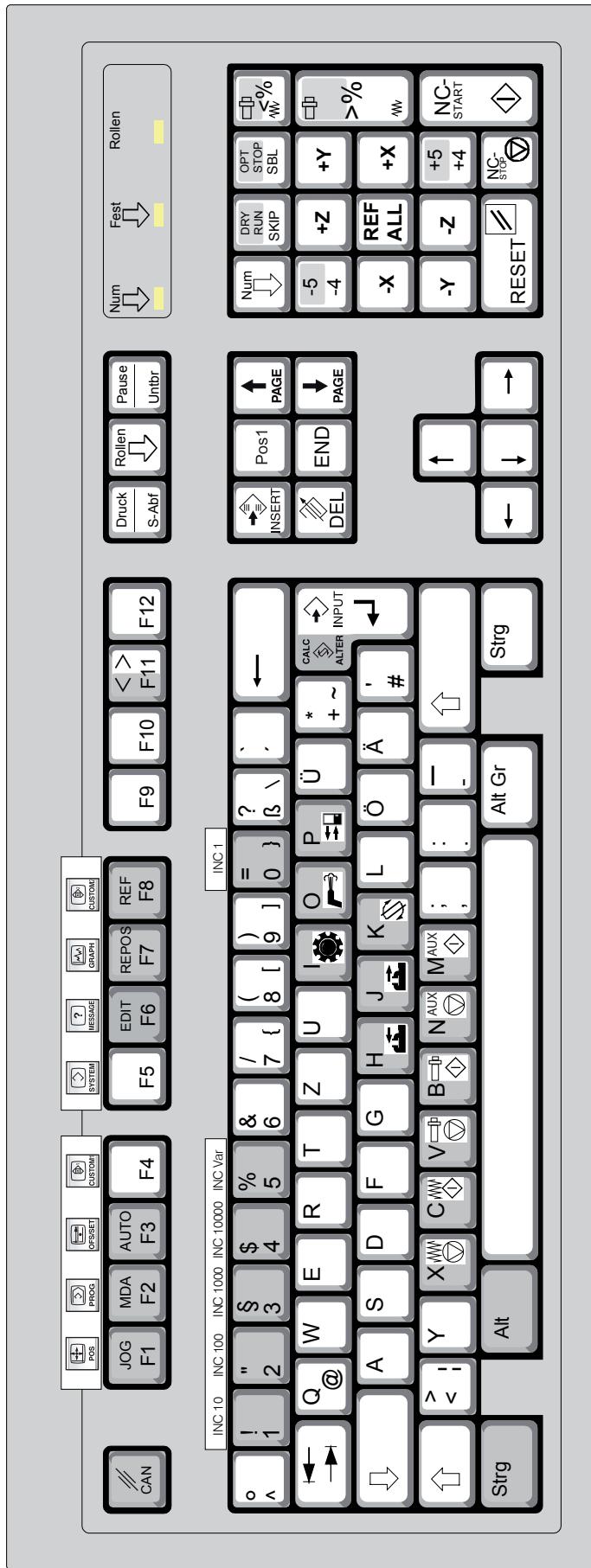
- | | |
|-------------------|--|
| 1 Mode | 11 Spindle speed |
| 2 Alarm status | 12 M-commands |
| 3 Program mode | 13 Feed display |
| 4 Axis position | 14 Display of active G-functions |
| 5 Residual travel | 15 Program window |
| 6 Spindle speed | 16 Current line number in ISO program window |
| 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 keypad 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

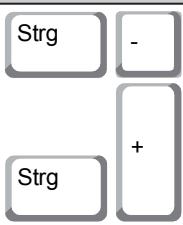
Overview Button Assignment Machine Operating Elements

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 OFF
Alt M		Switch auxiliary drive off AUX ON
Enter		NC start
,		NC stop
5		Approach reference point

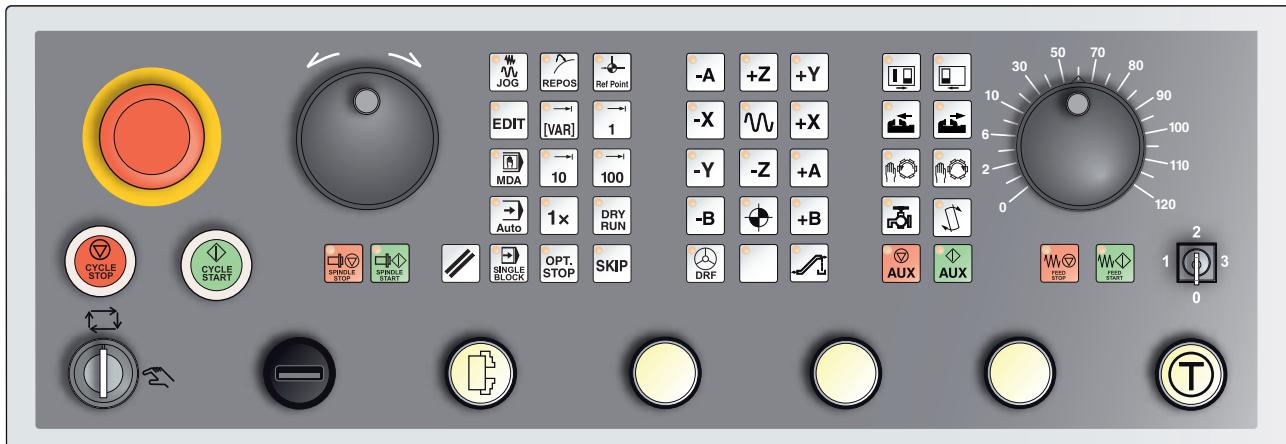
Instructions:

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 function 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



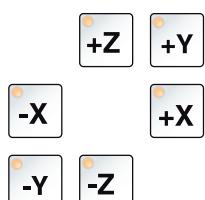
After pressing the Cycle stop button after the controller has taken over the function, machining of the current parts program is suspended

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.

Metric 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.

Metric 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.

Metric 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.

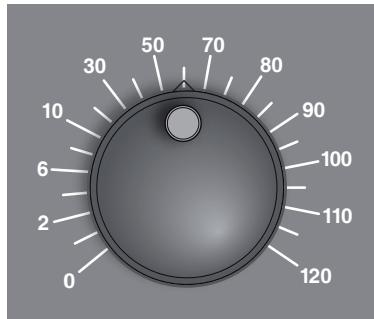
Freewheeling before referencing

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%).

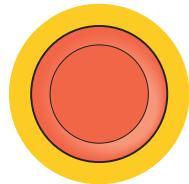
The set feed value F in % will be shown on the screen.

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

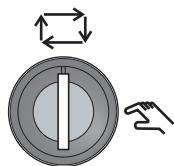
Press the red button only in an emergency.

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.

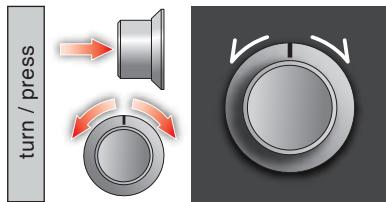
Keep the chip guard door closed even in Set-up mode.

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

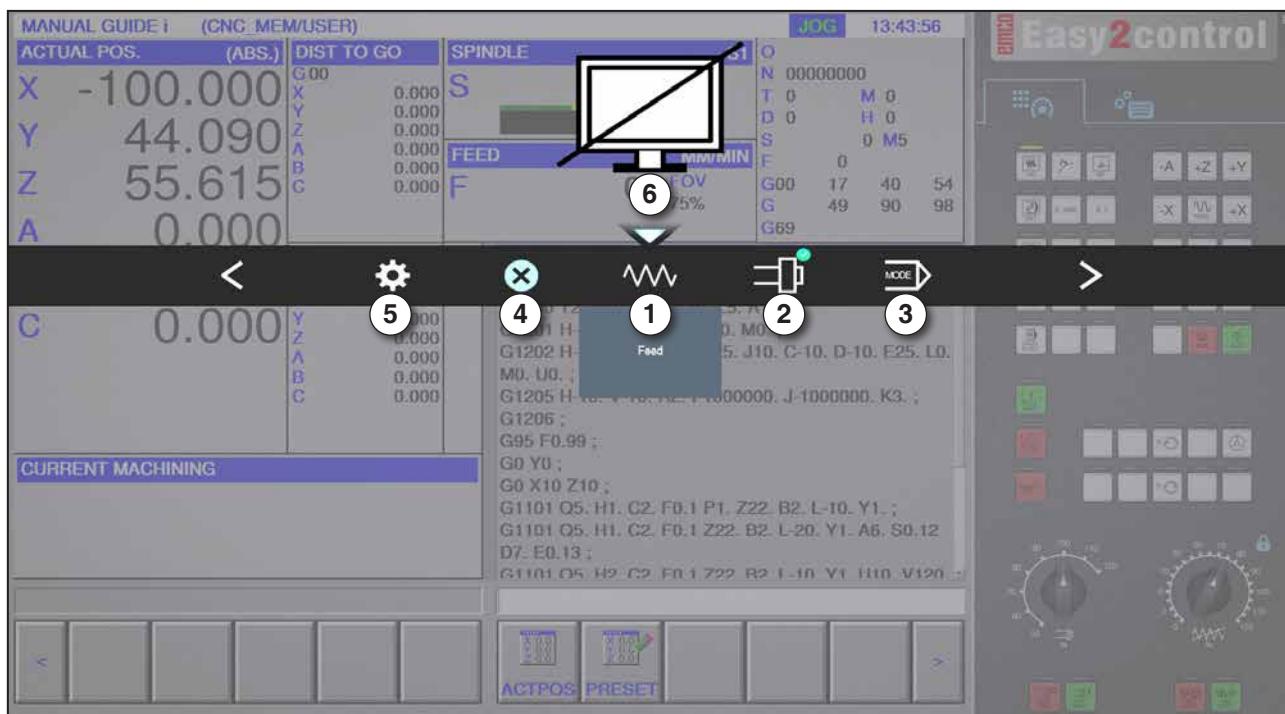


The multi-function switch is designed as a rotary switch with a press feature.

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

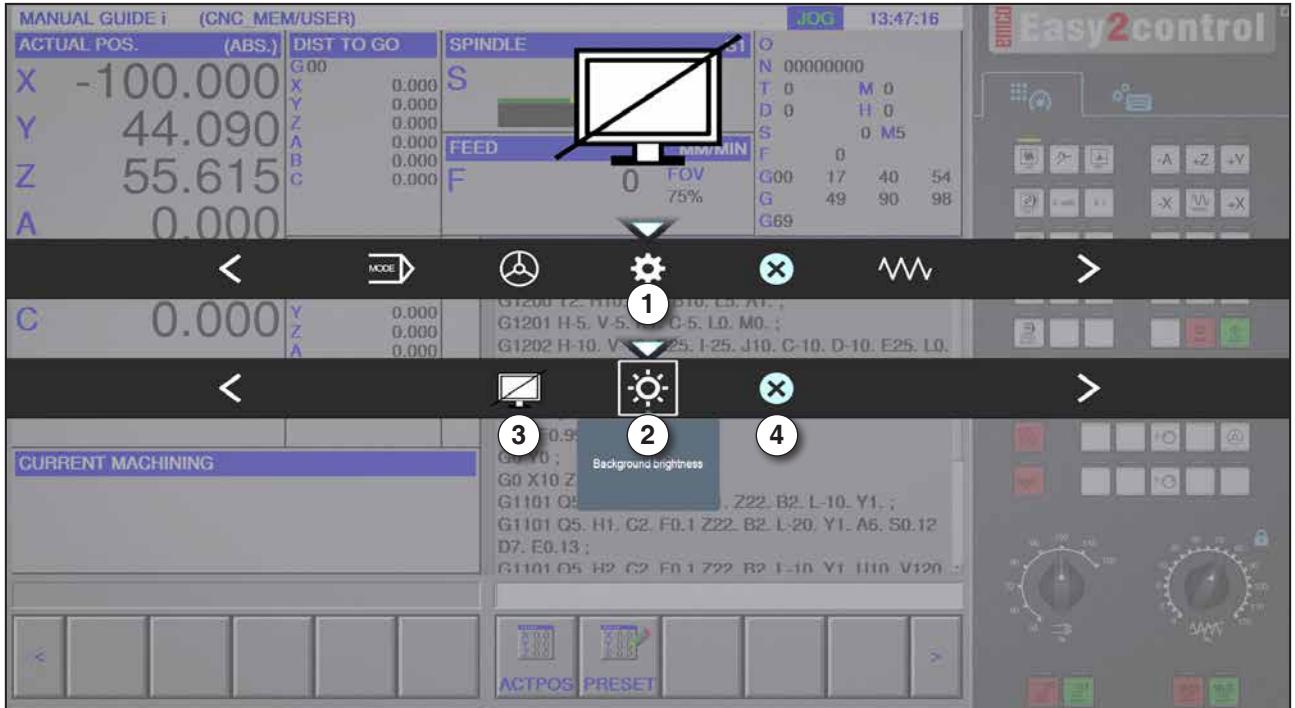
- 1 Feed override: controls the feed rate equivalent to conventional feed override
- 2 Spindle override: controls the spindle speed equivalent to conventional spindle override
- 3 Modes: allows you to select the operating mode using the multifunction operation

4 Close: The user interface is closed. The menu disappears, return to the control surface

- 5 Settings: opens another level with settings
- 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

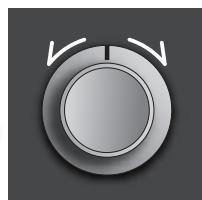
3 Lock screen: pressing again unlocks the screen.

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

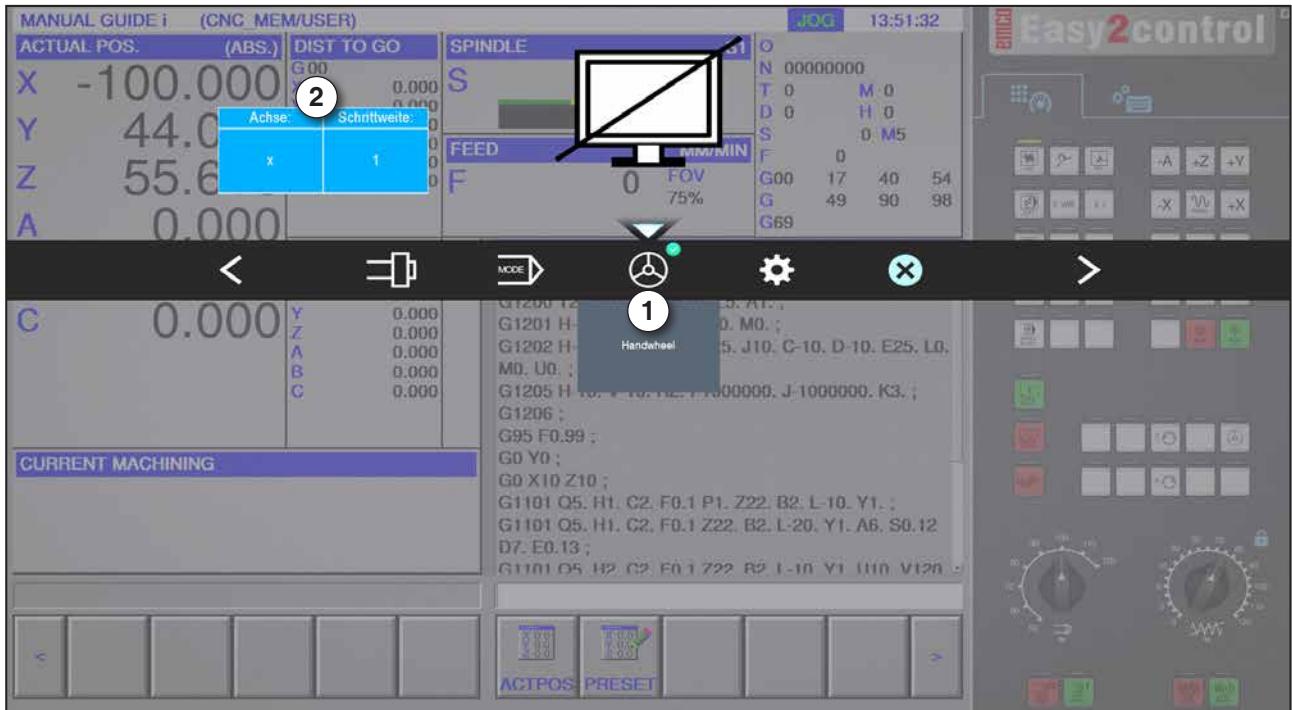
Setting the background transparency



- 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.

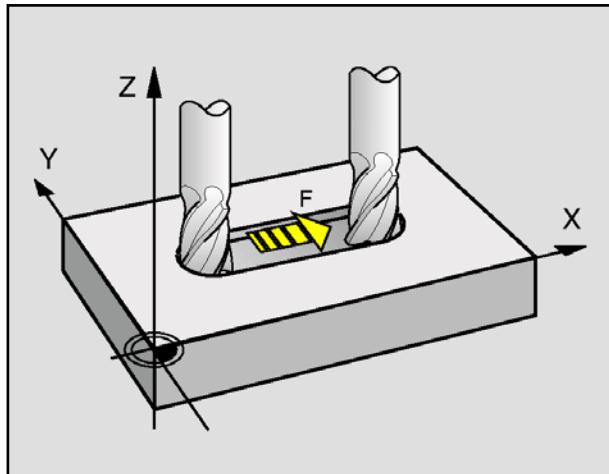


Enable button

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).

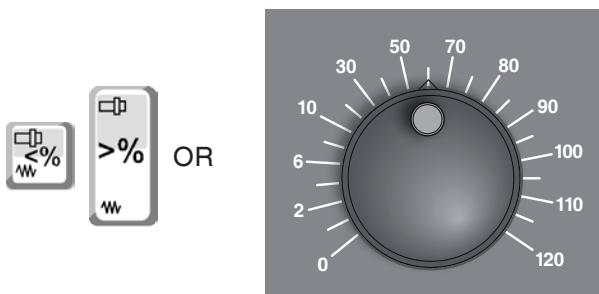
In machines with automated doors (option) pressing the enable switch opens the machine doors.

C: Operation



Feed F [mm/min]

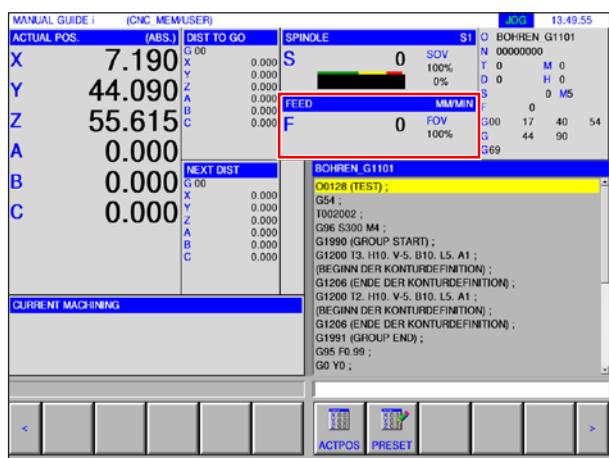
The feed F is the speed in mm/min (feet/min) with which the tool centre point moves on its path. The maximum feed can be different for each machine axis and is defined by machine parameters.



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 %.



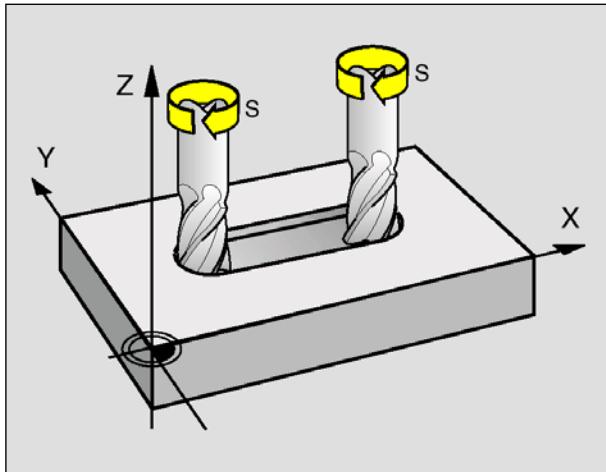
Adjustment range:

0% to 120% of the programmed feed.

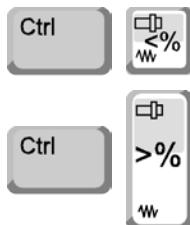
Only the changed percentage and not the resulting effective value will be shown.

In rapid traverse 100% of the maximum rapid traverse feed will not be exceeded.

Spindle speed S [rev/min]



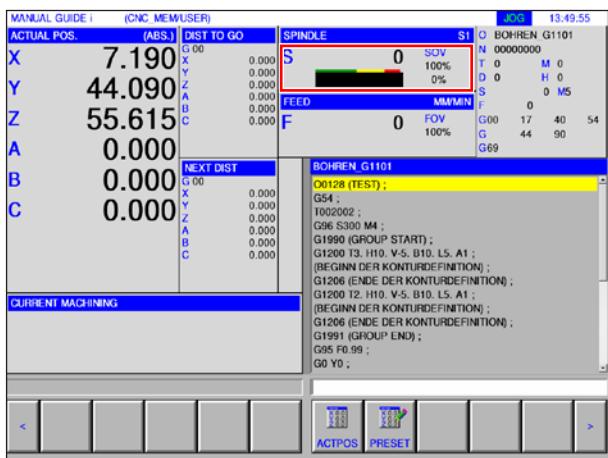
Indicate the spindle speed S in revolutions per minute (rpm).



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 1µm
Imperial (inch-based) measurement system: Inc 1 equals 0.1 µinch

Inc 10 - Incremental feed



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

Metrical measurement system: Inc 10 equals 10µm
Imperial (inch-based) measurement system: Inc 10 equals 1 µinch

Inc 100 - Incremental feed



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

Metrical measurement system: Inc 100 equals 100µm
Imperial (inch-based) measurement system: Inc 100 equals 10 µinch

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 1000µm
Imperial (inch-based) measurement system: Inc 1000 equals 100 µinch

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.

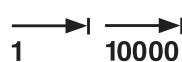


- If the button is pressed simultaneously, the carriages move in rapid traverse.

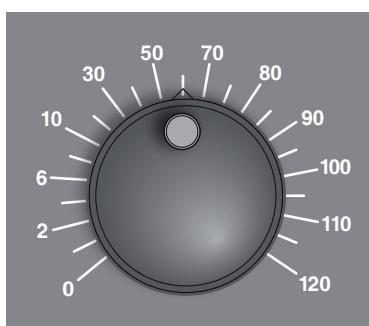
Move slides in step-movement

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.

MANUAL GUIDE i (CNC_MEM/USER)

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

FEED MM/MIN

F 0 FOV 100%	G 44 90
	G 69

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

Buttons 1-10:

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

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

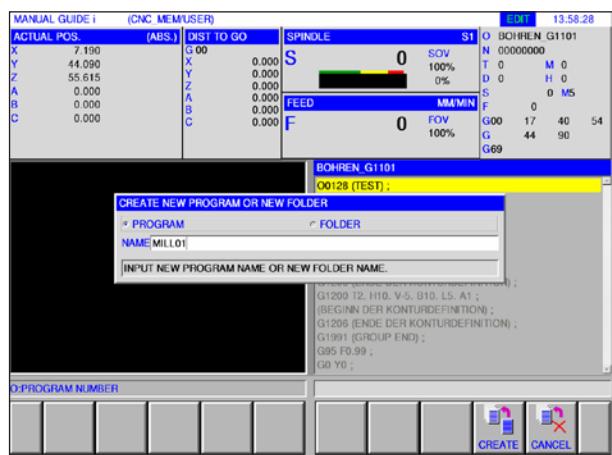
1 Select "Edit" mode.



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.

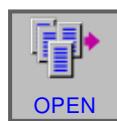
Instructions:

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)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	EDIT 14:07:18
X 7.190		G 00	0.000	O BOHREN_G1101	
Y 44.090		X	0.000	N 00000000	
Z		V		T 0 M 0	

PROGRAM LIST (//CNC_MEM/USER/LIBRARY/)

NAME	COMMENT	MODIFIED DATE	SIZE(CHAR)
RETURN UPPER FOLDER		<FOLDER>	10
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>	
SCHEIBEN_AUSSEN		<FOLDER>	

SELECT PROGRAM NAME AND SELECT SOFT KEY.

O:PROGRAM NUMBER

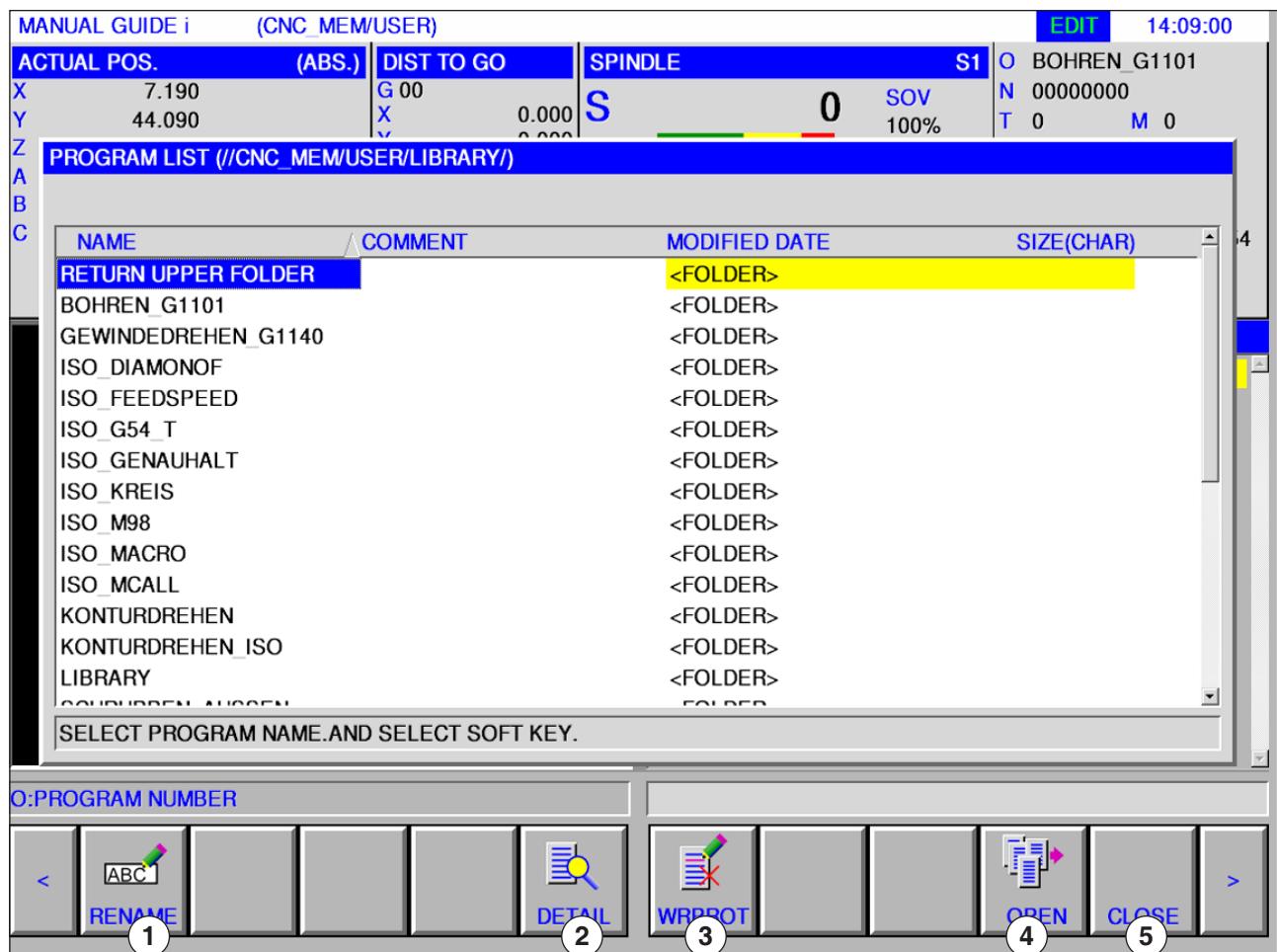
< 11	NEW 1	COPY 2	DELETE 3	EDTCOM 4	SEARCH 5	M CARD 6	MLTDEL 7	SRTORD 8	OPEN 9	CLOSE 10	> 11
------	-------	--------	----------	----------	----------	----------	----------	----------	--------	----------	------

- 1 Creating a new machining program
- 2 Copy programs
- 3 Delete programs
- 4 Enter a comment for a program
- 5 Search for programs
- 6 Output of a program to an external memory card
- 7 Delete a number of programs
- 8 Change the sorting sequence
- 9 Open program
- 10 End program listing
- 11 Forward and Back (extension keys)

Instructions:

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





- 1 Rename program or folder names
- 2 Program properties
- 3 Program protection
- 4 Open program
- 5 End program listing

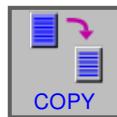
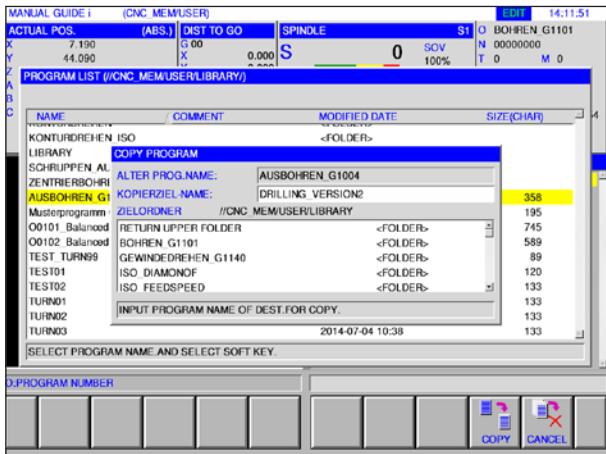
Copy program

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.



5 Press the softkey to copy.

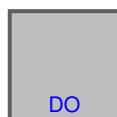
Delete program

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.

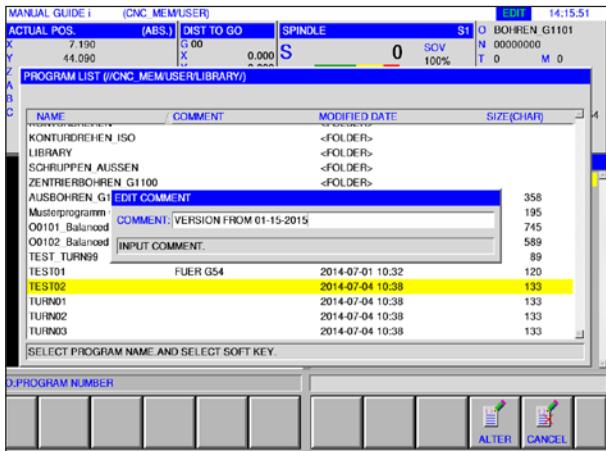
Enter comment

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.



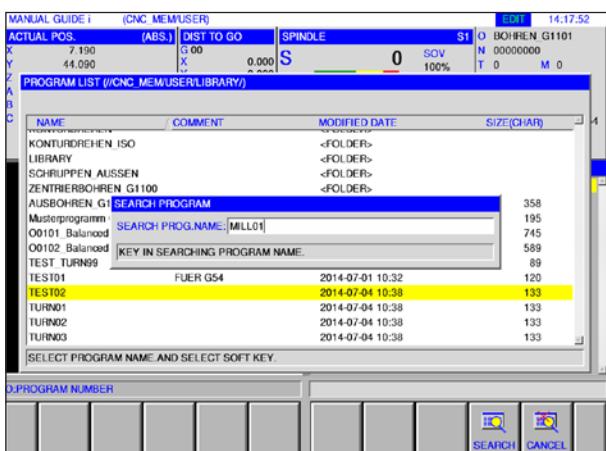
5 Press the softkey to change.

Search program

1 Select "Edit" mode.

2 Search program.

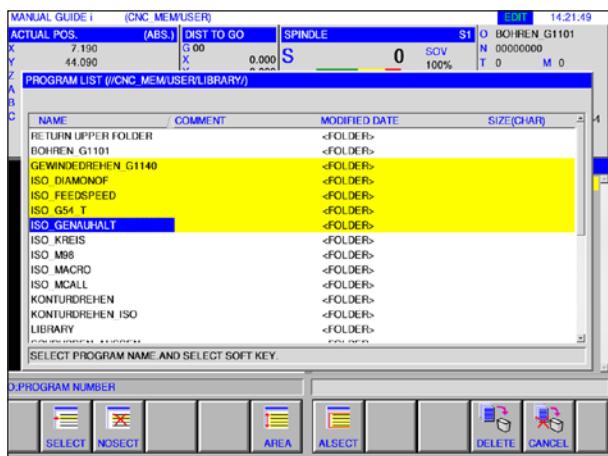
3 This softkey calls the window for searching programs. After a program name has been entered for the program to be found (target program name), press the "SEARCH" softkey to look for the given program.



4 Press the softkey to search.

Delete several 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.



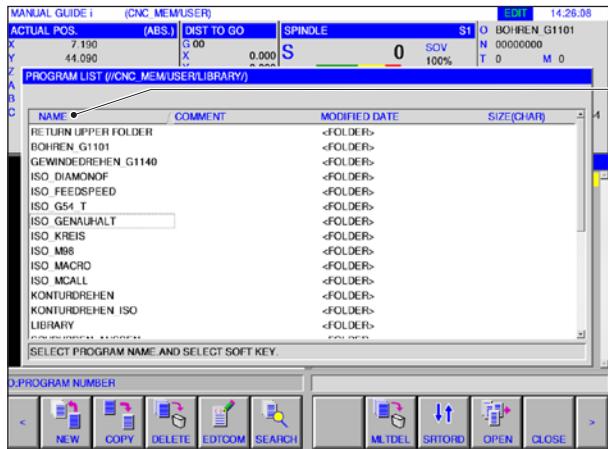
7 This softkey selects all programs. Pressing the softkey deletes all programs in the relevant folder.



8 Press the softkey to delete.

Change the sorting sequence

1 Select "Edit" mode.



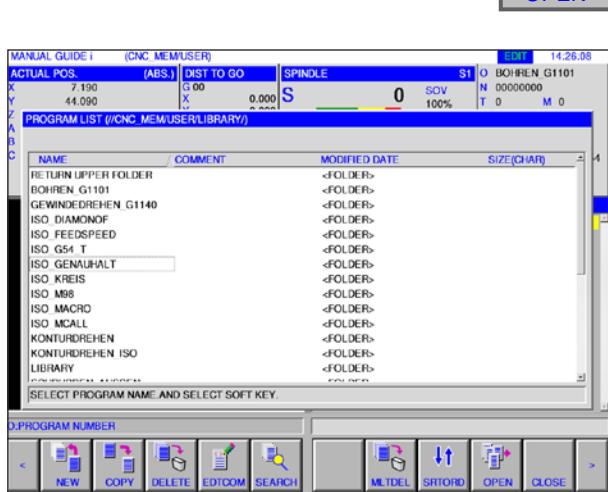
2 Change the sorting sequence.

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.

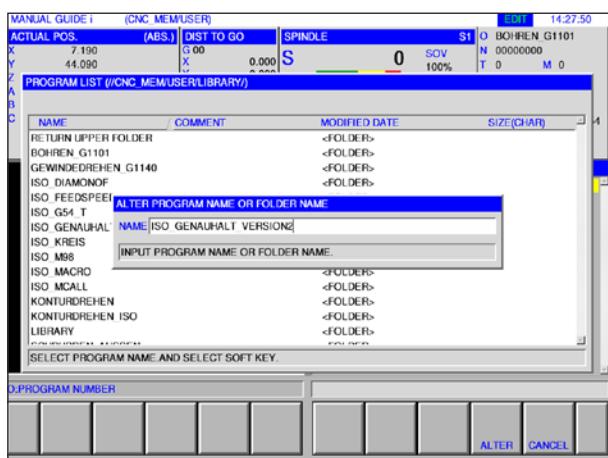


2 Open program.

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.



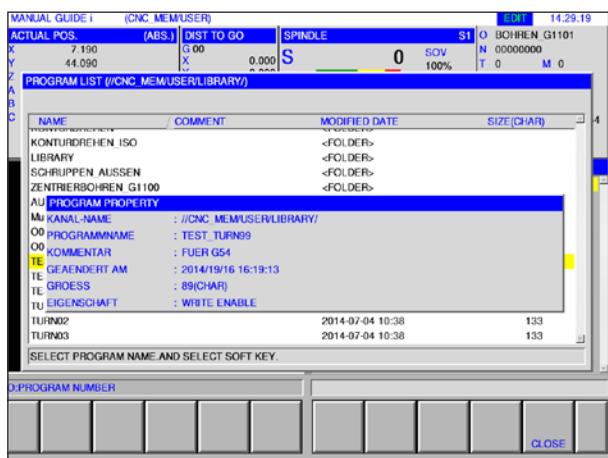
- 5 Change the program or folder name.



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.

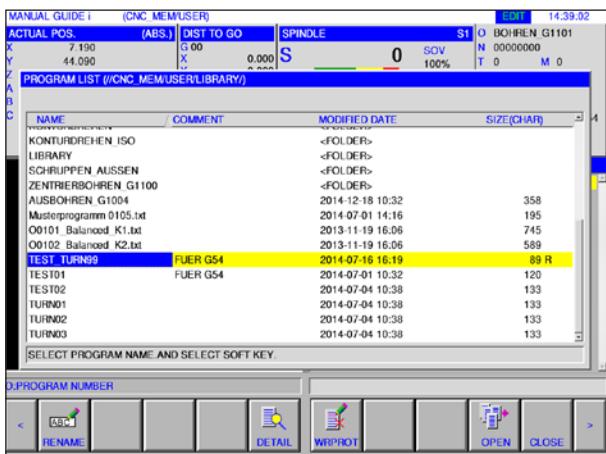
Program protection

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.



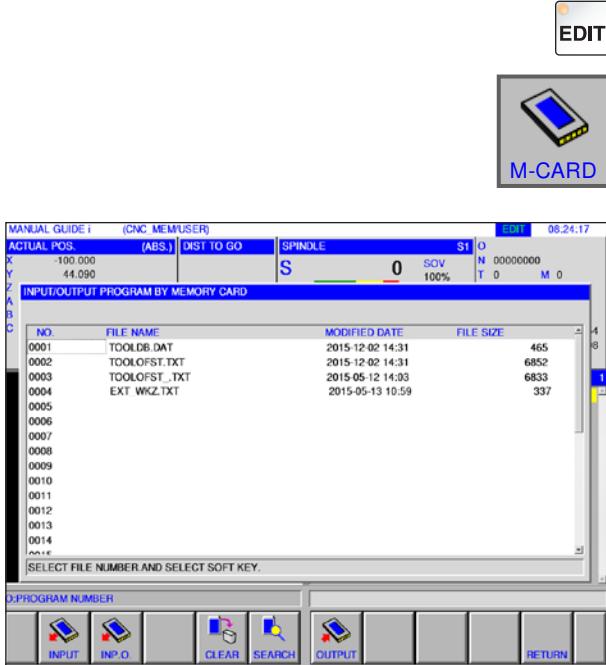
Program input and output on a memory card

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

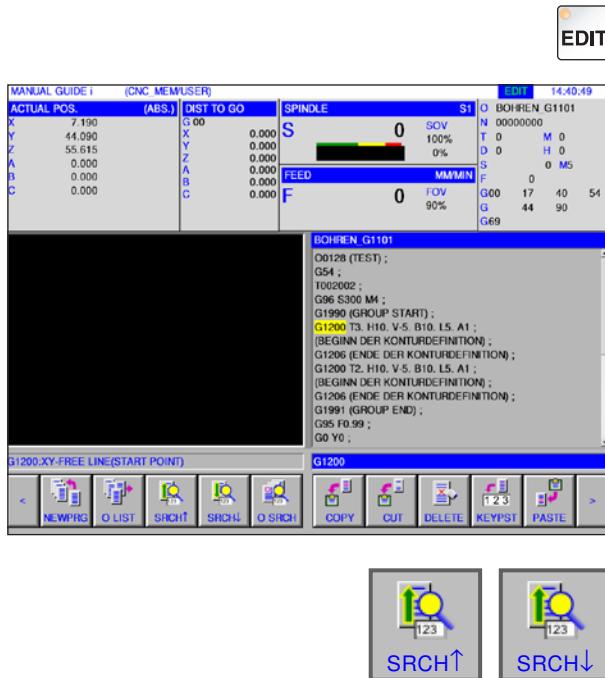


Note:

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

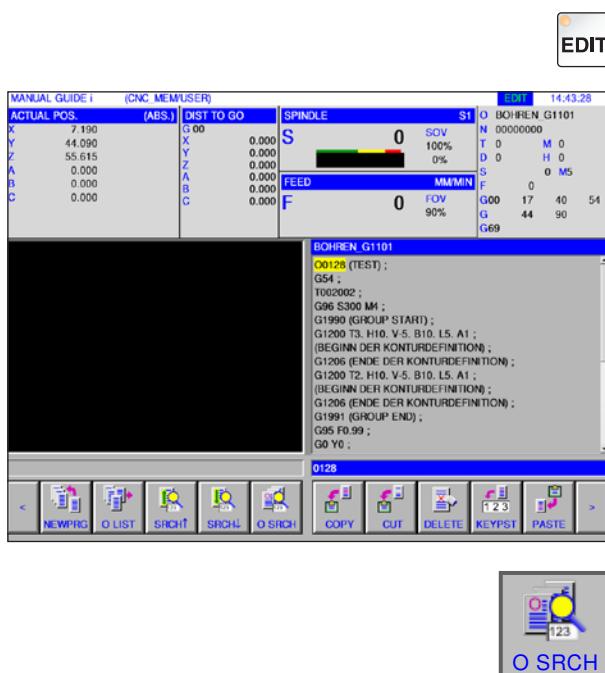


1 Select "Edit" mode.

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



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.

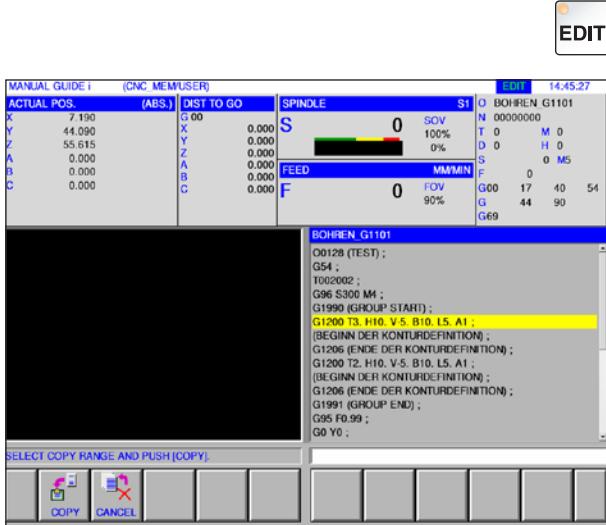
Instructions:

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

Copy highlighted text to clipboard

1 Select "Edit" mode.

2 Set the area for copying with the cursor keys (the marked area will be yellow).



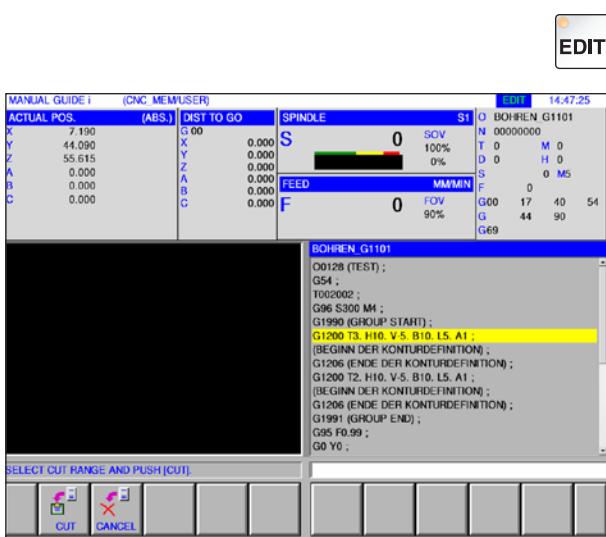
3 Press the "COPY" softkey to save the text in the Clipboard.

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

Move highlighted text to clipboard

1 Select "Edit" mode.

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



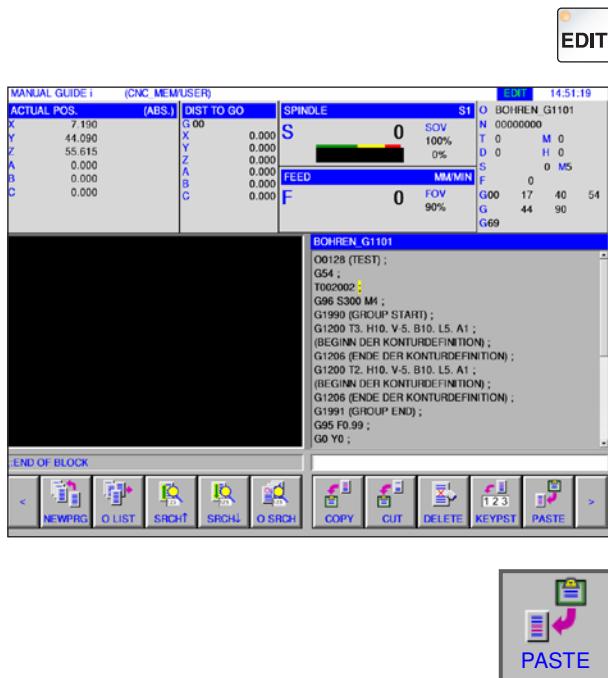
3 Press the "CUT" softkey to move the text from the program into the Clipboard.

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.

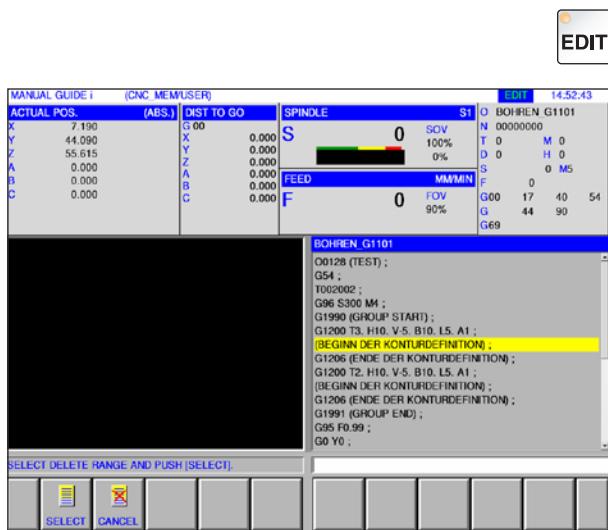


3 This softkey inserts the content of the clipboard at the point after the cursor.

Delete highlighted text

1 Select "Edit" mode.

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



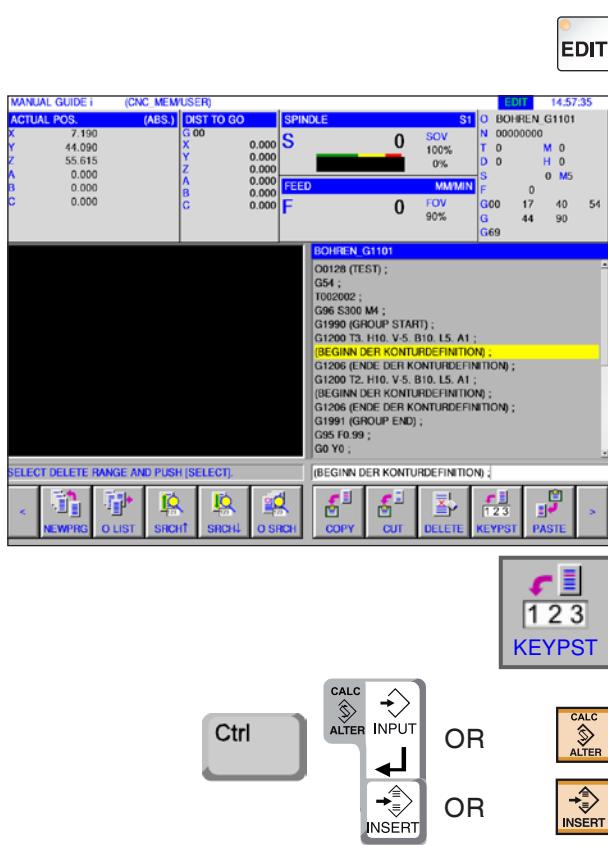
3 This softkey calls the window for deleting a number of programs.



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.

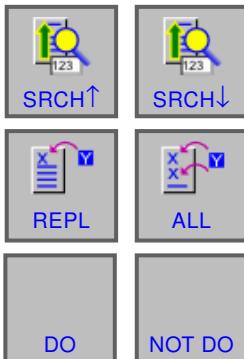
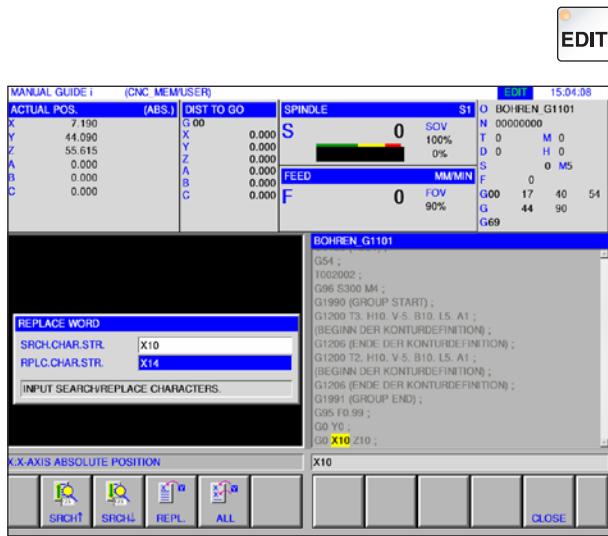


2 Press the "UNDO" softkey to revert to an editing process.

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.

MANUAL GUIDE i (CNC_MEM/USER)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	MEM 15:12:58
X	7.190	G 00 X 0.000 Y 0.000 Z 0.000 A 0.000 B 0.000 C 0.000	S 0 SOV 100% 0%	O BOHREN_G1101 N 00000000 T 0 M 0 D 0 H 0 S 0 M5 F 0 G00 17 40 54 G 44 90 G69	
Y	44.090		FEED F 0 FOV 90%		
Z	55.615				
A	0.000				
B	0.000	NEXT DIST G 00 X 0.000 Y 0.000 Z 0.000 A 0.000 B 0.000 C 0.000	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 ;		
C	0.000				

CURRENT MACHINING

O:PROGRAM NUMBER

< 11	SETTOP 1	O LIST 2	BGEDIT 3	N SRCH 4	O SRCH 5	ACTPOS 6	PRESET 7			SIMLAT 8	> 11
------	----------	----------	----------	----------	----------	----------	----------	--	--	----------	------

<					WK SET 9	T-OFS 10				SETING 12	>
---	--	--	--	--	----------	----------	--	--	--	-----------	---

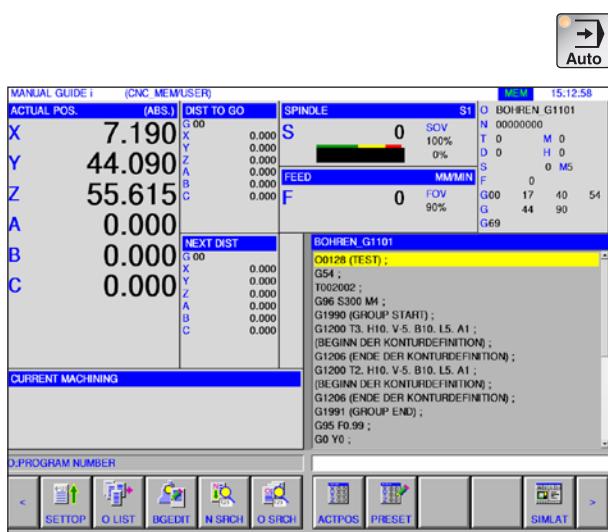
- 1 Go to the start of the selected program
- 2 End program listing
- 3 Background edit
- 4 Search for block number
- 5 Search for and open the program
- 6 Display Actual position ABSOLUTE, RELATIVE and MACHINE
- 7 Preset relative position
- 8 Simulation
- 9 Workpiece coordinates system
- 10 Tool settings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

Instructions:

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

Background Editing

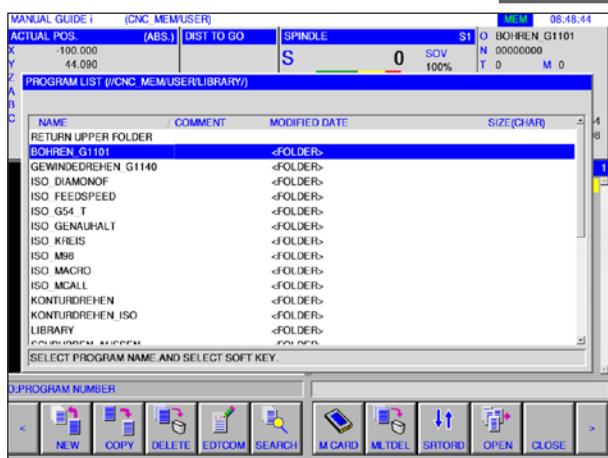
1 Select "Auto" mode.



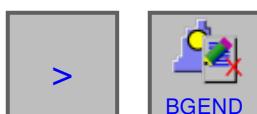
2 Press "BGEDIT" softkey.

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.



Semi-automatic mode



Select "MDA" mode.

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

MANUAL GUIDE i (CNC MEM/USER)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	MDI	15:22:33
X	7.190	G00 X 0.000 Y 0.000 Z 0.000 A 0.000 B 0.000 C 0.000	S 0 SOV 100% 0%		O 00000 N 00000000 T 0 M 0 D 0 H 0 S 0 M5 F 0 G00 17 40 54 G 44 90 G69	
Y	44.090		FEED MM/MIN	F 0 FOV 90%		
Z	55.615					
A	0.000					
B	0.000	NEXT DIST G00 X 0.000 Y 0.000 Z 0.000 A 0.000 B 0.000 C 0.000		O0000(MDI PROGRAM) O0000 ; %		
C	0.000					

CURRENT MACHINING

O:PROGRAM NUMBER

<	0123 ↑ REWIND 1	SRCH↑ 2	SRCH↓ 3	COPY 4	CLT 5	DELETE 6	KEXPST 7	PASTE 14	>
14				3	4	5	6	7	14

<	UNDO 8	REDO 9	WK SET 10	T-QFS 11	SETING 12	>
			9	10	11	

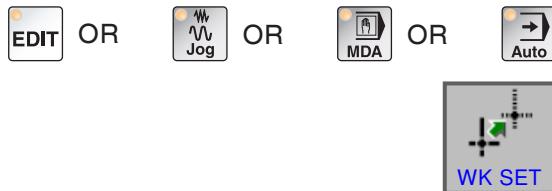
<	0123 ↑ REWIND 1	ACTPOS 12	PRESET 13	>
1		12	13	

**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.

MANUAL GUIDE i (CNC, MEM/USER)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	SI	M01	15:32:34
X	7.190	G00 X 0.000	S 0	SOV 100%	O 00000	
Y	44.090	Y 0.000	Z 0.000	0% T 0 M 0	N 000000000	
Z	55.615	Z 0.000	A 0.000	F 0 H 0	D 0 S 0 M 5	
	0.000		B 0.000	FEED MM/MIN F 0	G 000 17 40 54	
			C 0.000	G 44 90	G 44 90	

WORK COORDINATE SYSTEM

WORKPIECE ORIGIN

NO.	WRK CO	X	Y	Z	A	B
00	EXT	0.000	0.000	0.000	0.000	0.000
01	G54	100.000	200.000	300.000	0.000	0.000
02	G55	0.000	0.000	0.000	0.000	0.000
03	G56	0.000	0.000	0.000	0.000	0.000
04	G57	0.000	0.000	0.000	0.000	0.000
05	G58	0.000	0.000	0.000	0.000	0.000
06	G59	0.000	0.000	0.000	0.000	0.000

KEY IN NUMERALS.

O:PROGRAM NUMBER

	ACTPOS	MEASURE	+INPUT	NEXT	TAB →		NO.SRH	CLOSE
--	--------	---------	--------	------	-------	--	--------	-------

Workpiece zero point offset

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.

Measuring



MANUAL GUIDE i (CNC MEM/USER)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	MOT 15:34:02
X	7.190	G00 X 0.000	S 0 SOV 100% 0%	O 00000 N 00000000 T 0 M 0 D 0 H 0 S 0 M5 F 0 G00 17 40 54 G 44 90	
Y	44.090	Y 0.000			
Z	55.615	Z 0.000			
-	-	A 0.000 B 0.000 C 0.000	FEED MM/MIN F 0 FOV 90%		
WORK COORDINATE SYSTEM					
WORKPIECE ORIGIN					
NO.	WRK CO	X	Y	Z	A B
00	EXT	0.000	0.000	0.000	0.000 0.000
01	G54	CALCULATE OFFSET(MEASURE)			
02	G55				
03	G56	MACHINE COORD.	WORK COORD.		
04	G57	CURRENT	TARGET	RESULT	
05	G58	X 7.190 -X		7.190	0.000 0.000
06	G59	INPUT TARGET ON WORK COORD.			
KEY IN NUMERALS.					
O:PROGRAM NUMBER					

1 2

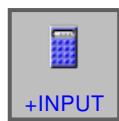


- Conclude measurement with "INPUT".

"MESURE" performs subsequent calculations.

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

Compute



MANUAL GUIDE i (CNC MEM/USER)

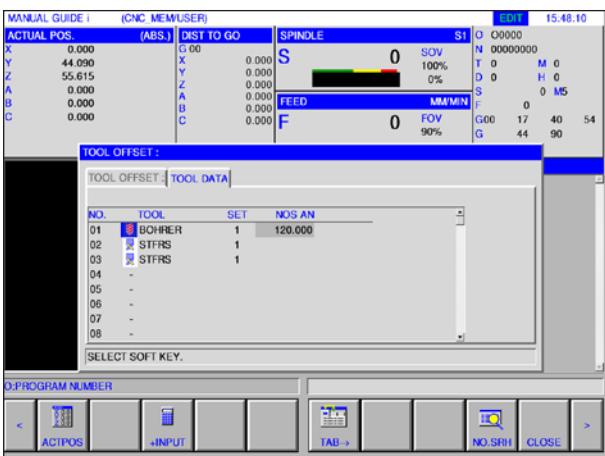
ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	MOT 15:38:55
X	0.000	G00 X 0.000	S 0 SOV 100% 0%	O 00000 N 00000000 T 0 M 0 D 0 H 0 S 0 M5 F 0 G00 17 40 54 G 44 90	
Y	44.090	Y 0.000			
Z	55.615	Z 0.000			
-	-	A 0.000 B 0.000 C 0.000	FEED MM/MIN F 0 FOV 90%		
WORK COORDINATE SYSTEM					
WORKPIECE ORIGIN					
NO.	WRK CO	X	Y	Z	A B
00	EXT	7.190	0.000	0.000	0.000 0.000
01	G54	CALCULATE OFFSET(+INPUT)			
02	G55				
03	G56	CURRENT	OFFSET	RESULT	
04	G57	X 7.190	+X	-X 7.190	0.000 0.000
05	G58	INPUT OFFSET.			
06	G59				
KEY IN NUMERALS.					
O:PROGRAM NUMBER					

1 2



- End computation with "INPUT".

Current value (1) + offset (2).

*Raw part definition**Tool definition*

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.

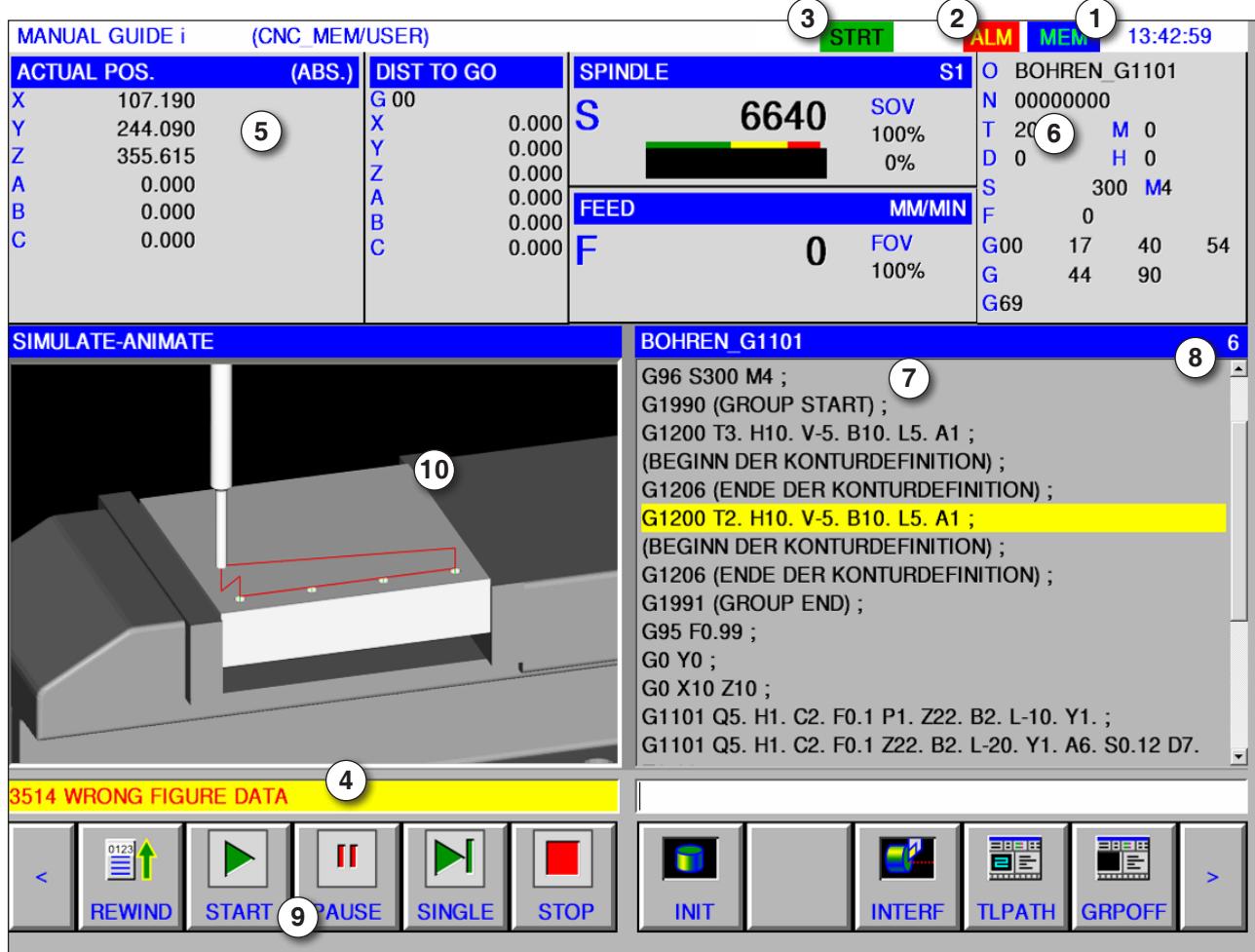
There is a precise description of the unmachined parts in Chapter D. Programming Manual Guide i" and the tool definition in chapter "F: Tool Management".

Instructions:

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



Screen layout graphical simulation



- 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-IMATE screen.

Softkey functions

To start the simulation



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



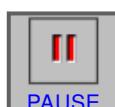
Start simulation

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.



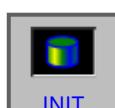
Stop simulation

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



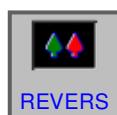
LARGE



SMALL



AUTO



REVERS



←MOVE



→MOVE



↑MOVE



↓MOVE



CENTER

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.

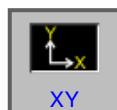
The "↓ MOVE" softkey moves the viewing point down. Accordingly, the marked tool path is moved up.

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.



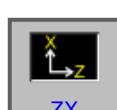
This softkey selects the ZY plane.



This softkey selects the YZ plane.



This softkey selects the ZX plane.



This softkey selects the ZY plane.



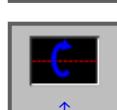
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis.



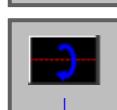
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis. The viewing point is opposite to that mentioned above.



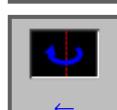
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis.



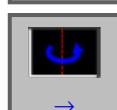
This softkey executes a counterclockwise rotation with the screen as the central axis.



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.

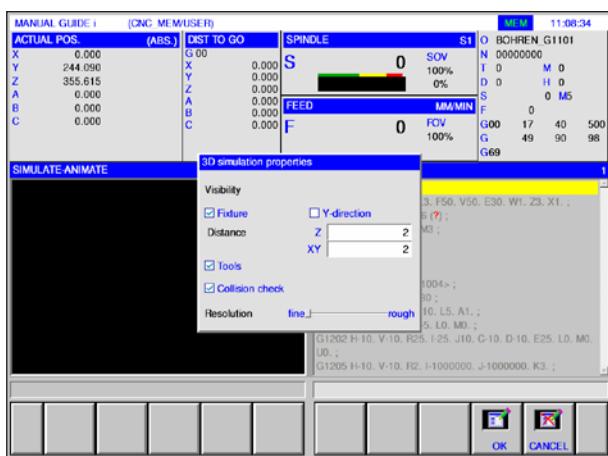


This softkey selects the simulation playback speed. There are 5 steps available for the speed.



3D configuration

This softkey starts 3D configuration.



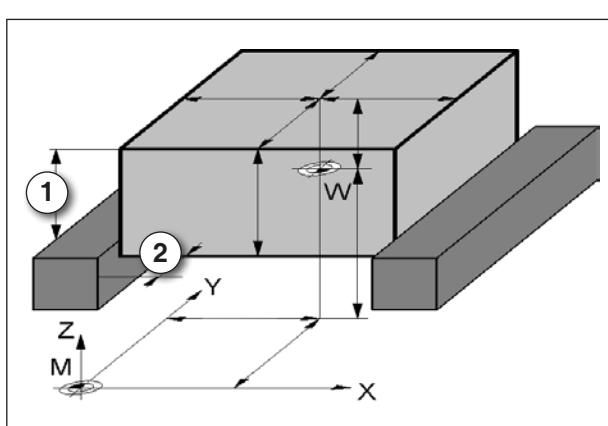
3D configuration

Input distance rawpiece to clamping device

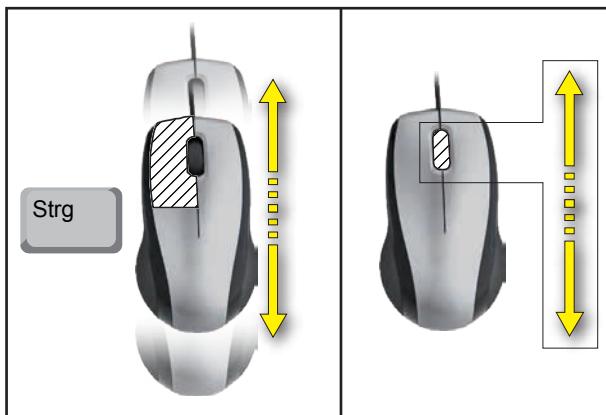
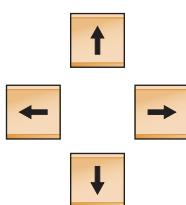
- in Z-direction (1)
- in XY-direction (2)
depending on the clamping situation
- Collision check on or off
- Set resolution: set from fine to rough.

Instructions:

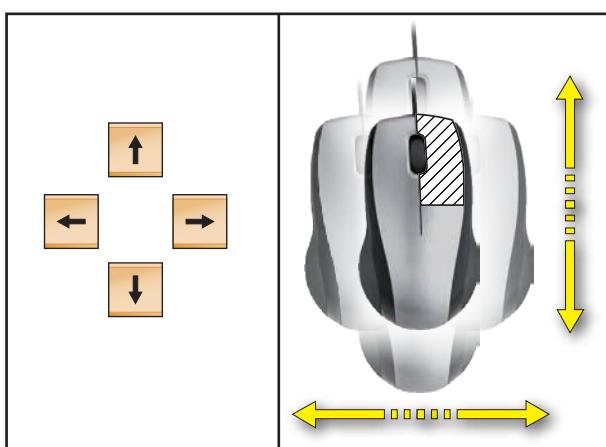
The selection of available setting options depends on whether a 3D viewing licence is available or not.



Distance rawpiece to clamping device

**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
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		

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.

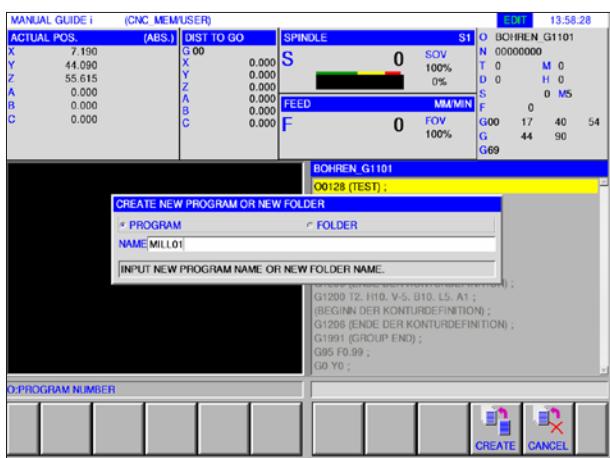
2 Press softkey.

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

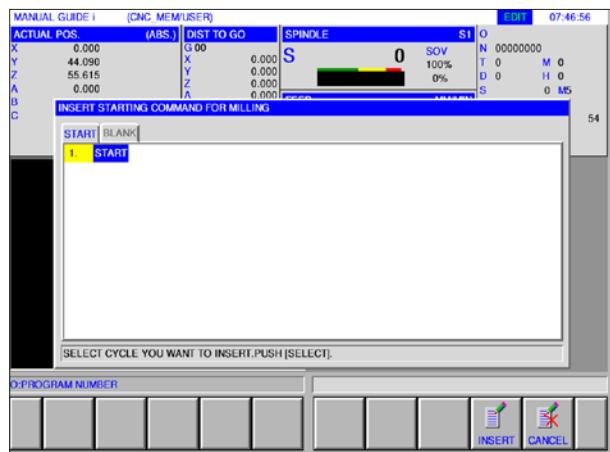
A cycle is divided into 2 parts:

- Machining conditions
- Geometric data

Program start START



- 1 Press "START" softkey.



Instructions:

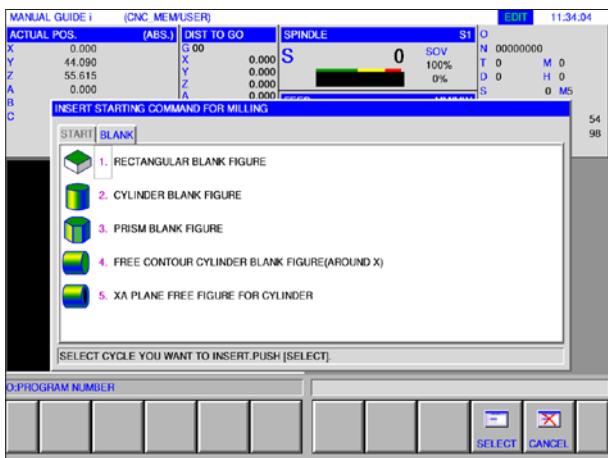
The fixed shapes from form 1 are available for "START".



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

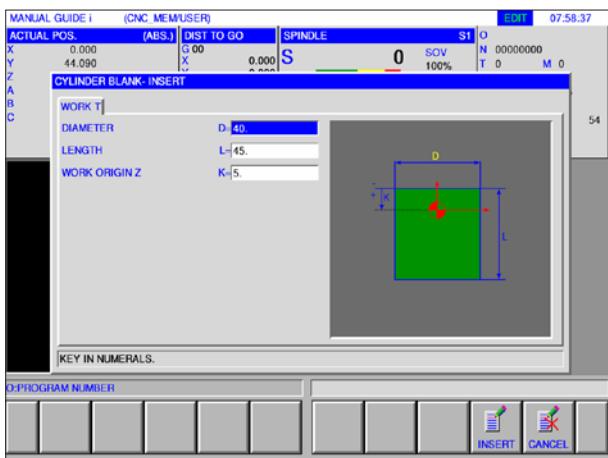
Select unmachined part

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



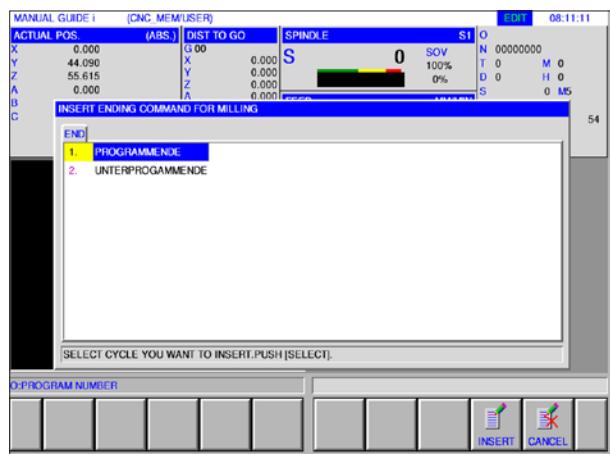
1 Press "CYCLE" softkey.

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

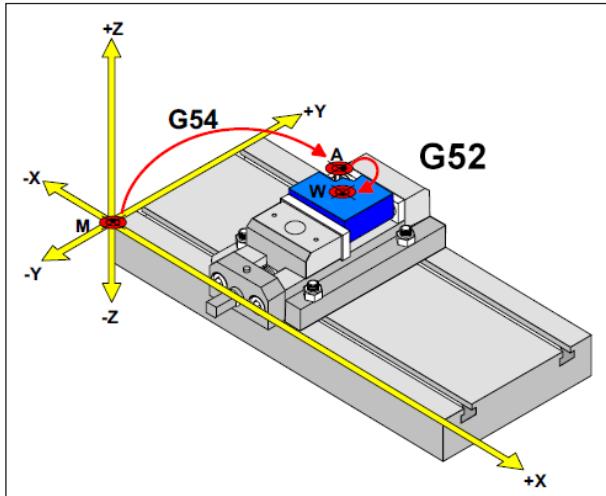
Program end END



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

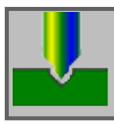
A= Stop point

W= Workpiece zero point

Cycle overview

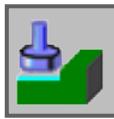
The cycle groups with the Fanuc ManualGuide *i* cycles defined within them are listed here.

Hole machining



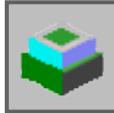
- Centre Drilling G1100
- Drilling G1101
- Tapping G1002
- Reaming G1103
- Boring G1104

Face machining



- Facing rough G1020
- Facing finish G1021

Contouring



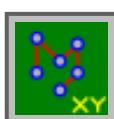
- Outer wall countering rough G1060
- Outer wall countering bottom finish G1061
- Outer wall countering side finish G1062
- Outer wall countering chamfer G1063
- Inner wall countering rough G1064
- Inner wall countering bottom finish G1065
- Inner wall countering side finish G1066
- Inner wall countering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

Pocketing

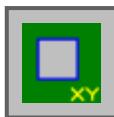


- Pocketing rough G1040
- Pocketing bottom finish G1041
- Pocketing side finish G1042
- Pocketing chamfer G1043

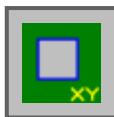
Figure: Hole position



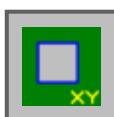
- Random points G1210
- Linear points G1211
- Grid points G1213
- Rectangle points G1214
- Circle points G1215
- Arc points G1216
- A-axis hole in cylinder (arc) G1772
- A-axis hole in cylinder (free) G1773

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

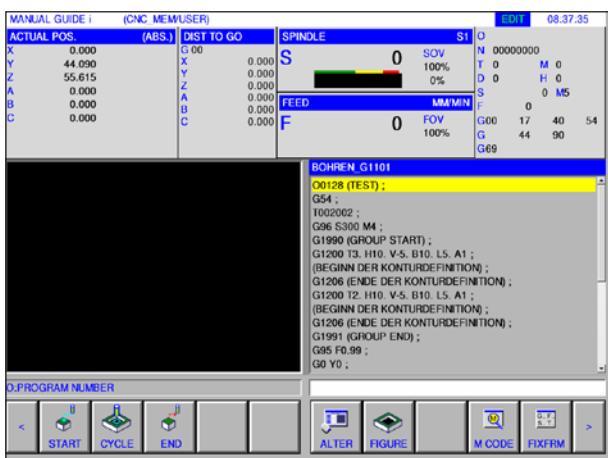
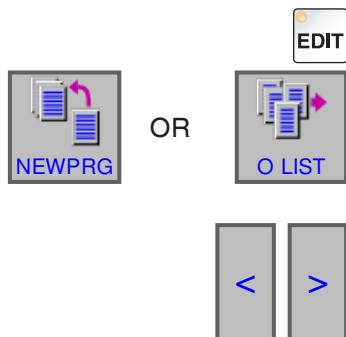
- Square concave G1220
- Circle concave G1221
- Track concave G1222
- Polygon concave G1225
- Free concave figure
- XA-plane free form concave contour for cylinder G1700

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.

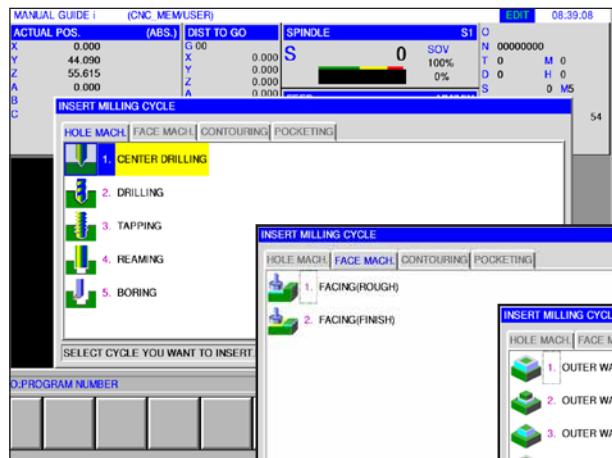
Define cycle

- 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.





Drilling cycles

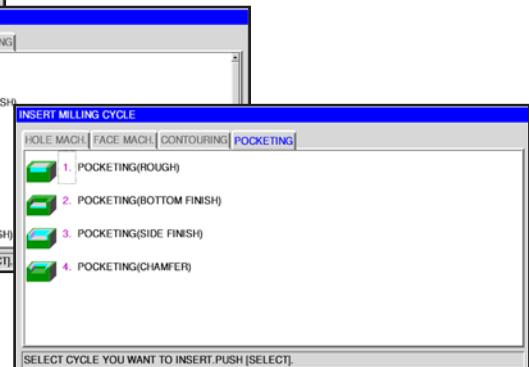
Plane machining

The dropdown window shows the different cycle groups.

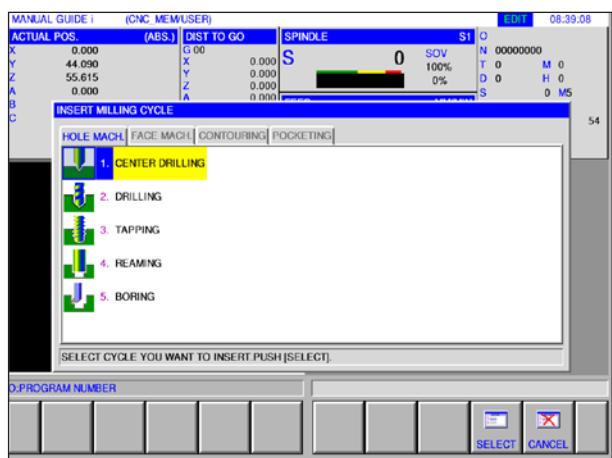
- Drilling
- Plane machining
- Contour machining
- Pocket milling



Contour machining

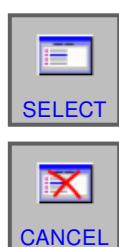


Pocket milling



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

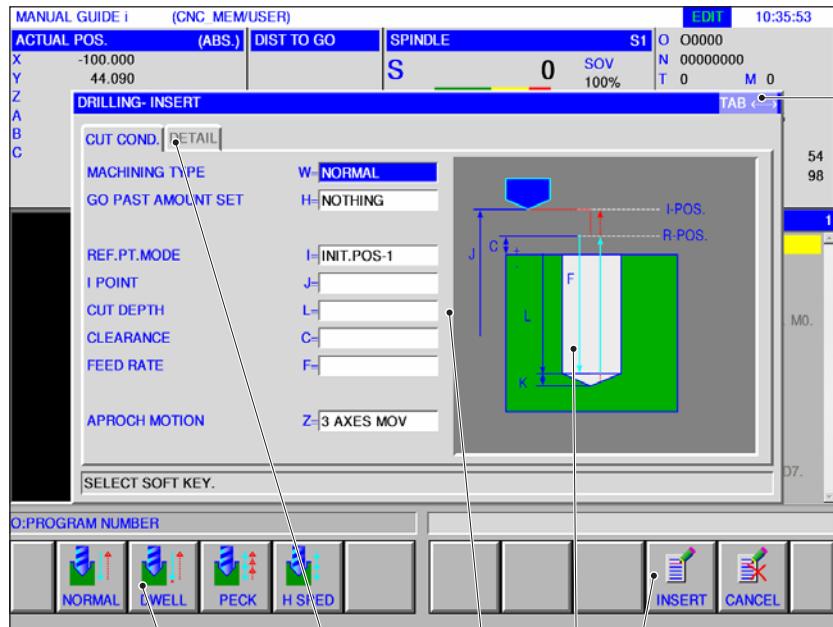
- "SELECT" softkey to confirm.



- "CANCEL" softkey to cancel.



Data Entry for Machining Cycles



For switching tabs with the cursor keys, "TAB $\leftarrow\rightarrow$ " 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:

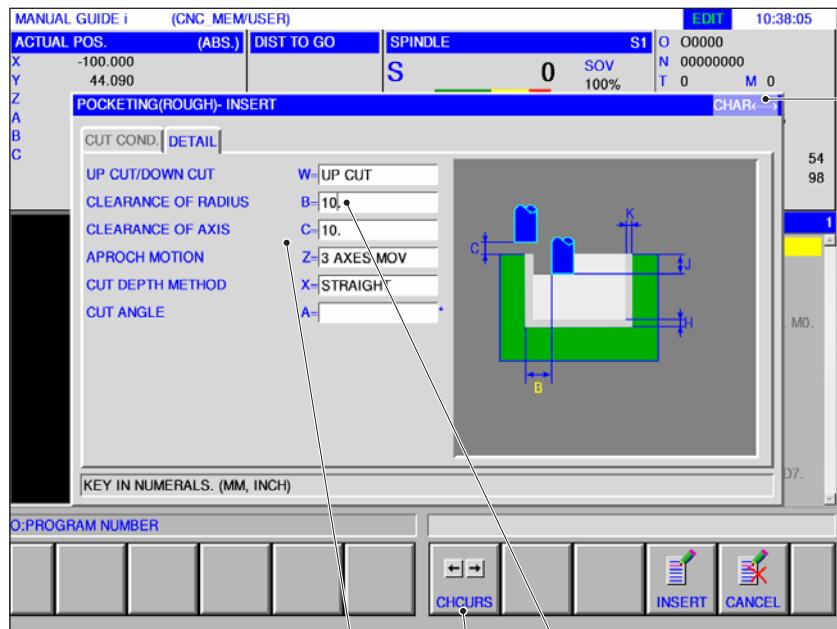
- Red positioning movement = tool moves in rapid traverse.
- Green positioning movement = tool moves in machining feed.

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 $<\rightarrow>$ or $<\rightarrow>$ 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 $\leftarrow\rightarrow$ " 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 $\leftarrow\rightarrow$ and $\uparrow\downarrow$ 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 $\uparrow\downarrow$ or $\leftarrow\rightarrow$, you can set the cursor on the desired data entry field.

- There are 2 options for data entry:

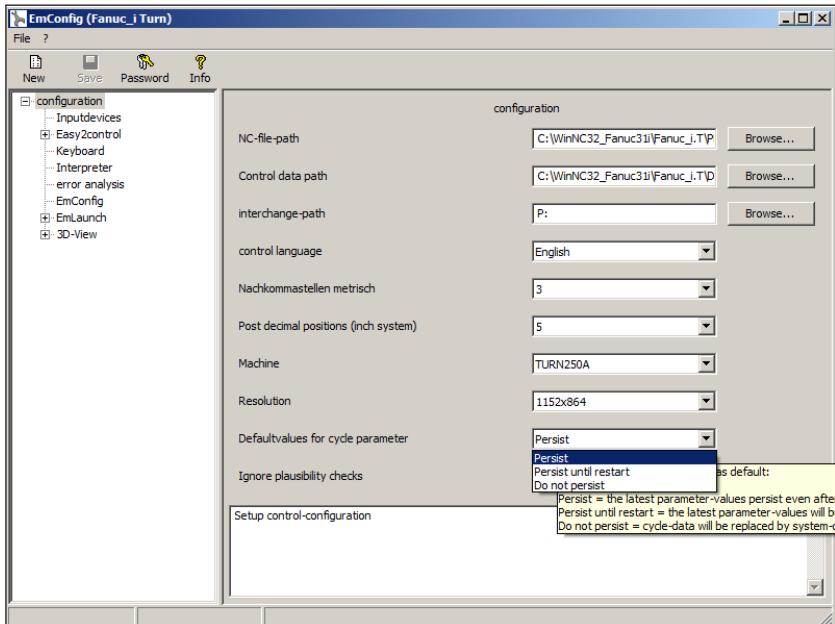
1. Entering data as numerals.

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



Default values for cycle parameters

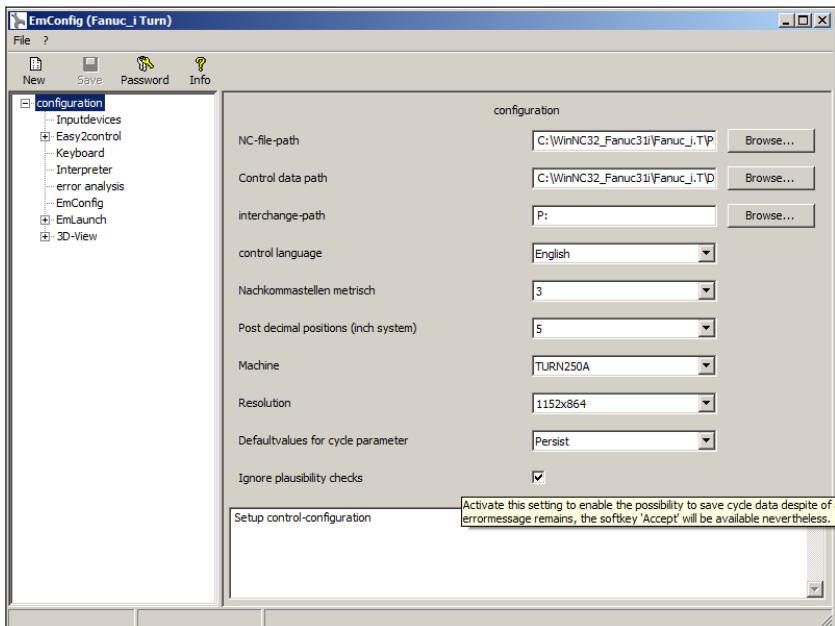
Instructions:

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

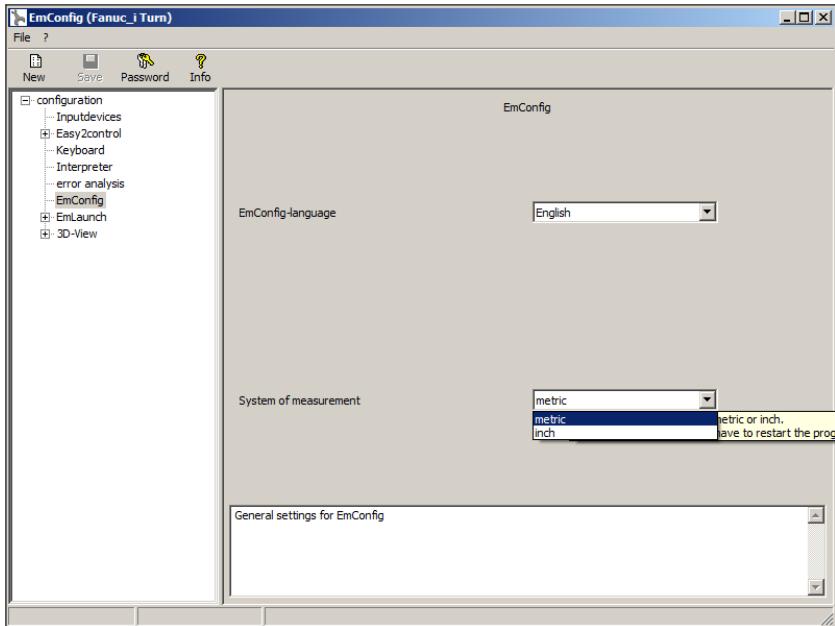


Set plausibility check for saving

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.

Select dimensions system



Set metric or imperial measuring system

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

Instructions:

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



Units table

Length dimensions imperial			
feet ^{*)}	inch	mm	m
1	12	304.5	0.304
inch ^{°)}	feet	mm	m
1	0.83	25.4	0.0254

Length dimensions metric			
m	mm	inch	feet
1	1000	39.37008	3.28084
mm	m	inch	feet
1	0.001	0.0393701	0.0032808

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

°) inch: Standard input

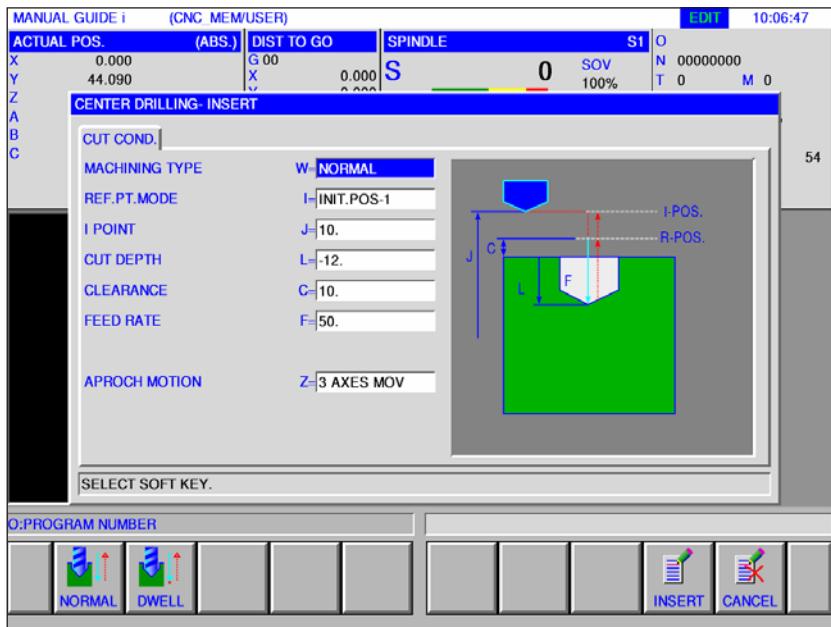
Hole machining



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



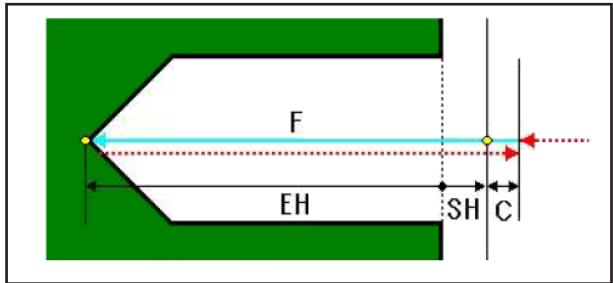
Centre drilling G1100



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"> [NORMAL]: Without dwell time. (Initial value) [DWELL]: With dwell time.
I	Ref.point mode	<ul style="list-style-type: none"> [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). [INIT-2]: All movements between borings, including the last return, are made to point I. [REF]: All movements between borings, including the last return, are made to point R.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



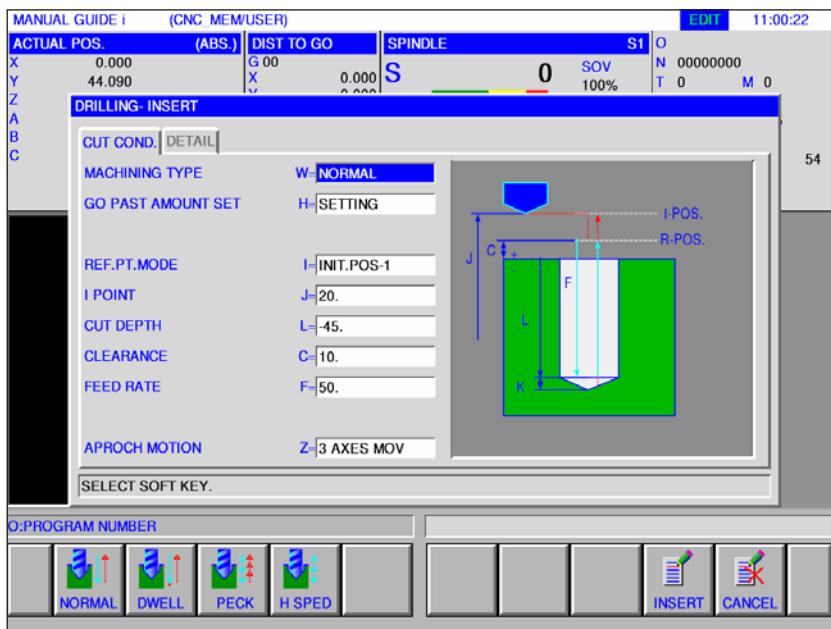
Tool track

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

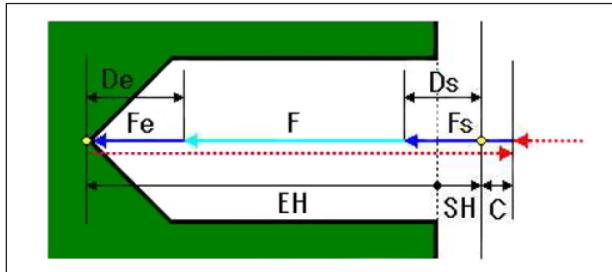


Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		Meaning
W	Machining type	<ul style="list-style-type: none"> [NORMAL]: Without dwell time. (Initial value) [DWELL]: With dwell time. [PECK]: Deep hole drilling [H SPED]: With dwell time.
H	Go past amount set	<ul style="list-style-type: none"> [NOTHIN]: Drilling depth relative to the tool shaft [SETING]: Drilling depth relative to the tool tip <p>For the setting [SETING] the parameters U, V, K and the softkey [CALC] are available under Details.</p>
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"> [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). [INIT- 2]: All movements between borings, including the last return, are made to point I. [REF.] : All movements between borings, including the last return, are made to point R.
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.

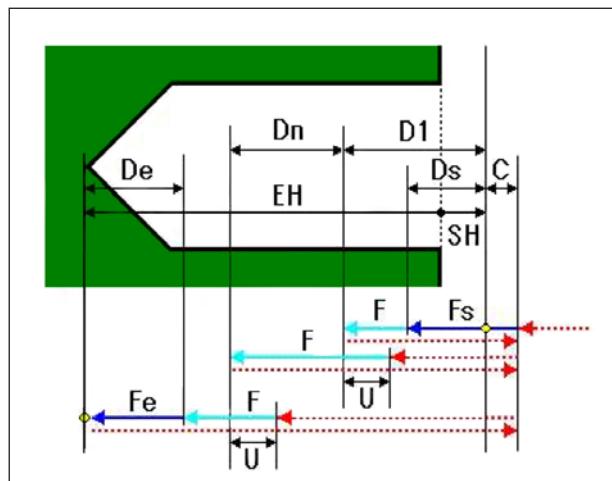
Detail		
Data item		Meaning
A*	Start cut depth	The start feed S applies for the start cut depth A
S*	Start feed rate	Feed speed at start
D*	End cut depth	The end feed E applies for the end cut depth D
E*	End feed rate	Feed speed at end
U	Tool diameter	Tool diameter entry
V	Nose angle	Cutting angle entry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



tool track

Cycle description: with/without dwell time

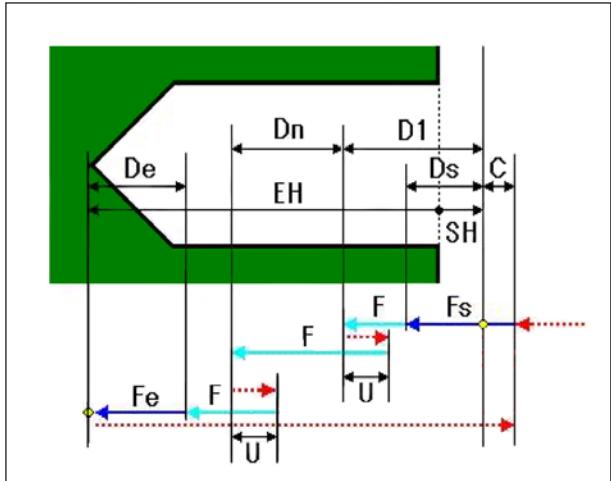
- 1 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".
- 2 The tool is moved with feed speed (F) into the machining end position.
- 3 The tool is moved in rapid traverse into the position "Machining start position + clearance (C)".



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 (D_1)".
- 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 (D_n)".
- 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)".



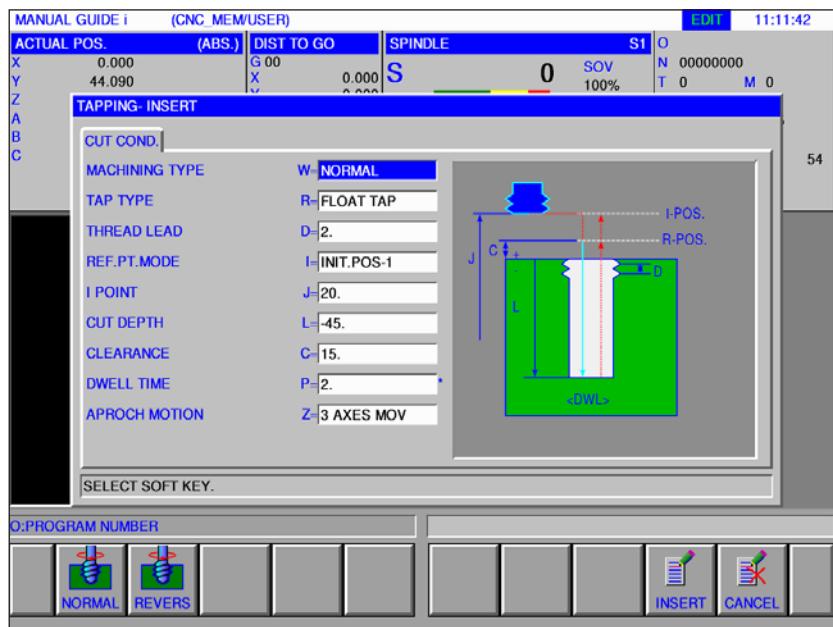
tool track

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)".



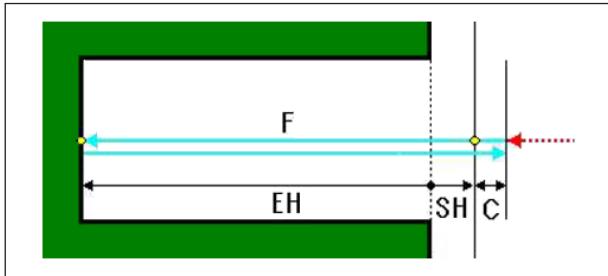
Tapping G1002



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"> [NORMAL]: Clockwise tapping. [REVERS]: Counterclockwise tapping.
R	Tap type	<ul style="list-style-type: none"> [FLOAT]: Thread tapping with compensating chuck. [RIGID]: Thread tapping without compensating chuck.
D	Thread lead	Tap pitch (radius value, positive value).
I	Ref.point mode	<ul style="list-style-type: none"> [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). [INIT-2]: All movements between borings, including the last return, are made to point I. [REF]: All movements between borings, including the last return, are made to point R.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



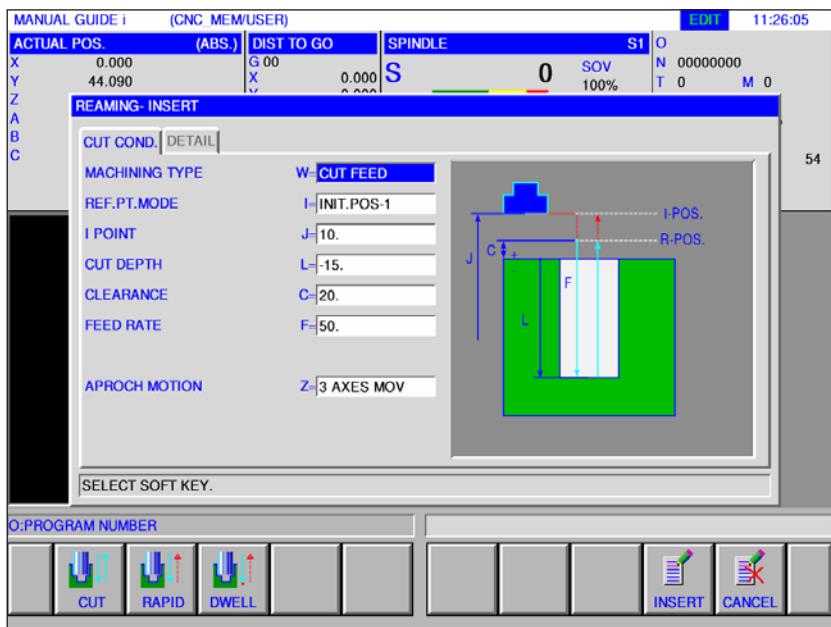
tool track

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.



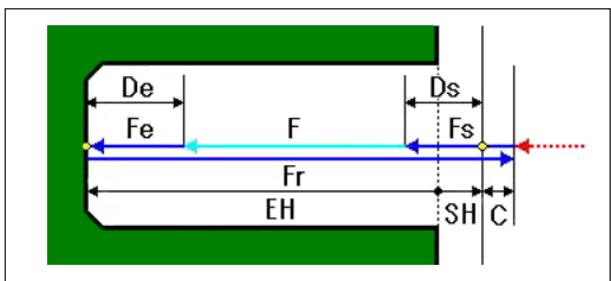
Reaming G1003



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"> [CUT]: The tool is retracted with feed speed from the bottom of the drilling. [RAPID]: The tool is retracted in rapid traverse from the bottom of the drilling. [DWELL]: After dwelling at the bottom of the drilling, the tool is retracted with feed speed.
I	Ref.point mode
	<ul style="list-style-type: none"> [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). [INIT-2]: All movements between borings, including the last return, are made to point I. [REF]: All movements between borings, including the last return, are made to point R.
J	I Point
L	Cut depth
C	Clearance
F	Feed rate
P*	Dwell time
Z	Approach motion

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



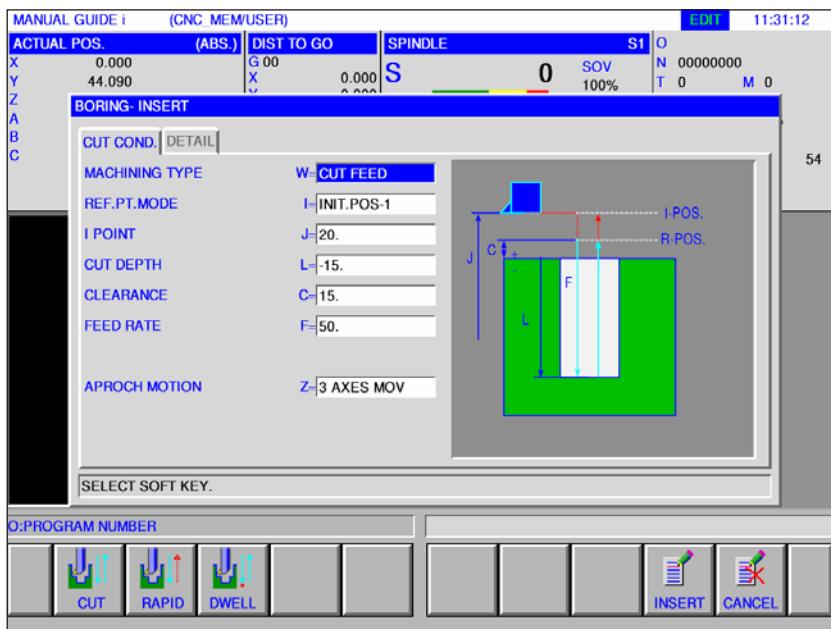
tool track

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

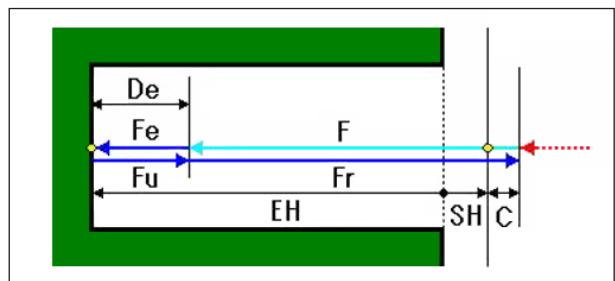


Entry fields marked with * are optional and should not be populated.

Cutting condition

Data item		Meaning
W	Machining type	<ul style="list-style-type: none"> [CUT]: The tool is retracted with feed speed from the bottom of the drilling. [RAPID]: The tool is retracted in rapid traverse from the bottom of the drilling. [DWELL]: After dwelling at the bottom of the drilling, the tool is retracted with feed speed.
I	Ref.point mode	<ul style="list-style-type: none"> [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). [INIT-2]: All movements between borings, including the last return, are made to point I. [REF]: All movements between borings, including the last return, are made to point R.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.

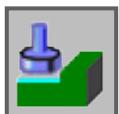
Detail		
Data item		Meaning
A*	Start cut depth	The start feed S applies for the start cut depth A
S*	Start feed rate	Feed speed at start
D*	End cut depth	The end feed E applies for the end cut depth D
E*	End feed rate	Feed speed at end



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)".

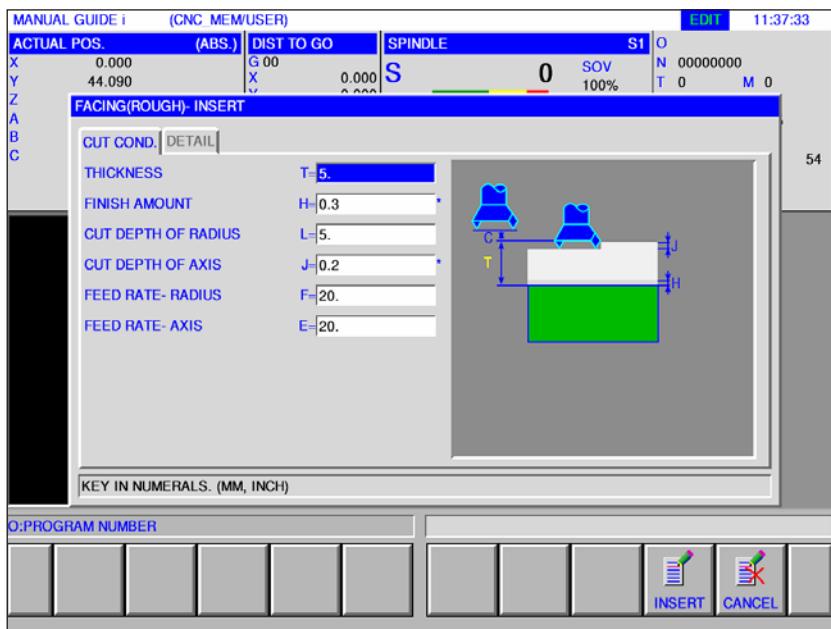
Face machining



- Facing rough G1020
- Facing finish G1021



Facing (rough) G1020



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

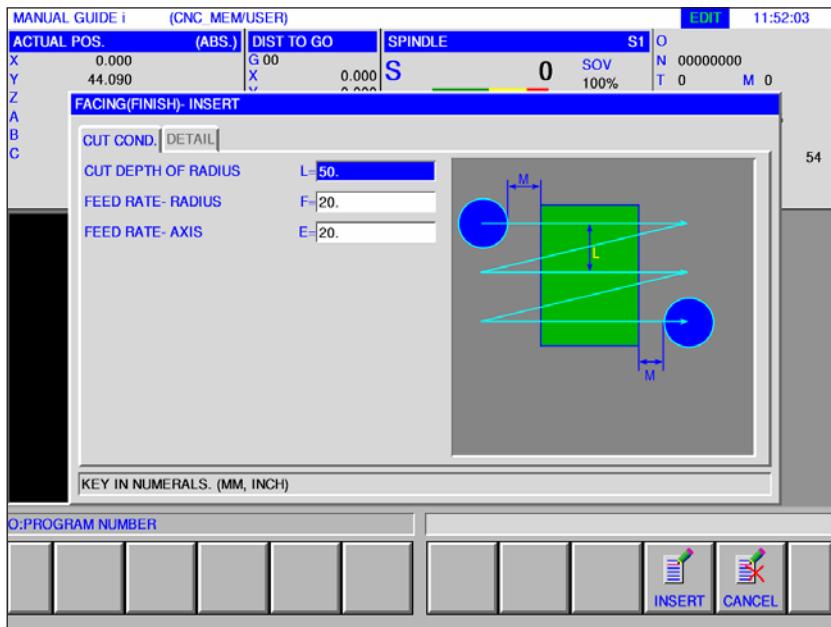
Cutting condition

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 st 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"> [SINGLE]: Cutting in tool radius direction is always done in the same direction. [ZIGZAG]: Cutting in tool radius direction is done forwards and backwards.
P	Path move method	<ul style="list-style-type: none"> [PULL]: Return to point R before approaching the start point for the next cutting track (in tool axis direction). [KEEP]: Direct approach to the start point of the next cutting track without returning to point R. <p>Only if machining method [ZIGZAG]</p>
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"> [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. [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. [UP]: Cutting is done upwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track upwards. [DOWN]: Cutting is done downwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track downwards. <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"> [RIGHT]: Cutting is done to the right during the change between two cutting tracks, as shown in the sketch. [LEFT]: Cutting is done to the left during the change between two cutting tracks, as shown in the sketch. [UP]: Cutting is done upwards during the change between two cutting tracks, as shown in the sketch. [DOWN]: Cutting is done downwards during the change between two cutting tracks, as shown in the sketch. <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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Facing (finish) G1021



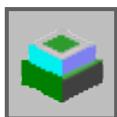
Entry fields marked with * are optional and should not be populated.

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"> • [SINGLE]: Cutting in tool radius direction is always done in the same direction. • [ZIGZAG]: Cutting in tool radius direction is done forwards and backwards.
P	Path move method	<ul style="list-style-type: none"> • [PULL]: Return to point R before approaching the start point for the next cutting track (in tool axis direction). • [KEEP]: Direct approach to the start point of the next cutting track without returning to point R. 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"> • [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. • [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. • [UP]: Cutting is done upwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track upwards. • [DOWN]: Cutting is done downwards as shown in the sketch. If both directions are selected, cutting is done in the first cutting track downwards. <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"> • [RIGHT]: Cutting is done to the right during the change between two cutting tracks, as shown in the sketch. • [LEFT]: Cutting is done to the left during the change between two cutting tracks, as shown in the sketch. • [UP]: Cutting is done upwards during the change between two cutting tracks, as shown in the sketch. • [DOWN]: Cutting is done downwards during the change between two cutting tracks, as shown in the sketch. <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"> • [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.

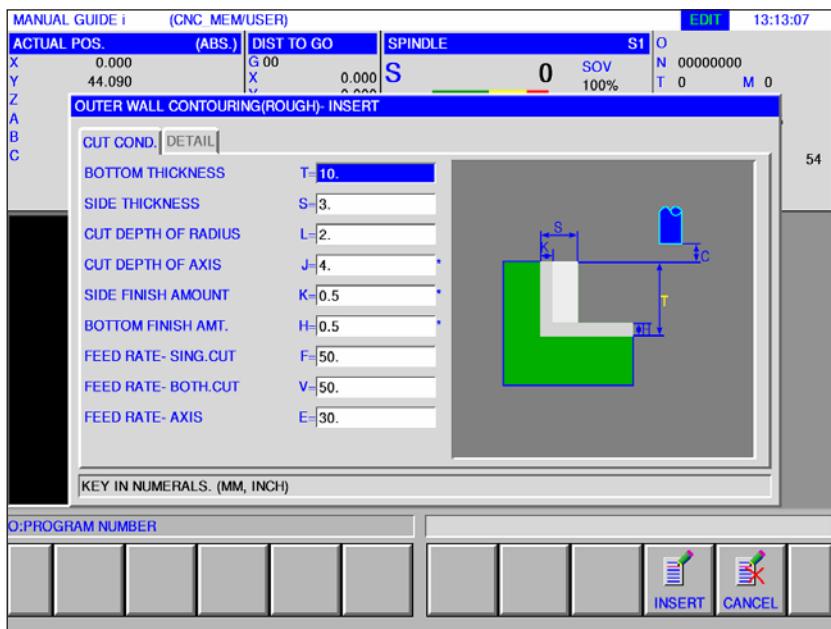
Contouring



- 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

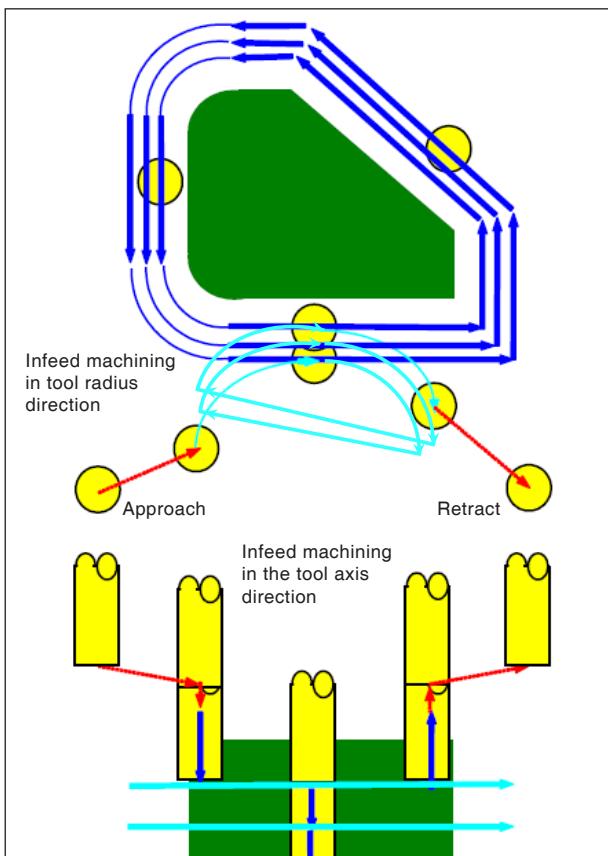


Entry fields marked with * are optional and should not be populated.

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 st 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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIK]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.

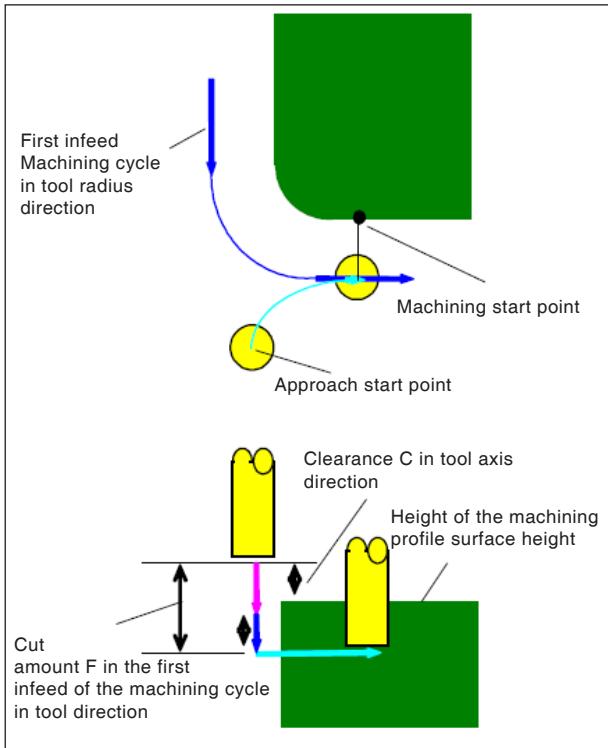
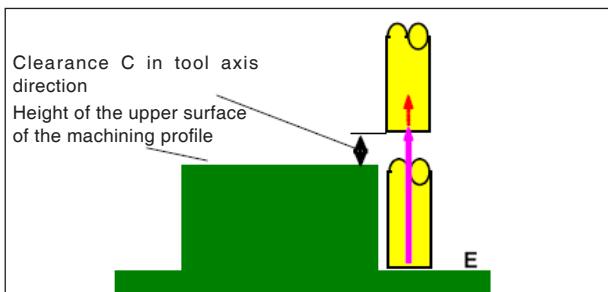


Tool track G1060, G1064, G1068

Cycle description for G1060, G1064 and G1068:

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

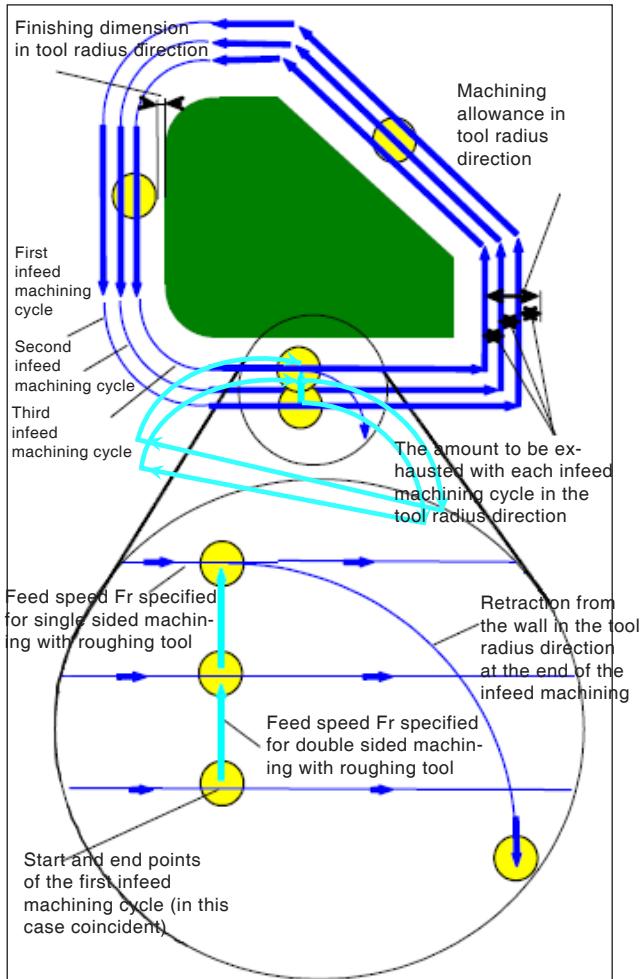
- 1 The tool is moved to the above approach start point.
- 2 The tool is moved to the height of the machining surface.
- 3 The tool cuts along the side surface contour of the machining profile.
The tool cuts by infeed machining in the tool radius direction until the machining allowance in the tool radius 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**Retraction 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 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:

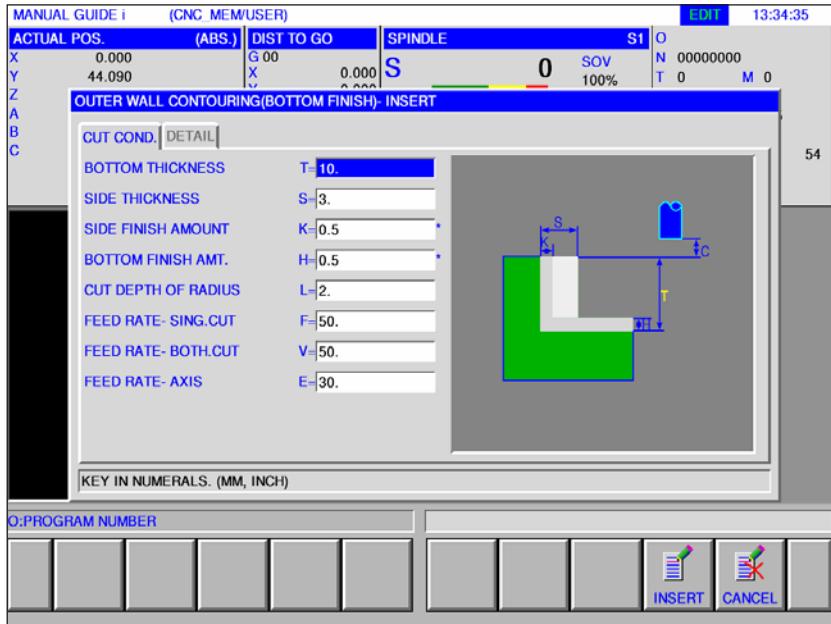
- 1 The tool is moved for cutting along the contour from the first infeed machining cycle start point to the end point with the feed speed (F) specified for single side machining with roughing tool.
- 2 The tool approaches as follows.

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.



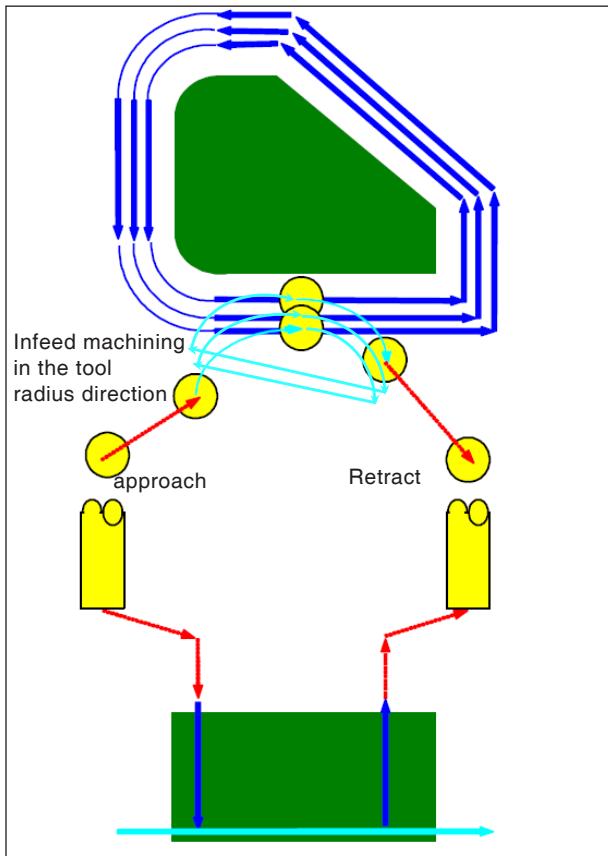
Outer wall (bottom finish) G1061



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"> [UPCUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWNCUT] Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Tool track G1061, G1065, G1069

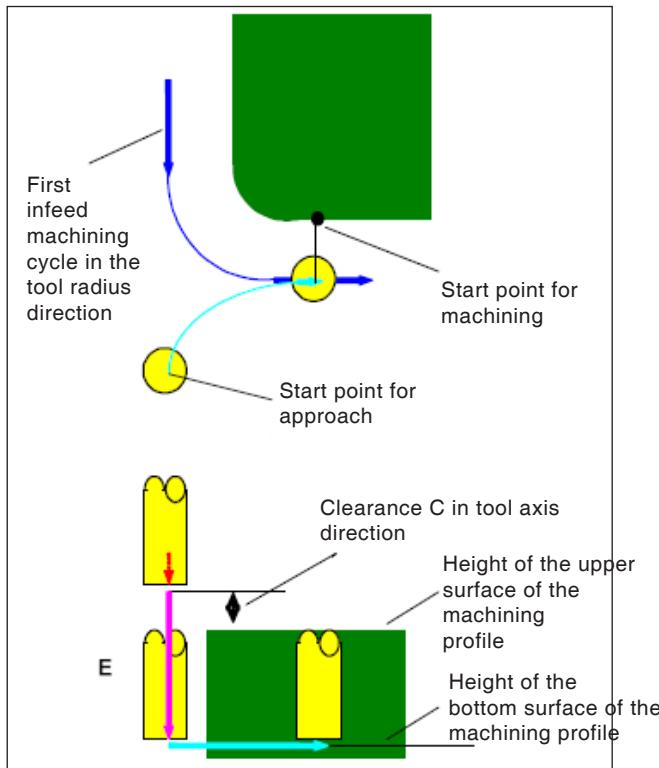
Cycle description for G1061, G1065 and G1069:

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

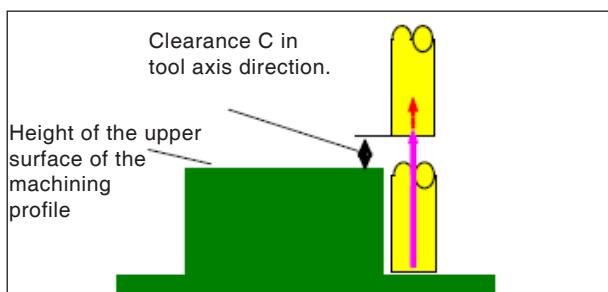
- 1 The tool is moved to the machining profile approach start point.
- 2 The tool is moved to the height of the machining surface of the machining profile.
- 3 The tool is moved to cut along the side surface contour of the machining profile.

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



Retraction track G1061, G1065, G1069

Approach cycle description:

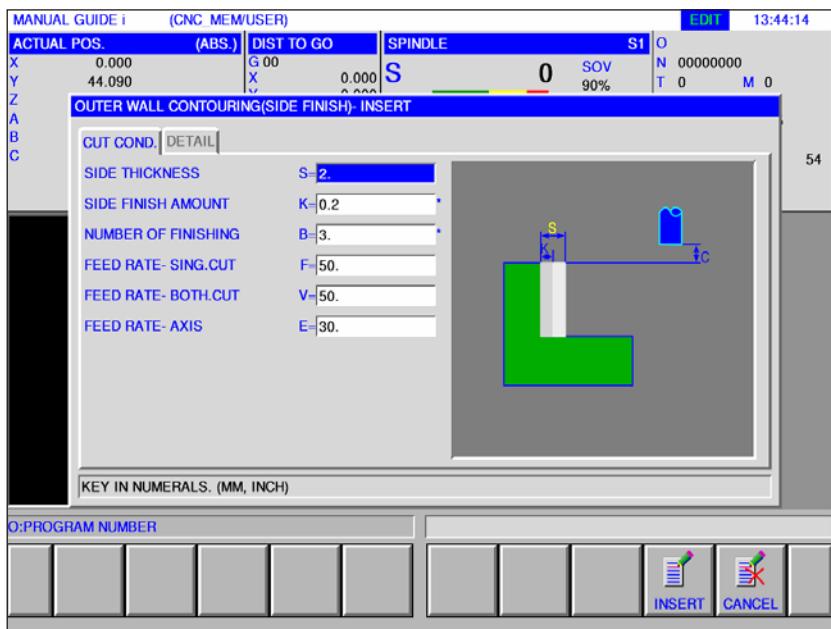
- 1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".
- 2 The tool traverses into the position "Machining profile bottom surface + machining allowance (Vt) in the tool axis direction" with the feed speed (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 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

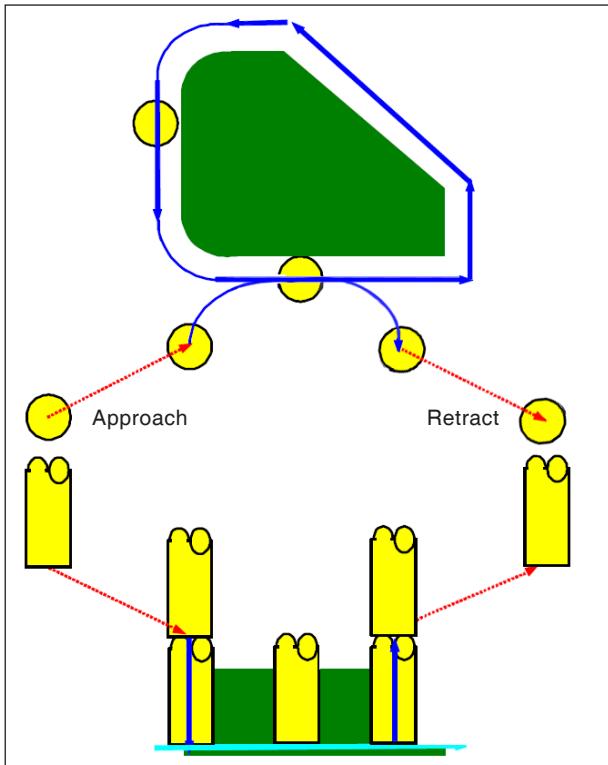


Entry fields marked with * are optional and should not be populated.

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"> [UPCUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWNCUT] Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



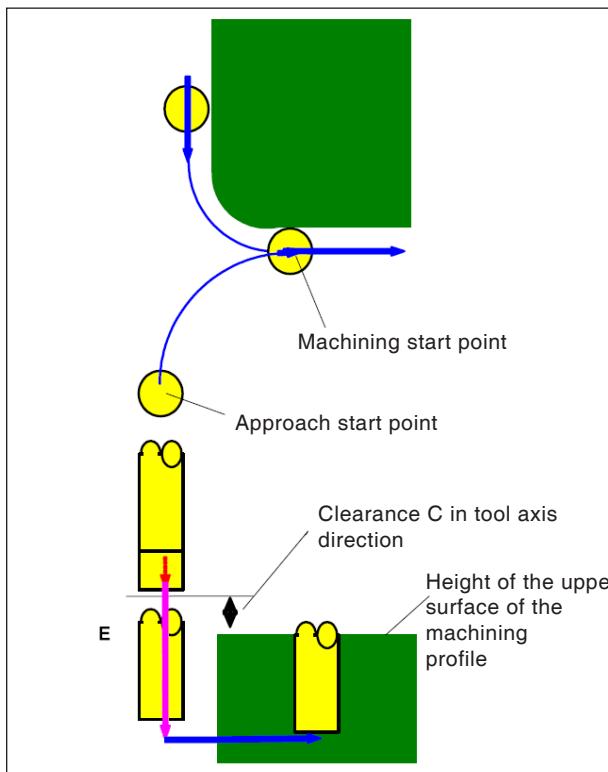
Tool track G1062, G1066, G1070

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.

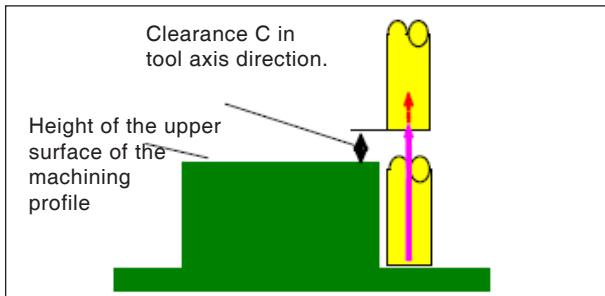
- 4 The tool retracts.



Approach G1062, G1066, G1070

Approach cycle description:

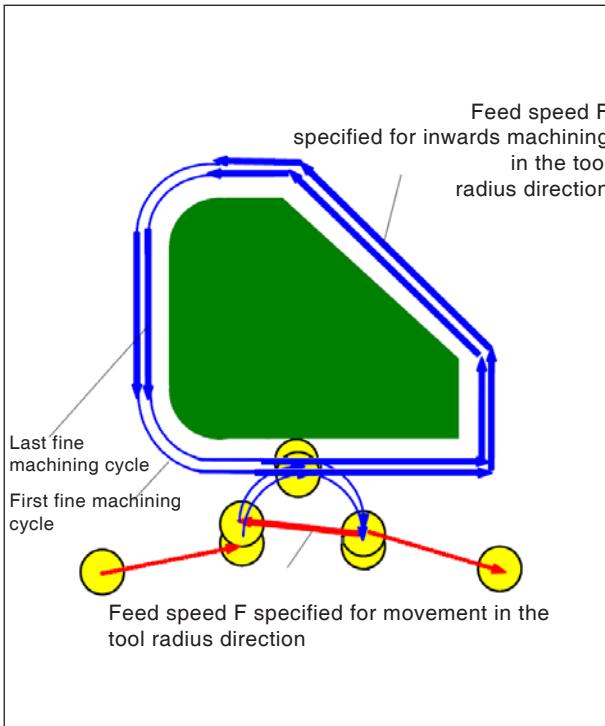
- 1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".
- 2 The tool traverses into the position "Machining profile bottom surface + finishing dimension (S) in the tool axis direction + clearance (C) in 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

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".

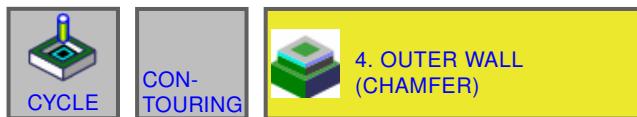


Retraction G1062, G1066, G1070

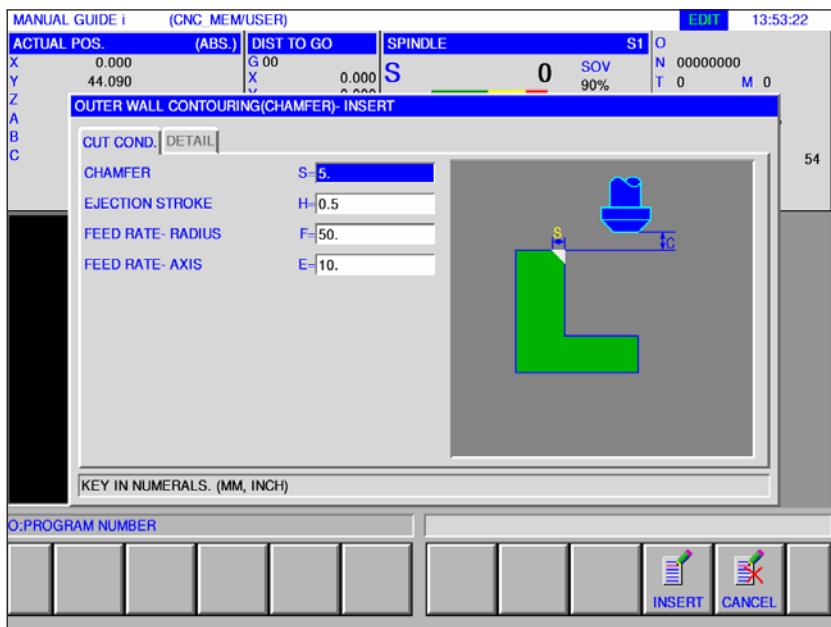
Infeed machining in tool radius direction:

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

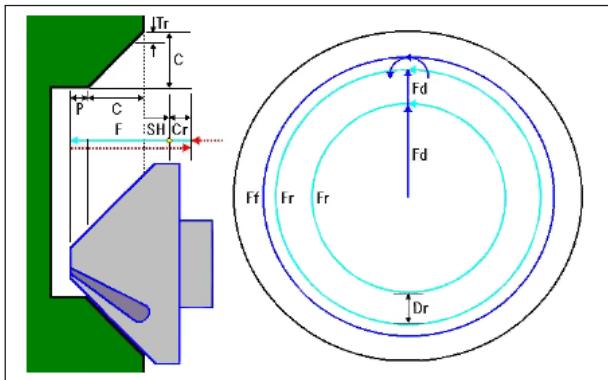


Entry fields marked with * are optional and should not be populated.

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"> [UPCUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWNCUT] Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



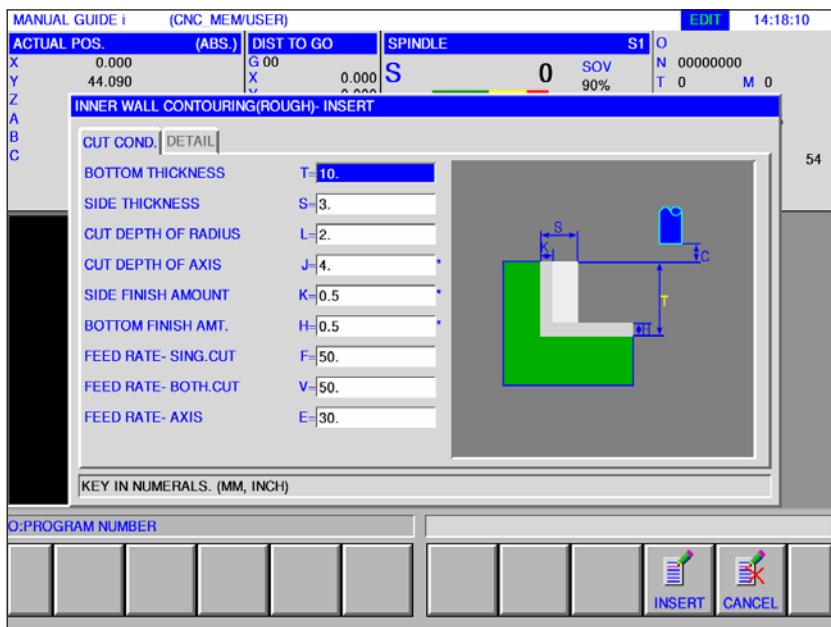
Tool track G1063, G1067, G1071

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.

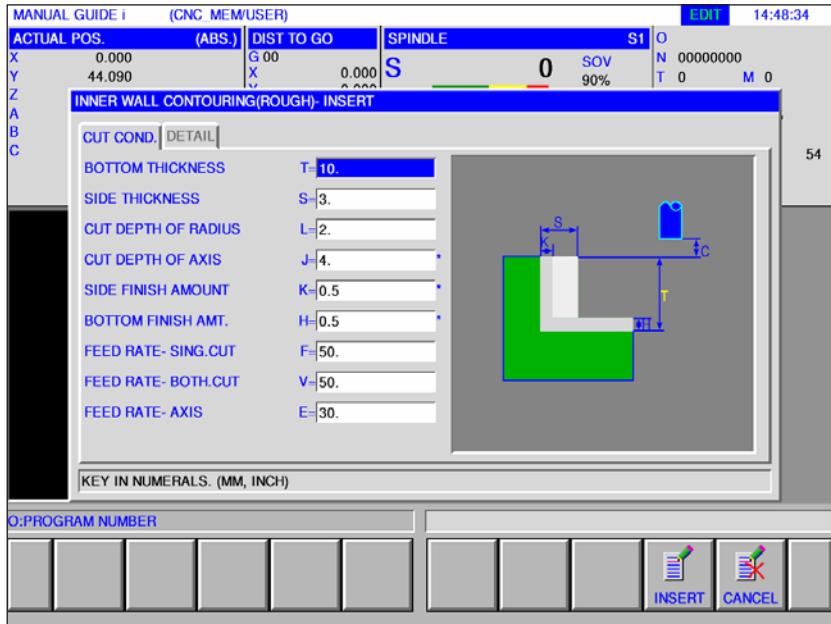
Cutting condition

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 st 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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Inner wall (bottom finish) G1065



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

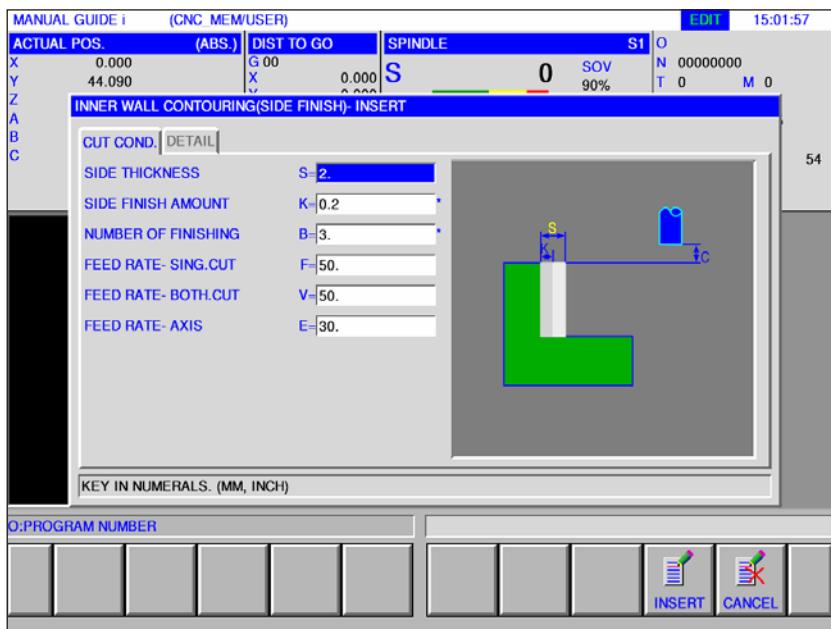
Cutting condition

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Inner wall (side finish) G1066



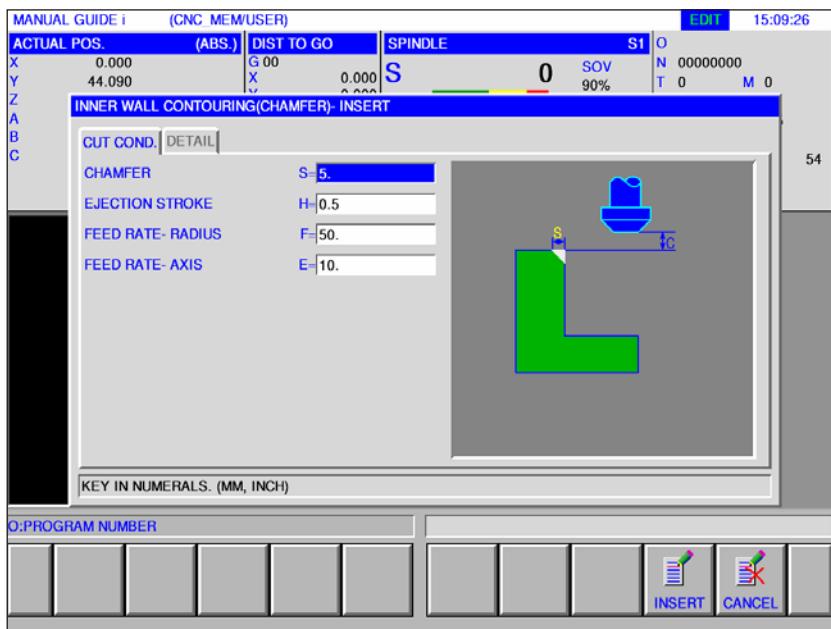
Entry fields marked with * are optional and should not be populated.

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Inner wall (chamfer) G1067



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

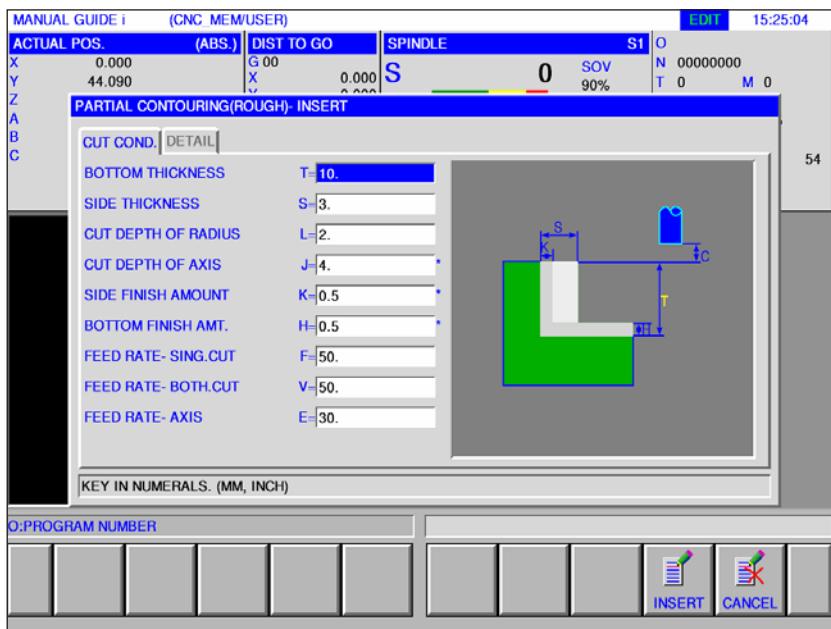
Cutting condition

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Partial (rough) G1068



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

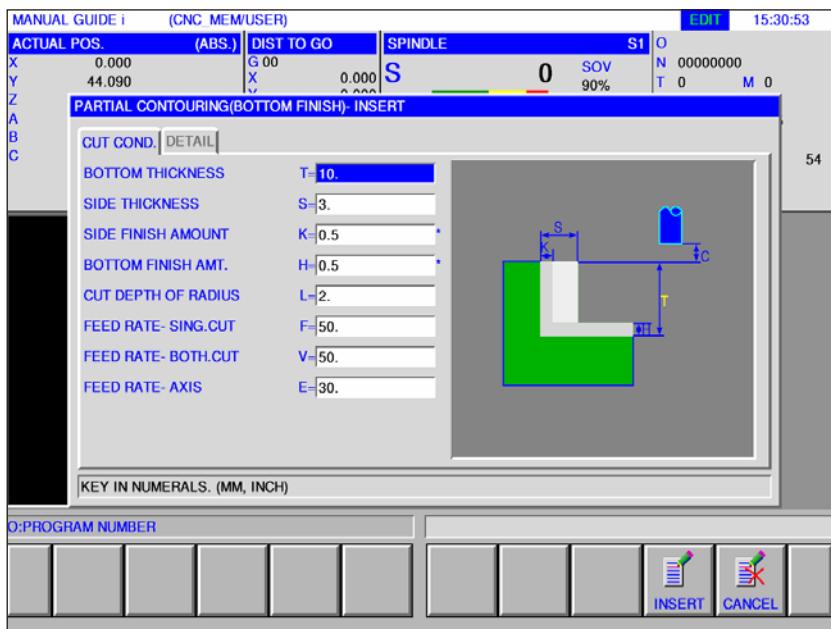
Cutting condition

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 st 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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Partial (bottom finish) G1069



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

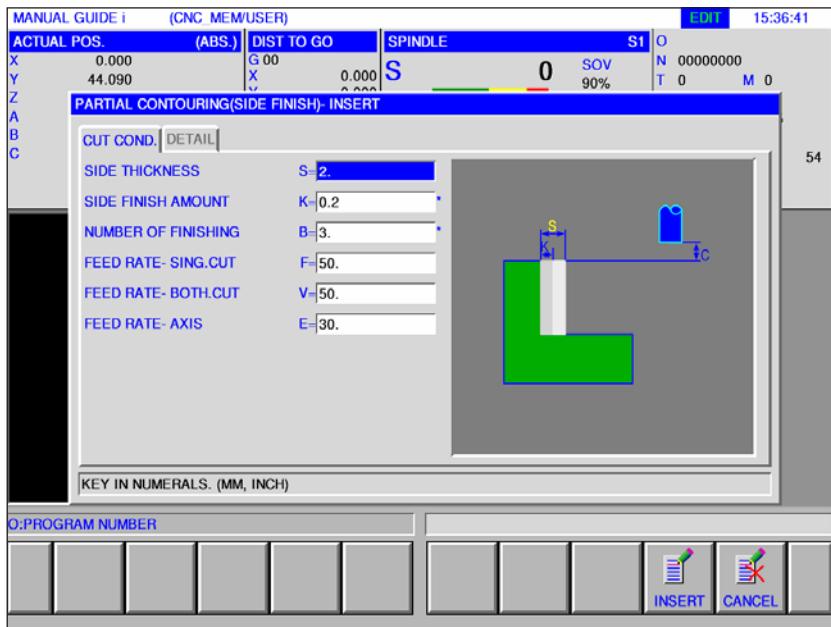
Cutting condition

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Partial (side finish) G1070



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

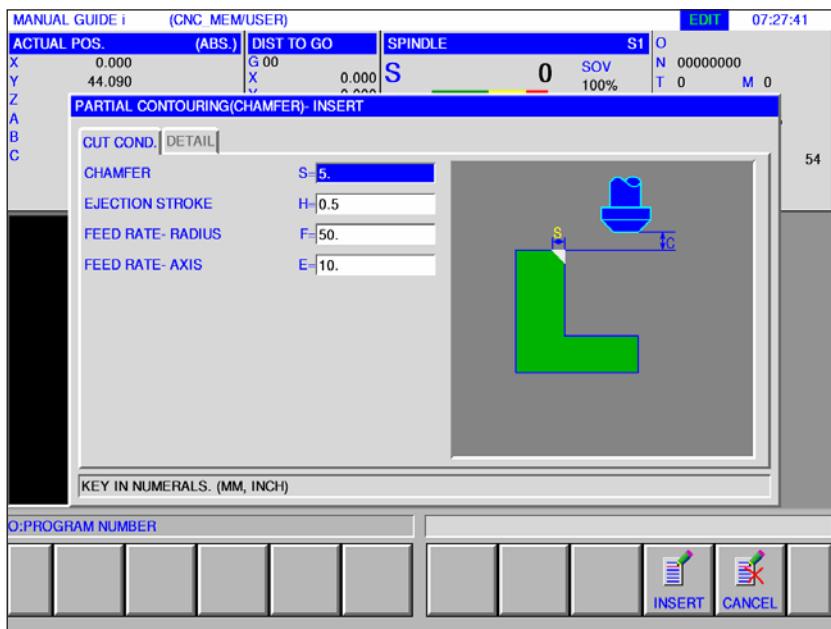
Cutting condition

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Partial (chamfer) G1071



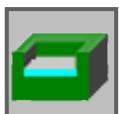
Entry fields marked with * are optional and should not be populated.

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. [TANGEN]: The approach to the side surface takes the form of a straight line tangential to the first contour in side surface machining. [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.
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"> [ARC]: Retraction from the side surface takes the form of an arc. [TANGEN]: Retraction from the side surface takes the form of a straight line tangential to the first contour in side surface machining. [VERTIC]: Retraction from the side surface takes the form of a straight line at right angles to the first contour in side surface machining.
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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.

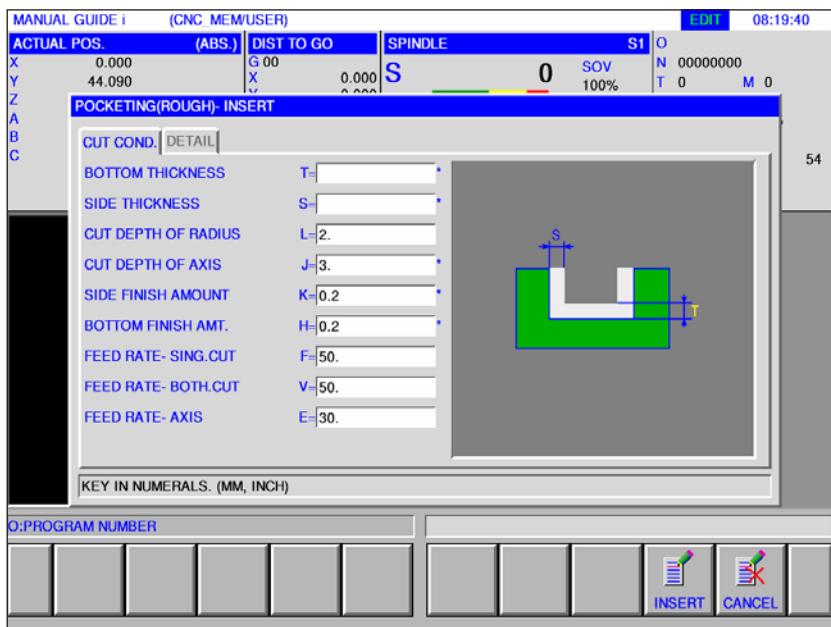
Pocketing



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



Pocketing (rough) G1040

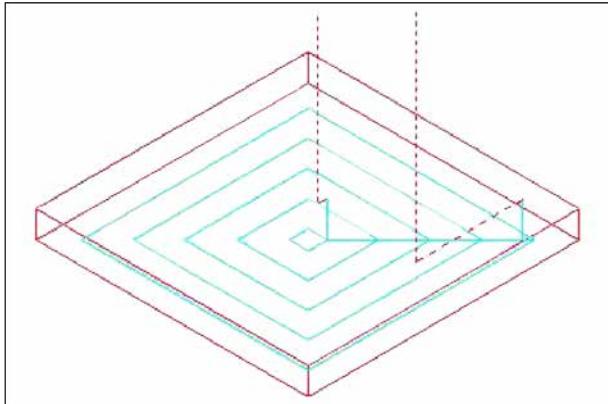


Entry fields marked with * are optional and should not be populated.

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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	<ul style="list-style-type: none"> [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"> [STRAIT]: Plunging in the form of a straight line. [HELICL]: Plunging on spiral path (helix).
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.

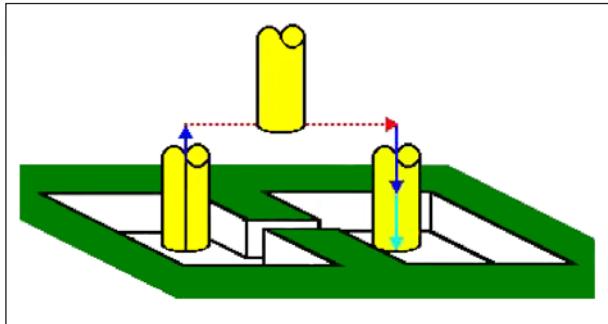
The following tool track is generated.

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.

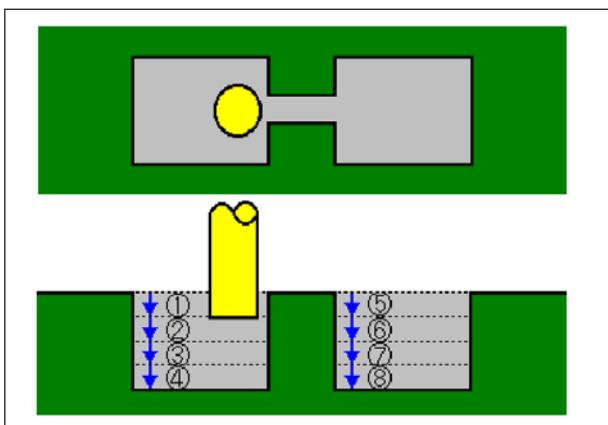
It can be cut at any angle in the tool axis direction. This also determines the machining start point automatically.

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.

In the following pocket machining profile, where the tool can move through the pocket, the tool is lifted off automatically, do 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.

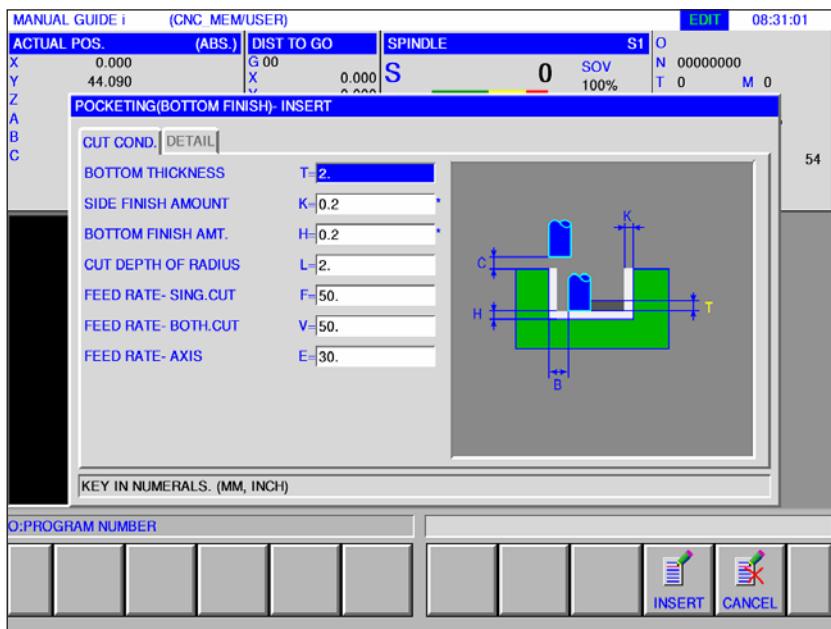
If a number of cuts have to be made in the tool axis direction, each area is machined completely before another is started.



Tool track



Pocketing (bottom finish) G1041

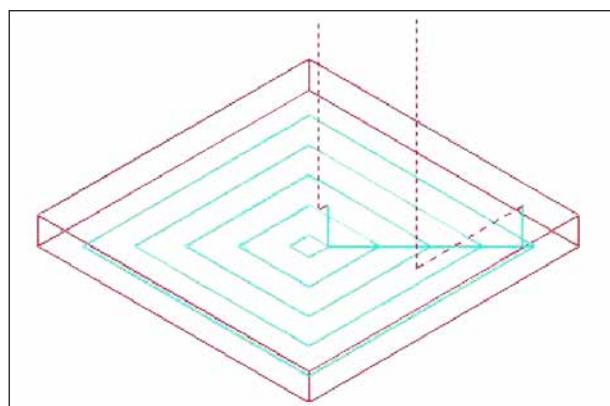


Entry fields marked with * are optional and should not be populated.

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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	<ul style="list-style-type: none"> [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"> [STRAIT]: Plunging in the form of a straight line. [HELICL]: Plunging on spiral path (helix).
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

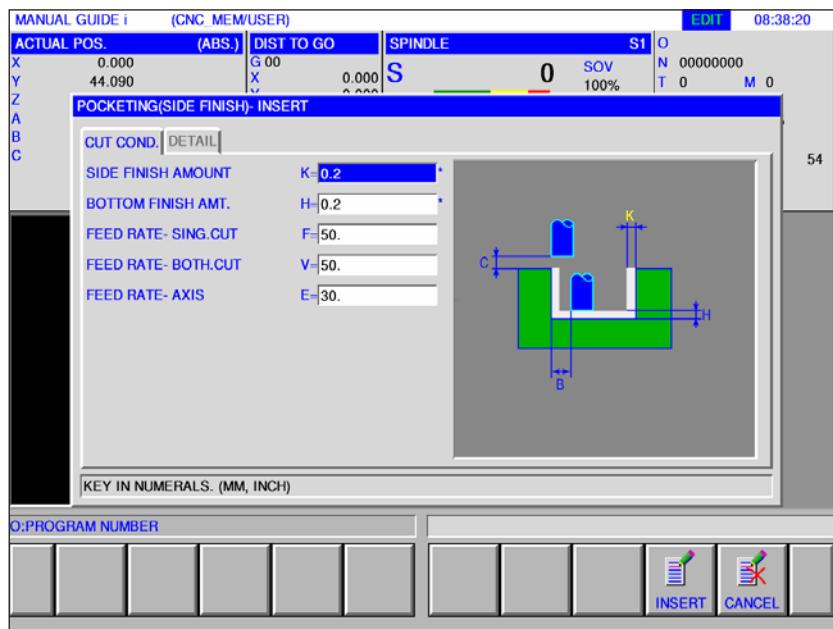
The bottom surface of a pocket machining profile is finished spirally. The tool track for this is the same as with pocket milling (roughing).

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

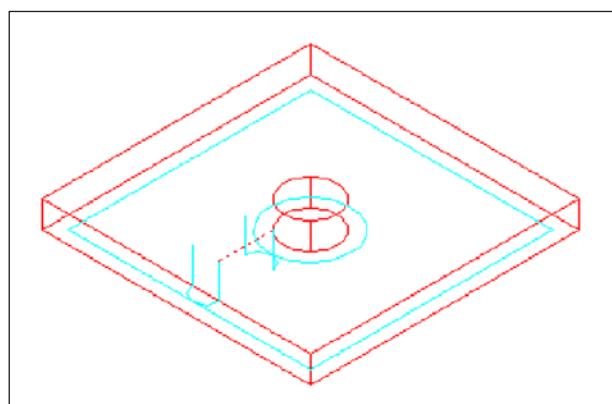


Entry fields marked with * are optional and should not be populated.

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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. 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"> [ARC]: Retraction from the side surface takes the form of an arc. 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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.



Tool track

Cycle description

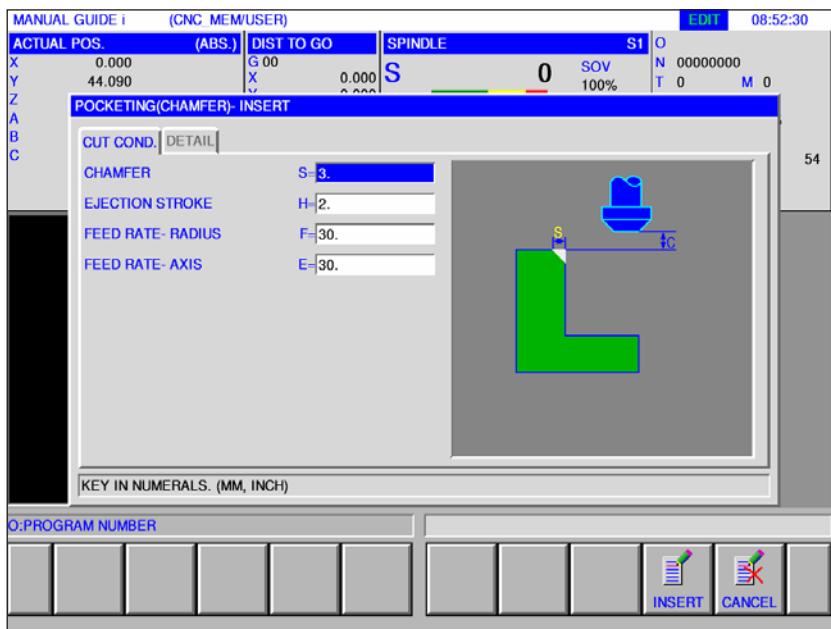
The side contour of the pocket and of the island machining profile are finished. The tool track for this is the same as for contour machining (Finishing the side surface).

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

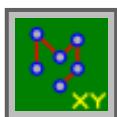
Data item		Meaning
S	Chamfer	Chamfer length (radius value, positive value).
H	Ejection stroke	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value).
F	Feed rate radius	Feed speed when cutting in tool radius direction
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"> [UP CUT]: Machining is done by upcut milling, in which the tool turns clockwise. [DWN CUT]: Machining is done by downcut milling, in which the tool turns clockwise.
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"> [ARC]: The approach to the side surface takes the form of an arc. 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"> [ARC]: Retraction from the side surface takes the form of an arc. 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"> [3 AXES]: The tool is moved from the instantaneous position to the machining start point in 3-axis synchronous mode.

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

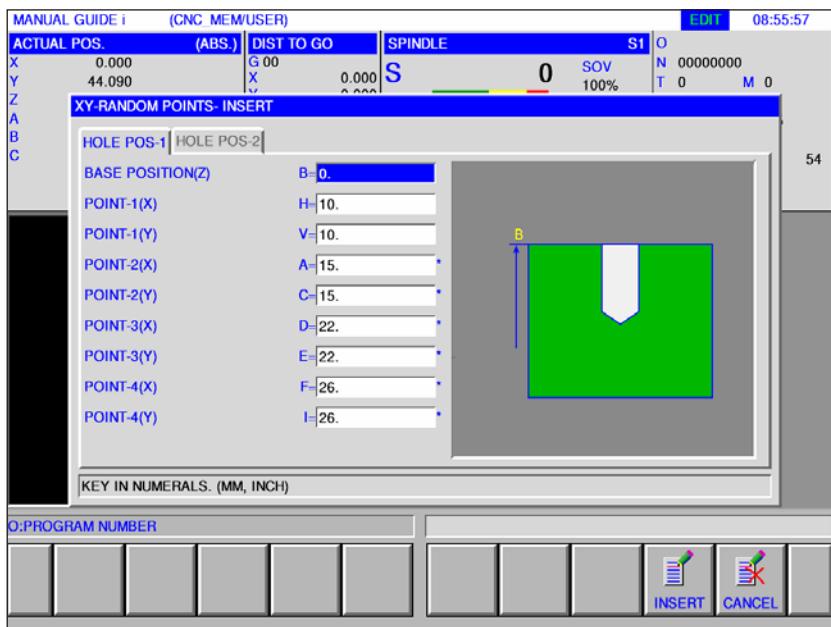


- Random points G1210
- Linear points G1211
- Grid points G1213
- Rectangle points G1214
- Circle points G1215
- Arc points G1216
- A-axis hole in cylinder (arc) G1772
- A-axis hole in cylinder (free) G1773



1. XY RANDOM POINTS

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.

MANUAL GUIDE I (CNC MEM/USER)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	S1	0	08:58:02
X	0.000	G00	X	0.000	S	
Y	44.090				SOV	100%
Z					N	00000000
A					T	0 M 0
B						
C						

XY-LINEAR POINTS(SAME INTERVAL)- INSERT

HOLE POSITION [SKIP]

BASE POSITION(Z)	B=0.
START POINT(X)	H=10.
START POINT(Y)	V=10.
ANGLE	A=45.
HOLE POS.TYPE	D=LENGTH
LENGTH/PITCH	E=35.
NUMBER OF HOLE	C=5.

KEY IN NUMERALS.

O:PROGRAM NUMBER

54

INSERT CANCEL

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"> [LENGTH]: Gives the distance between the first and last drilling and the number of drillings. [PITCH]: Gives the distance between two adjacent drillings and the number of drillings.
E	Length / Pitch	<ul style="list-style-type: none"> [LENGTH]: Distance between the first and last drillings (if [LENGTH] is selected for point D). [PITCH]: Distance between two adjacent drillings (if [PITCH] is selected for point D).
C	Number of hole	Number of drillings

Skip

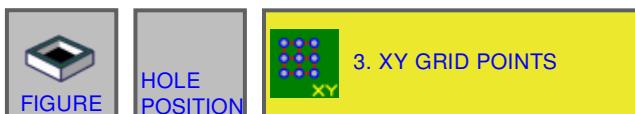
Data item		Meaning
F*	Omitting point 1	Point at which no hole is drilled (1)
I*	Omitting point 2	Point at which no hole is drilled (2)
J*	Omitting point 3	Point at which no hole is drilled (3)
K*	Omitting point 4	Point at which no hole is drilled (4)

Instructions:

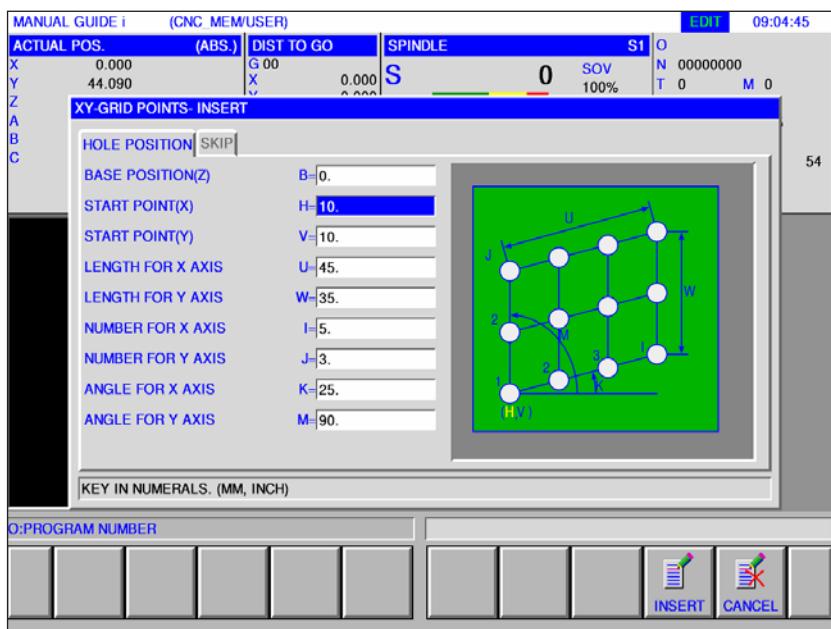
It is unnecessary to enter a value for a drainage position.

This applies also for cycles G1213 to G1216.





XY Grid points G1213



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

Hole position

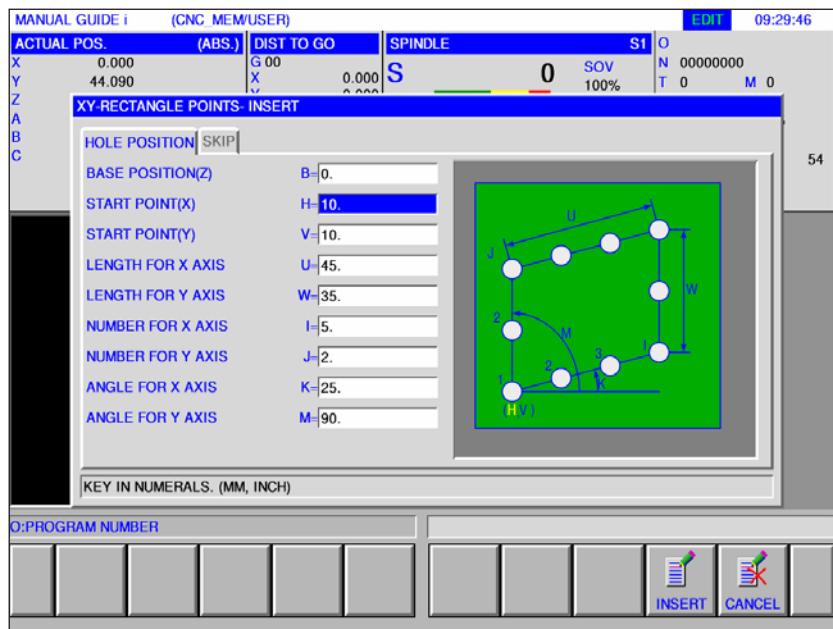
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Start point X	X-coordinates of the start point (first drilling) on a straight line.
V	Start point Y	Y-coordinates of the start point (first drilling) on a straight line.
U	Length for X-axis	Length of the first side of the 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

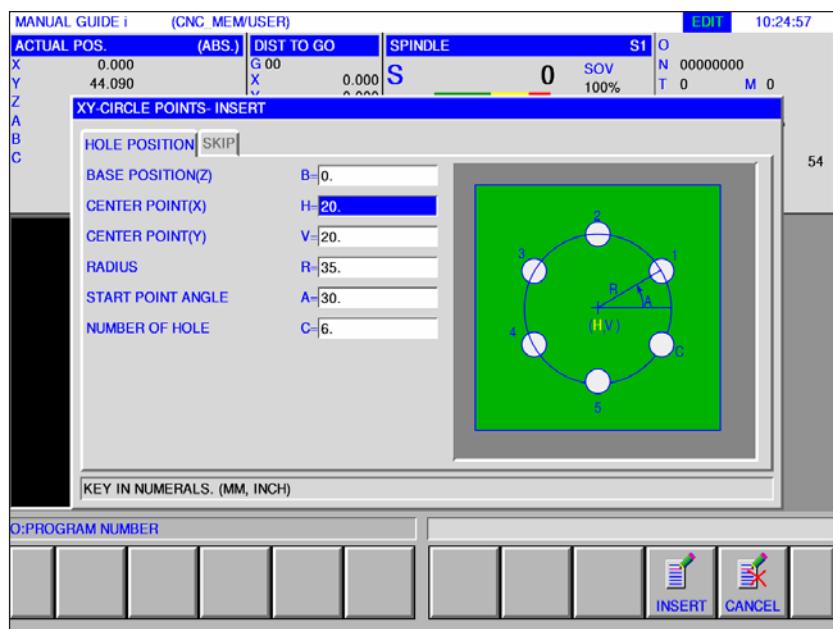
Data item		Meaning
B	Base position Z	Z-coordinates of the workpiece surface.
H	Start point X	X-coordinates of the start point (first drilling) on a straight line.
V	Start point Y	Y-coordinates of the start point (first drilling) on a straight line.
U	Length for X-axis	Length of the first side of the 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



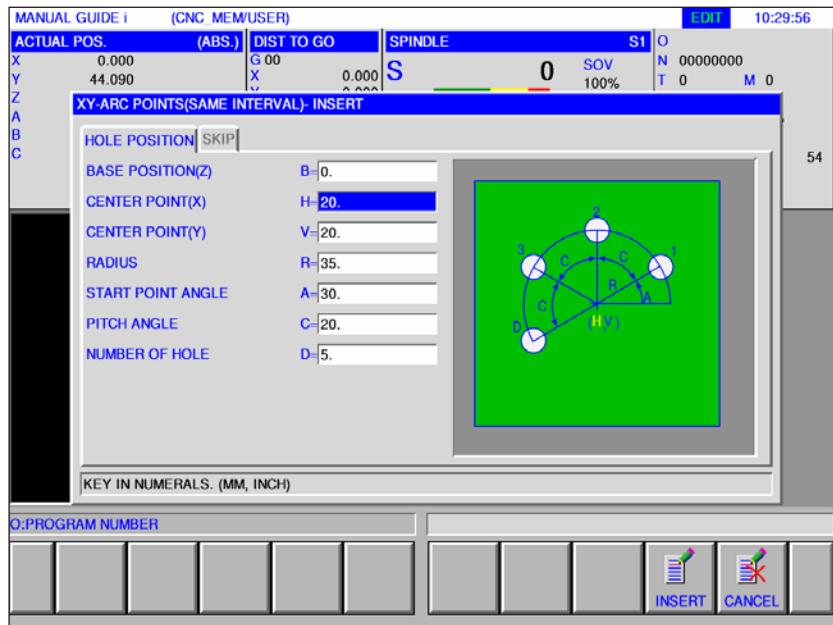
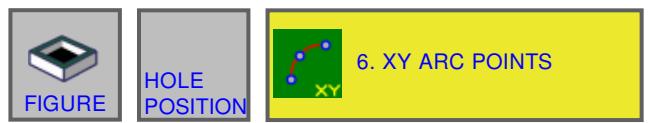
Entry fields marked with * are optional and should not be populated.

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

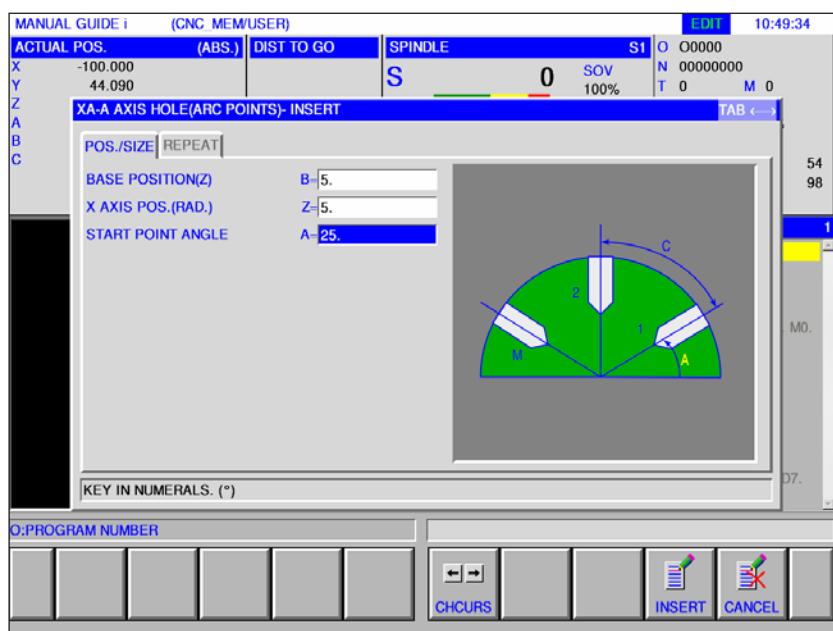
Entry fields marked with * are optional and should not be populated.

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).

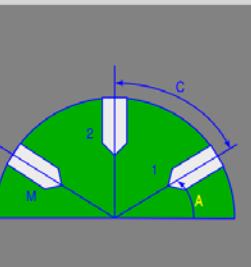
Skip

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.

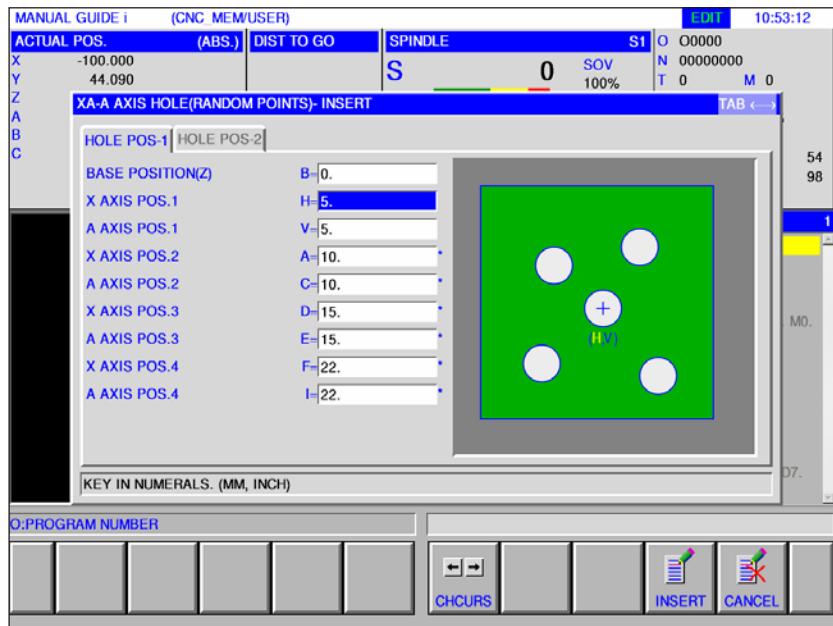


Machining Hole Position		
Data element		Meaning
B	Basis position Z	Z-coordinates of the work piece surface.
Z	X-axis position radius	X-coordinate of the hole position (positive value).
A	Start point angle	Centre angle to the X-axis of the first hole (positive or negative value) (initial value = 0).

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



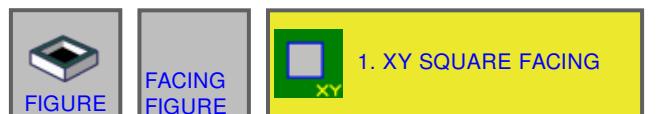
Entry fields marked with * are optional and are not required to be populated.

Machining Hole Position

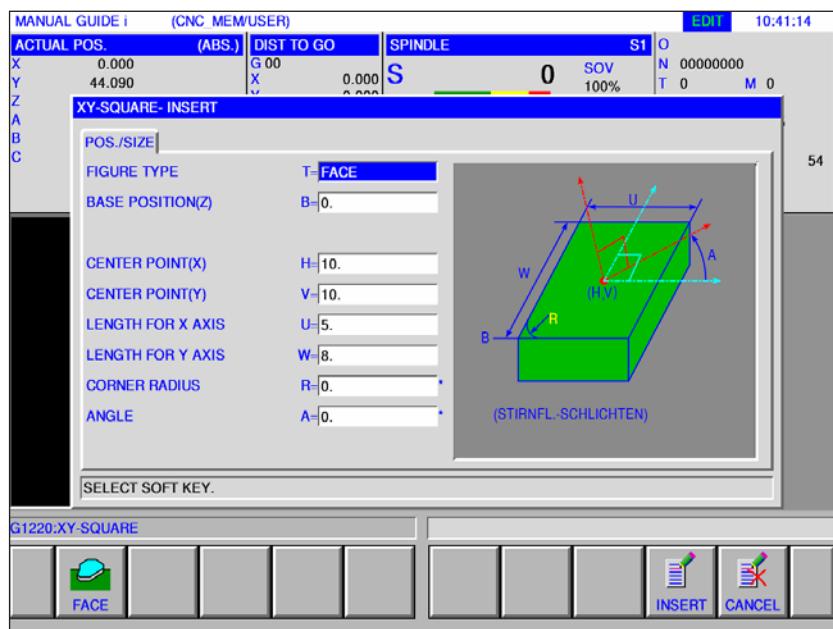
Data element		Meaning
B	Basis position Z	Z-coordinates of the work piece surface.
H	X-axis position 1	X-coordinates of the first hole.
V	A-axis position 1	A-coordinate of the first hole.
A*	X-axis position 2	X-coordinate of the second hole.
C*	A-axis position 2	A-coordinate of the second hole.
D*	X-axis 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



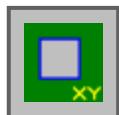
XY Square facing G1220



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).

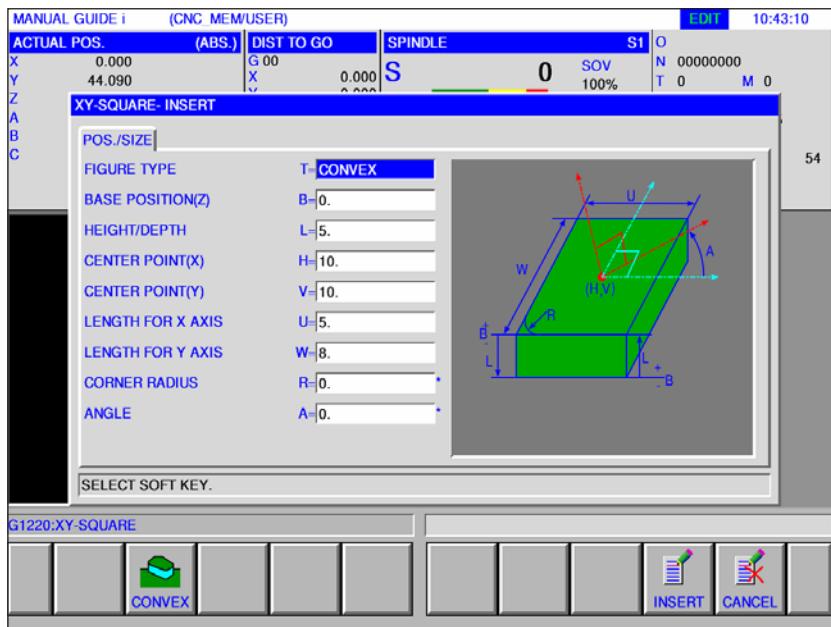
Figure: Contouring figure



- 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

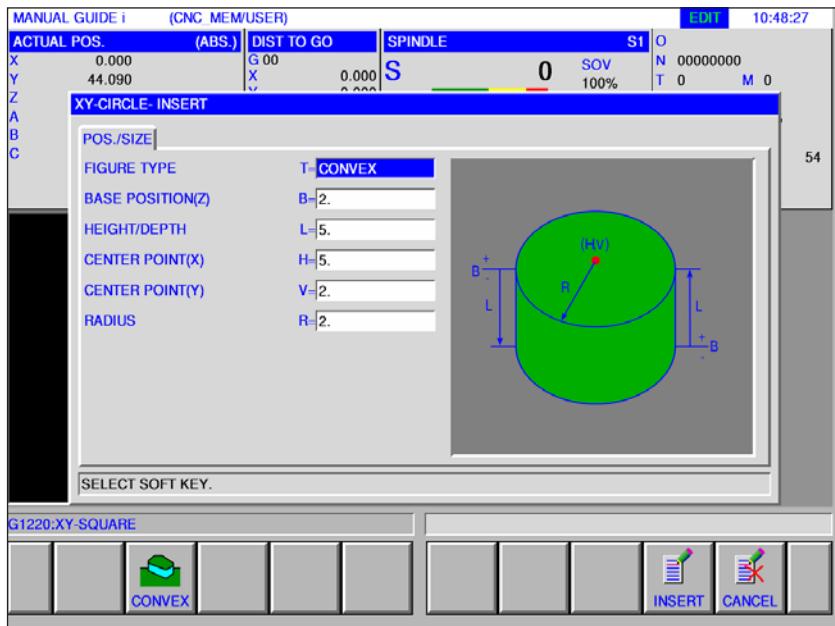


Entry fields marked with * are optional and should not be populated.

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).



XY Circle convex G1221

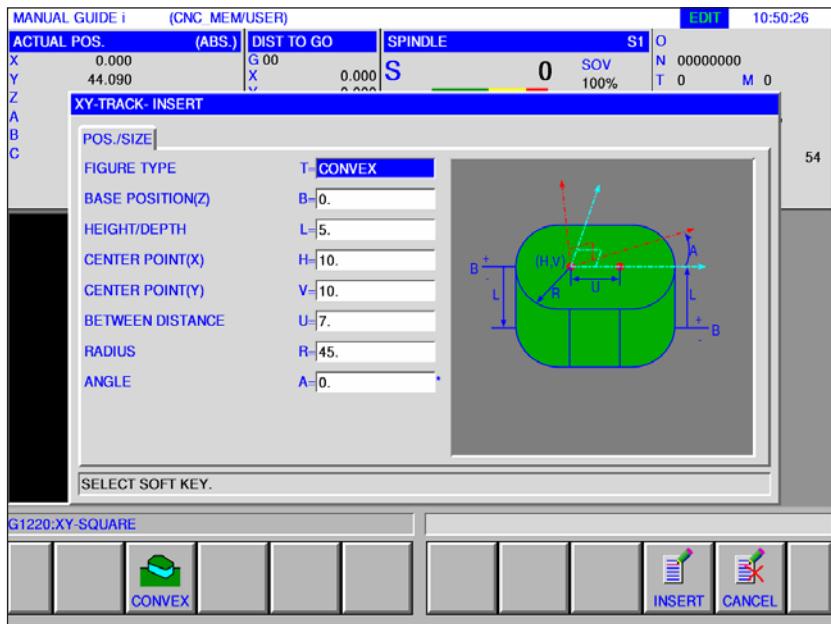


Entry fields marked with * are optional and should not be populated.

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

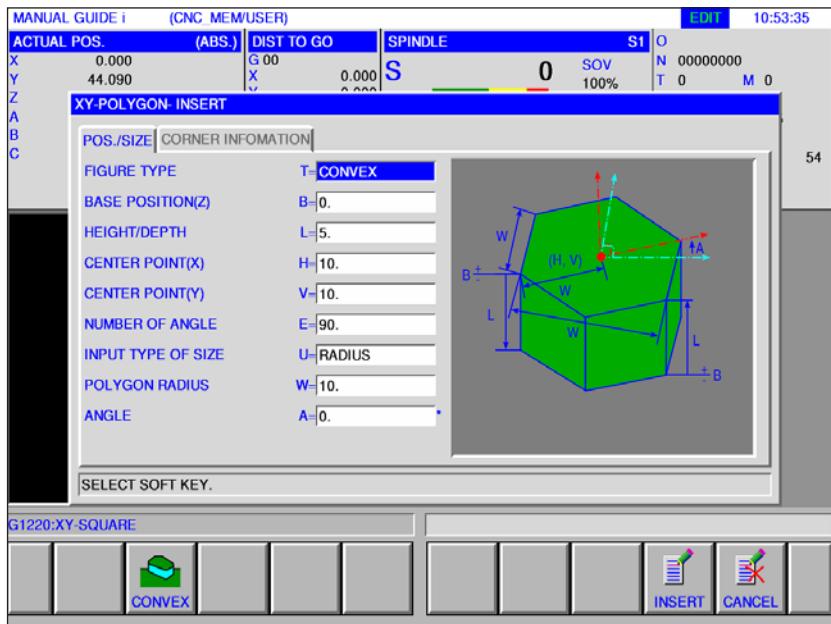


Entry fields marked with * are optional and should not be populated.

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)



XY Polygon convex G1225



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

Position / Size

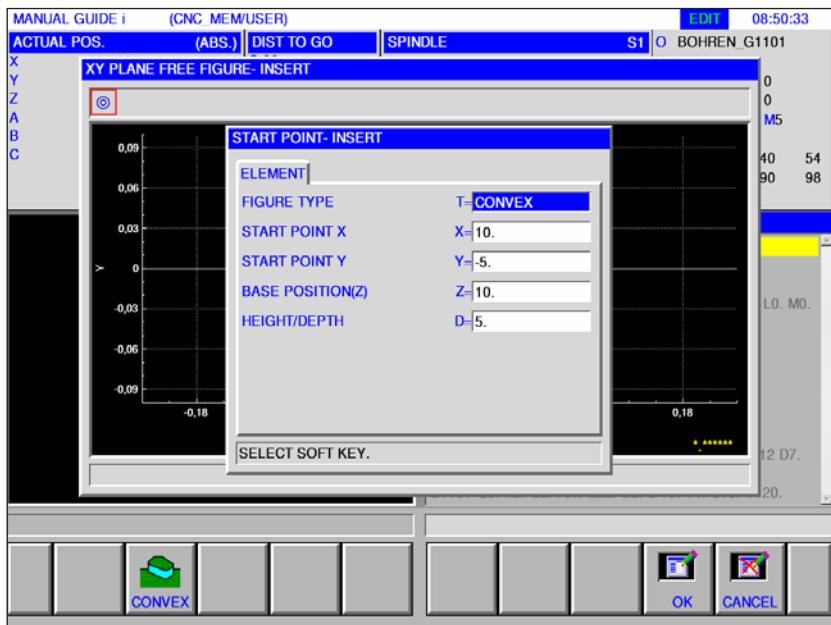
Data item		Meaning
T	Figure type	• [CONVEX]: used as outside contour for contour machining.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, the distance to the bottom 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.
E	Number of angle	Number of edges, positive integer between 3 and 99.
U	Input type of size	• [RADIUS]: Radius • [LENGTH]: Edge length • [WIDTH]: Key width
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 st axis.

Corner information

Data item		Meaning
C	Corner type	• [NOTHIN]: • [CHAMFR]: • [ARC]:
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

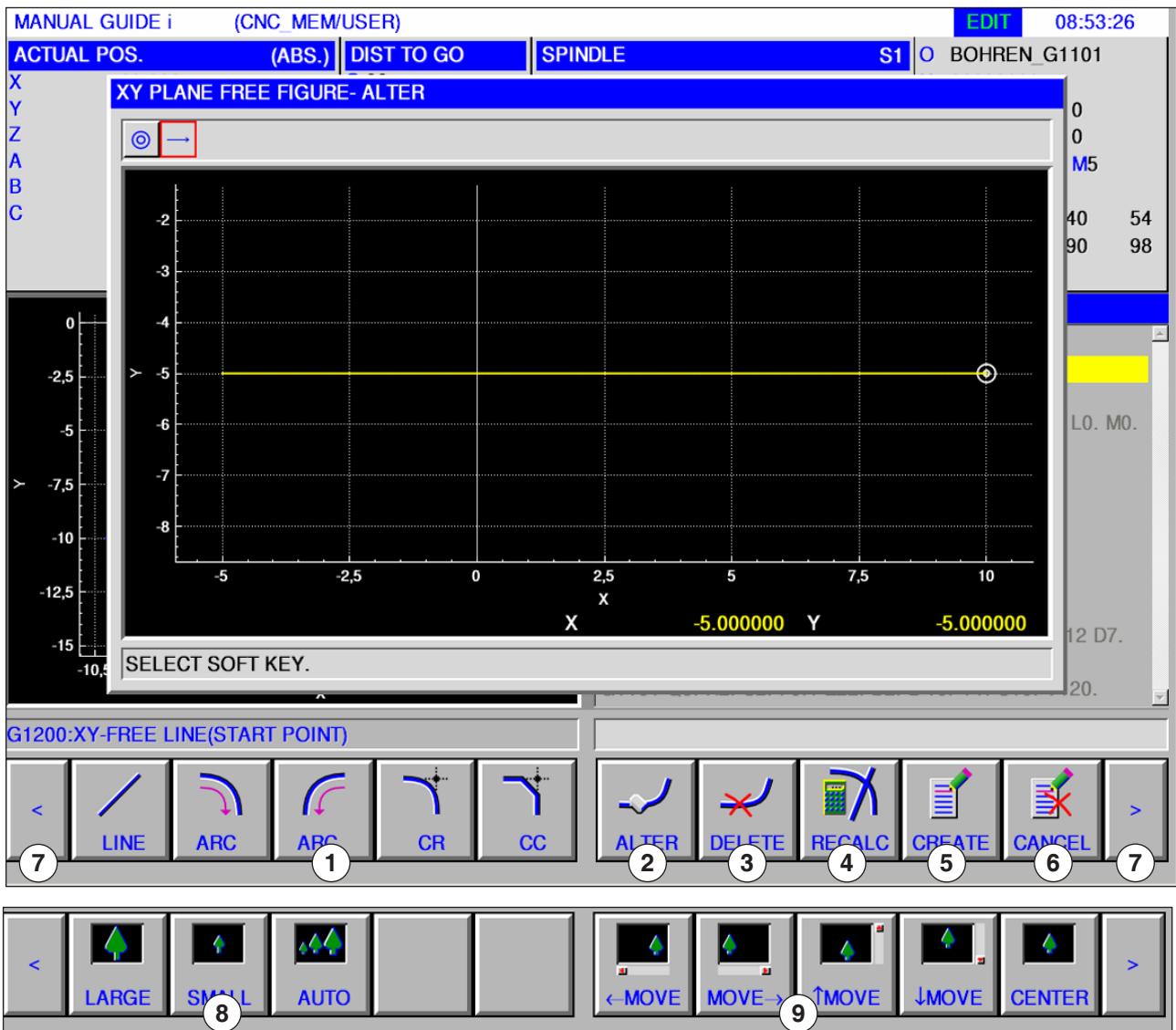
Instructions:

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

Instructions:

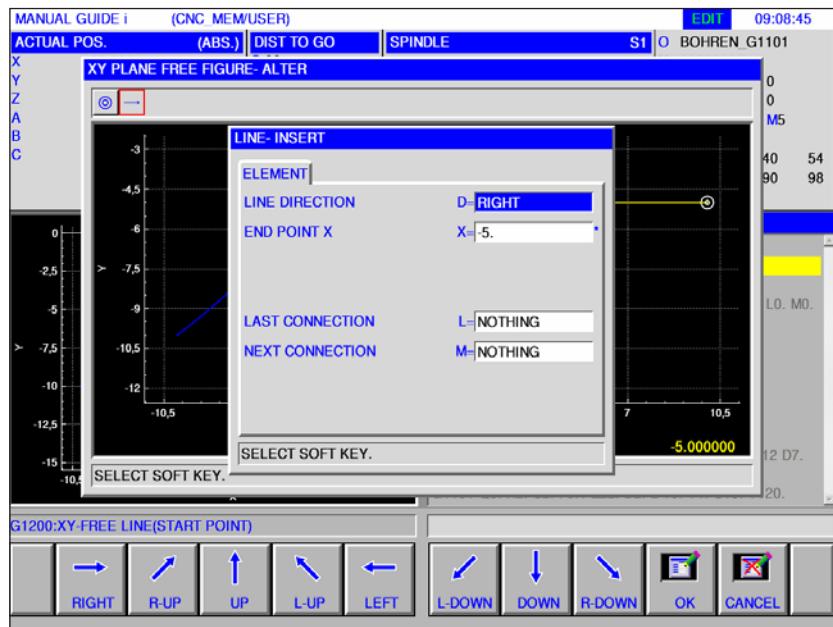
OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can call these up only for program display purposes.
This applies also for cycles G1201 to G1206.

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

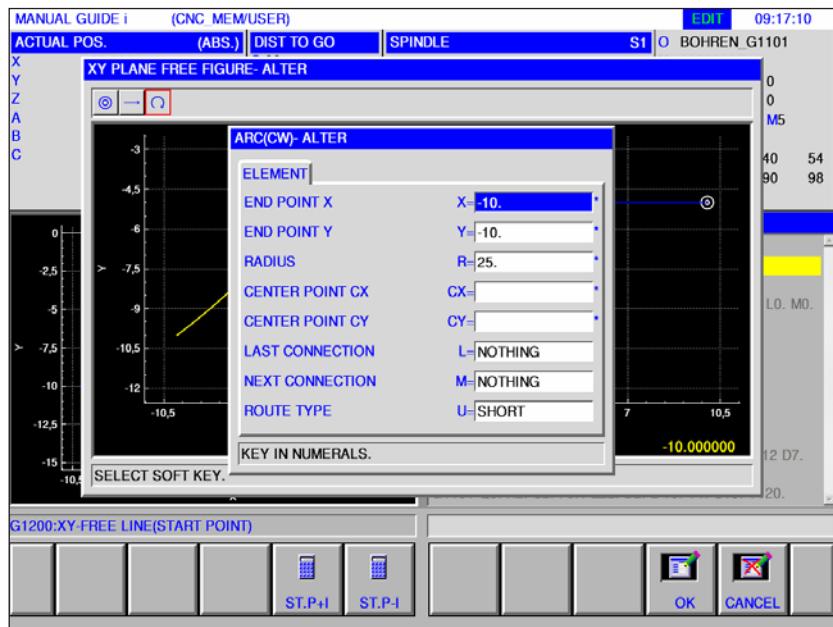


Entry fields marked with * are optional and should not be populated.

Element data input

Data item		Meaning
D	Line direction	The line direction is selected by means of a softkey in the toolbar. • [RIGHT]: • [R-UP]: • [UP]: • [L-UP]: • [LEFT]: • [L-DOWN]: • [DOWN]: • [R-DOWN]:
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"> • [TANGNT]: Contact with the immediately preceding contour. • [NO SET]: No contact with the immediately preceding contour (initial value).
M	Next connection	<ul style="list-style-type: none"> • [NO SET]: No contact with the immediately preceding contour (initial value).

Input items for arc (XY plane) G1202, 1203

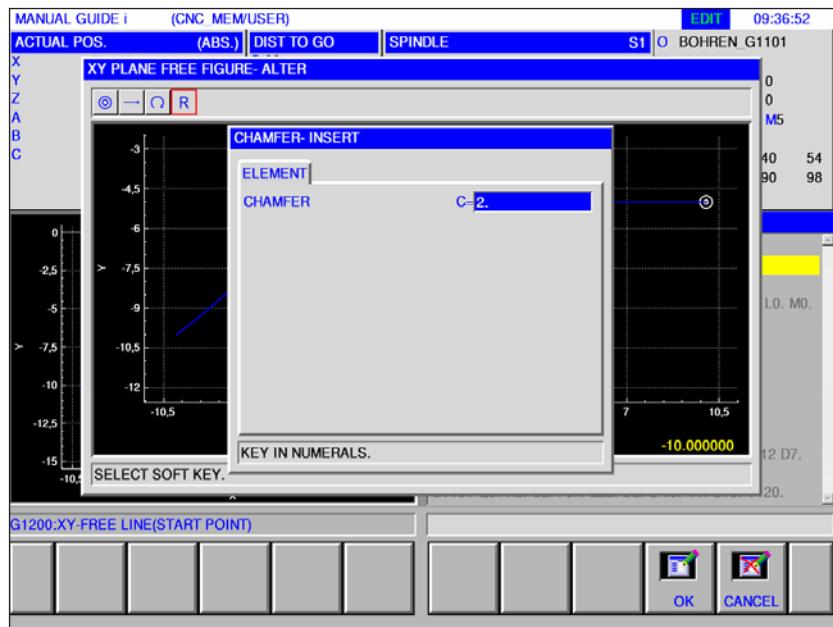


Entry fields marked with * are optional and should not be populated.

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"> [TANGNT]: Contact with the immediately preceding contour. [NO SET]: No contact with the immediately preceding contour.
M	Next connection	<ul style="list-style-type: none"> [NO SET]: No contact with the immediately preceding contour (initial value).
U	Route type	<ul style="list-style-type: none"> [SHORT]: An arc with a long route is generated. [LONG]: An arc with a short route is generated.

Input items of chamfer (XY plane) G1204

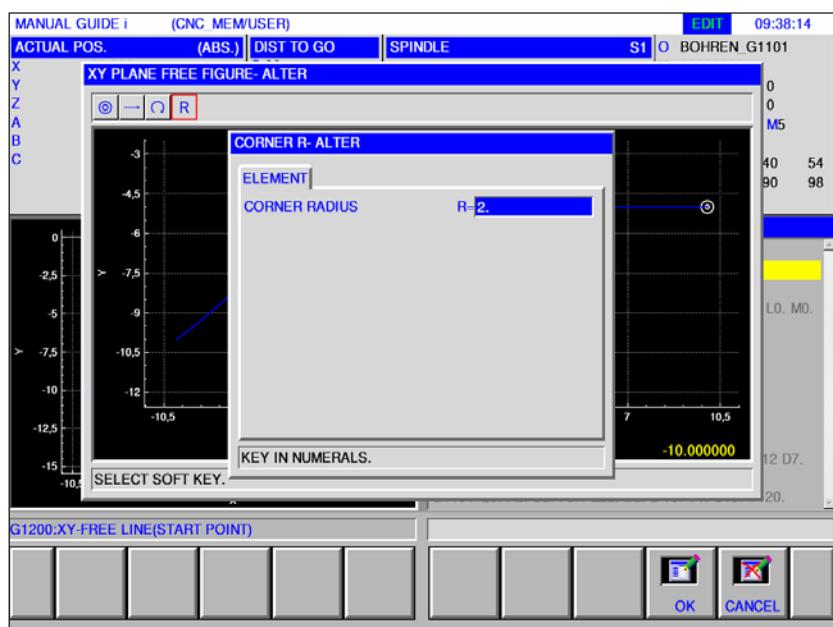


Entry fields marked with * are optional and should not be populated.

Element data input

Data item		Meaning
C	Chamfer	Chamfer, positive value

Input items of corner (XY plane) G1205

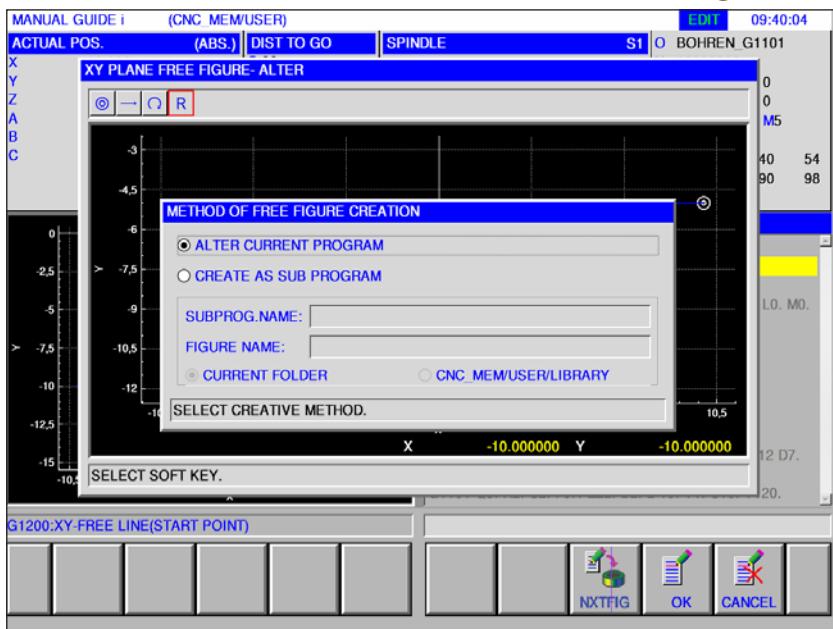


Entry fields marked with * are optional and should not be populated.

Element data input

Data item		Meaning
R	Corner radius	Radius value, positive value

Input items of end of arbitrary figures G1206



Close contour

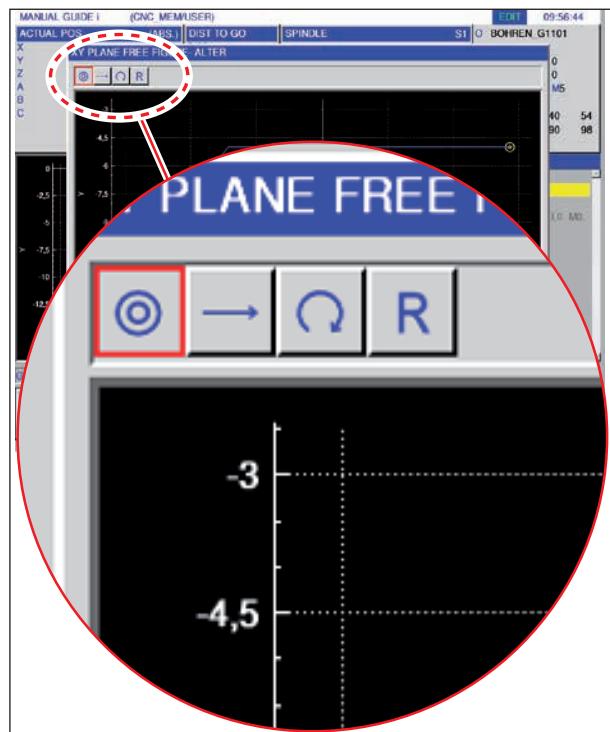
- Press softkey.

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

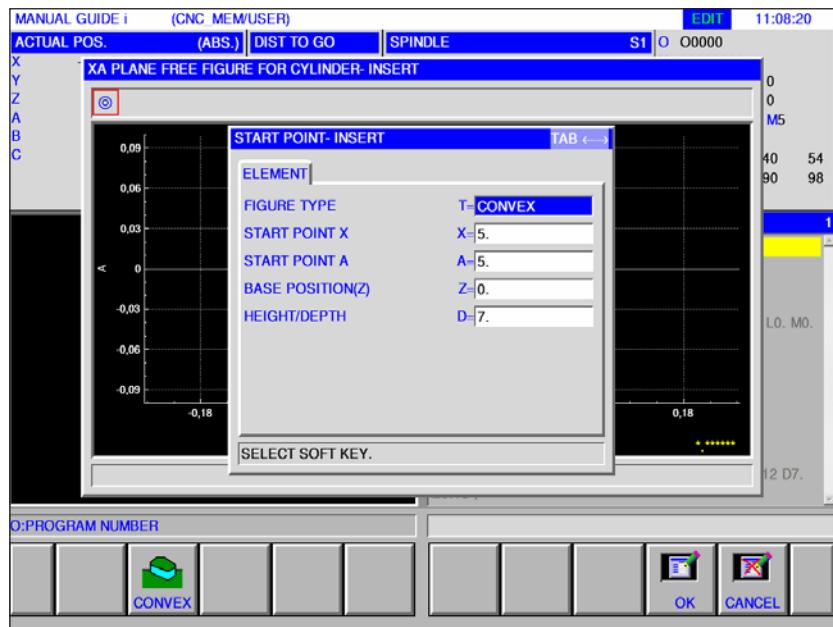


Contour element	Symbol	Meaning
Start point		Start point of the figure
Straight up Straight down		Straight line in 90° grid
Straight left Straight right		Straight line in 90° grid
Straight line anywhere		Straight line with arbitrary pitch
Curve right Curve left		Arc
Radius		
Chamfer		

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

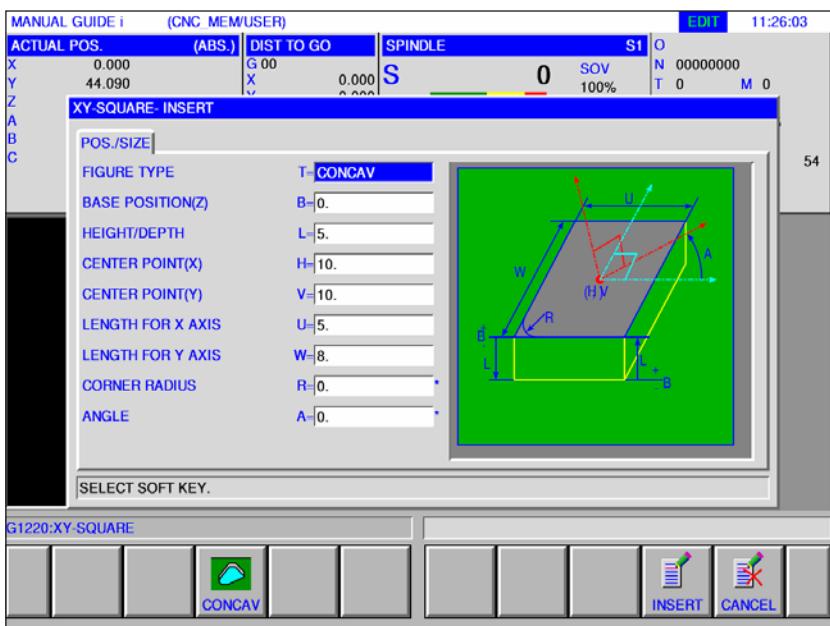
Element		Meaning
T	Figure type	• [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position Z	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the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

Note:

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



XY Square concave G1220

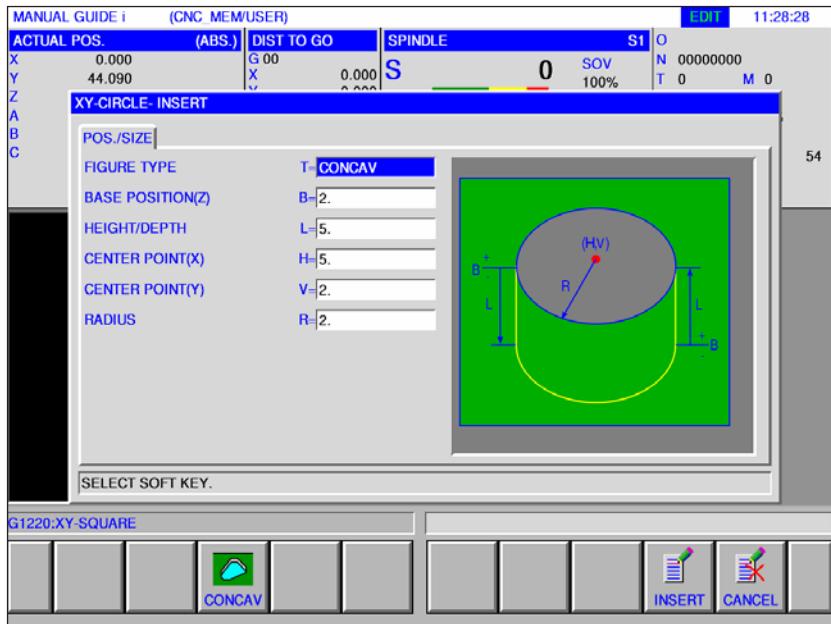


Entry fields marked with * are optional and should not be populated.

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



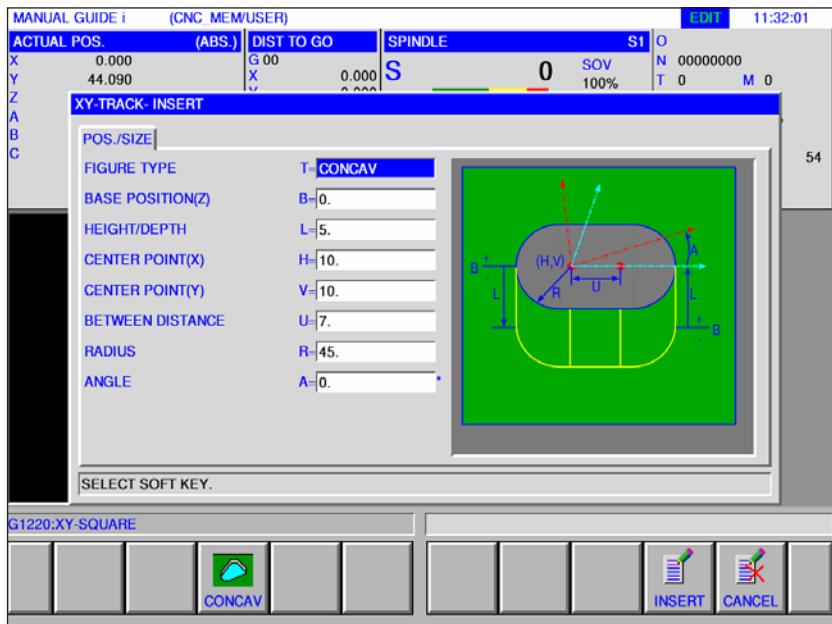
Entry fields marked with * are optional and should not be populated.

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



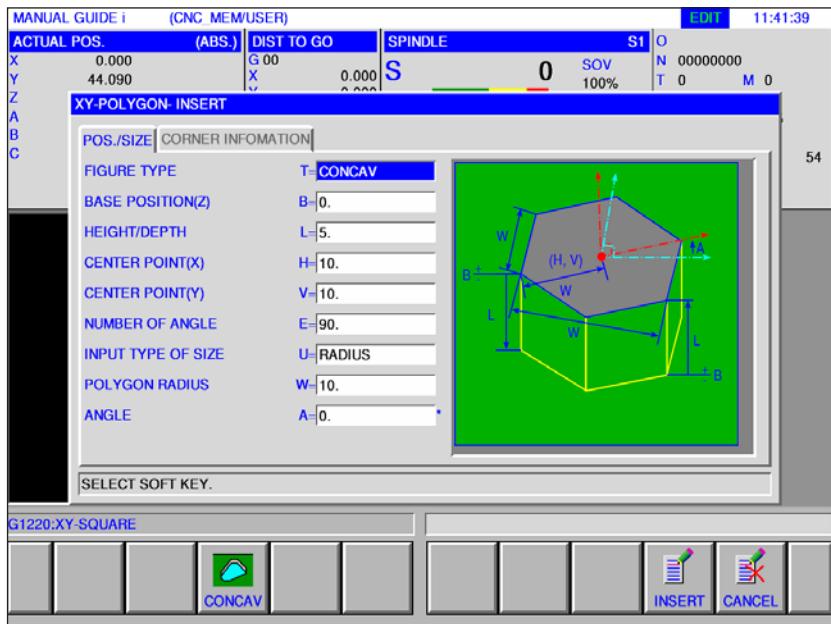
Entry fields marked with * are optional and should not be populated.

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



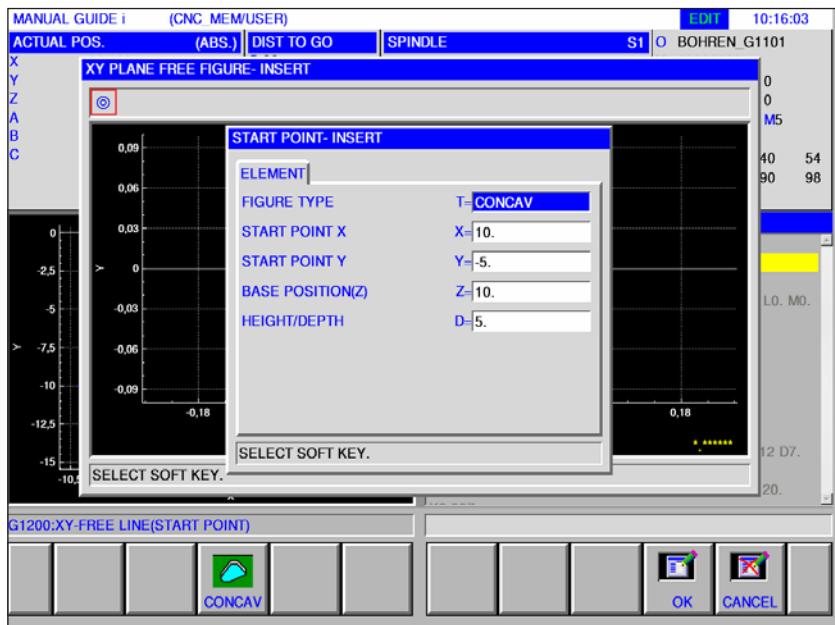
Entry fields marked with * are optional and should not be populated.

Position / Size	
Data item	Meaning
T	Figure type
B	Base position Z
L	Height / Depth
H	Centre point X
V	Centre point Y
E	Number of angle
U	Input type of size
W	Polygon radius, U1
A*	Angle

Corner information	
Data item	Meaning
C	[NOTHIN]: [CHAMFR]: [ARC]:
R*	Corner size



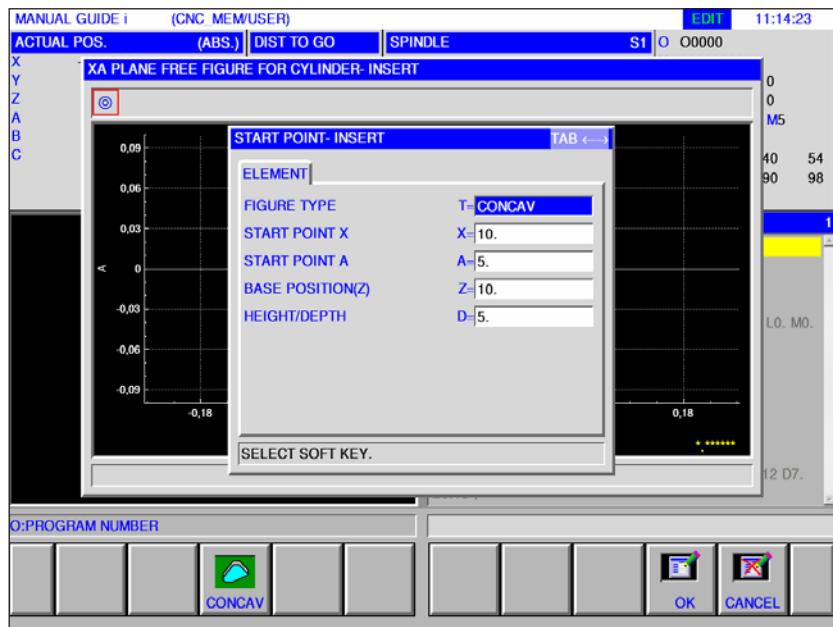
XY Free concave figure



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

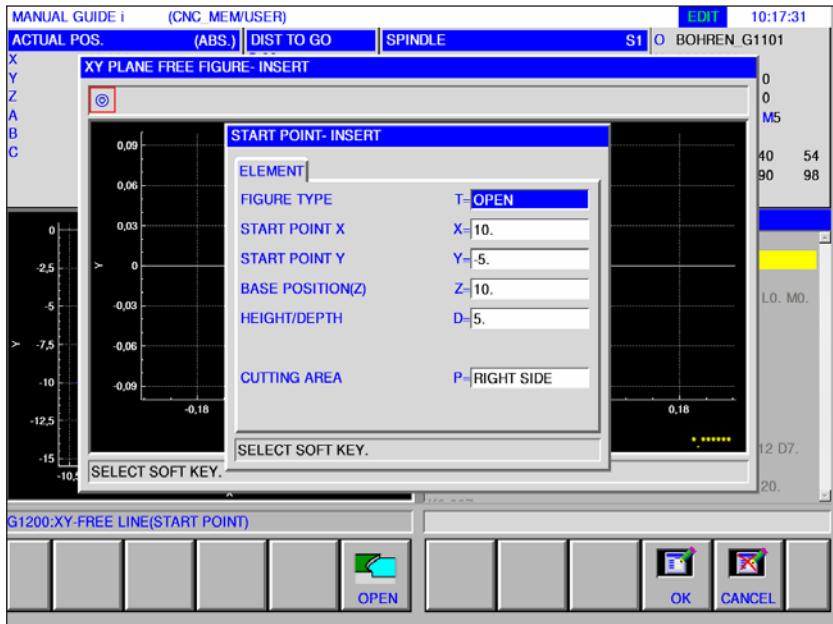
Element		Meaning
T	Figure type	• [OPEN]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position Z	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis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

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





XY Free open figure

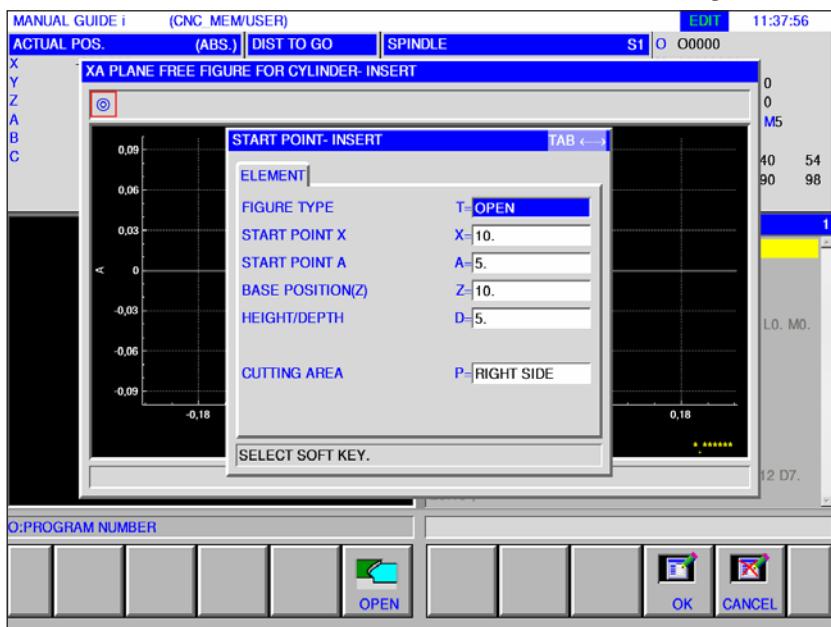


Entry fields marked with * are optional and should not be populated.

Start point		
Element		Meaning
T	Figure type	<ul style="list-style-type: none"> [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	<ul style="list-style-type: none"> [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

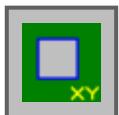
Data element		Meaning
T	Figure type	<ul style="list-style-type: none"> [CONCAV]: is specified and cannot be changed.
X	Start point X	X-coordinates of the contour startpoint.
Y	Start point Y	Y-coordinates of the contour startpoint.
Z	Base position (Z)	Position of the machining surface for any contour.
D	Height / Depth	Height of depth of the basis position to the cut surface
P	Cutting area	<ul style="list-style-type: none"> [RIGHT]: right side [LEFT]: left side

Note:

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



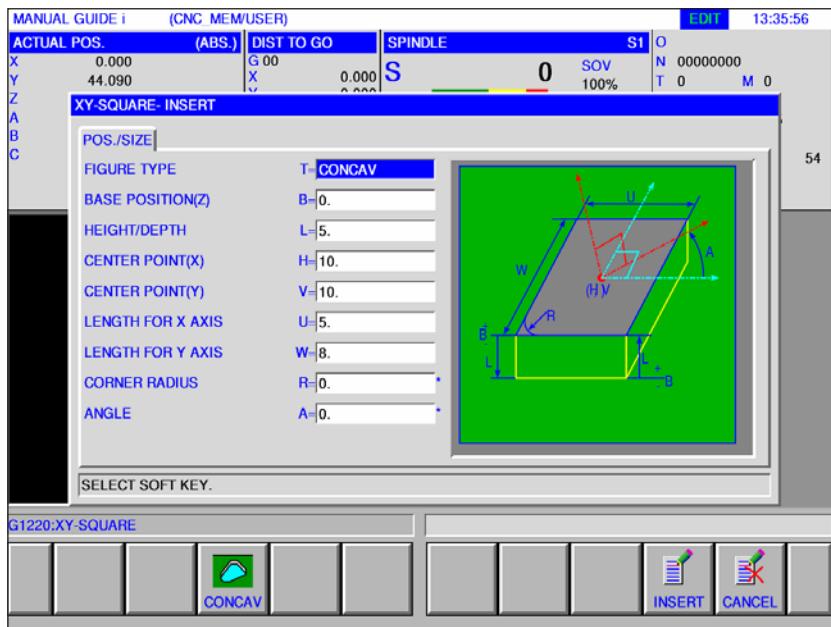
Figure: Pocket figure



- 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



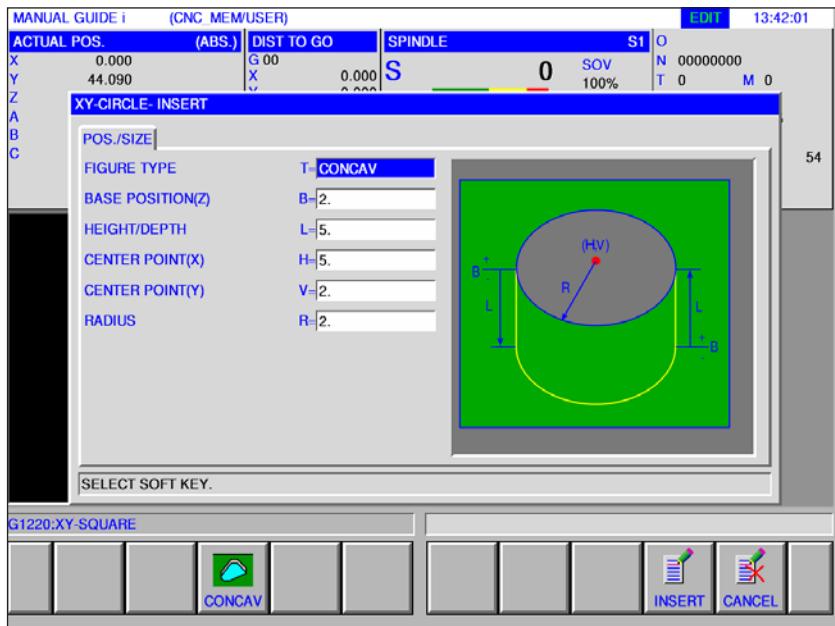
Entry fields marked with * are optional and should not be populated.

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



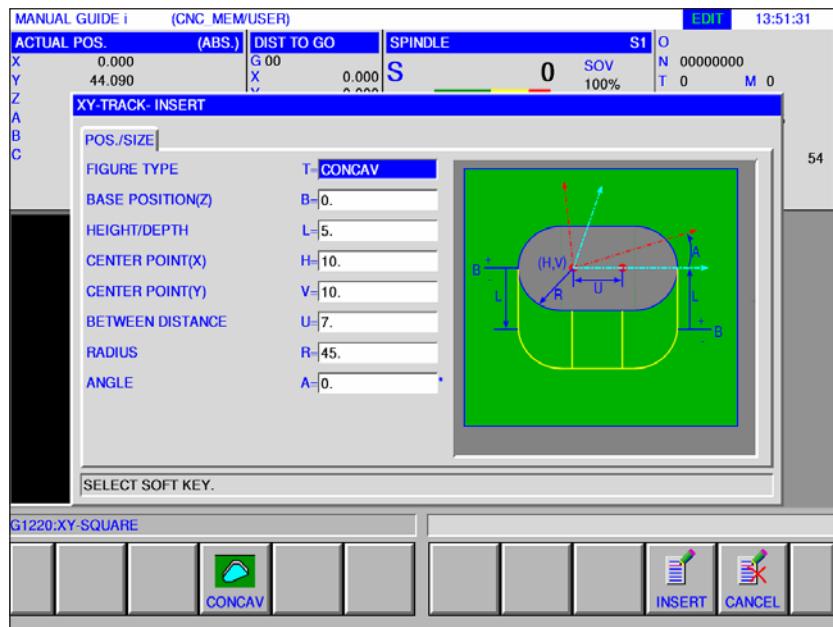
Entry fields marked with * are optional and should not be populated.

Position / Size

Element		Meaning
T	Figure type	<ul style="list-style-type: none"> [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

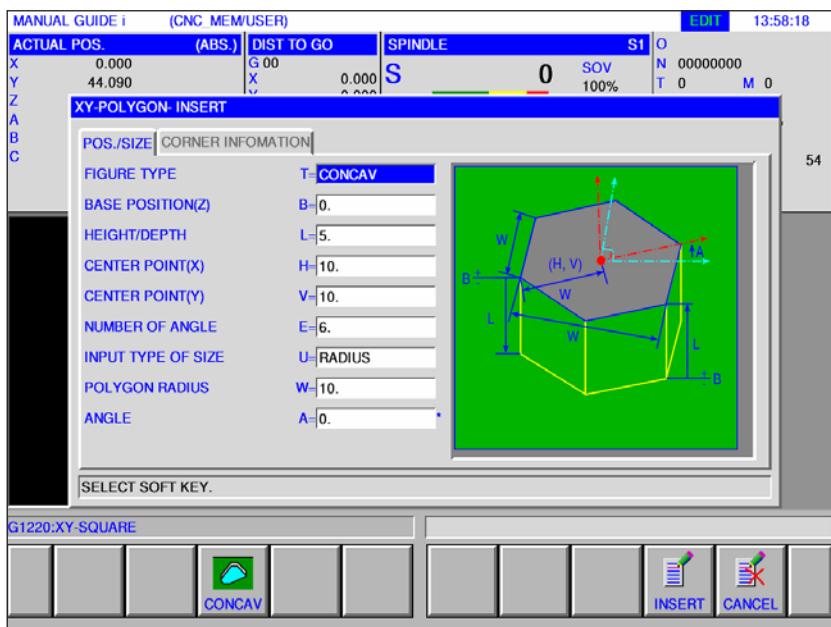


Entry fields marked with * are optional and should not be populated.

Position / Size	
Element	Meaning
T	Figure type
B	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	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	X-coordinates of the centre point of the left semicircle.
V	Y-coordinates of the centre point of the left semicircle.
U	Distance between the centre points of the right and left semicircles (radius value, positive value).
R	Radius of the left and right semicircles (Radius value, positive value)
A*	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

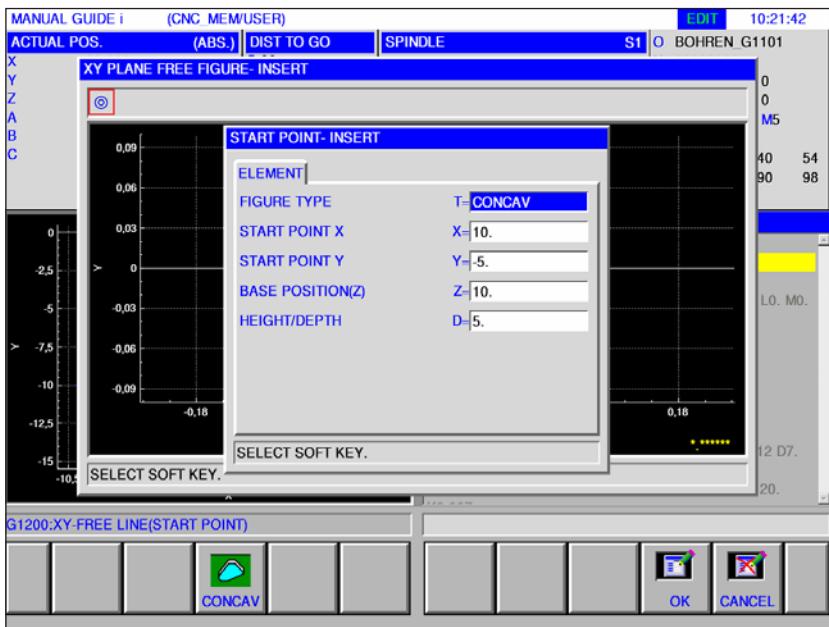
Element		Meaning
T	Figure type	• [CONCAV]: used as inside contour for contour machining or as contour for pocket milling.
B	Base position Z	Z-coordinates of the final surface when plane machining (in the tool axis direction).
L	Height / Depth	If the top surface of a workpiece is selected as BASISPOSITION, 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.
E	Number of angle	Number of edges, positive integer between 3 and 99.
U	Input type of size	• [RADIUS]: Radius • [LENGTH]: Edge length • [WIDTH]: Key width
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 st axis.

Corner information

Element		Meaning
C	Corner type	• [NOTHIN]: • [CHAMFR]: • [ARC]:
R*	Corner size	Chamfer radius



XY Free concave figure



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

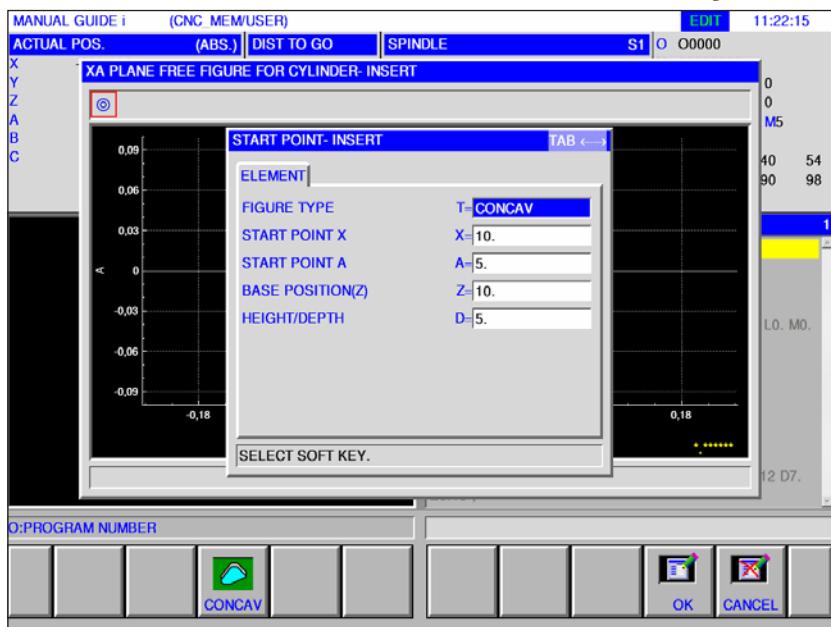
Insert startpoint

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".





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

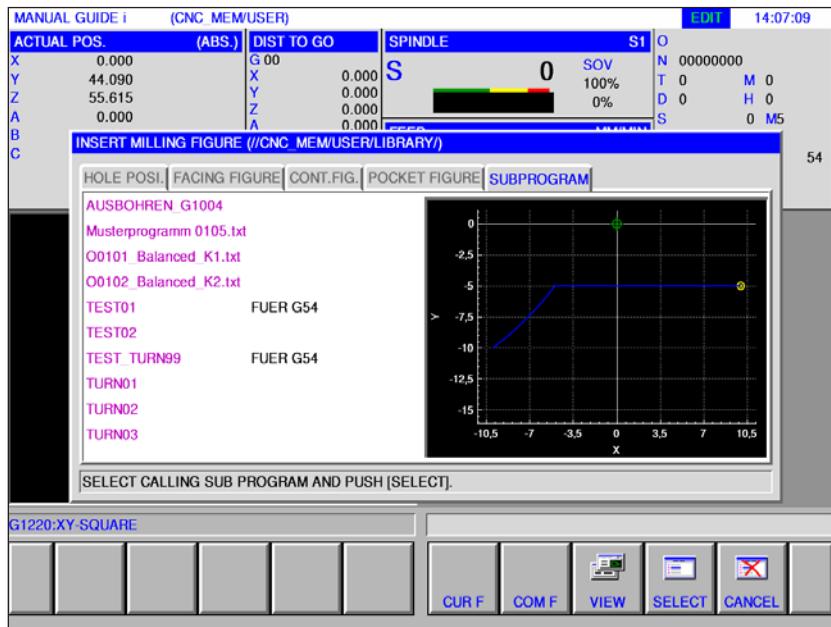
Note:

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

**Subroutines****Fixforms****M-code menu**



Sub-routines



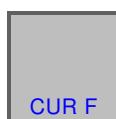
1 Select "Edit" mode.



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 suprogram. Press the softkey again to turn off the preview.

Programming

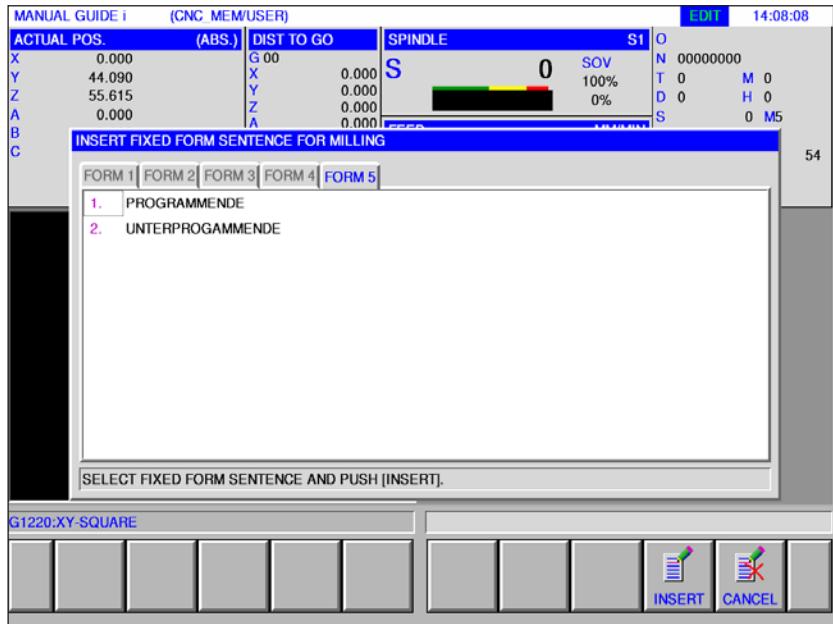
M98 Call subroutine

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.



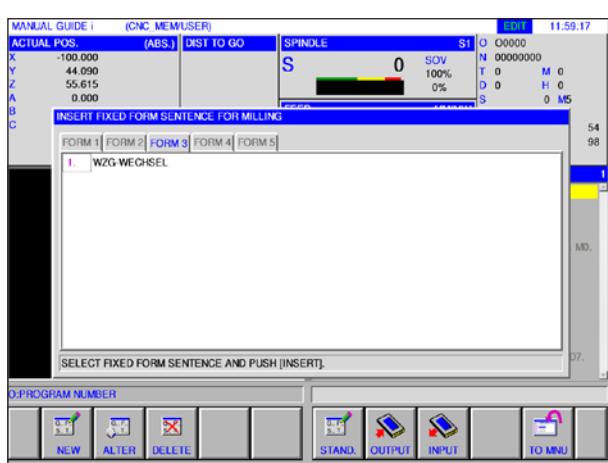
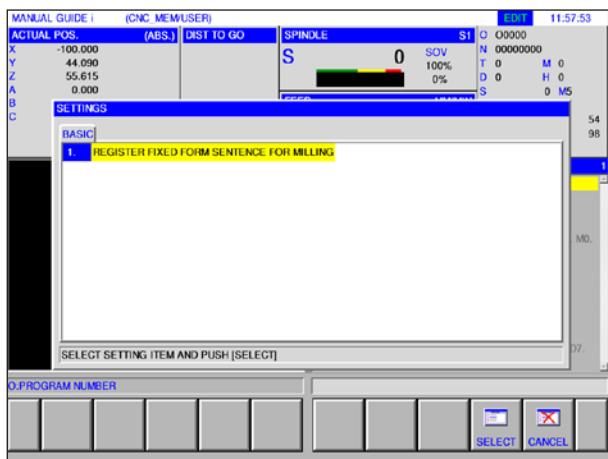
3 Select the desired Fixform with "INSERT" and insert it into the program.

Creating fixed forms

- 1 Press softkey "SETING" 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:

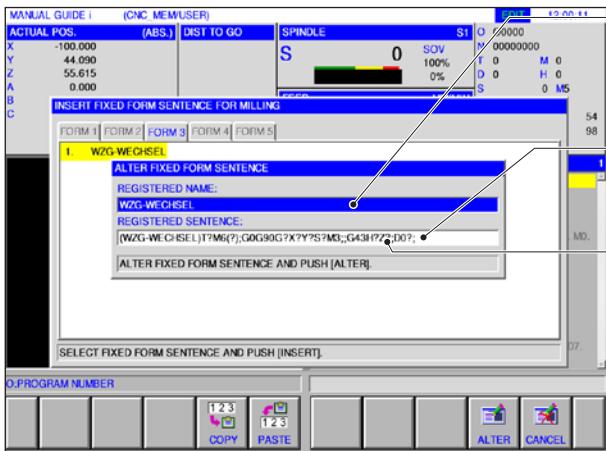
- The fixed form set menu displayed under the tab "FORM 1" has the same content as the menu displayed under "START".
- The fixed form set menu displayed under the tab "FORM 5" has the same content as the menu displayed under "END".



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



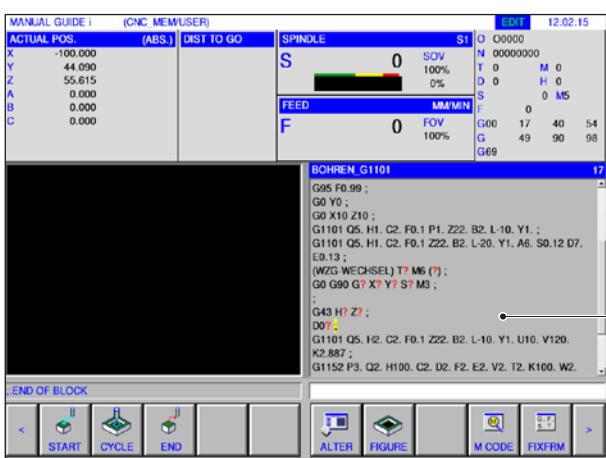
- 5 Press softkey to delete a fixed form.



6 Enter or change registration name.

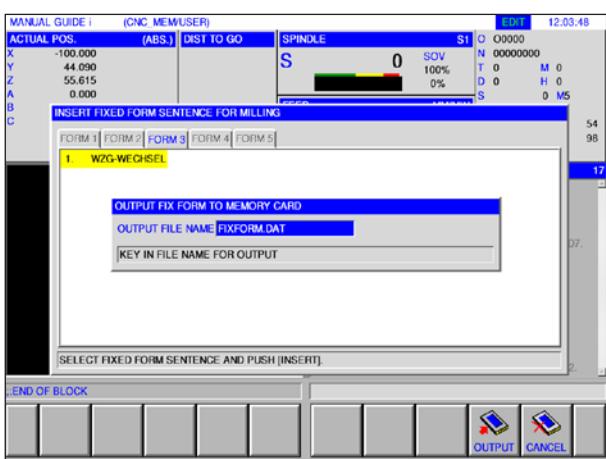
7 Enter or change registration set.

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

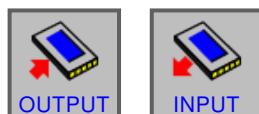


Example: Tool change for milling

(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		

Command abbreviations overview

Part 1 valid for turning and milling

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		Absolute dimension programming
G91		Incremental dimension programming
G94	10	Feed in mm/min
G95		Feed in mm/revolution

Brief description of G-commands

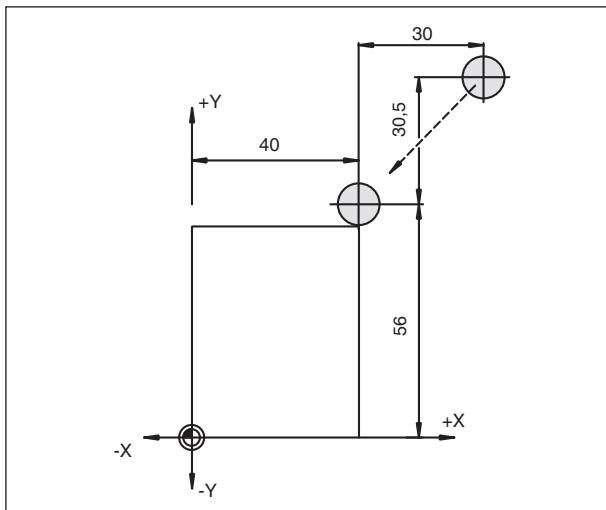
This description is an extract from the programming instructions for the WinNC for Fanuc 31i controller and is considered in first line as a programming aid.

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).



Absolute and incremental dimensional data

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

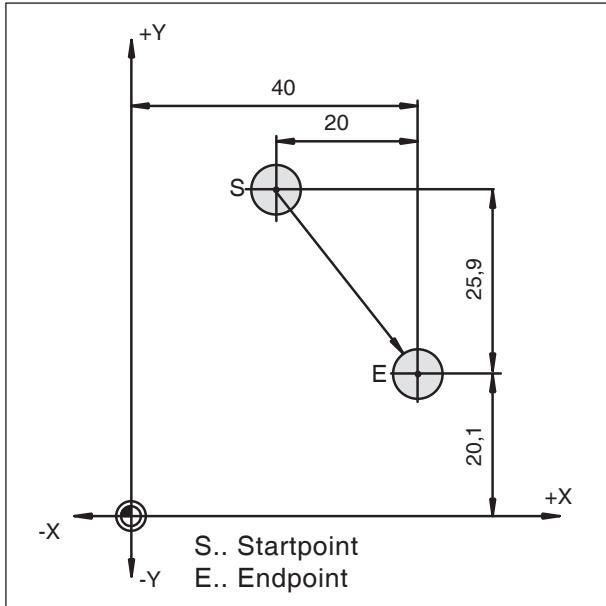
absolute G90

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

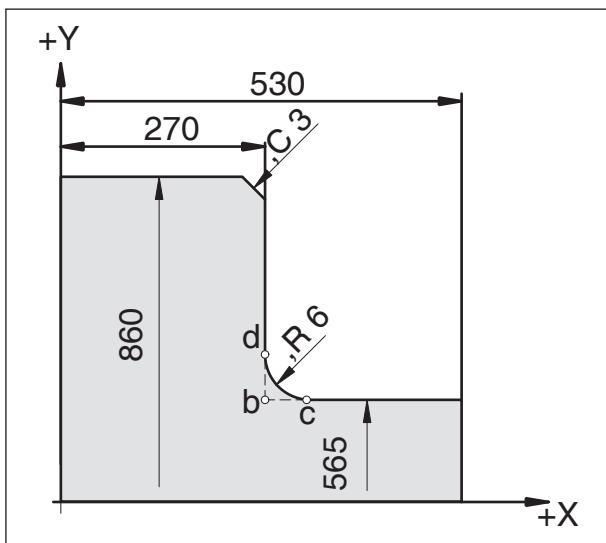
N.. G94

.....
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

Entering chamfers and radii

Format

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

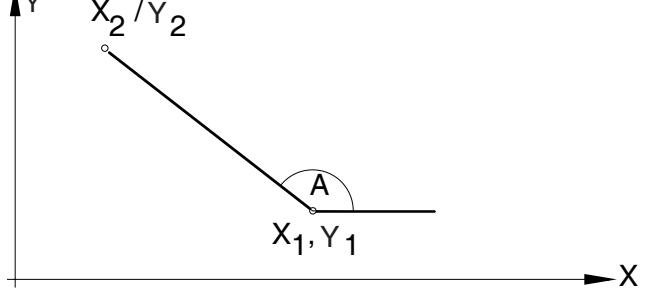
Notes

- 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

Direct drawing dimension input

	Commands	Tool movements
1	$X_2 \dots (Y_2 \dots) , A \dots$	 <p>The diagram illustrates a tool movement in a 2D coordinate system. A horizontal line represents the X-axis and a vertical line represents the Y-axis. A straight line segment connects two points: one labeled (X_1, Y_1) and another labeled (X_2, Y_2). The angle between the positive X-axis and the line segment is labeled A. The origin of the coordinate system is indicated by a small circle.</p>

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:

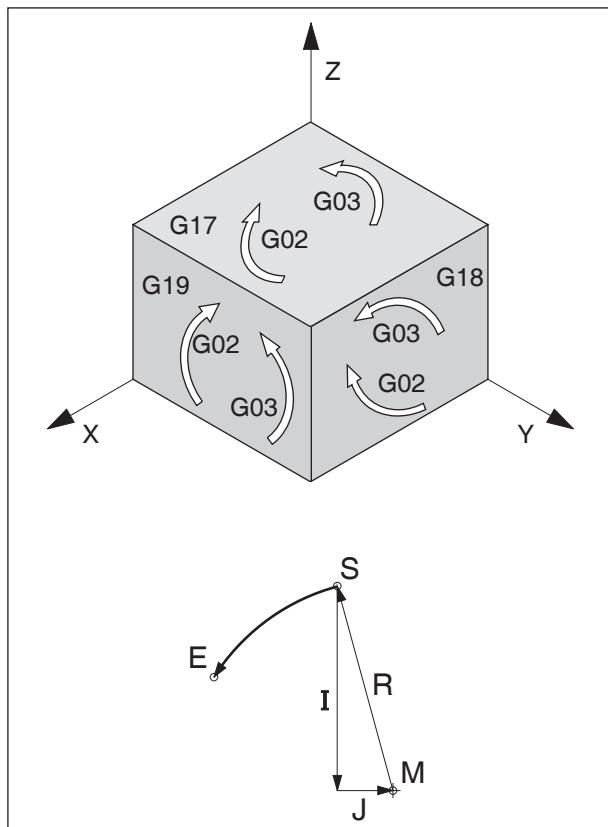
G02, G03,

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



G02 and G03 turning directions

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

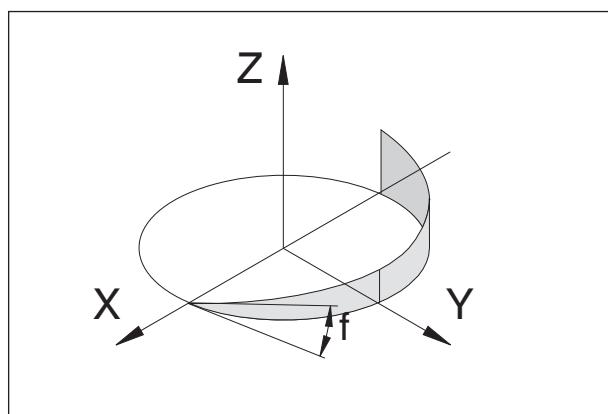
I, J, K Incremental circular parameter
(Distance from start point to the centre of the circle, I is assigned to the X-axis, J to the Y-axis, K to the Z-axis)

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.



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]

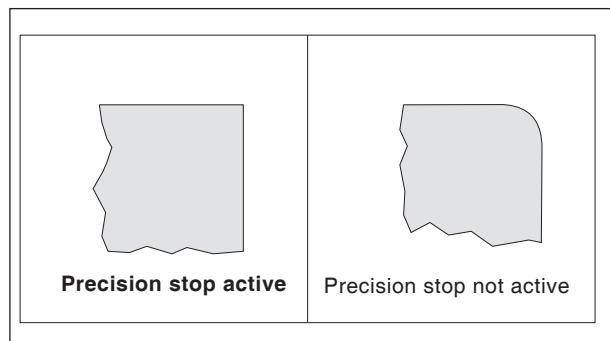
The tool is held for the time defined under X or P (in the last position reached) - sharp edges - clean transfers, mill base, precise stop.

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)

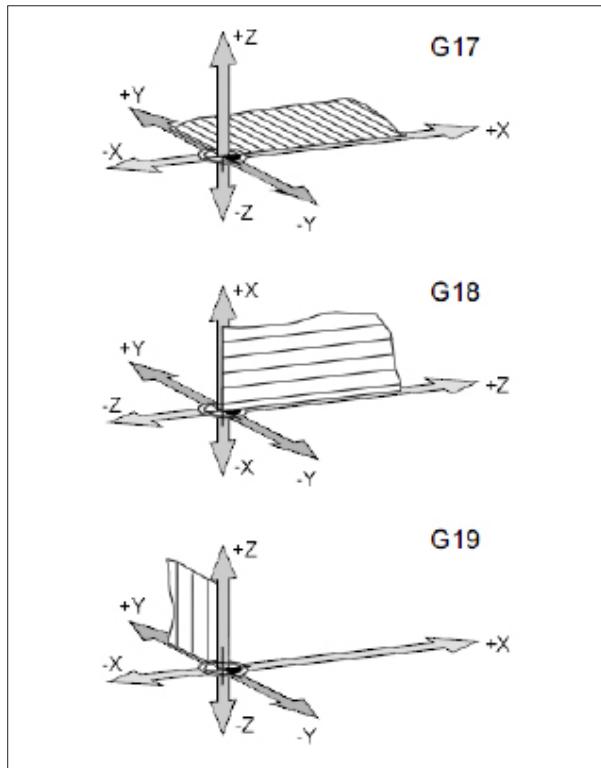
Format

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.

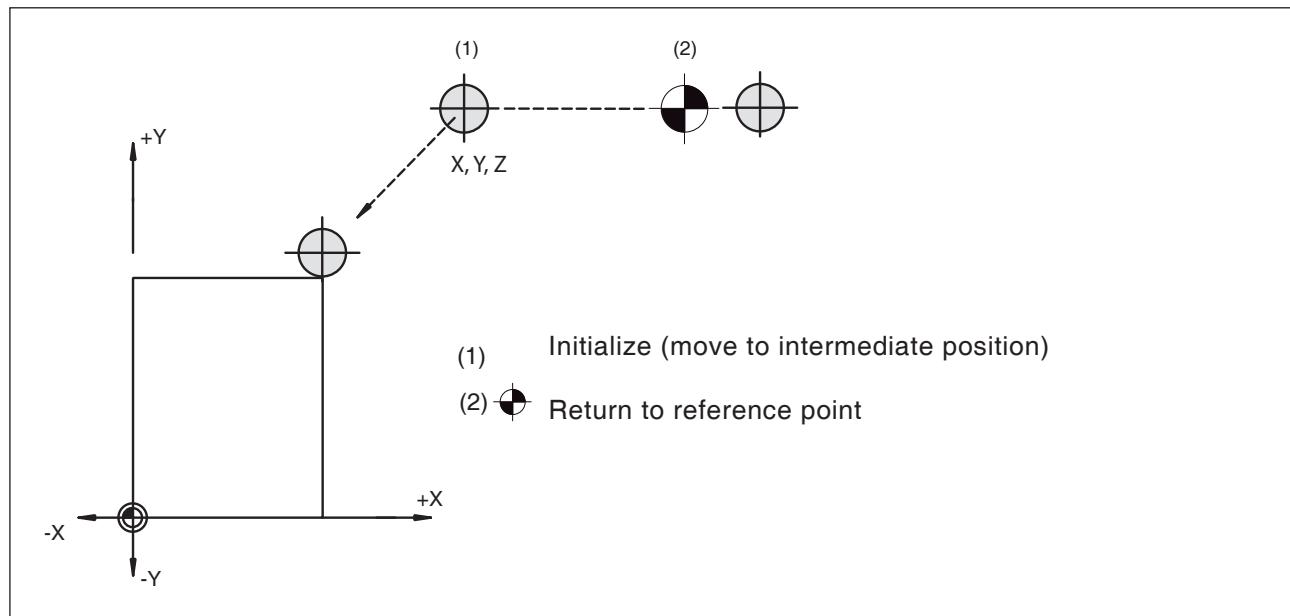
Both movements with G0.

The movement to the intermediate position can be programmed incrementally:

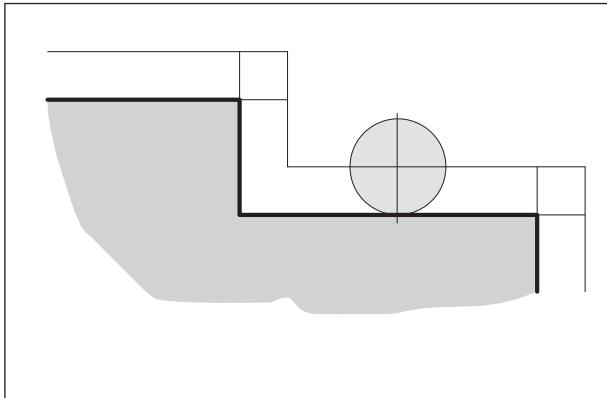
G-Code group B/C

G91 G28 X10 Y10 Z10

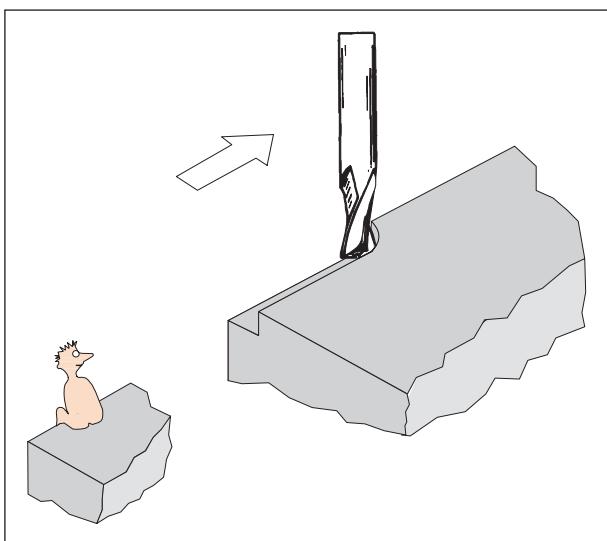
G90



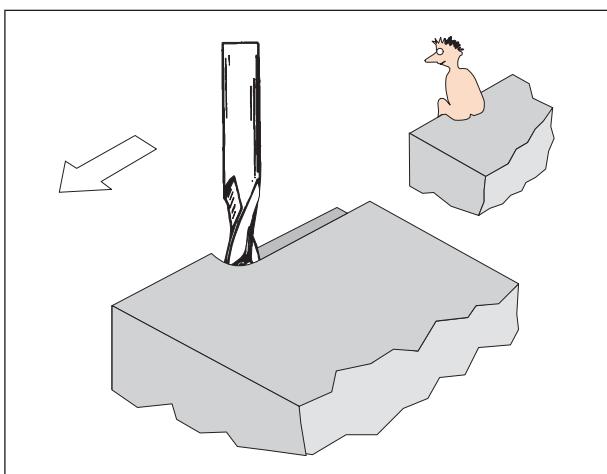
Return to reference point



Radius offset tool path



Definition of G41 Mill radius correction left



Definition of G42 Mill radius correction right

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.

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

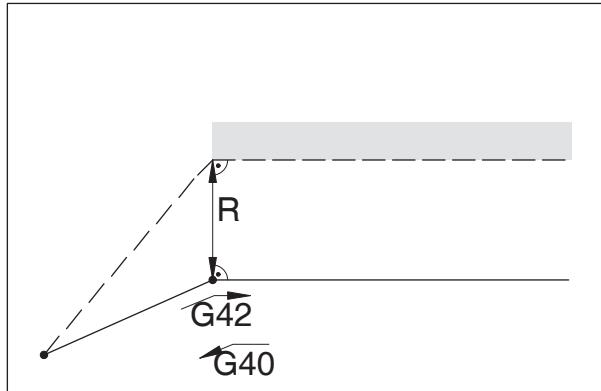
- A direct switch between G41 and G42 is not allowed - prior deselection with G40.
- Approach in connection with G00 or G01 required.
- The mill radius data is essential, the H parameter is effective only until it is deselected with H0 or a different H parameter is programmed.

G42 Mill radius offset right

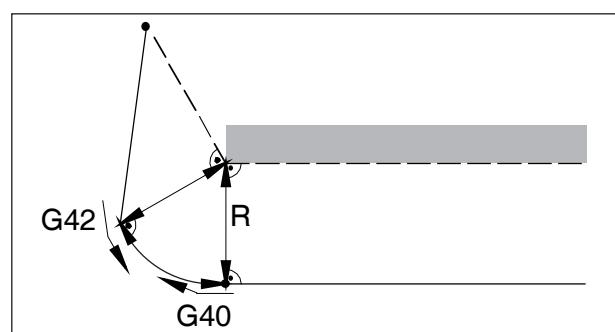
If the tool (viewed in the feed direction) is to the **right** of the contour to be machined, G42 must be programmed.

For instructions see G41!

Tool paths when selecting/deselecting mill radius offset

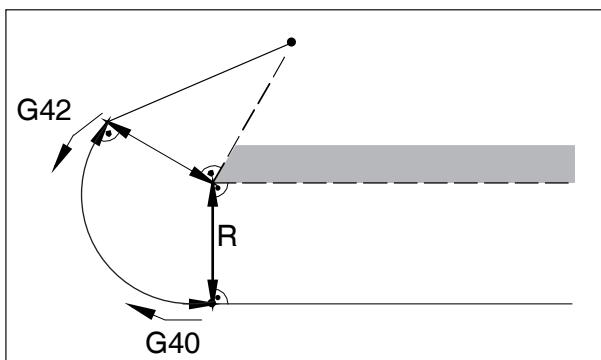


Approach or move away from a corner point from the front



Approach or move way from the side to the rear

— Programmed tool path
— Actual tool path



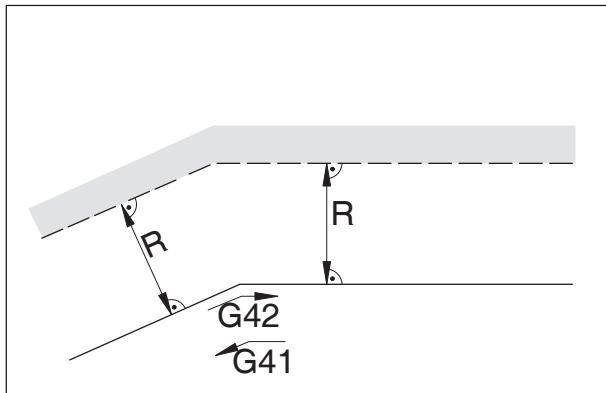
Approach or move away from a corner point from behind

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

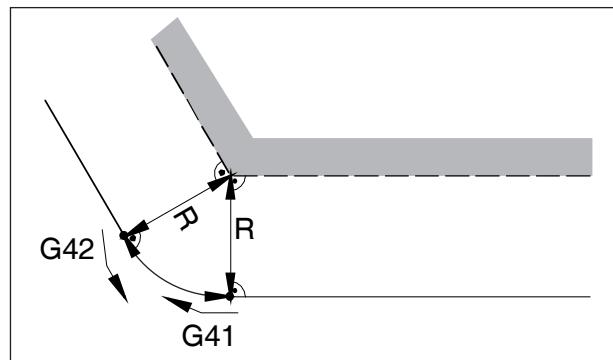
The approach path to the contour and the path away from the contour must be greater than the mill radius R, otherwise the program will stop with an alarm.

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

Tool paths in the program sequence with mill radius offset

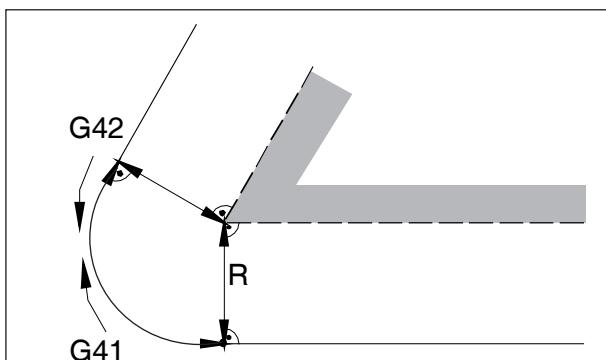


Tool path at an inside corner



Tool path at an outside corner $> 90^\circ$

— Programmed tool path
— Actual tool path



Tool path at an outside corner $< 90^\circ$

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.

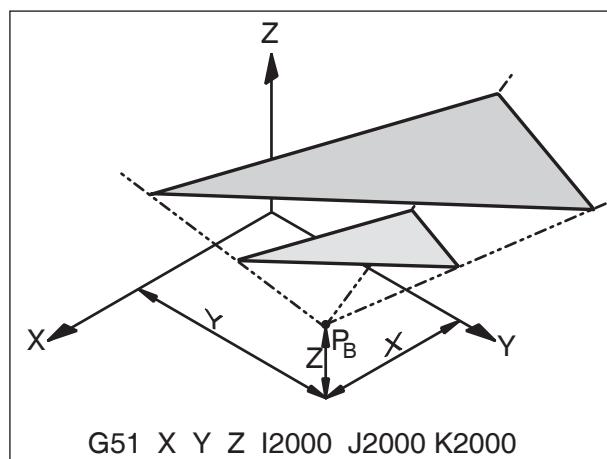
Example

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.



Enlarging a contour

G50 Deselection scaling factor

G51 Scaling factor

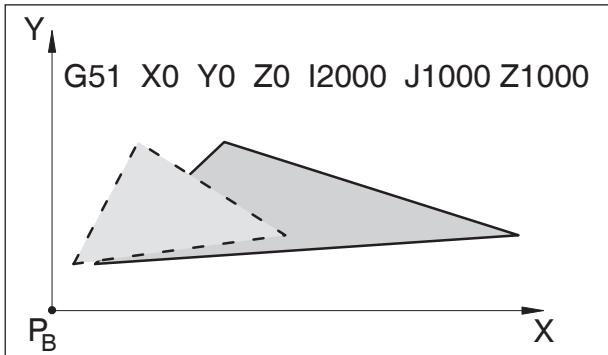
Format

N.. G50

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

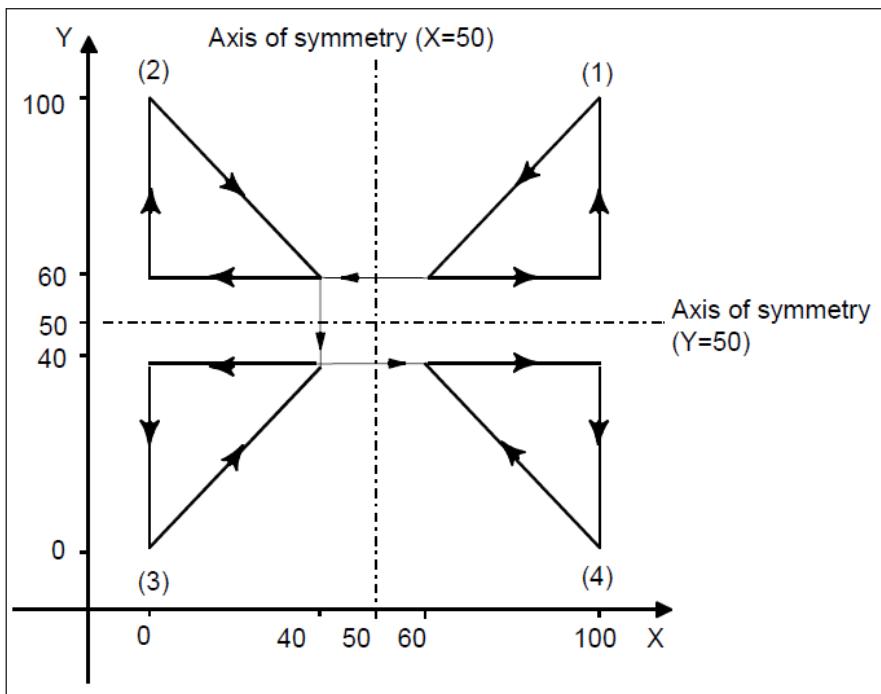
G51 computes all position data to scale, until the scale is deselected with G50. X, Y and Z determine a reference point P_B , from which all the dimensions are calculated.

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

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).

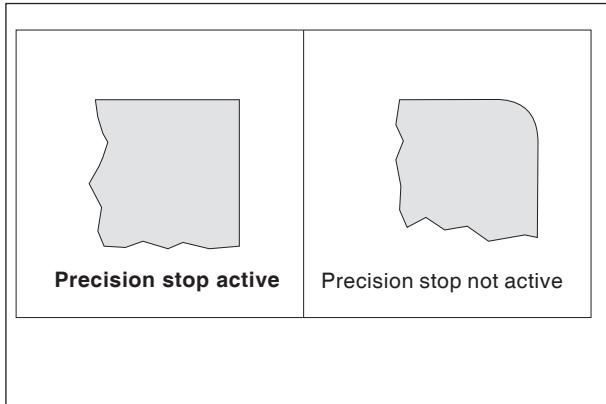
Certain work steps (tool change, measurement position, etc.) are always executed at the same position in the workspace.

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)

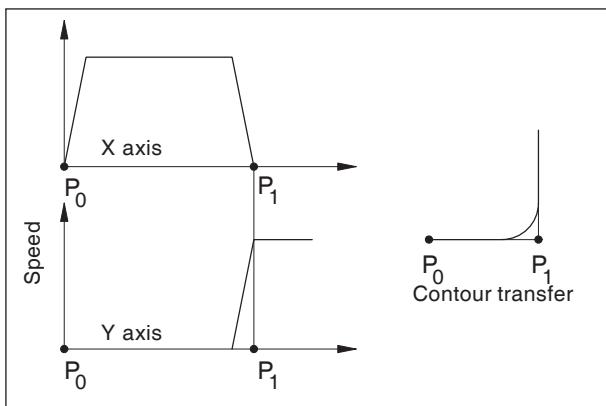
Format

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#26 Y#1 Z-#2

This call will produce

G1 X4 Y5 Z-3

Arguments for macro calls			
Address	Variable number	Address	Variable number
A	#1	Q	#17
B	#2	R	#18
C	#3	S	#19
D	#7	T	#20
E	#8	U	#21
F	#9	V	#22
H	#11	W	#23
I	#4	X	#24
J	#5	Y	#25
K	#6	Z	#26
M	#13		

G66 Macro call (modal)

Format

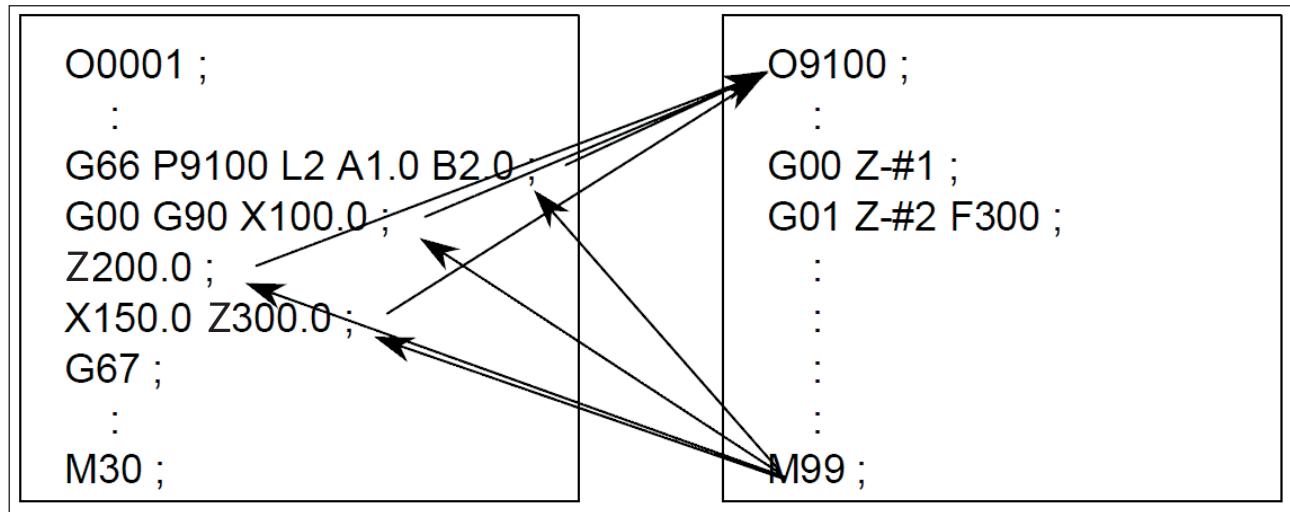
N.. G66 P.. L.. Argument

..

P Program number

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

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



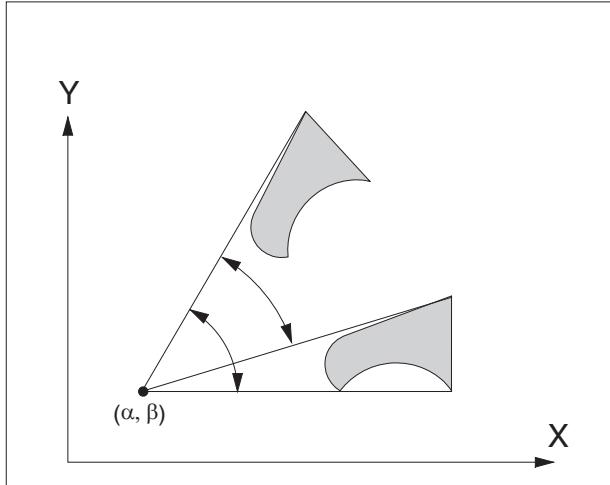
Example of macro call programming

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

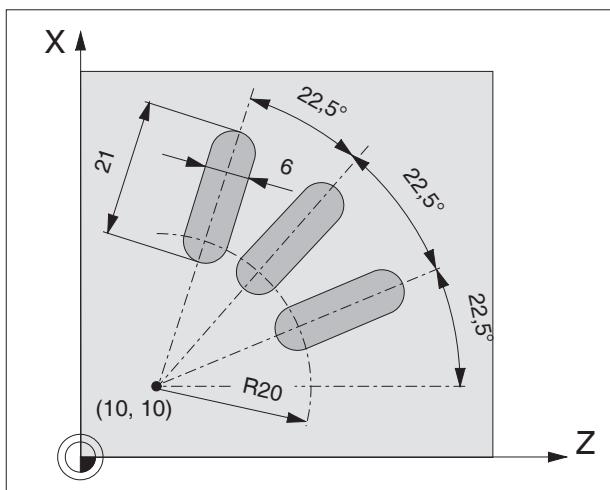
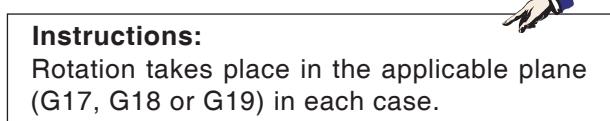
G68 Coordinates system rotation ON

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.



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

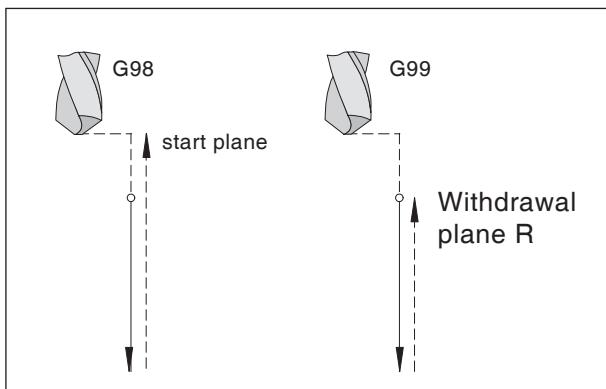
N20 G1 Z-2

N25 X45

N30 G0 Z5

N35 M99

Drilling Cycles G73 - G89



Movements with G98 and G99

Systematic G98/G99

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

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

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

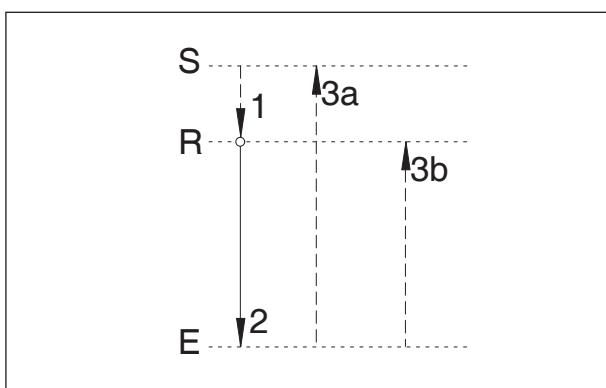
The computation of the R parameter is different with incremental and absolute programming:

Absolute programming (G90):

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

Incremental programming (G91):

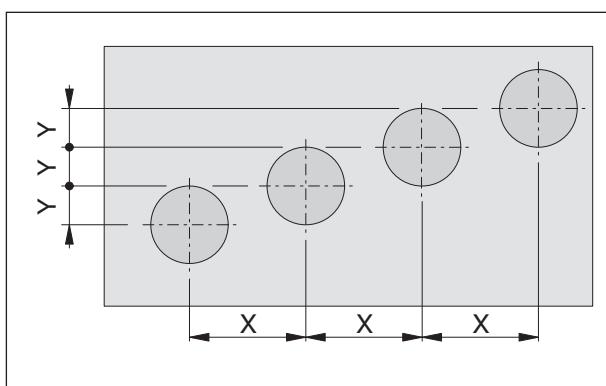
R defines the height of the withdrawal plane related to the last Z position (start position of the drilling cycle). With a negative value for R the withdrawal plane will be below the start position, with a positive value the withdrawal plane will be over the start position.



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.



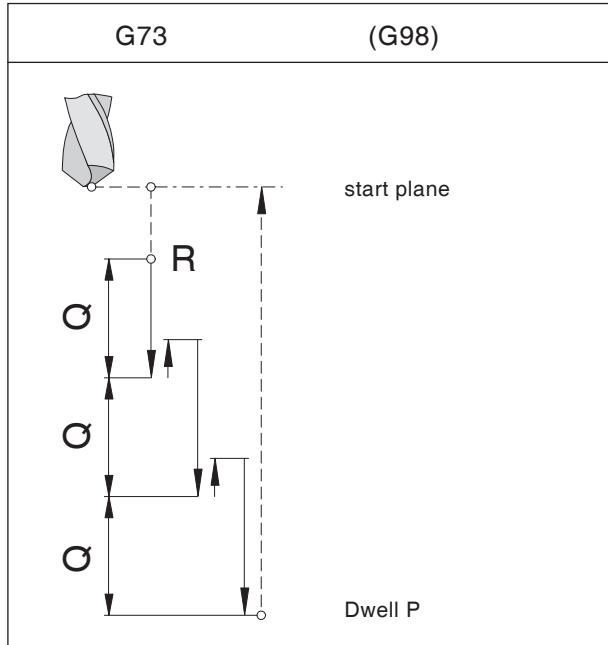
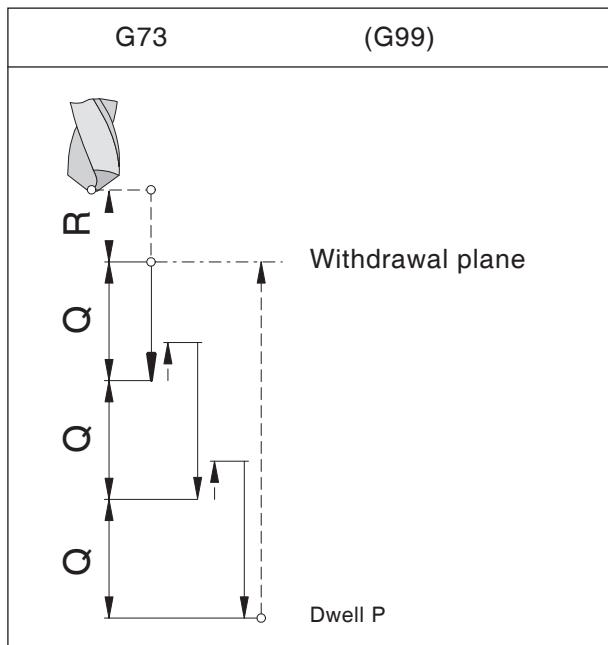
Cycle repetition for a row of holes

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

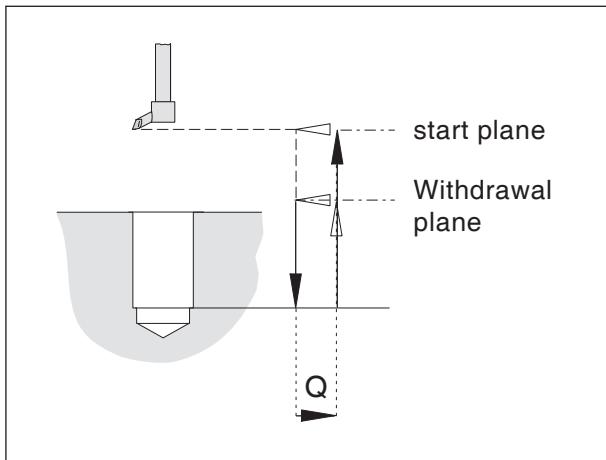
G74 Left Tapping Cycle

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.

G76 Fine Drilling Cycle



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

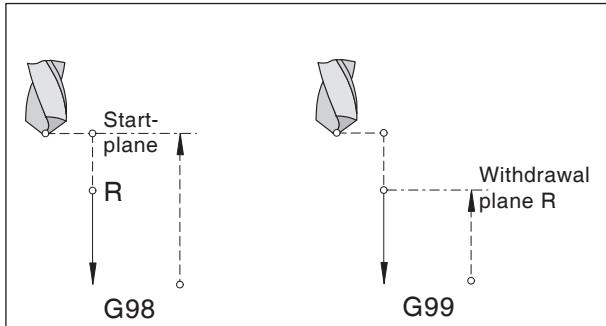
K Number of repetitions

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)

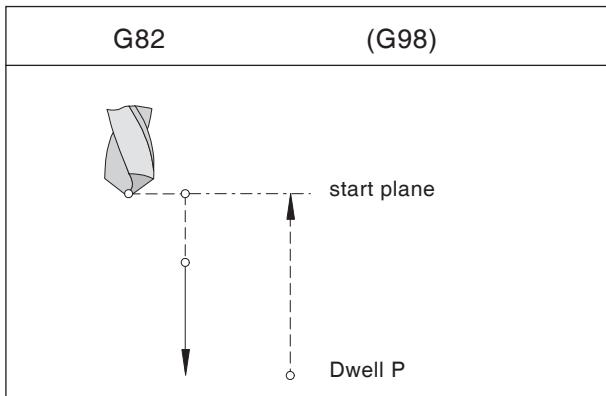
X, Y Hole position

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

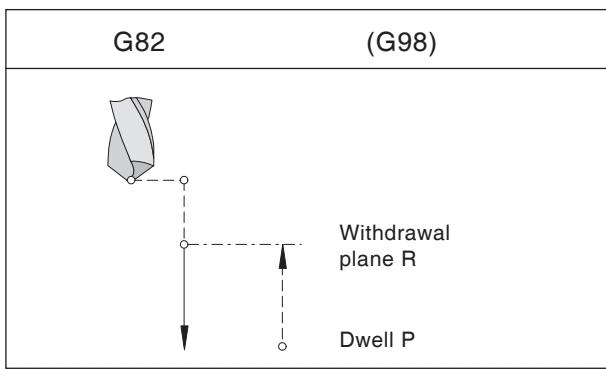
Z Absolute (incremental) drilling depth

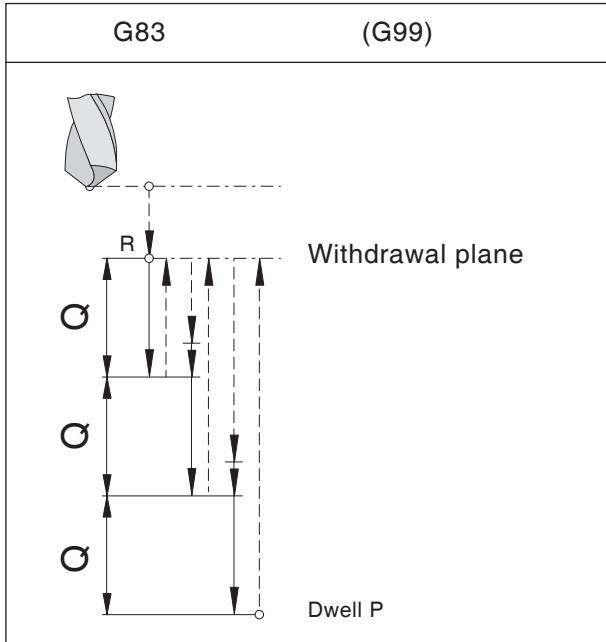
R [mm]..... Absolute (with G91 incremental) value of the withdrawal plane

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

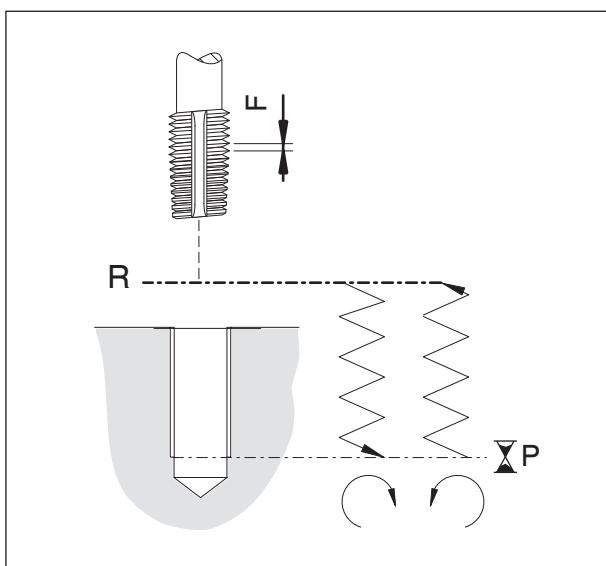
P [msec]..... Dwell at the hole bottom

P1000 = 1 sec

F Feed rate

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

K Number of repetitions



Tapping cycle (with G99)

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

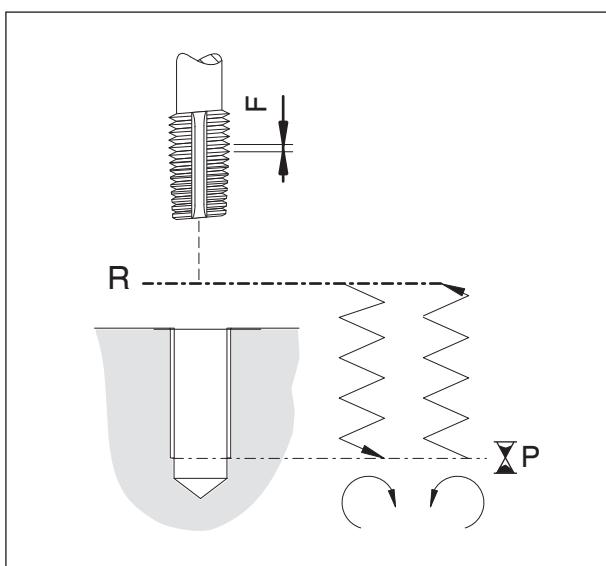
R [mm]..... Absolute (with G91 incremental) value of the withdrawal plane

F Feed rate with G94
Thread pitch with G95

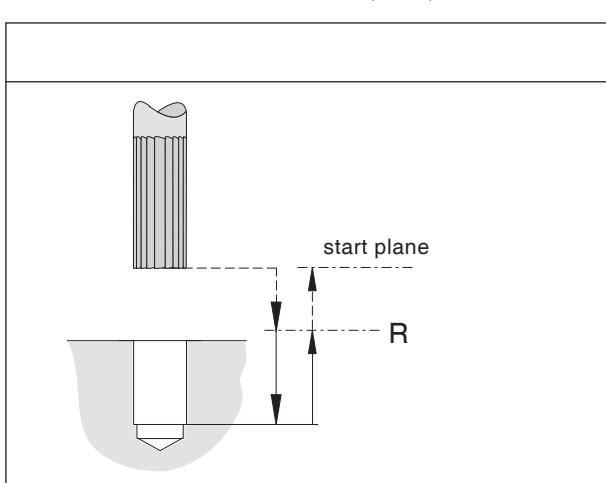
P [msec]..... Dwell at thread ground

P 1000 = 1 sec

K Number of repetitions



Tapping cycle (with G99)



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...

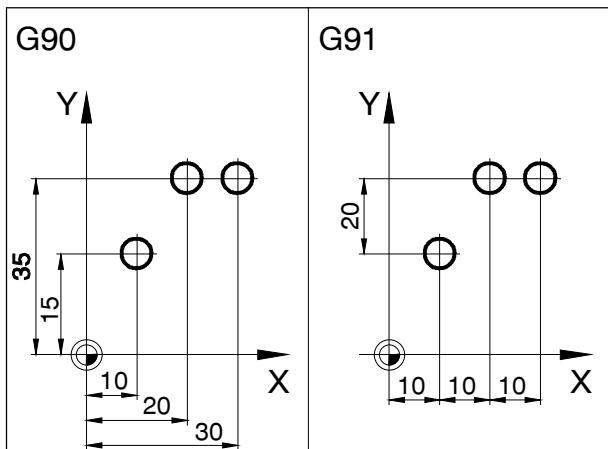
The tool traverses down to end depth with feed speed and retracts to the withdrawal plane with feed. Retraction to withdrawal plane with rapid feed depending on G98.

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

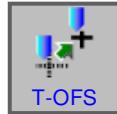
G95 means all values defined under F (feed) as mm/rev.

Format

N.. G95 F...

F: Tool management

Tool settings

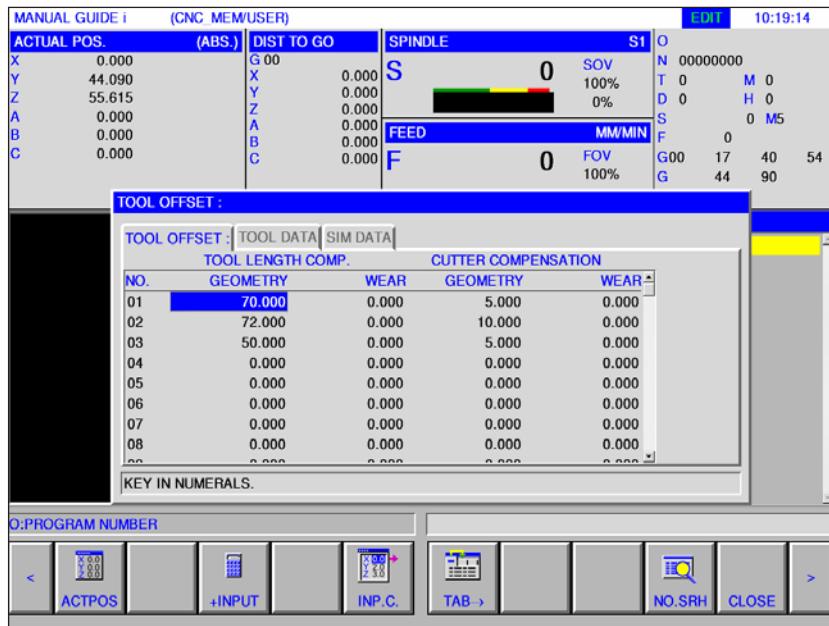


- Press softkey.

- "T-OFS" displays the tool offset data window.

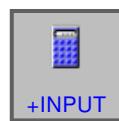
3 register cards are available for selection:

- 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
Tool length offset	
Geometry	Entry of the measured geometry data (with default setting device)
Wear	Tool wear entry
Tool radius compensation	
Geometry	Entry of the measured geometry data (with default setting device)
SCHN-wear	Tool wear entry

Tool length offset entry



- Press softkey.

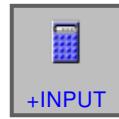
"+INPUT" opens the entry mask.

MANUAL GUIDE i (CNC MEM/USER)		SPINDLE		S1		EDIT	10:20:39
ACTUAL POS.	(ABS.)	DIST TO GO		S	0	SOV	
X	0.000	G 00				100%	
Y	44.090	X 0.000				0%	
Z	55.615	Y 0.000					
A	0.000	Z 0.000					
B	0.000	A 0.000	FEED	MW/MIN			
C	0.000	B 0.000	F	0	FOV		
		C 0.000				100%	
TOOL OFFSET:							
TOOL OFFSET : TOOL DATA SIM DATA							
TOOL LENGTH COMP. CUTTER COMPENSATION							
NO.	GEOMETRY	WEAR	GEOMETRY	WEAR			
01	70.000	0.000	5.000	0.000			
02							
CALCULATE OFFSET(+INPUT)							
03	CURRENT	OFFSET	RESULT				
04	70.000	+	70.000				
05							
06							
07	INPUT OFFSET.						
08	0.000	0.000	0.000	0.000			
09	0.000	0.000	0.000	0.000			
KEY IN NUMERALS.							
O:PROGRAM NUMBER							
<input type="button" value="INPUT"/> <input type="button" value="CANCEL"/>							

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.

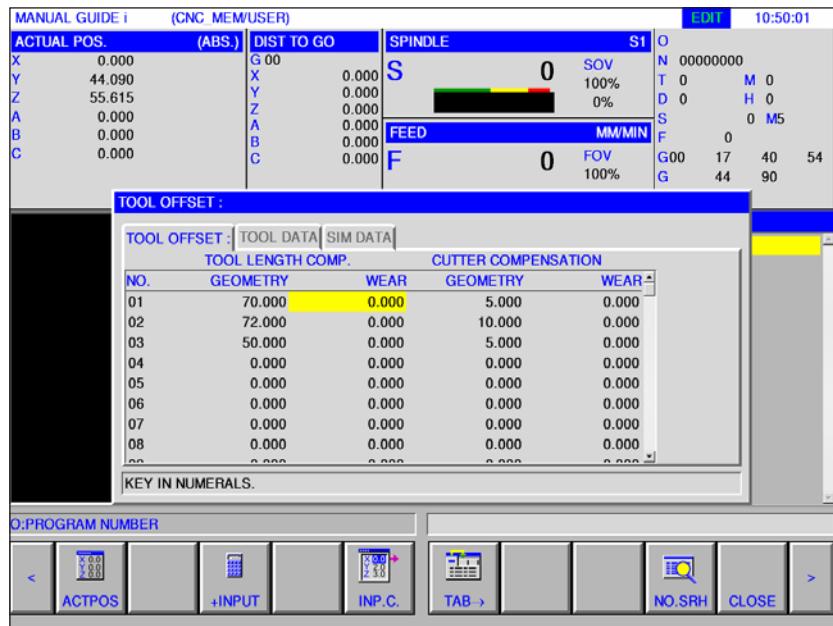
MANUAL GUIDE i (CNC MEM/USER)		SPINDLE		S1		EDIT	10:48:42
ACTUAL POS.	(ABS.)	DIST TO GO		S	0	SOV	
X	0.000	G 00				100%	
Y	44.090	X 0.000				0%	
Z	55.615	Y 0.000					
A	0.000	Z 0.000					
B	0.000	A 0.000	FEED	MW/MIN			
C	0.000	B 0.000	F	0	FOV		
		C 0.000				100%	
TOOL OFFSET:							
TOOL OFFSET : TOOL DATA SIM DATA							
TOOL LENGTH COMP. CUTTER COMPENSATION							
NO.	GEOMETRY	WEAR	GEOMETRY	WEAR			
01	70.000	0.000	5.000	0.000			
02							
CALCULATE OFFSET(+INPUT)							
03	CURRENT	OFFSET	RESULT				
04	5.000	+	5.000				
05							
06							
07	INPUT OFFSET.						
08	0.000	0.000	0.000	0.000			
09	0.000	0.000	0.000	0.000			
KEY IN NUMERALS.							
O:PROGRAM NUMBER							
<input type="button" value="INPUT"/> <input type="button" value="CANCEL"/>							

Current value + workpiece geometry offset compensation is displayed.

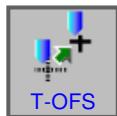
Enter the value calculated previously for the geometry offset.
"INPUT" closes the data entry.

Tool wear offset

Tool wear offset entries should be made in the same way as those for tool length and tool radius offsets.

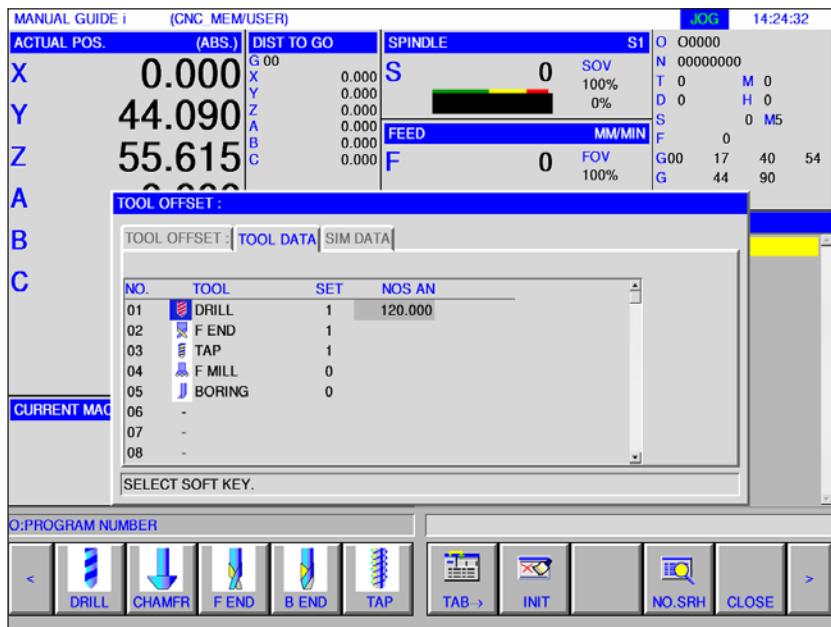


Tool data



- 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:

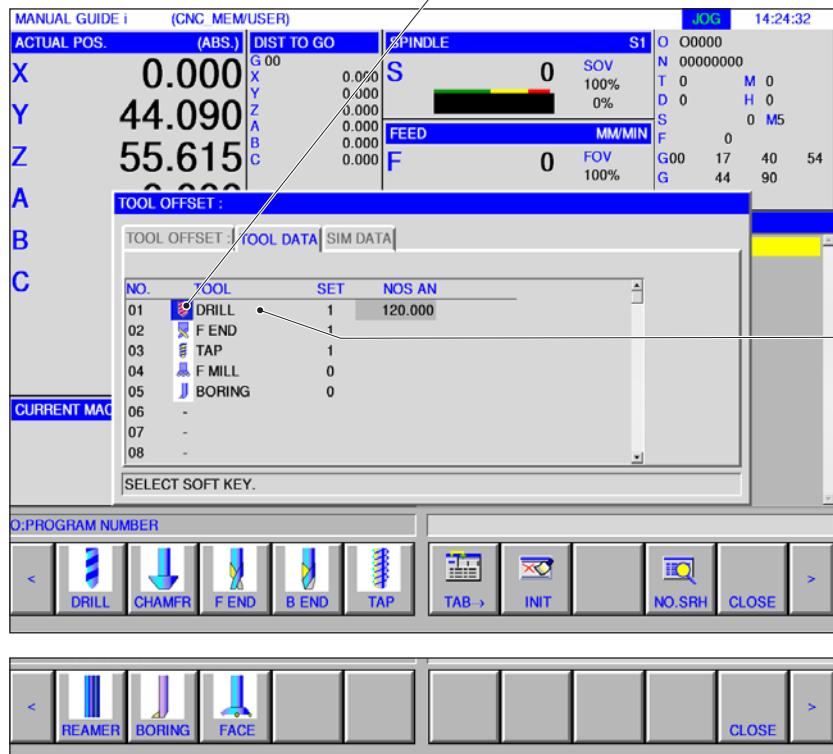
The available parameters can differ, according to the tool selected.



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.

Tool setting number

Instructions:

The available parameters can differ, according to the tool selected.



- 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)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	FEED	EDIT 10:53:03
X 0.000		G 00	S 0	F 0	O N 00000000 T 0 M 0 D 0 H 0 S 0 M5 F 0 G 00 17 40 54 G 44 90
Y 44.090		X 0.000	Y 0.000	Z 0.000	SOV 100% 0% FOV 100%
Z 55.615		Z 0.000			
A 0.000		A 0.000			
B 0.000		B 0.000			
C 0.000		C 0.000			

TOOL OFFSET :

TOOL OFFSET : TOOL DATA SIM DATA		
NO.	TOOL	SET
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)

ACTUAL POS.	(ABS.)	DIST TO GO	SPINDLE	FEED	EDIT 10:53:47
X 0.000		G 00	S 0	F 0	O N 00000000 T 0 M 0 D 0 H 0 S 0 M5 F 0 G 00 17 40 54 G 44 90
Y 44.090		X 0.000	Y 0.000	Z 0.000	SOV 100% 0% FOV 100%
Z 55.615		Z 0.000			
A 0.000		A 0.000			
B 0.000		B 0.000			
C 0.000		C 0.000			

TOOL OFFSET :

TOOL OFFSET : TOOL DATA SIM DATA		
NO.	TOOL	SET
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 >

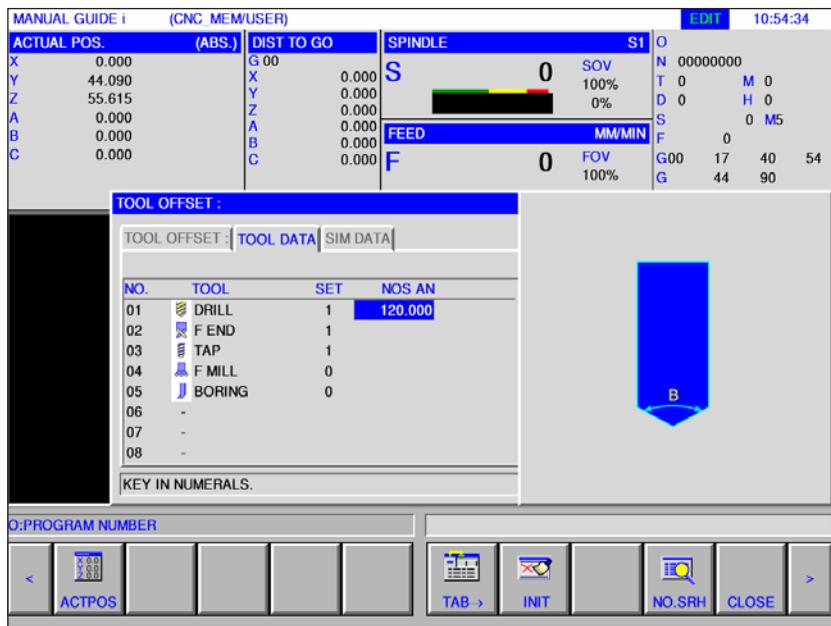
Settings for a milling tool

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.



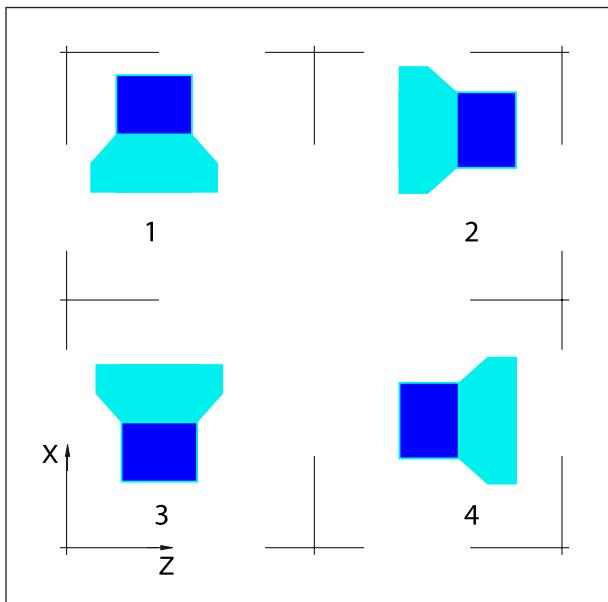
The descriptions and numbers of the elements are dependent on the tool type. Details are in the table below. Tools that do not need tool data settings are not included. No tool data is required to execute milling cycles.

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 ^{*)}	Mill diameter	

^{*)} the cycle can be machined without a setting.



Cutting position of drilling and milling tools

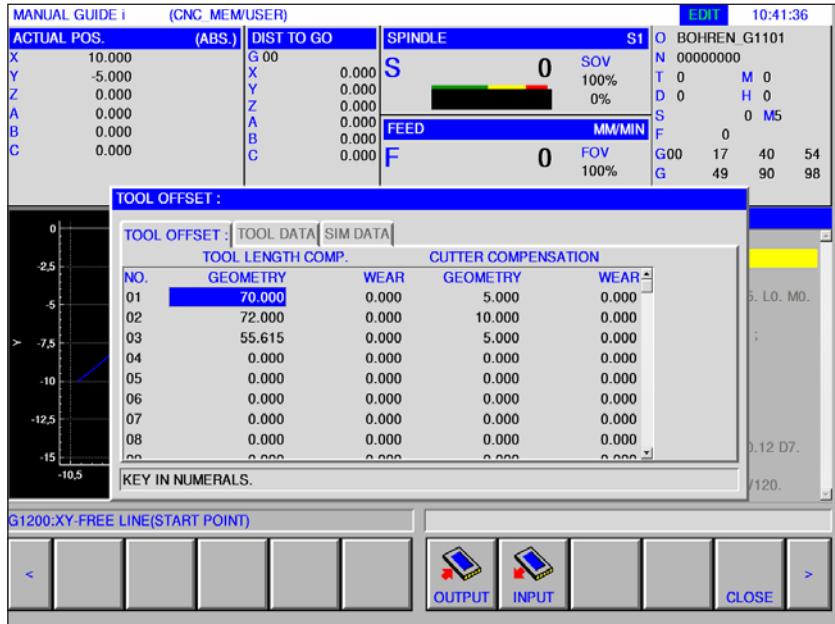
Cutting position of milling and drilling tools

Milling tools

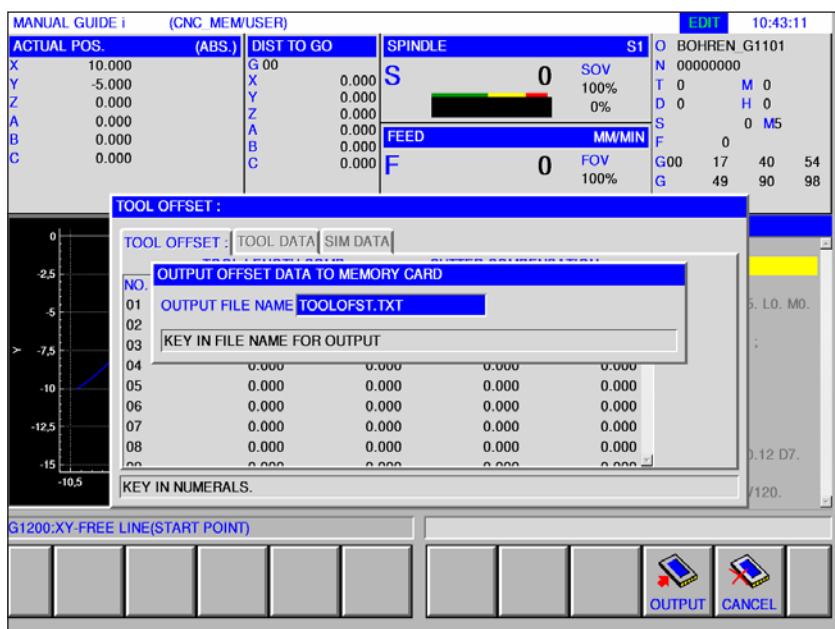
Countersink
Face milling cutter
High speed face milling cutter
Tap
Surface milling cutter

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

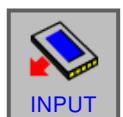


The default save for tool correction data is in the file TOOLOFST.TXT.

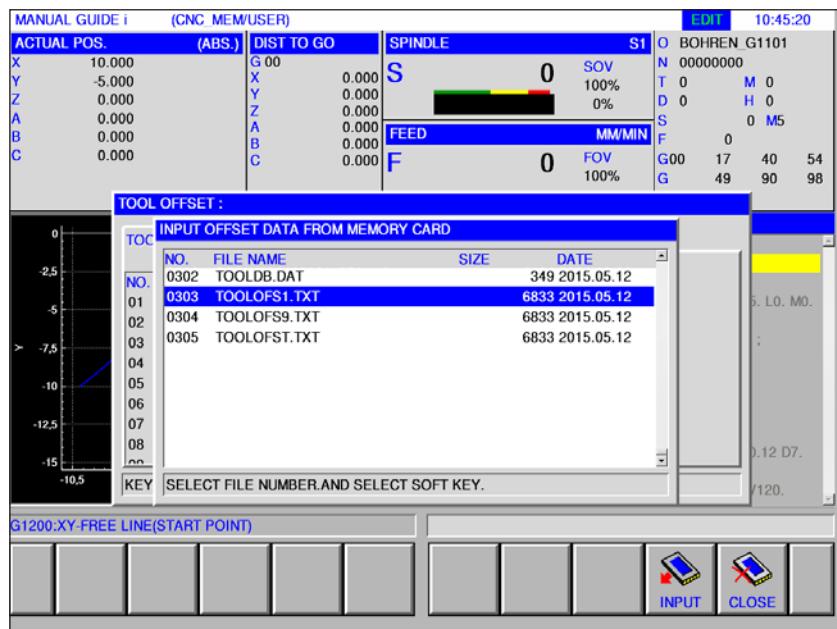
You can change the file name, subject to obeying the following restrictions:

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

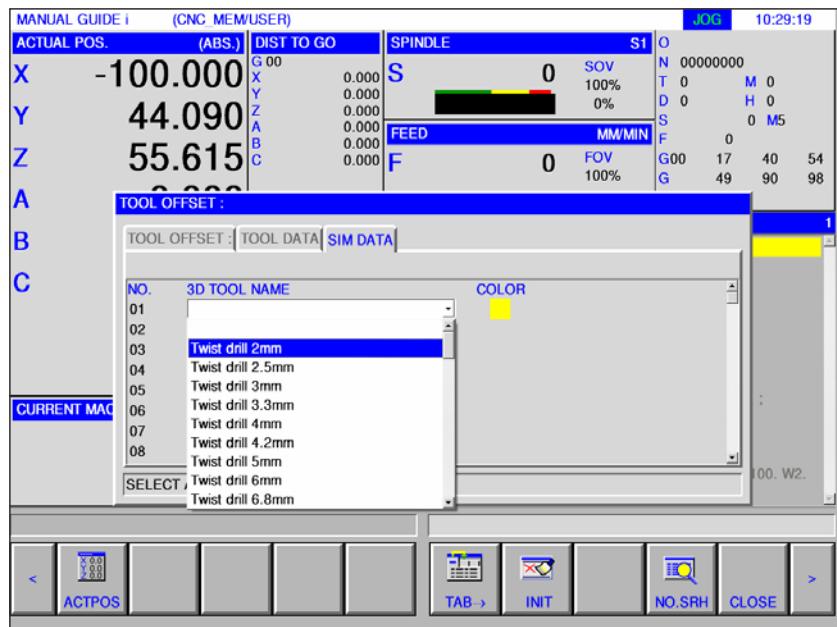


Select the desired file and open with "INPUT".

Simulation data

3D tool

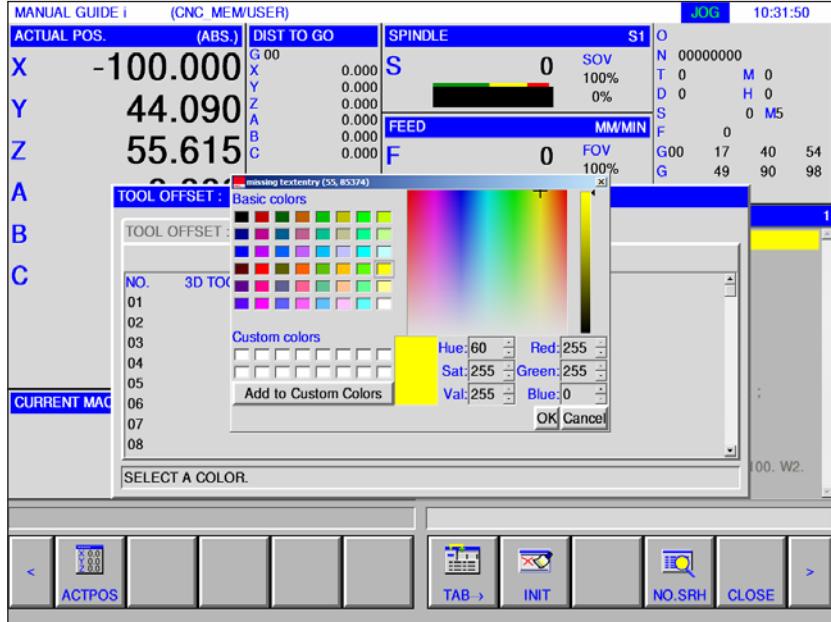
3D tools from the tool manager can be imported into the tool list. An independent color assignment for the individual tools can be made.



- 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.



Instructions:

If no color is selected, it is taken over from the 3D tool manager. Otherwise a selected color has priority.

- 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.

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.

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.

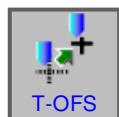
5 Press the "PRESET" softkey and enter the value 0 in the Z axis.



6 Accept changes with the "ALTER" softkey.



7 Tilt the workpiece to be measured.



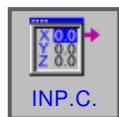
8 Scratch the tool on the workpiece or the meter.

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

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.



Instructions:

During the program sequence the tool data for the tools used may not be changed.

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.



- 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



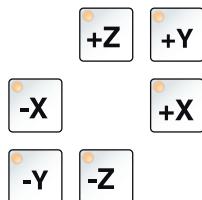
If a program interruption occurs in automatic mode, e.g. after a tool break, then the tool can be moved away from the contour in manual mode. To avoid later collisions the axes must be moved to a safe position.

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 different 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 auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED

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

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 toothings while the machine was switched off.

A running CNC program will be stopped.

Contact EMCO service.

6047: TOOL DISK UNLOCKED

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

A running CNC program will be interrupted.

Contact EMCO service.

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

Turn the drum into locking position manually
Change into MANUAL (JOG) mode.

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

6048: DIVIDING TIME EXCEEDED

Dividing head stuck, 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 stuck mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic door

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 stuck mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.

Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY

Cause: locking switch defective
cabling defective
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic dividing device
lock the dividing device

6074: DIVIDING TIME EXCEEDED

Cause: dividing device stuck mechanically
locking switch defective
cabling defective
fuses defective
insufficient compressed-air supply.

A running program will be aborted.

The auxilliary 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 auxilliary 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 openend/ 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 xxxx.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.

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.

6001: PLC-CYCLE TIME EXCEEDING

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

6011: DRIVE Z-AXIS NOT READY

see 6010.

6012: DRIVE Z-AXIS NOT READY

see 6010.

6002: PLC - NO PROGRAM CHARGED

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

6013: MAIN DRIVE NOT READY

Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.

A running program will be stopped, the auxilliary drives will be switched off.

Check fuses or contact EMCO Service.

6003: PLC - NO DATA UNIT

The auxiliary drives will be switched off.
 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.

6004: PLC - RAM MEMORY FAILURE

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

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.

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.

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 stuck (collision?), fuse defective, hardware defective.

A running CNC program will be stopped.

Check for collisions, check fuses or contact EMCO service.

6042: TOOL TURRET OVERHEAT

Tool turret motor too hot.

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

6043: TOOL CHANGE TIMEOUT

Tool drum stuck (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: Unexpecd 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 work-piece 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 openend/ 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 an 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 CONVEYOR 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.

7099 QUILL 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 "back position".

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.msd 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 License for Easy2control could not be found!

Cause: USB-Dongle or a valid license key 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 manny axis for thread
Remedy: Program max. 2 axes for thread.

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

8198 Internal error (to manny threads)
see 8175.

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

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

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

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

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8212 Rotation axis not allowed
see 8199.

8213 Circle and rotation axis can't be interpolated

8214 Thread and rotation axis can'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 configurated 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 inTransmit.

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: Turnoff 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: Turnoff 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: Turnoff 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: Turnoff 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 EMCConfig.

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 ist 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 shutdownpath 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 ≤ 0 .

Remedy: Program the radius > 0 .

4079 pitch of helix too small

Cause: The radius of the helix is ≤ 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 ≤ 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 co-ordinates.

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".

0011 FEED ZERO

Declaration: 1) The cutting feedrate instructed by an F code has been set to 0.
2) This alarm is also generated if the F code instructed for the S code is set extremely small in a rigid tapping instruction as the tool cannot cut at the programmed lead.
3) During continuous circle motion-based groove cutting, correct Q or F value is not specified or the acceleration clamp value for continuous circle motion in parameter No. 3490 is invalid.

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.

0045 ADDRESS Q NOT FOUND (G73/G83)

Declaration: In a high-speed peck drilling cycle (G73) or peck drilling cycle (G83), the amount of each-time cutting is not specified by address Q, or Q0 is specified. Modify the program.

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.

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.

3510 NO MACHINING CYCLE BLOCK

Declaration: No machining cycle is found. No machining type block is found; only a figure block is specified.

Remedy: Modify the machining program by, for example, adding the necessary machining type block.

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.

3533 WRONG DWELL TIME

Explanation: The dwell time specification is invalid.

Remedy: For example, a negative value may be entered as the dwell time. Modify the machining program to specify an appropriate dwell time.

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 per-

mitted 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.

3559 WRONG ESCAPE AMOUNT

Declaration: The return clearance is invalid.

Remedy: A value not specifiable as turning or other return clearances is entered. Modify the machining program to specify an appropriate value.

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.

3581 WRONG GROOVE WIDTH

Declaration: The groove width specification is invalid.

Remedy: A value not specifiable as the groove width of a figure block is entered. Modify the machining program to specify an appropriate value.

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.

- (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.

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.

3587 WRONG GROOVE DEPTH

Declaration: The groove depth specification is invalid.

Remedy: A value not specifiable as the groove depth of a figure block is entered. Modify the machining program to specify an appropriate value.

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.

3593 FIGURE IS NOT CLOSED

Declaration: All figure elements of an arbitrary figure are specified as parts.

Remedy: All figure elements of an arbitrary figure entered for turning are specified as "parts." Modify the machining program to specify the figure elements corresponding to actual blanks as "blanks."

3594 ALL FIG. ELEMENTS ARE PART

Declaration: The end face specification is invalid.

Remedy: A value not specifiable as the end face of the cycle machining is entered. Modify the machining program to specify an appropriate value.

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).

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.

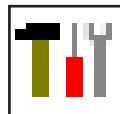
Win3D View

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.

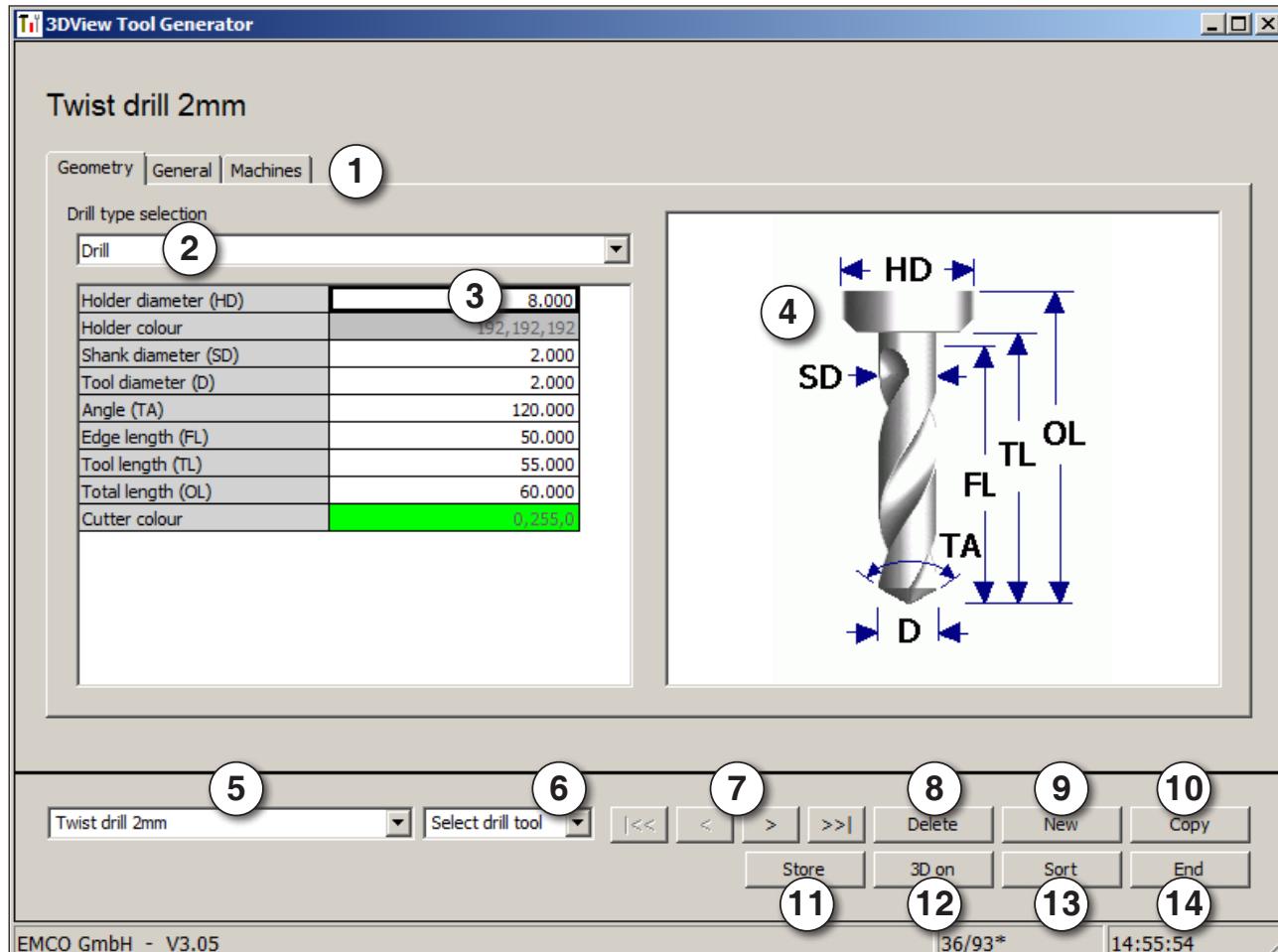
Win3D View offers the following advantages:

- 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



Tool modelling with the 3D-Tool Generator

With the 3D-ToolGenerator you can modify existing tools and create new tools.

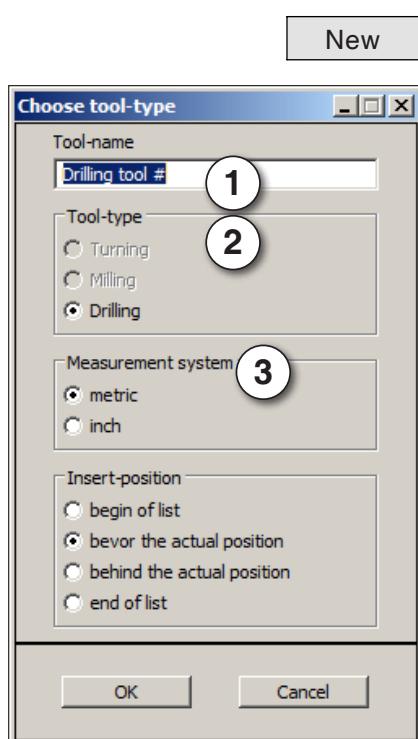


- 1 Register cards for "Geometry", "General" and "Machines" for drilling and milling tools and "Tip", "Holder", "General" and "Machines" for turning tools.
- 2 Selection of tool types
- 3 This window enables the input of tool dimensions.
- 4 Graphical support for the tool dimensioning
- 5 Choice of tools for the selected tool type
- 6 Choice of tool types (here: only drill)
"Turning tool", "Milling tool" and "Drilling tool" reduce the tool choice to the respective type (here: only drilling tools are listed).
"All" does not reduce the tool choice.

- 7 Buttons for quickly browsing through the tools
|<< go to first tool in the group
>>| go to last tool in the group
< go forward in the list by one tool
> go back in the list by one tool
- 8 Button to delete tools
- 9 Button to create new tools
- 10 Button to copy tools
- 11 Button to save changes
- 12 Button for 3D visualization
- 13 Button to sort tools
- 14 Button to terminate the 3DView tool generator

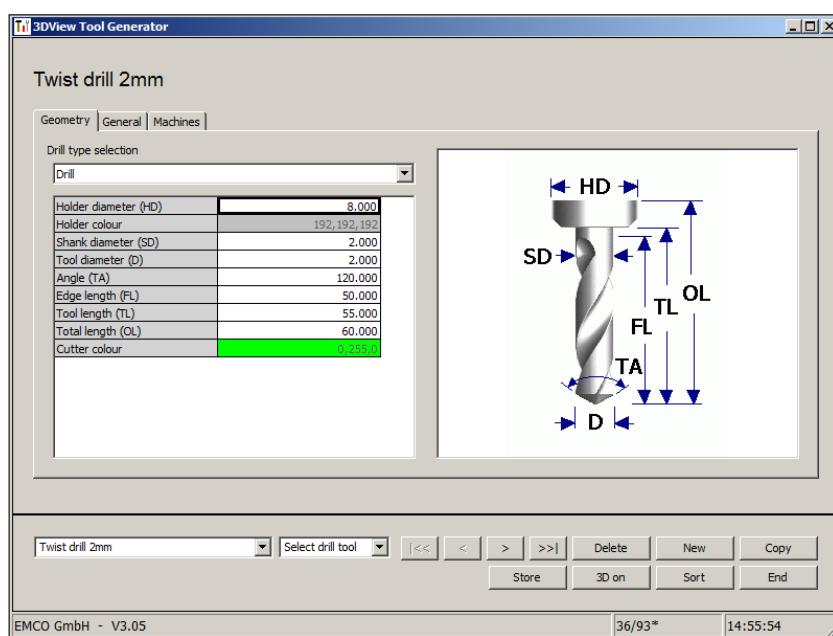
Generating a new tool

- Set the selection for tool types to "Selection all".
- Press the button to generate new tools.
- Select the tool name (1), the tool type (2) and the measurement system (3).



OK

- Confirm the entries with "OK".



Store

- Confirm the entries with "Store".

Copying a tool

Copy

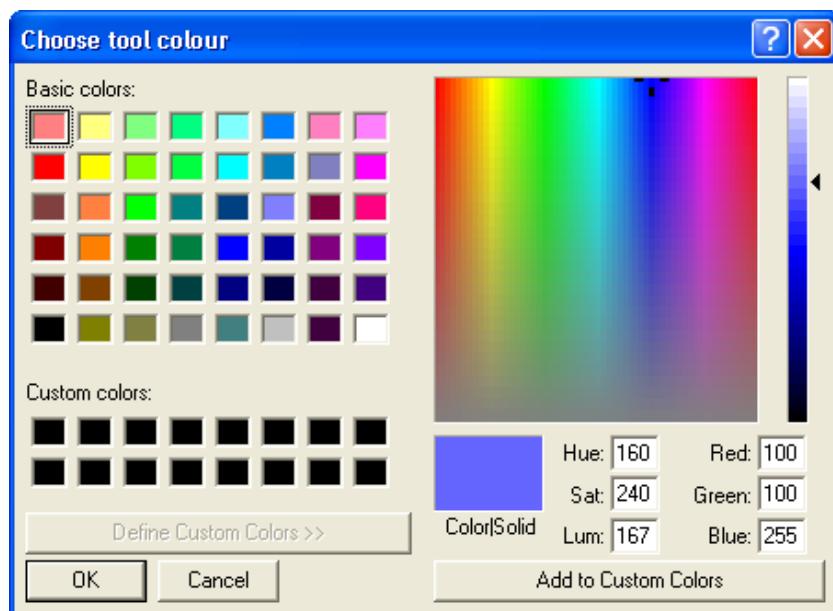
- Call the tool you wish to copy.
- Press the button to copy tools.
- Enter the new tool name.
- Confirm the inputs with "Save".

Store

Changing an existing tool

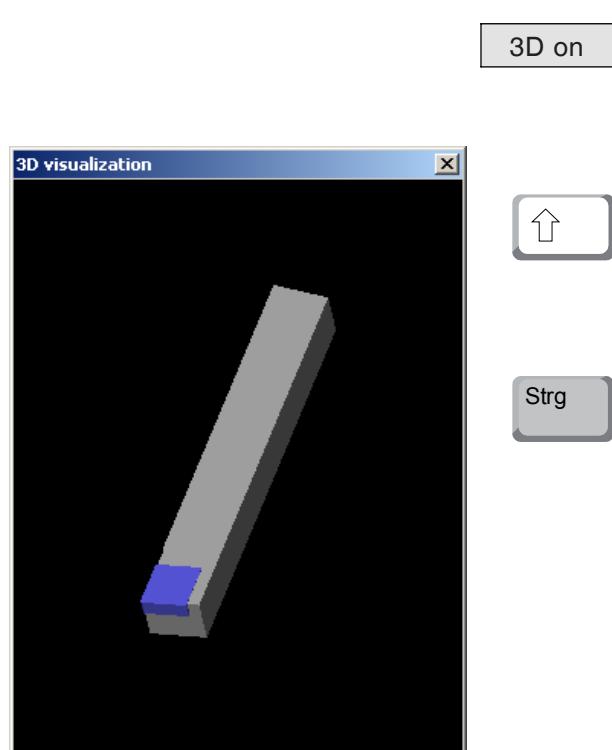
- Call the tool you wish to change.
- Change the values.
- Confirm the entries with "Store".

Selecting a tool colour



OK

- Confirm the entries with "OK".



Visualizing a tool

- Press the button for 3D visualization.

Rotating image

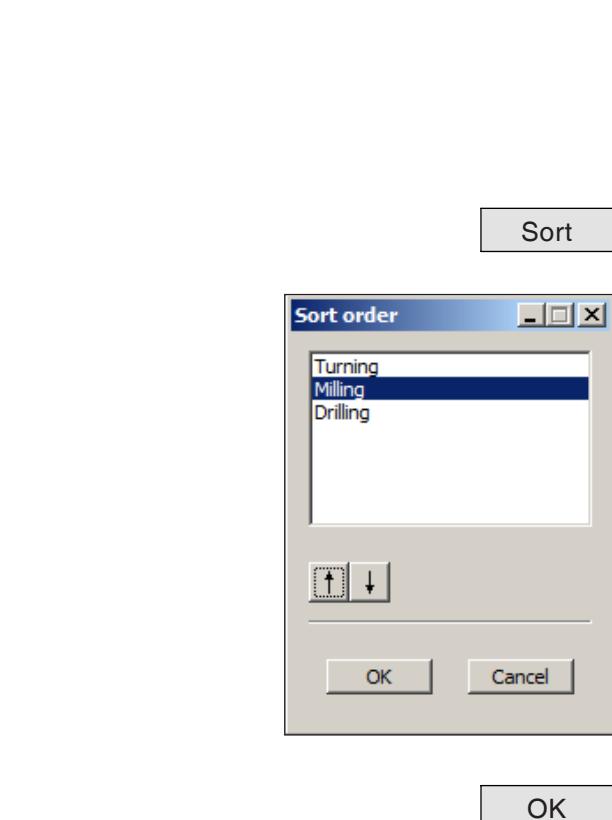
At any time you can rotate the simulation image in one plane as required by pressing and holding the left mouse button. For movements around the Z axis press "Shift" + left mouse button + mouse movement to the right or to the left.

Zooming

You can zoom the tool simulation image in or out by means of „Ctrl“ + left mouse button + mouse movement upwards or downwards

Shifting

Press the right mouse button + mouse movement in the required direction to shift the simulation image.



Sorting function

The sorting sequence makes it possible to display the tools being sorted according to tool types. Every time the sorting sequence is changed, the selection for tools will be updated.

- Press the button for sorting.

- Define new sorting sequence.

- Confirm the entries with "OK".

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

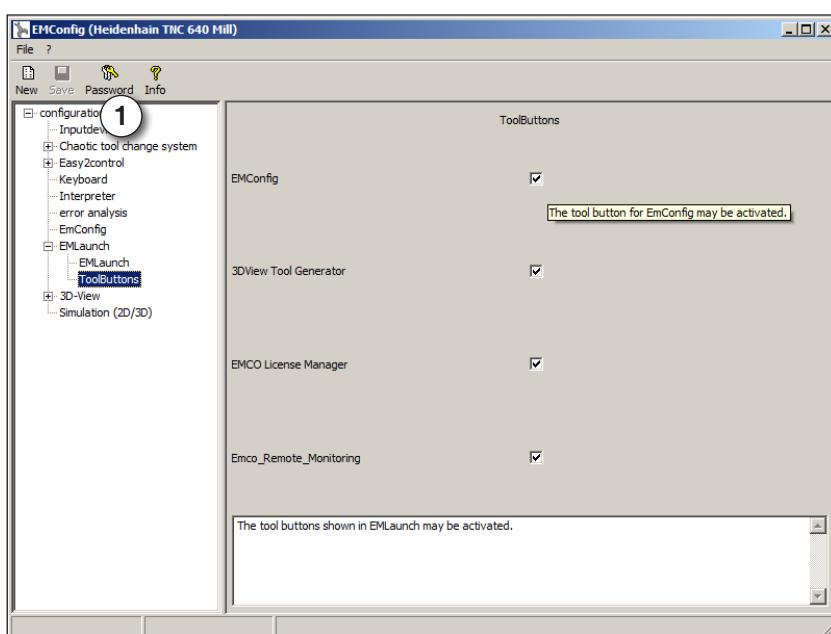
Using EMConfig you can also activate diagnostic functions in case of troubles - that way you get help immediately.



Safety-related parameters are protected by a password. They can only be activated by set-up technicians or by technical support representatives.

Note:

In order to be able to make changes in EMConfig, the password "emco" must be entered (1).



EMLaunch configuration

The following ToolButtons for EMLaunch can be activated or deactivated, e.g.:

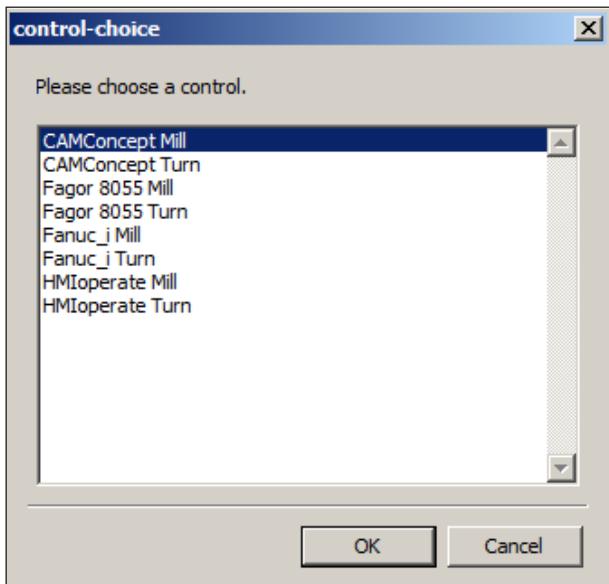
- EMConfig
- 3DView Tool Generator
- EMCO License Manager
- Emco_Remote_Monitoring



Icon for EMConfig

How to start EMConfig

Open EMConfig.



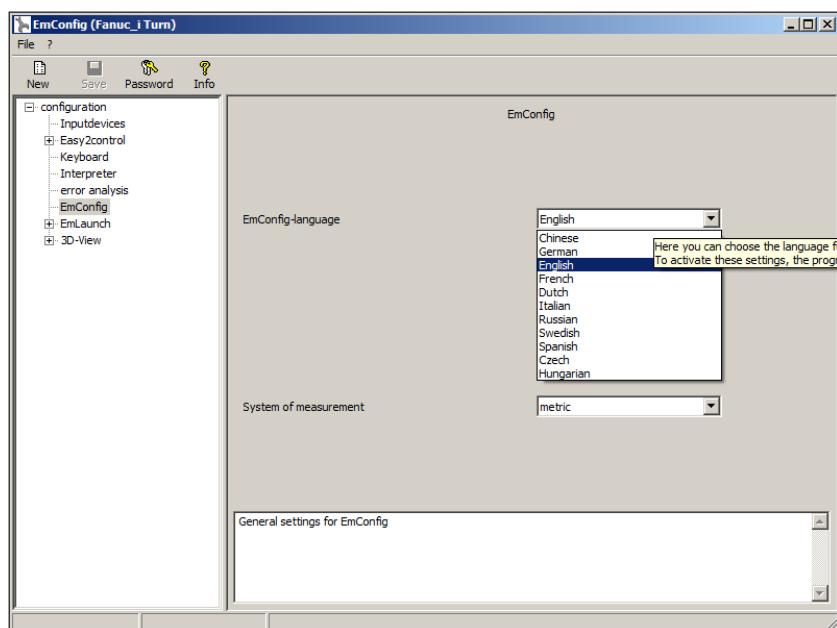
Selection box for control type

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.



Change the language of EMConfig

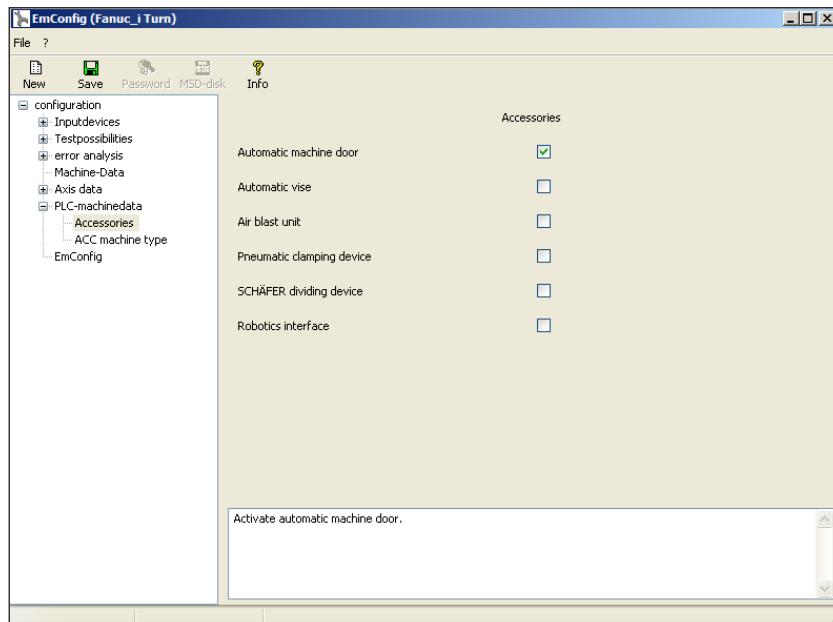
Here you can change the language of EMConfig. In order to activate the settings, restart the program.

Note:

Select the desired menu item. The appropriate function is explained in the text box.

How to activate accessories

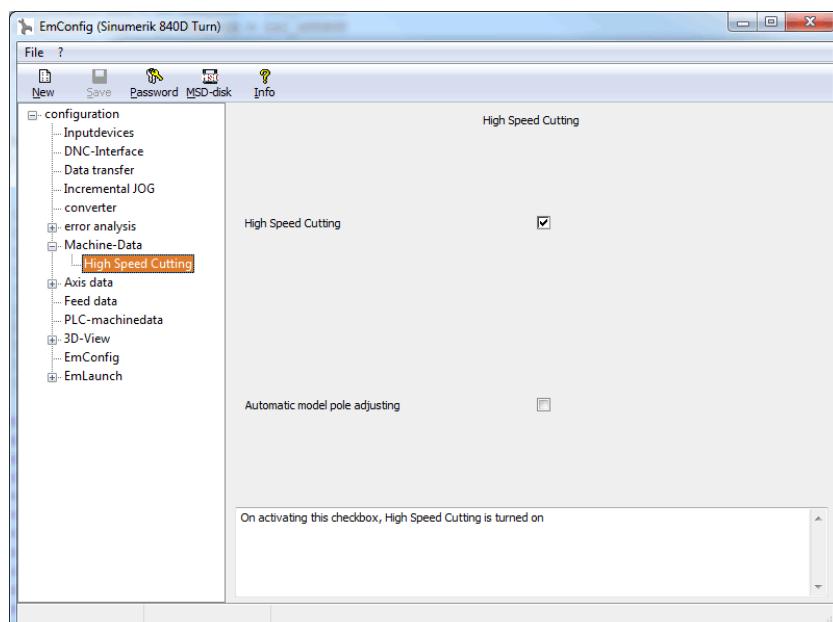
When you install accessories on your machine, you need to activate them here.



Activate accessories

High Speed Cutting

On activating this checkbox, High Speed Cutting is turned on.



Activate High Speed Cutting

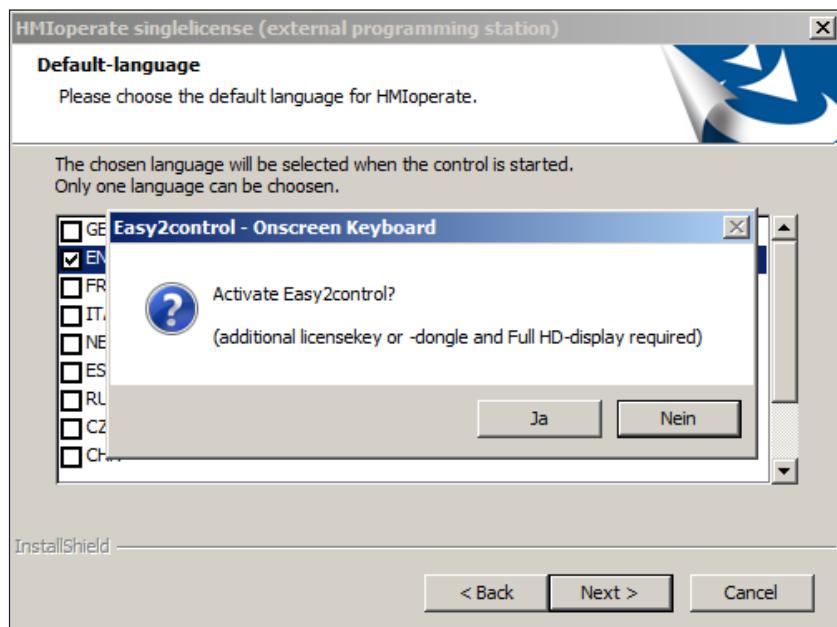
By using high speed cutting, the setting of the axis controller is adjusted. This gain is only effective until the programmed feed rate of 2500 mm/min and allows contour faithful retraction of the tool path and generating of sharp edges.

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.

Easy2control on screen operation

**Note:**

If Easy2control is used after demo version without the hardwaredongle or a valid license key, the controls are deactivated and an appropriate alarm is output by the controller. However, the virtual keyboard is displayed completely.



Activating Easy2control

When installing the software WinNC you will be prompted to activate Easy2control. In order to use the Software for a programming station without restriction, the supplied dongle or a valid license key must be connected to a free USB port.

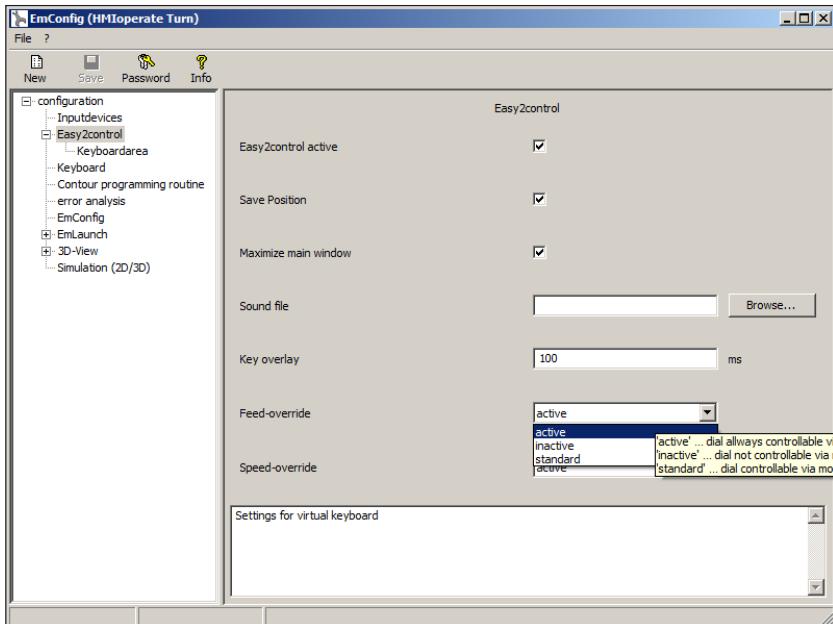
For lathes and milling machines of the Concept series 55/60/105 with connected Easy2operate keyboard no license dongle is necessary.

Using Easy2operate on a programming station requires either

- a license dongle, or
- a license key, or
- a connected Easy2Operate keyboard.

Easy2control settings

This mask allows you to enable or disable Easy2control and make settings.



Easy2control settings

Dial feed-override and dial speed-override:

- Aktive:** dial always controllable via mouse/touchscreen (even with available hardware-version).
- Inaktive:** dial not controllable via mouse/touchscreen.
- Standard:** dial controllable via mouse/touchscreen when no hardware-version is available.

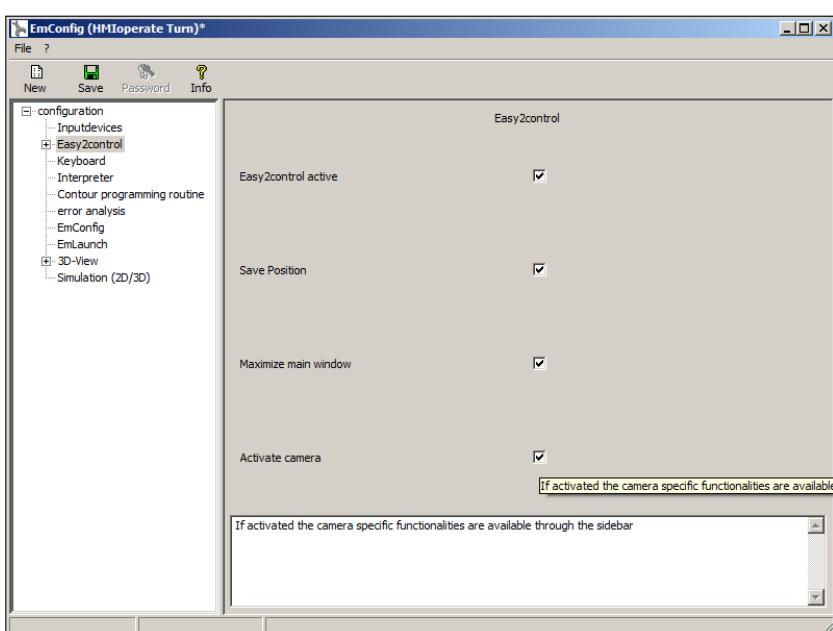


Danger:

Make sure the machine room camera is placed properly in the work area that collisions with the tool turret and the axes are avoided.

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 the camera.

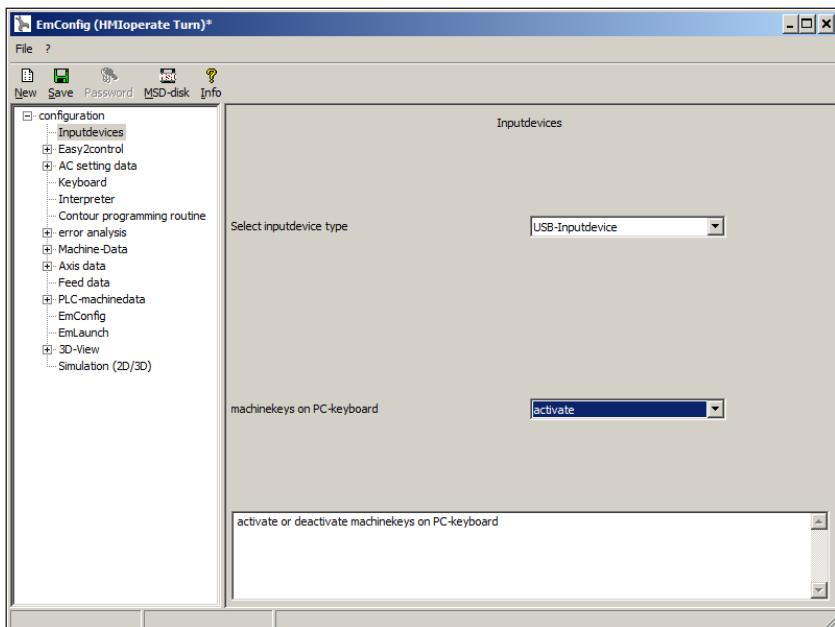


Attention:

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.

Machinekeys on PC keyboard



Setting for the machinekeys

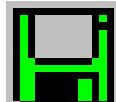
You can activate or deactivate the machinekeys on the PC keyboard.

The setting is available for the following controls:

- Sinumerik Operate
- Fanuc 31i
- Emco WinNC for Heidenhain TNC 640

How to save changes

After the settings, the changes must be saved.



Select "Save" or click on the icon.

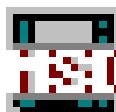
Note:

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

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.

Single license: Best. Nr.: X9C 120
Multiple license Best. Nr.: X9C 130

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

A connected Easy2operate keyboard unlocks Easy2control even without a separate license.

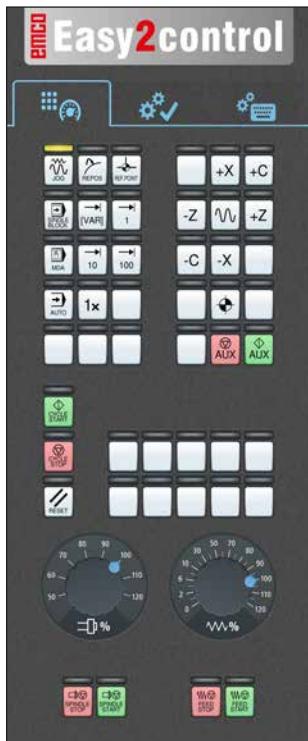


Note:

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

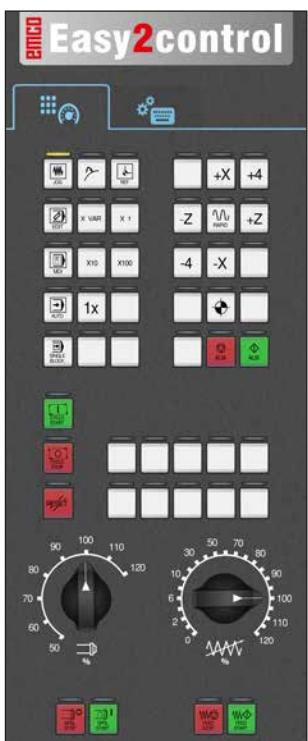


Controller-specific operation



Control operation complete

Fanuc 31i



Machine control panel



Control operation complete

Emco WinNC for Heidenhain TNC 640



Machine control panel

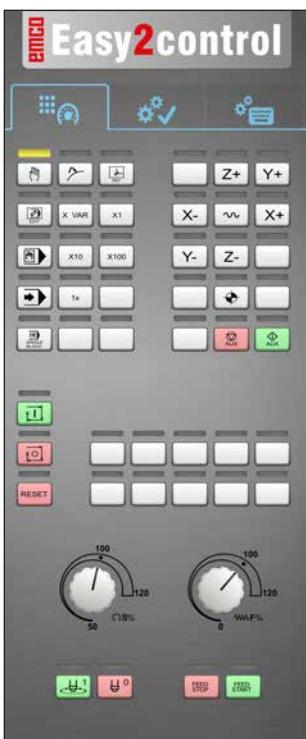


Controller-specific operation



Control operation complete

Heidenhain TNC 426



Machine control panel

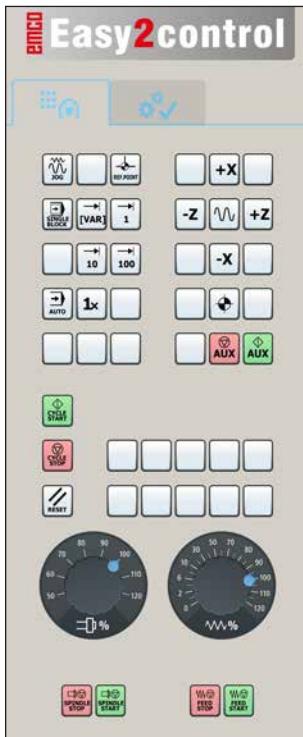


Controller-specific operation

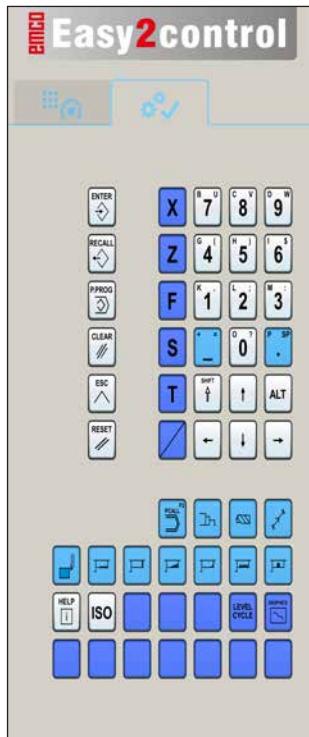


Control operation complete

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:

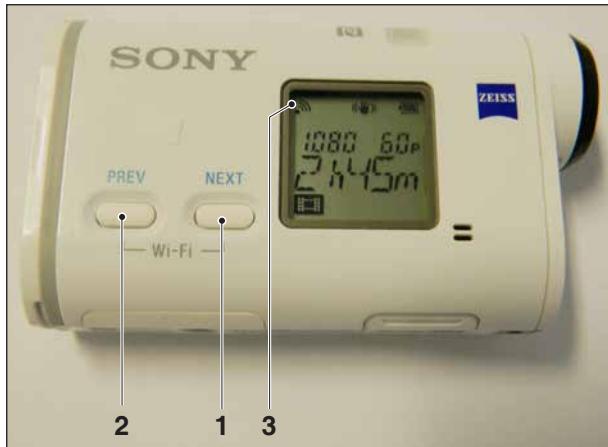
The screen display, based on customer-specific configurations, may look different.



Machine room camera

This accessory can be ordered using the following order number:

Ref. No.: S4Z750



Activating WLAN



Connect WLAN

5 4

Installing the camera

Requirement

USB WLAN adapter for the machine.

Setup WLAN

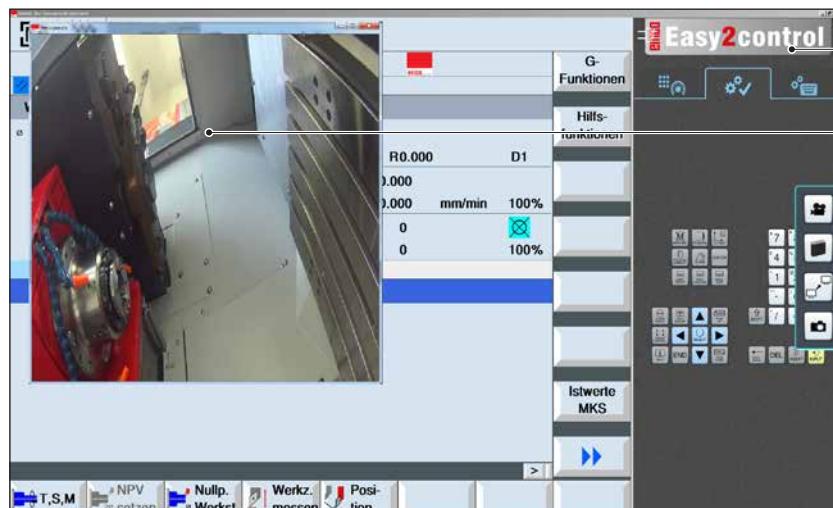
- press NEXT (1) or PREV (2) until an operating mode appears which supports WLAN, e.g. MOVIE. The symbol for WLAN (3) appears in the corner on top left.
- Open EMConfig and activate the camera.
- Connect the WLAN adapter to the USB port to the machine.
- Open windows network and sharing centre (4).
- 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.

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 2 GHz
- Working memory 4 GB RAM
- free hard drive space 2 GB
- Interface:
easy2operate: 1x USB for machine keyboard
Machine connection:
1x LAN (cable connection), only for machine licence
optional: LAN or WLAN for network connection

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".

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-liscence 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.

Note:

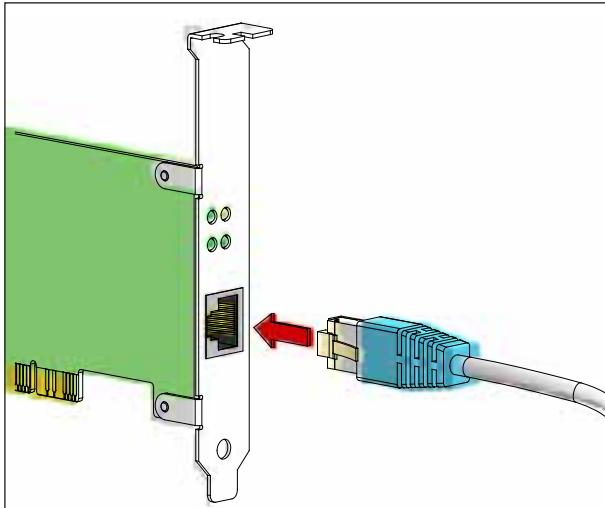
PC TURN and PC MILL have to be equipped with the conversion kit for ACC in order to operate EMCO WinNC.

**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.



Selection menu EMLaunch



Note:
EMLaunch displays all WinNC und CAMConcept controls that are installed in the same directory.



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, the installed folder, and start WinNC Launch.
- 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.
- 11 Tool Buttons (1) can be configured in EMConfig.

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.

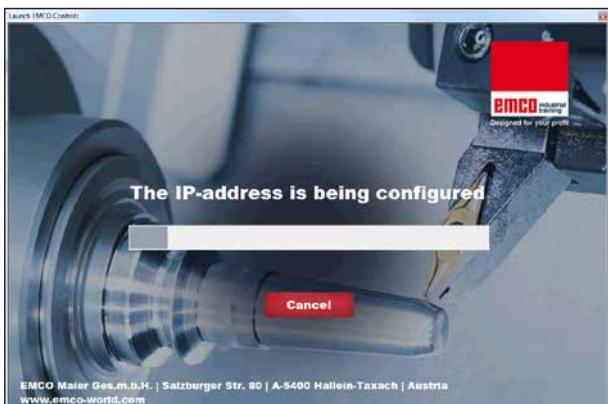
Checks by EmLaunch

In the ACC / ACpn-machine version EmLaunch is checking if a machine is available:

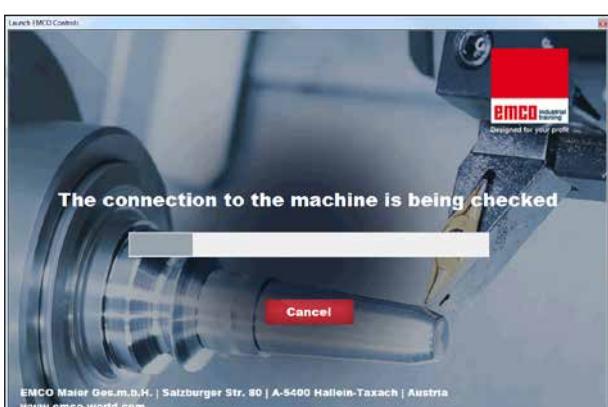
During the network configuration, the IP address is not configured correctly and DHCP for automatic configuration of the IP address is disabled. Connection to the machine is not possible.



DHCP disabled



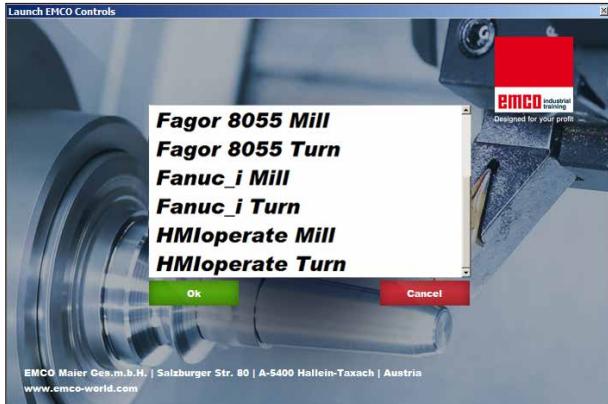
IP-address configuration



Setup the connection to the machine

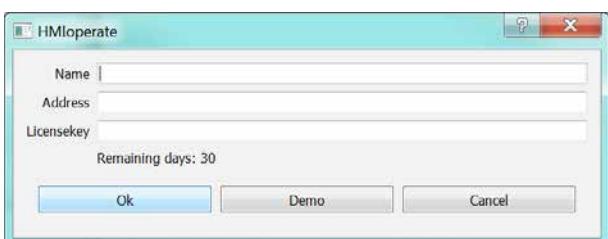
It is attempt to configure the IP address automatically via DHCP.

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.



The connection to the machine is completed and the corresponding control can be started.

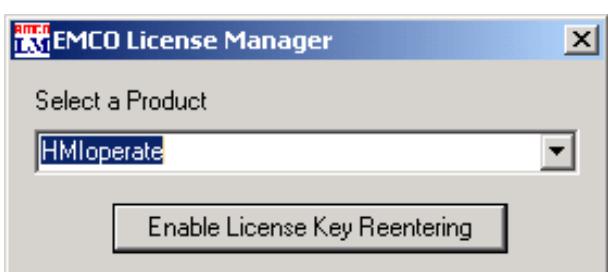
Connection to the machine is OK



Input window licence key enquiry



Run EMCO licence manager upon licence key input



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. If an Emco USB-Stick is connected, this data is taken from the USB-Stick.

UAC dialogue appears upon saving licence input. In order to complete the licence input successfully, the dialogue must be confirmed.

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

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 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.

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 "HMIoperate".