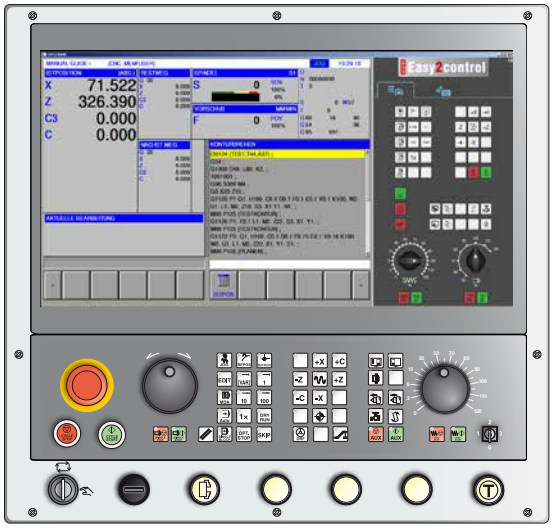
EMCO WinNC for Fanuc 31i Turn
Software description, software release effective from 01.02



Software description
EMCO WinNC for Fanuc 31i Turn

Ref. no. EN 1847
Edition C 2016-02

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

Original operating instructions

EMCO GmbH

P.O. Box 131

A-5400 Hallein-Taxach/Austria

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

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

Internet: www.emco-world.com

E-Mail: service@emco.at

emco group

Designed for your profit



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

Preface

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

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

A cycle is programmed interactively, with graphics support. A large number of processing cycles, which can be combined freely with each other to form a single program, is available to the user.
Individual cycles or the NC programs produced can be simulated graphically on screen.

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

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

EMCO GmbH
Technical Documentation Department
A-5400 HALLEIN, Austria

Designed for your profit

EC compliance

CE 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 lathe reference points A1
M = Machine zero point A1
R = Reference point A1
N = Tool housing reference point A1
W = Workpiece zero point A1
Lathe reference system A2
Absolute and incremental workpiece positions A3
Zero point offset A4

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* *B12*
PC keyboard B13
Overview button assignment control keyboard B14
Overview Button Assignment Machine Operating Elements B15
Machine control panel B17
Key description B17
Skip (block mask) B17
Dry run (test run feed) B17
Individual piece mode B18
Optional stop B18
Edit B18
Handwheel mode (optional) B18
Reset key (Reset) B18
Feed Stop B18
Feed Start B18
Single block B19
Cycle stop B19
Cycle start B19
Arrow keys B19
Rapid traverse B19
Reference point B19
Chip conveyor (Option) B19
Tool turret B20
Tailstock forwards, tailstock backwards B20
Clamping devices B20
Coolant B20
Automatic machine doors B20
Spindle Stop B20
Spindle Start B20
Types of operation B21
Auxiliary OFF B22
Auxiliary ON B22
Override switch (feed rate override) B23
EMERGENCY SHUTDOWN B23
Key Switch Special Operations Mode B23
Multifunction switch for operating modes B24
Key switch B27
Additional left button clamping means B27
USB connection (USB 2.0) B27
Enable button B27

C: Operation

Feed F [mm/min] C1
Speed fundamentals C2
Spindle speed S [rev/min] C3
Types of operation C4
Approach reference point C6
Move slides manually C7
Move slides in step-movement C7
Program management C10
Create program C11
Where programs are saved C11
Programming list C12
Copy program C14
Delete program C14
Enter comment C15
Search program C15
Delete several programs simultaneously C16
Change the sorting sequence C17
Open program C17
Change program name C18
Program properties C18
Program protection C19
Program input and output on a memory card C19
Search for text from the input line in the program C20
Search for and open the program C20
Copy highlighted text to clipboard C21
Move highlighted text to clipboard C21
Insert text C22
Delete highlighted text C22
Insert highlighted text into an input line C23
Revocation and withdrawal C24
Search and replace C24
End program listing C25
Program mode C26
Background Editing C27
Semi-automatic mode C28
Zero point table C30
Setting up workpiece coordinates data C30
Measuring: workpiece zero point offset C31
Compute: workpiece zero point offset C31
Measuring: workpiece coordinates system offset data C32
Calculating: workpiece coordinates system offset data C32
Graphical simulation C33
Screen layout graphical simulation C34
softkey functions C35
3D configuration C39
Shift graphics C40

D: Programming with MANUAL GUIDE *i*

Overview D2
Produce MANUAL GUIDE ***i*** program D2
Programming set-up D3
Raw part definition D6
Cycle overview D7
Working with cycles D10
Data Entry for Machining Cycles D12
Default values for cycle parameters D14
Ignore plausibility check for saving D15
Set M-code system D15

Select dimensions system D16
Turning D17
Outer Rough G1120 ..... D18
Inner Rough G1121 ..... D22
Face Rough G1122 ..... D26
Outer Semi-Finish G1123 ..... D30
Inner Semi-Finish G1124 ..... D32
Face Semi-Finish G1125 ..... D34
Outer Finish G1126 ..... D36
Inner Finish G1127 ..... D38
Face Finish G1128 ..... D40
Outer Rough + Finish G1150 ..... D42
Inner Rough + Finish G1151 ..... D46
Face Rough + Finish G1152 ..... D50
Grooving D55
Turn grooving outside roughing G1130 ..... D56
Grooving Inside Roughing G1131 ..... D58
Grooving Face Roughing G1132 ..... D60
Turn Grooving Outside Coarse + Fine G1133 ..... D62
Turn Grooving Inside Coarse + Fine G1134 ..... D64
Turn Grooving Face Coarse + Fine G1135 ..... D66
Turn Grooving Outside Finishing G1136 ..... D68
Turn Grooving Inside Finishing G1137 ..... D70
Face Finishing G1138 ..... D72
Threading D75
Threading Outer G1140 ..... D76
Threading Inner G1141 ..... D82
Hole machining D85
Centre Drilling G1100 ..... D86
Drilling G1101 ..... D88
Tapping G1102 ..... D90
Reaming G1103 ..... D91
Boring G1104 ..... D92
Figure D93
Plane Turning Figure G1450 ..... D94
Overview of the input elements for free contour programming. D95
Input items of line (ZX plane) G1451 ..... D96
Input items of arc (ZX plane) G1452, 1453 ..... D97
Input items of chamfer (ZX plane) G1454 ..... D98
Input items of corner (ZX plane) G1455 ..... D99
Input items of end of arbitrary figures (ZX plane) G1456 D100
Unmachined part element connection: Closing a contour D101
Symbol representation of the contour elements D103
Groove Contour D105
ZX-Groove Outer Normal G1470 ..... D106
ZX-Trapezoidal Outside Groove G1471 ..... D108
ZX-Standard Inside Groove G1472 ..... D112
ZX-Trapezoidal Inside Groove G1473 ..... D114
ZX-Standard Face Groove G1474 ..... D118
ZX-Trapezoidal Groove Face G1475 ..... D120
ZX-Contour Groove G1456 ..... D124
Thread Contour D125
Thread General G1460 ..... D126
Thread Metric G1461 ..... D128
Thread Unify G1462 ..... D130
Thread Pipe T G1463 ..... D131
Thread Pipe F G1464 ..... D132
Milling D133
Centre Drilling G1110 ..... D134
Drilling G1111 ..... D136
Tapping G1112 ..... D138
Grinding G1113 ..... D139
Reaming G1114 ..... D140
Plane Machining D141
Face Milling Roughing G1020 ..... D142
Face Milling Finishing G1021 ..... D144
Contour Machining D147
Outer Wall (Roughing) G1060 ..... D148
Outer Wall (Z-Finishing) G1061 ..... D154
Outer Wall (Side Finishing) G1062 ..... D158
Outer Wall (Chamfer) G1063 ..... D162
Inner Wall (Roughing) G1064 ..... D164
Inner Wall (Finishing) G1065 ..... D166
Inner Wall (Side Finishing) G1066 ..... D168
Inner Wall (Chamfer) G1067 ..... D170
Partial Machining (Roughing) G1068 ..... D172
Partial Machining (Finishing) G1069 ..... D174
Partial Machining (Side Finishing) G1070 ..... D176
Partial Machining (Chamfer) G1071 ..... D178
Pocket Milling D181
Pocket Milling (Roughing) G1040 ..... D182
Pocket Milling (Finishing) G1041 ..... D186
Pocket Milling (Side Finishing) G1042 ..... D188
Pocket Milling (Chamfer) G1043 ..... D190
Special D193
Engraving G1025 ..... D194
Figure D197
C-axis Hole on Face (Arc Points) G1572 ..... D198
C-axis Hole on Face (Circle), Polar Coordinates G1573 . D199
C-axis Hole on Face (Cart. Coordinates) G1574 ..... D200
C-axis Hole on Cylindrical Surface (Arc Points) G1672 ... D201
C-axis Hole on Cylindrical Surface (Free Choice) G1673 D202
Front Surface Contour D203
XC Square Face Contour G1520 ..... D204
Contouring Figure D205
XC Square Convex G1520 ..... D206
XC Circle Convex G1521 ..... D207
XC Oval Convex G1522 ..... D208
XC Polygon Convex G1525 ..... D210
XC Free Contour Convex G1500 ..... D212
Overview of the input elements for free contour programming. D213
ZC Free Contour Convex on Cylindrical Surface G1600 . D214
XC Square Concave G1520 ..... D215
XC Circle Concave G1521 ..... D216
XC Oval Concave G1522 ..... D217
XC Polygon Concave G1525 ..... D218
XC Free Contour Concave G1500 ..... D220
ZC Free Contour Concave on Cylindrical Surface G1600 D222
XC Free Contour Open G1500 ..... D223
ZC Free Contour Open on Cylindrical Surface G1600 .... D224
Pocket Contour D225
XC Square Concave G1520 ..... D226
XC Circle Concave G1521 ..... D227
XC Oval Concave G1522 ..... D228
XC Polygon Concave G1525 ..... D230
XC Free Contour Concave G1500 ..... D232
ZC Free Contour Concave on Cylindrical Surface G1600 D233
Sub-routines D236
Insert fixed forms D237
Creating fixed forms D238
M-code menu D240

E: G code programming

Command abbreviations overview E2
Calculation operators for NC program E3
Overview of machine G-commands E4
Brief description of G-commands E6
G00 Rapid traverse E6
G01 Straight interpolation E7
Entering chamfers and radii E7
Direct drawing dimension input E8

G02 Circular interpolation, clockwise ... E9
G03 Circular interpolation, counterclockwise ... E9
G04 Dwell time ... E9
G7.1 Cylindrical interpolation ... E10
G10 Data setting ... E12
G12.1/G13.1
Polar coordinates interpolation ... E14
G17-G19 Plane selection ... E17
G28 Return to reference point ... E18
G33 Thread cutting ... E19
Cutting radius offset ... E20
G40 Deselection mill radius offset ... E22
G41 Mill radius offset left ... E22
G42 Mill radius offset right ... E22
G52 Local coordinates system ... E23
G53 Selecting the machine coordinates system ... E24
G54 to G59 Selecting a workpiece coordinates system ... E24
G61 Precision stop (modally effective) ... E27
G64 Path control mode ... E27
G65 Macro call ... E28
G66 Macro call (modal) ... E29
G67 Macro call (modal) End ... E29
G76 Deep Hole Drilling / Face Grooving Cycle ... E30
G77 Grooving cycle (X-axis) ... E31
G78 Multiple thread cutting cycle
(ID or AD cycle) ... E32
Boring cycles with power tools ... E33
G80 Delete boring cycle (G83 to G89) ... E34
G83 Plane boring cycle ... E34
G87 Side boring cycle ... E34
G84 Plane thread tapping cycle with/without compensating chuck ... E36
G88 Side thread tapping cycle with/without compensating chuck E36
G90 Absolute programming ... E39
G91 Incremental programming ... E39
G92 Setting maximum spindle speed ... E39
G94 Feed per minute ... E40
G95 Feed per revolution ... E40
G96 Constant cutting speed ... E40
G97 Constant spindle speed deselected ... E40
G98/G99 Return plane ... E40
Bar feeder - / bar loader interface (option) ... E41
Bar feeder general ... E41
M functions ... E41
Control signals (MACRO Variable) ... E41
Programming control signals ... E41
Used variables ... E42

F: Tool management

Tool settings ... F1
Measuring ... F2
Calculating ... F2
Radius offset ... F3
Cutting position ... F3
Tool wear offset ... F4
Tool data ... F5
Select tool ... F6
Tool setting number ... F7
Setting angle, corner angle, diameter, width and length ... F8
Geometry correction, wear correction and tool data input and output ... F11
Simulation data ... F13
3D tool ... F13
Color select ... F14
Measuring tool manually ... F15
Measure the tool with the optical default setting device at the machine ... F18

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
Set flow ... 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
Settings ... X4
Machine room camera ... X5
Machinekeys on PC keyboard ... X5
How to save changes ... X6
How to create machine data floppy disk or machine data USB flash drive ... X6

Y: External Input Devices

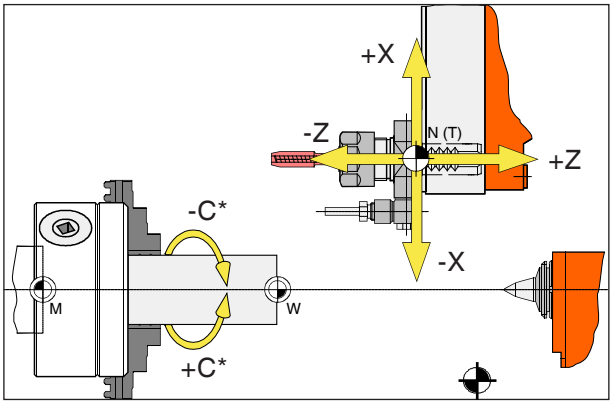
EMCO Control Keyboard USB Y1
Scope of supply Y1
Assembling Y2
Connection to the PC Y3
Settings at the PC software Y3
Easy2control On Screen operation Y4
Scope of supply Y4
Operating areas Y5
Machine room camera Y8
Installing the camera Y8
Operating the camera Y9

Z: Software Installation Windows

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

Blank page

A: Principles



Points on the machine

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

EMCO lathe reference points

M = Machine zero point

An unchangeable reference point defined by the manufacturer of the machine.
The entire machine is measured from this point.
At the same time, "M" is the origin of the coordinate system.

R = Reference point

A position in the machine's work area defined exactly by the limit switch.
Moving the slots to "R" notifies the slot position to the control system. Required after any power cut.

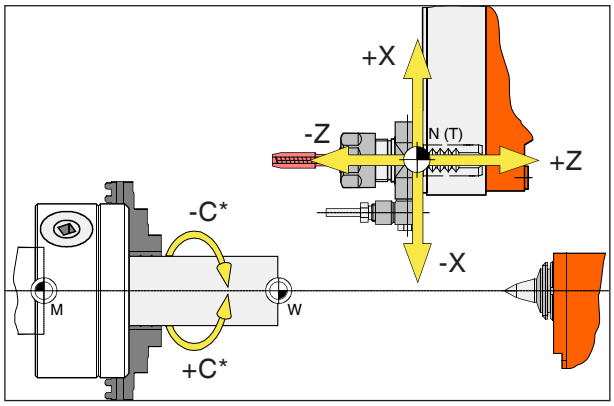
N = Tool housing reference point

Start point for tool measurement.
"N" is at an appropriate point of the tool holder system and is defined by the manufacturer of the machine.

W = Workpiece zero point

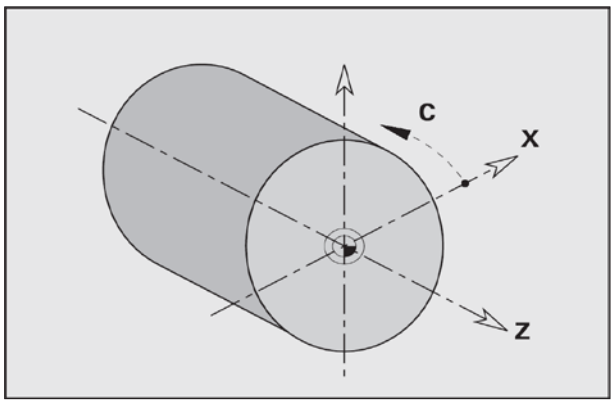
Start point for the dimensional details in the parts program.
Freely definable by the programmer and often movable at random within a parts program.

Lathe reference system



Absolute coordinates are related to a fixed position, incremental coordinates to the tool position.
** only for power 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!



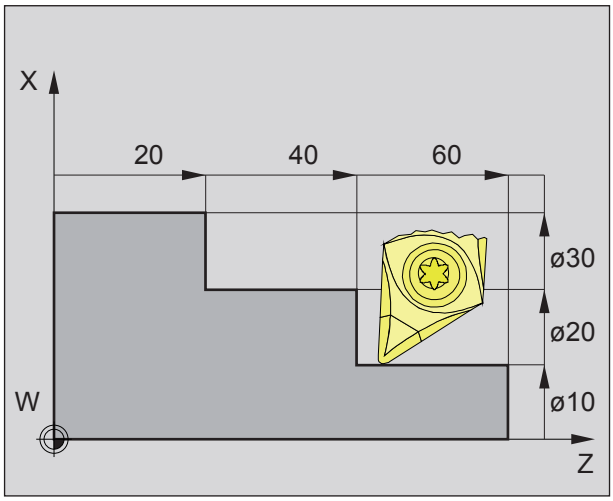
With a reference system, you define positions in a plane or in space unambiguously. Positional information always relates to a given point and is described by means of coordinates.
In the right-angled system (Cartesian system), three directions are defined as X, Y and Z axes. The axes are perpendicular 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.
The X-coordinate is in the direction of the transverse slot and the Z-coordinate in the direction of the longitudinal slot.
Coordinate details in the minus direction describe movements of the tool system towards the workpiece and details in the plus direction away from the workpiece.

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.

WinNC recognizes movements in straight lines or in circles between programmed points. You can program a workpiece process by entering coordinates and linear or circular movements sequentially.

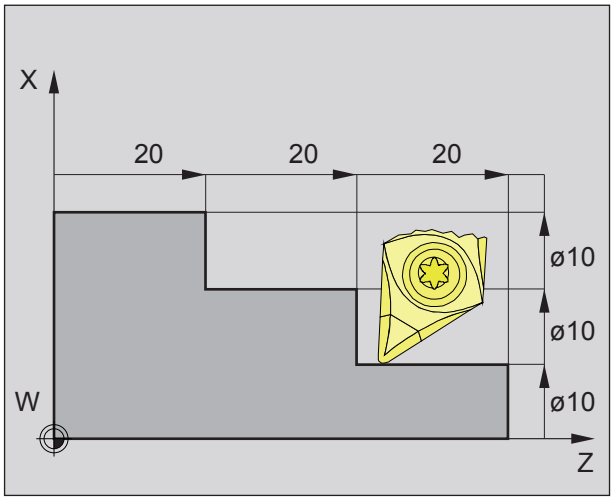
Angular information for the C-axis is related to the "C-axis zero point".

Absolute and incremental workpiece positions



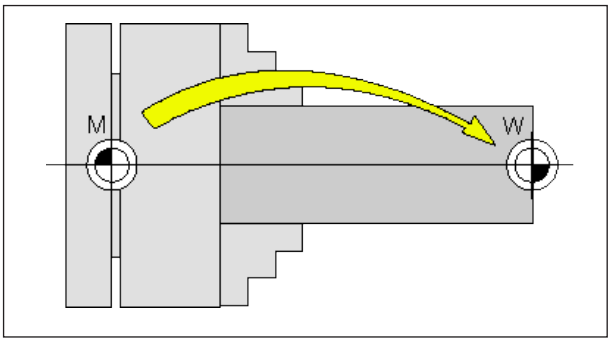
Absolute workpiece positions

If coordinates for a position are related to the workpiece zero point, they are described as absolute coordinates.
Each position of a workpiece is defined unambiguously by absolute coordinates (picture top left).
The origin of the coordinate system is in the machine zero point "M" or, after a programmed zero point shift, in the workpiece zero point "W".
All target points are described starting from the origin of the coordinate system, by stating the relevant X and Z distances.
X distances are given as diameters (as dimensioned on the drawing).



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 path travelled by the tool. Consequently, it is also described as ***chain measurement***.
Each position of a workpiece is defined unambiguously by incremental coordinates (picture left).
The origin of the coordinate system is in the tool housing reference point "N" or, after a tool call-up, in the knife point.
Incremental value programming describes the actual paths travelled by the tool (from point to point).
X is entered as a radius measurement.



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

Zero point offset

The machine zero point "M" is on the rotating axis and on the face of the spindle flange in EMCO rotating machines. 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
- Base Zero Point System (BZS)
- 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)

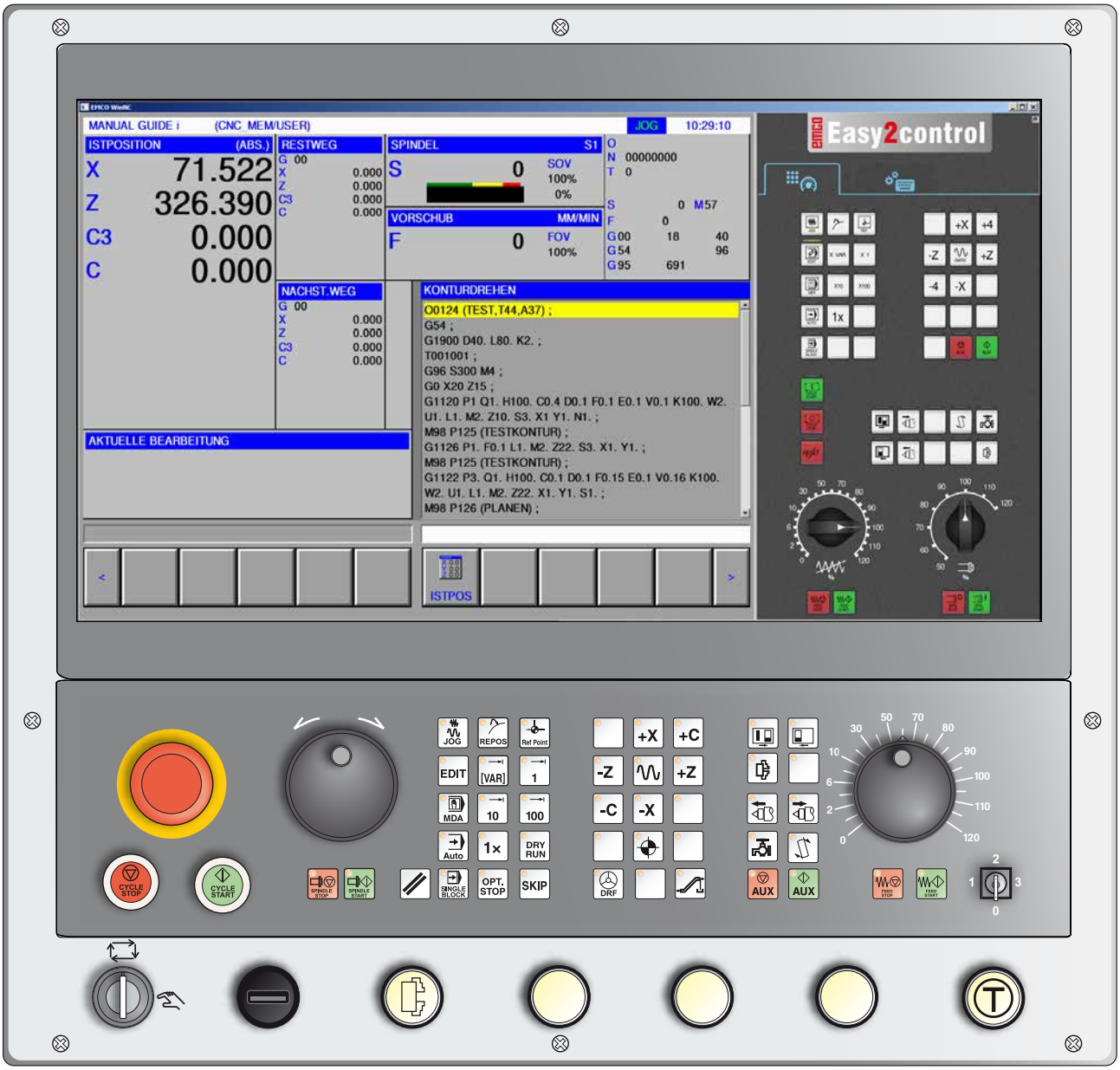
Programmable coordinate transformations (frames) shift the originally selected workpiece coordinate system to another position, to rotate, scale or mirror it.

Workpiece coordinate system (WCS)

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

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:



Y



Queries

Button functions



Satzende, 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).



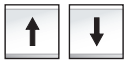
Enter word, accept data.



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.

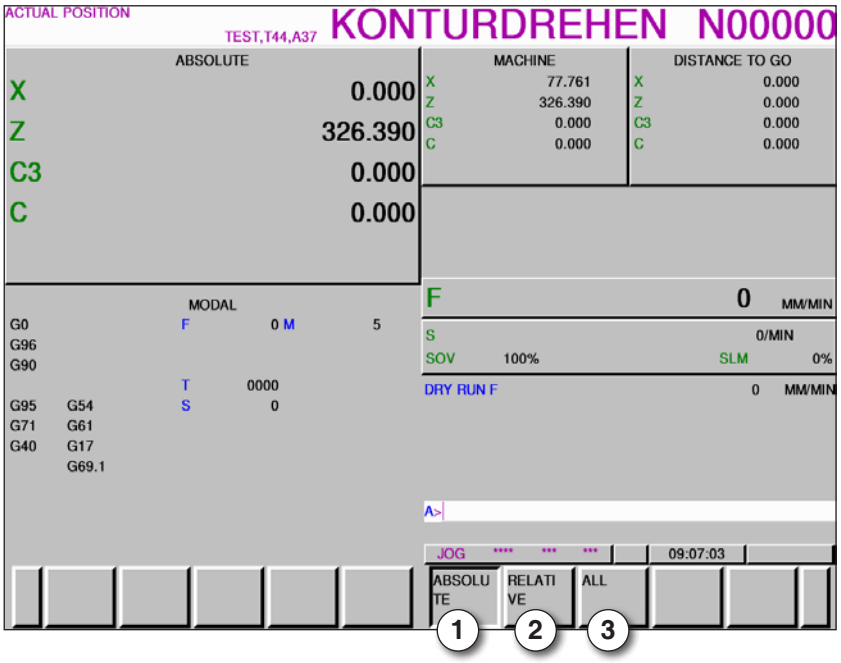


Switch to Manual Guide mode.

Key description ISO functions



Actual position display



1 Absolute position

2 Relative position

3 Indicate both positions together



Release overview display



Release overview

Displays the current WinNC software release



Alarm and message display.

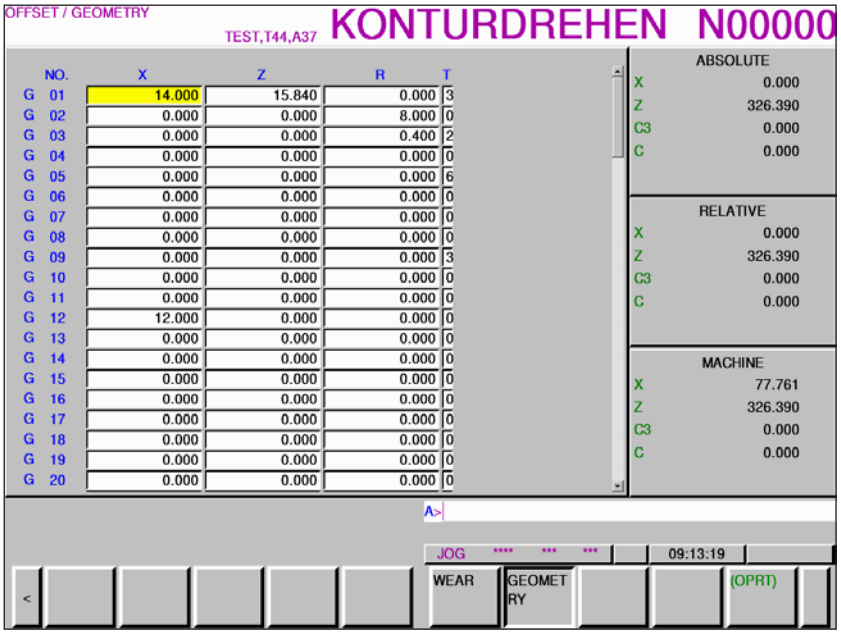


Alarm and report overview

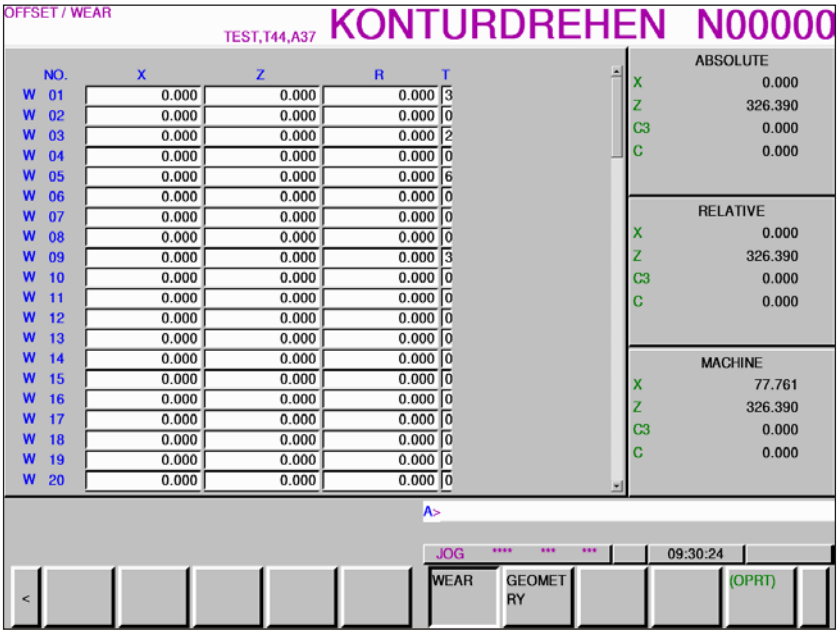
Displays all alarms and reports



Zero point offset setting and display,
Tool offsets, wear offsets and variables

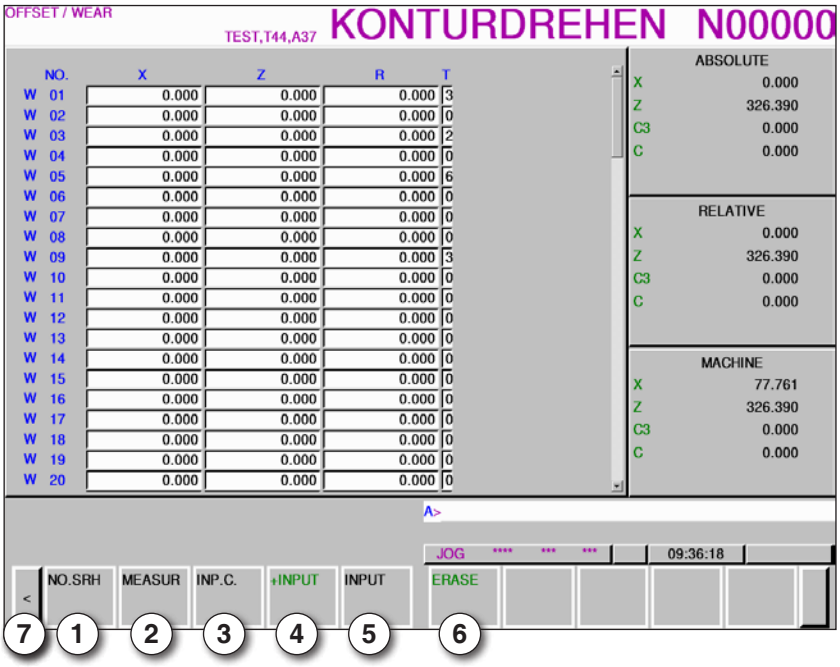


Tool correction geometry



Tool wear offset

The "OPRT" softkey opens further setting options



1 Search for tool number

2 Measure tool

3 Coordinates entry

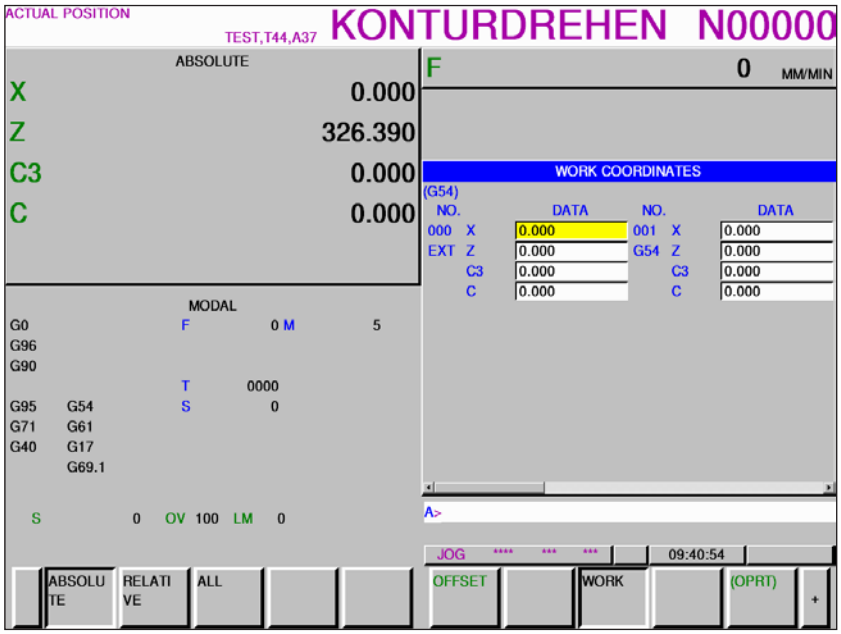
4 Computes current value + entry from entry line

5 Accept value from entry line

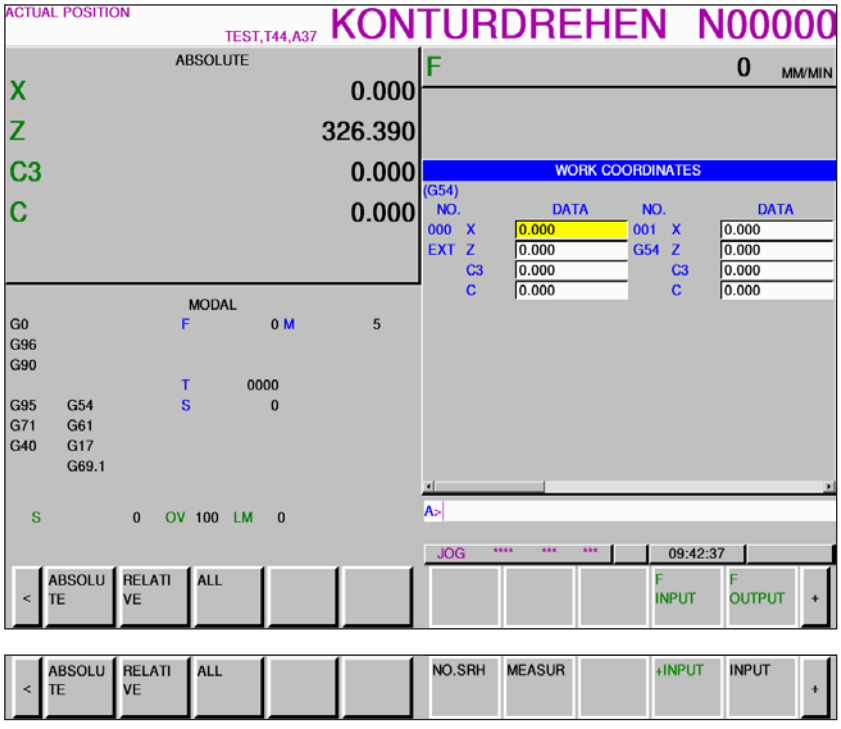
6 Delete

7 Expansion button, scroll to zero point offsets

Instructions:
You limit tool wear correction with "2".



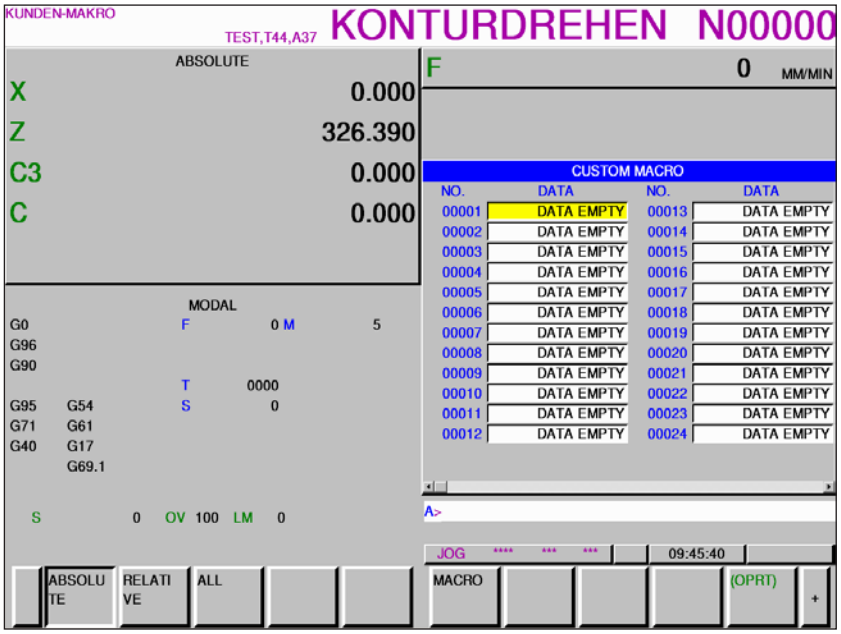
Zero point offsets



Zero point offsets

- 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.

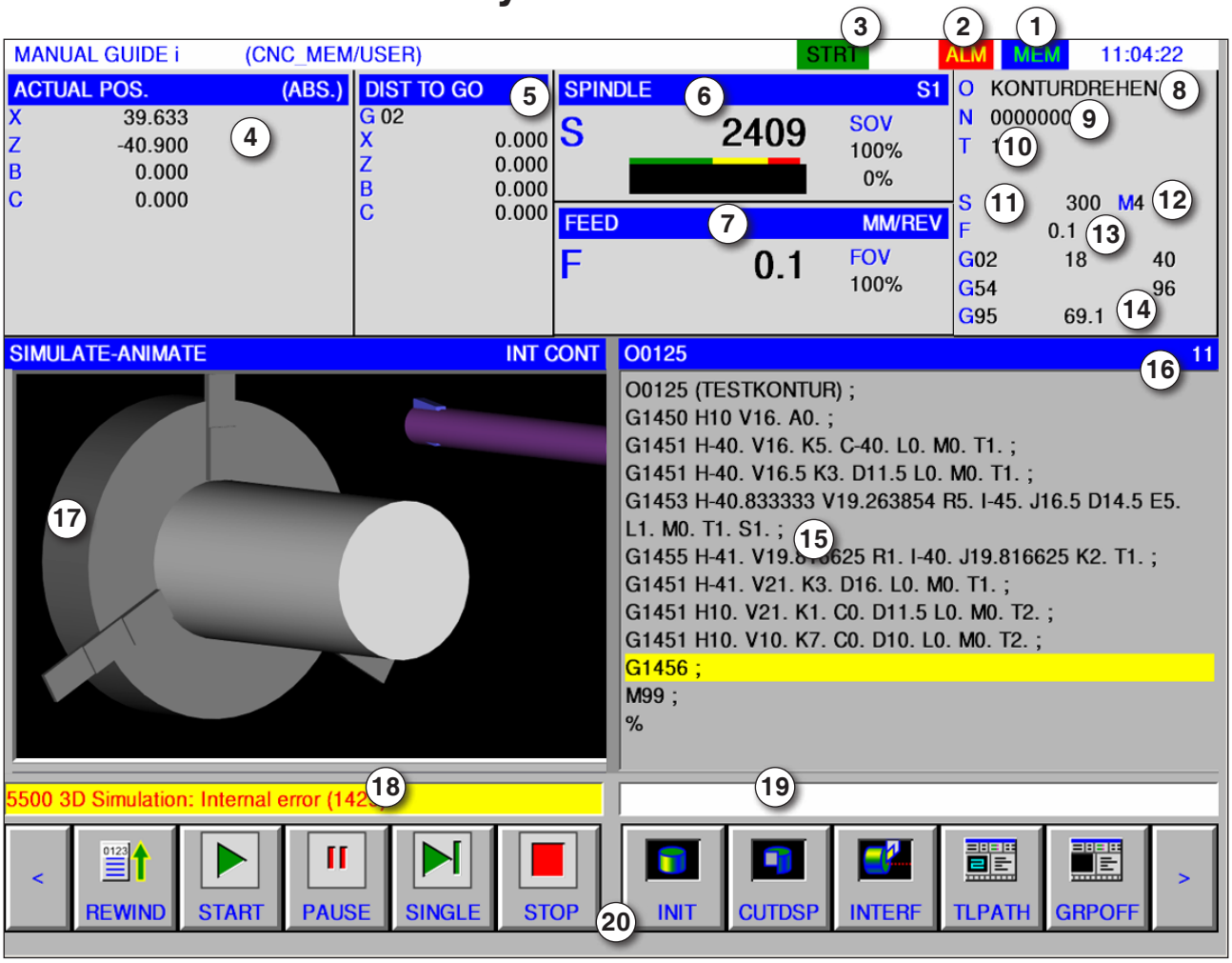
The "+" expansion button calls up the customer macro variables page



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

2 Alarm status

3 Program mode

4 Axis position

5 Residual travel

6 Spindle speed

7 Feed

8 Program name

9 Block number

10 Tool number

11 Spindle speed

12 M-commands

13 Feed display

14 Display of active G-functions

15 Program window

16 Current line number in ISO program window

17 Graphical simulation

18 Message window

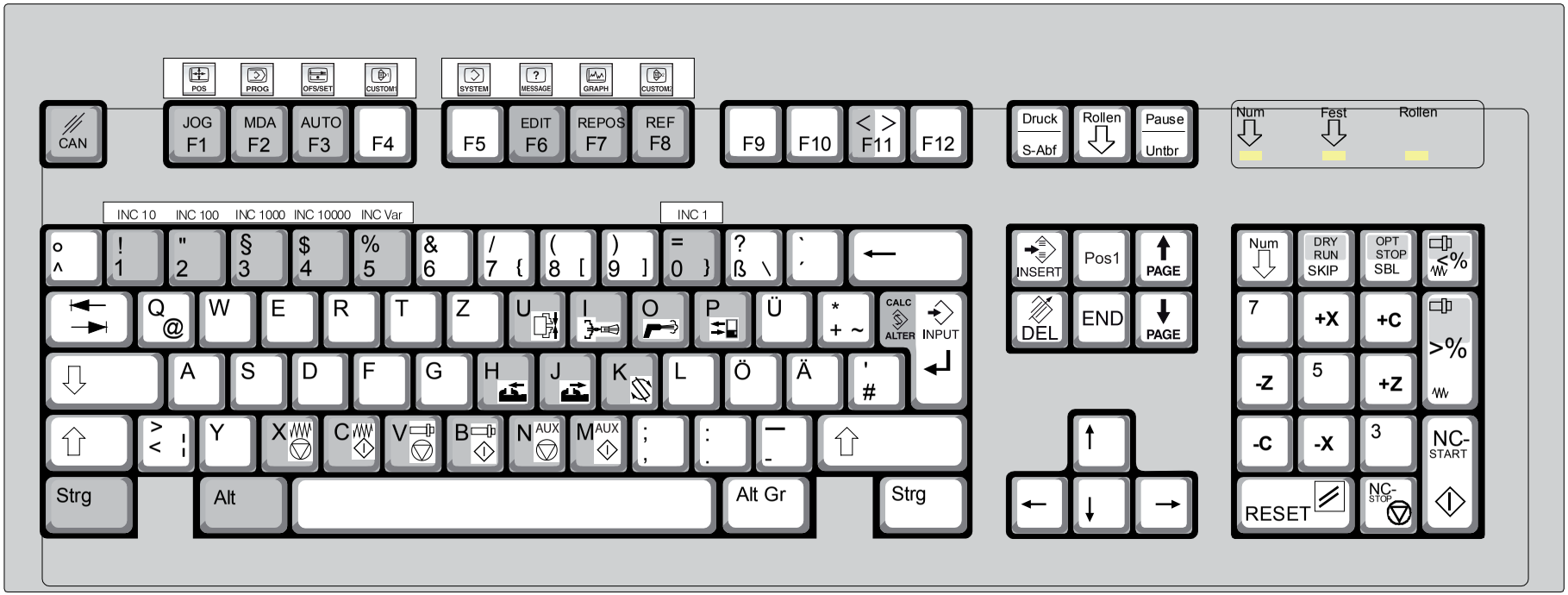
19 Keypad buffer

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



$ 4 = 4 ⇧ $ 4 = $ Alt § 3 = INC 1000 Alt X =

JOG F1 = JOG ⇧ JOG F1 = POS

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

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

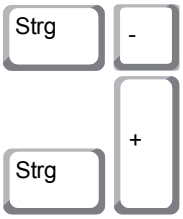
Overview button assignment control keyboard

PC button	Control system button	Check
Entf	DELETE	Delete input
Enter	INPUT	Conclude input and continue dialog.
↑ ← ↓ →	↑ ← → ↓	Move mark
⇧	SHIFT	Upper/lower case
x	SBL	Single block (SBL)
÷	SKIP	Skip (block mask)
0		Reset key (Reset)
Strg ÷	DRY RUN	Dryrun (test run feed)
Strg x	OPT. STOP	Optional stop
⇧ JOG F1	POS	Indicates the current position
⇧ MDA F2	PROG	Program functions
⇧ AUTO F3	OFS/SET	Setting and indication of the zero point shift, tool offsets, wear offsets and variables
⇧ F4	CUSTOM1	Not populated.
⇧ F5	SYSTEM	Parameter settings and indications, and troubleshooting indications.
⇧ EDIT F6	MESSAGE	Alarm and message display.
⇧ REPOS F7	GRAPH	Manual Guide mode
⇧ REF F8	CUSTOM2	Not populated.
Strg JOG F1	HELP	Call context-sensitive help

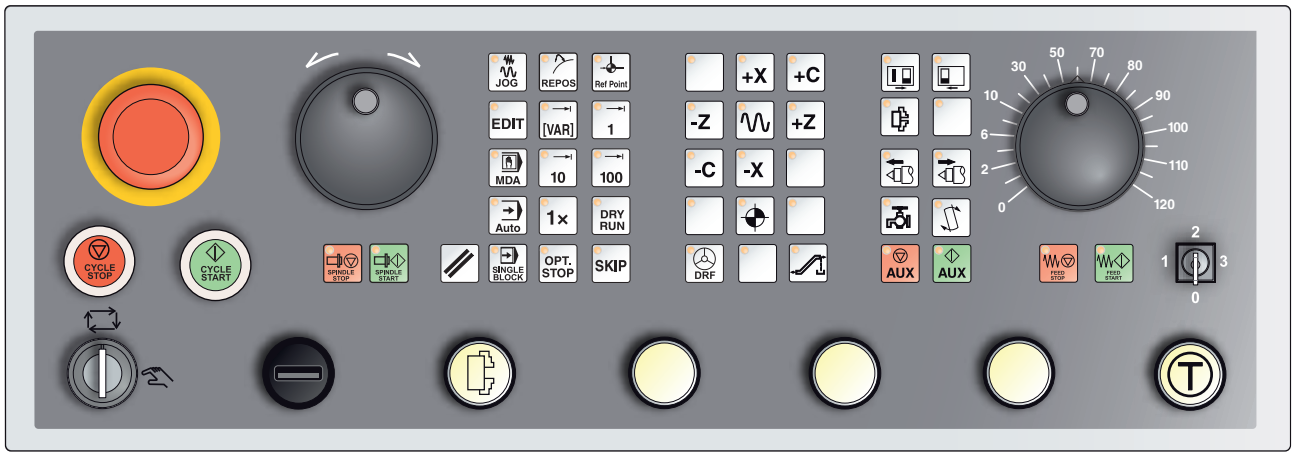
Overview Button Assignment Machine Operating Elements

PC button	Control elements	Check
Alt U		Clamping device open / closed
Alt I		Internal/External Clamping (Option Concept Turn 55)
Alt O		Coolant / Purge on / off
Alt P		Door open / closed
Alt H		Tailstock forwards
Alt J		Tailstock backwards
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 machining again with the NC Start button.
If the function is not active, the M01 special function (from the parts program) will be ignored.

Edit



Toggle into Edit mode.

Handwheel mode (optional)



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

Reset key (Reset)



Pressing the Reset key:
interrupts the machining of the current parts program.

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

Feed Stop



This button interrupts a programmed carriage movement.

Feed Start



This button continues a programmed but interrupted carriage movement.
If the main spindle motion was also interrupted, this must be switched on again first.

Single block

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



When single block processing is active:

- the current block of the parts program is only machined when you press the NC Start key.
- machining stops after a block is executed.
- the following block is executed by pressing the NC Start key again.

The function can be deselected by pressing the Single Block key again.

Cycle stop



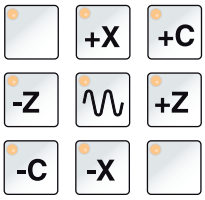
After pressing the Cycle stop button after the controller has taken over the function, machining of the current parts program is suspended
Machining can then be continued by pressing the Cycle start key.

Cycle start



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

Arrow keys



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

Depending on the machine configuration, various arrow keys are available

Rapid traverse



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

Reference point



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

Chip conveyor (Option)



Switch on chip conveyor:
Forwards: Hold the key down for less than 1 second.
Backwards: Hold the key down for longer than 1 second.

The chip conveyor will be switched off after a defined time (approx. 35 seconds).
This value is set in the factory.

Tool turret

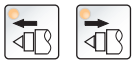


This button continues to swivel the tool turret by one tool position in JOG mode.



If the "AUX ON" button is pressed at the same time, it swivels back by one position.

Tailstock forwards, tailstock backwards



These buttons move the tailstock forwards or backwards.

Clamping devices



This button activates the clamping device.
For chuck/collet changeover see "Machine configuration".

Coolant



This button switches the coolant unit on and off.
The LED indicates that the coolant pump is running.
If the chip guard door is opened, the coolant pump continues to run only while the button is pressed.

If this button is pressed in AUTOMATIC mode, after the coolant has been switched on in the program with M8, the coolant pump is switched off and the LED flashes. Switch it on again by pressing the button once more.

Programming:
M8 Coolant on
M9 Coolant off

Automatic machine doors



To open and close the machine doors.

Spindle Stop



This key interrupts the running of the main spindle and the power tool. If this happens during a feed movement, that must be stopped first.

Spindle Start



This key continues the programmed spindle sequence.

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 metric 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:
  Millimetres 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

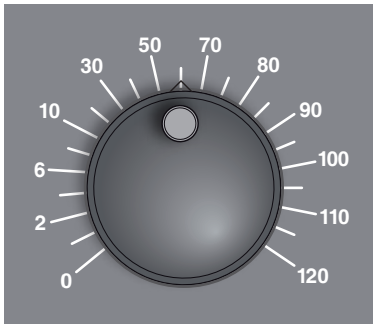
If the carriage has to be freewheeled before referencing (e.g. from a position where it is at risk of colliding), press the button 

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  .

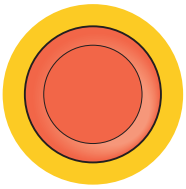


Override switch (feed rate override)

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

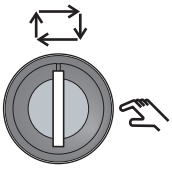


EMERGENCY SHUTDOWN

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.

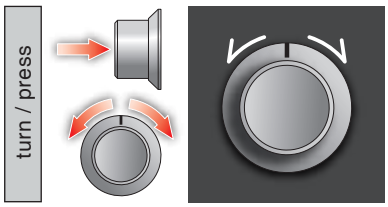


Key Switch Special Operations Mode

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 ....).



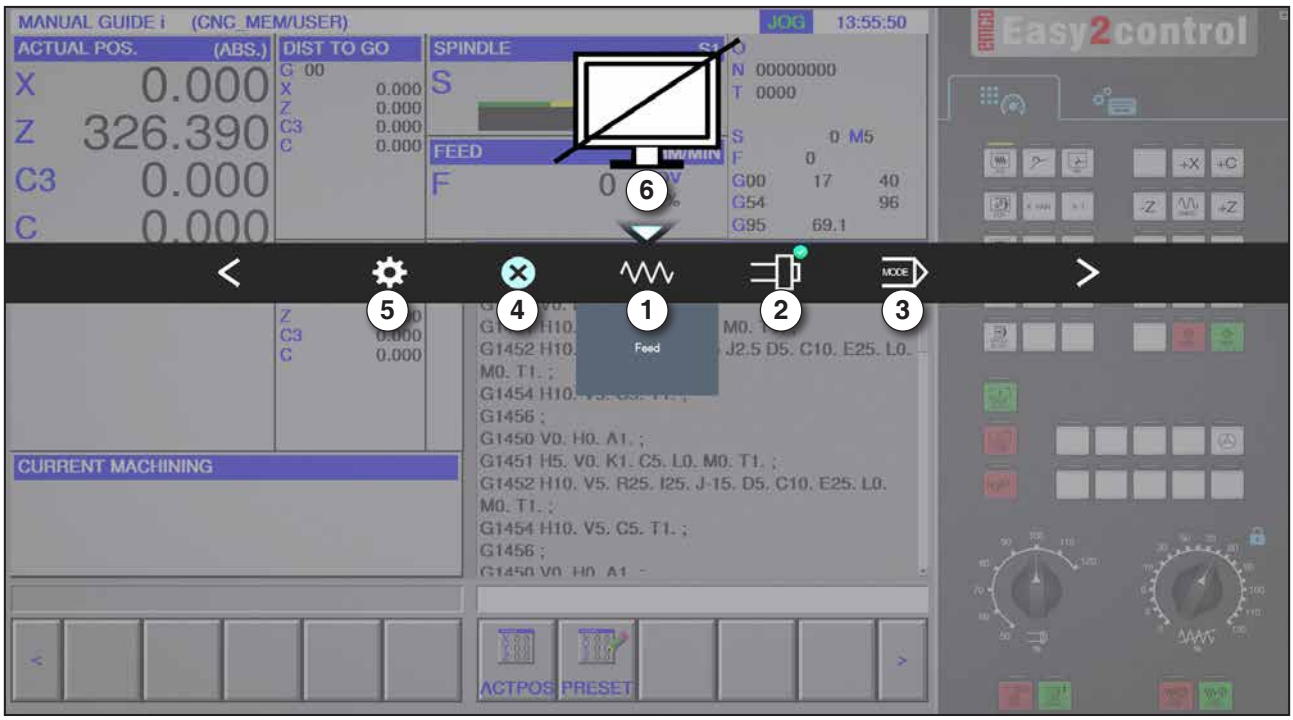
Multifunction switch for operating modes

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:



Overwiew

1 Feed override: controls the feed rate equivalent to conventional feed override

2 Spindle override: controls the spindle speed equivalent to conventional spindle override

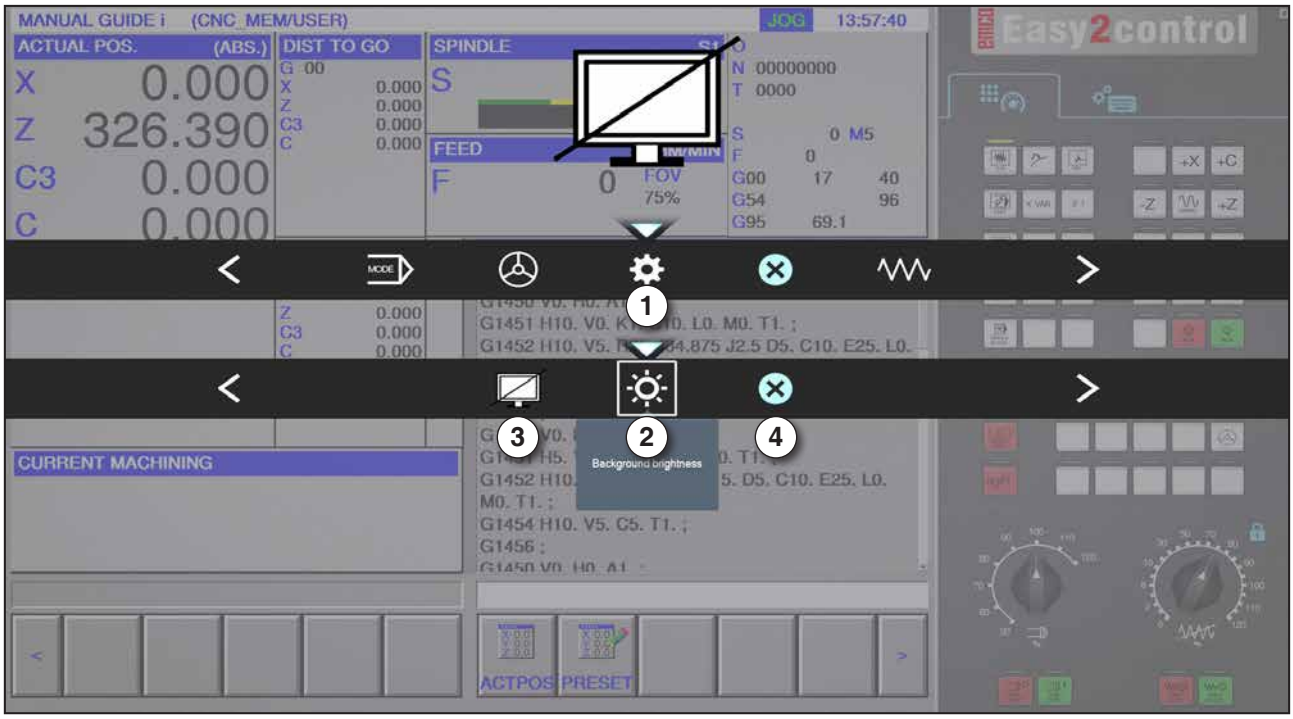
3 Modes: allows you to select the operating mode using the multifunction operation

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.

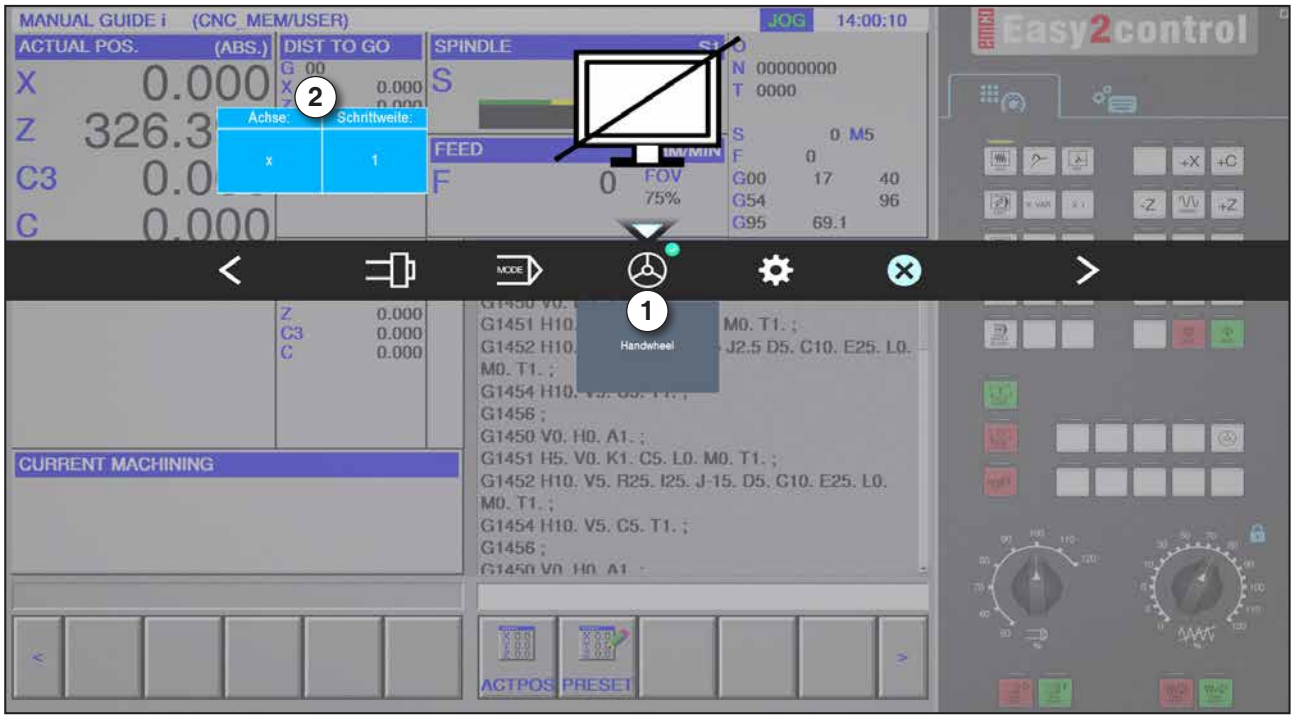
Setting the background trancparency



- 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 dissapears.



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 left button clamping means

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



USB connection (USB 2.0)

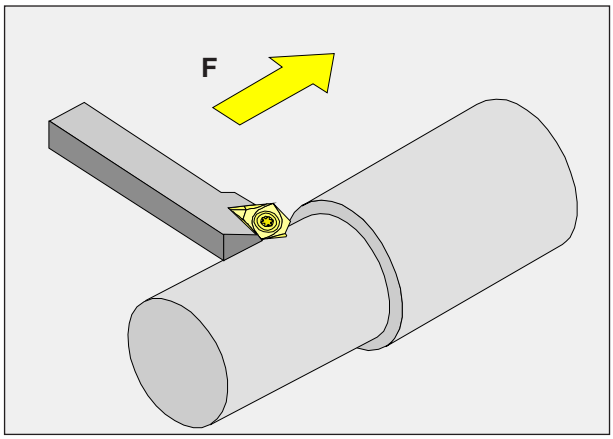
Data is exchanged with the machine (data copying, software installation) via this USB connection.



Enable button

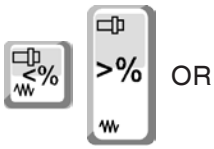
When the door is open, axis movements via the arrow keys and tool turret movements are authorized by pressing the enable button (precondition key switch in SET-UP position).
In machines with automated doors (option) pressing the enable switch opens the machine doors.

C: Operation

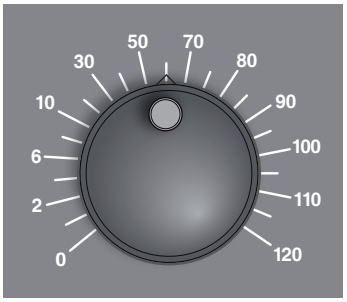


Feed F [mm/min]

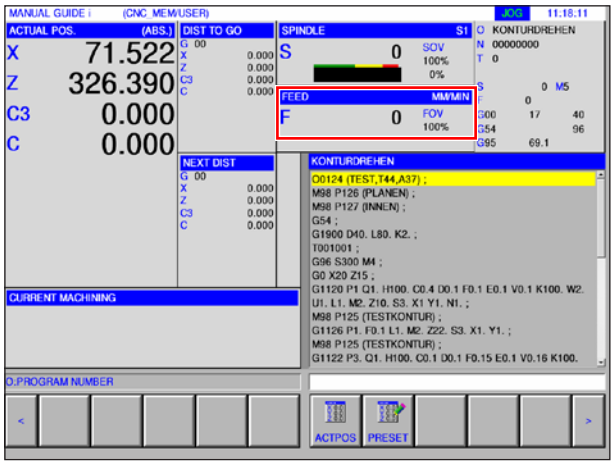
The feed F is the speed in mm/min (feet/min) with which the tool center point moves on its path. The maximum feed can be different for each machine axis and is defined by machine parameters.



OR



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 %.

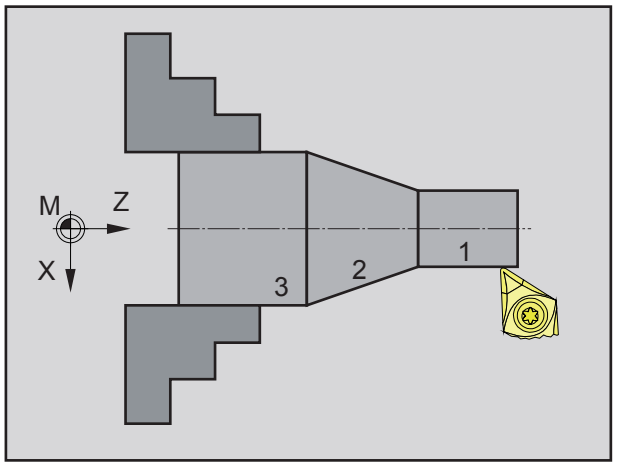


Adjustment range:
0% to 120% of the programmed feed.
Only the changed percentage and not the resulting effective value will be shown.
In rapid traverse 100% of the maximum rapid traverse feed will not be exceeded.

Speed fundamentals

You choose between the following procedures:

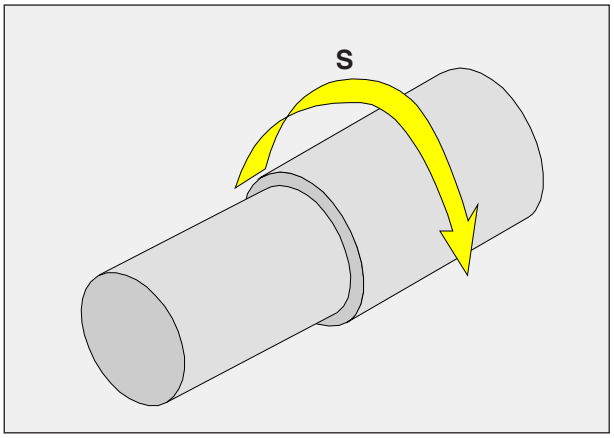
- **Spindle speed S:**
  You program the spindle speed directly. The spindle speed is independent of the diameter on which the tool works.
- **Constant Cutting Speed CCS:**
  You program the spindle speed indirectly. The control system changes the speed depending on which diameter the tool is working directly. In this way a constant cutting speed will be achieved.



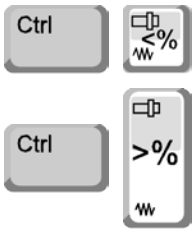
Example:

- **Spindle speed S:**
  Sections 1 to 3: same speed.
- **Constant Cutting Speed CCS:**
  Section 1: high speed.
  Section 2: constantly decreasing speed.
  Section 3: low speed.

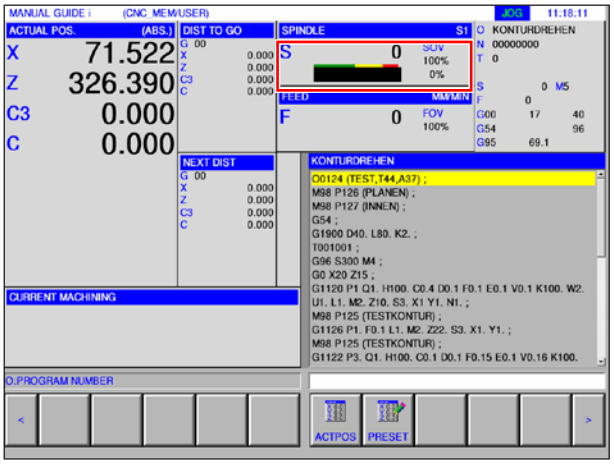
Spindle speed S [rev/min]



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



Spindle speed correction
The spindle speed S programmed by you corresponds to 100%.
This button 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.

Metric 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.

Metric 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.

Metric 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.

Metric 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:

Referencing 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

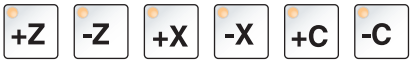
Pressing the button "Reference point" causes the axes to approach their reference points automatically one after another. First the axes then the tool changer will be referenced.

Move slides manually



The machine axes are moved manually using the arrow keys.

- Switch to JOG mode.



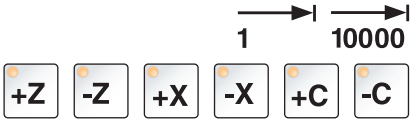
- The arrow keys buttons move the axes in the corresponding direction as long as the key is pressed.
- The feed speed is set with the override switch.



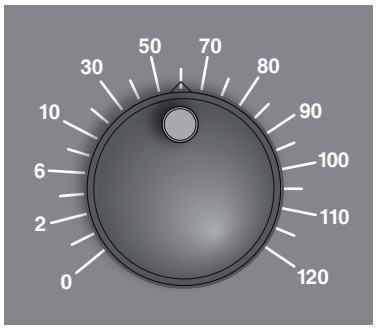
- If the button is pressed simultaneously, the carriages move in rapid traverse.

Move slides in step-movement

The machine axes can be moved in steps using the arrow keys.



- Switch to INC mode.
- The arrow keys move the axes in the corresponding direction by the set incremental 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).

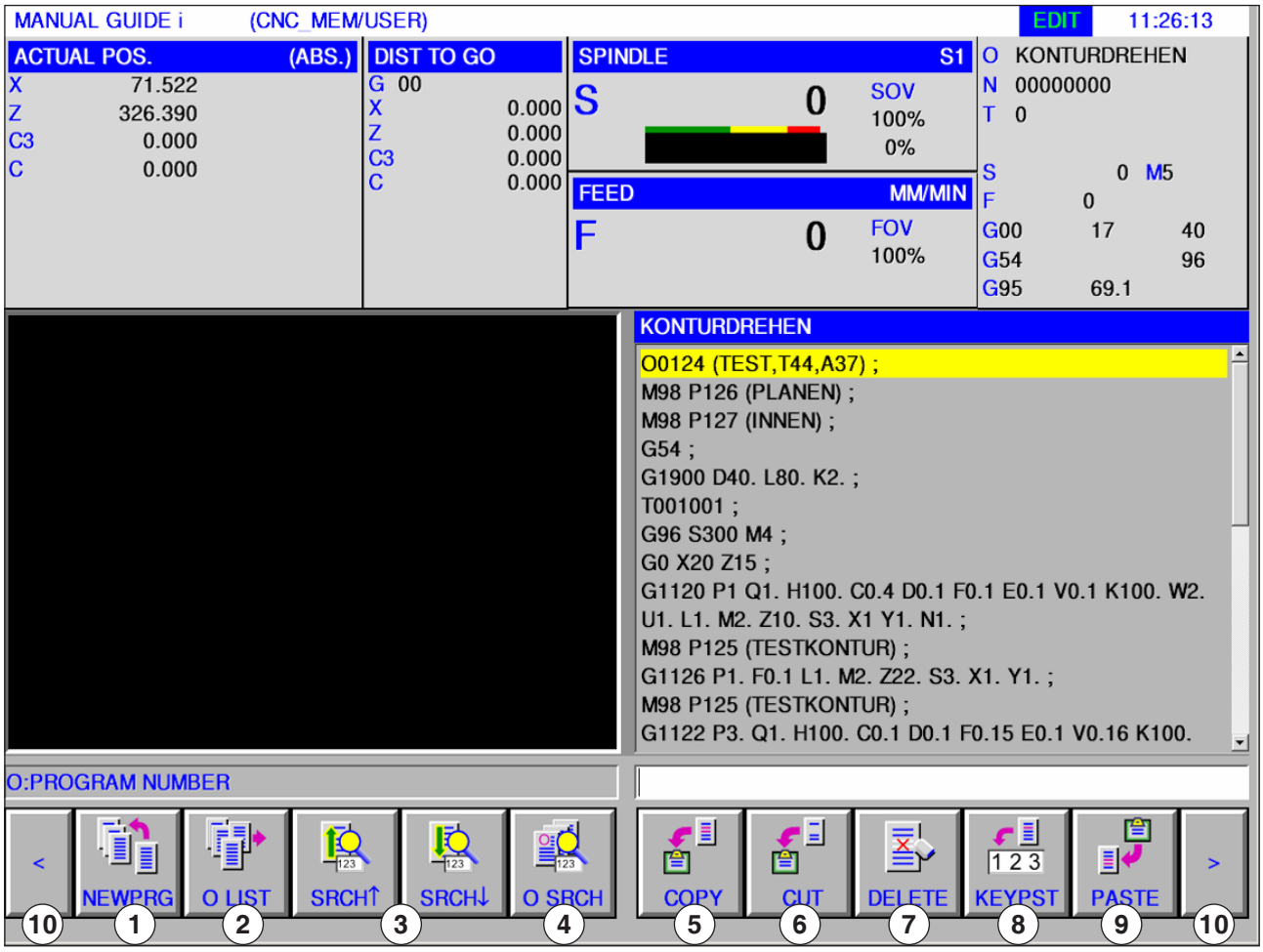
Options in Automatic mode:
- Program correction
- Block search run
- Overstore
- Program influence

(see Section G program sequence)

Program management

EDIT

- Select "Edit" mode.



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.



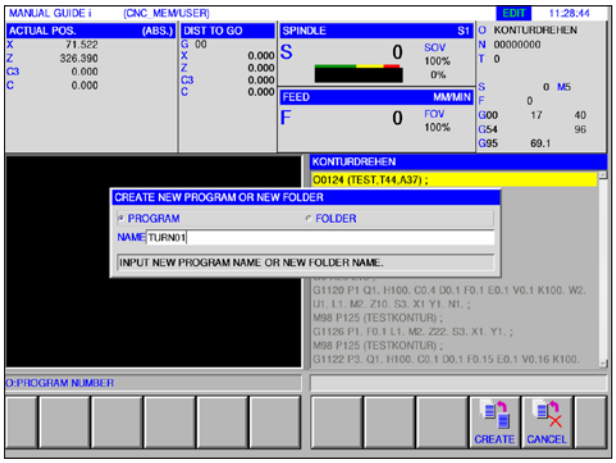
1 Select "Edit" mode.



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.T\PRG\LIBRARY

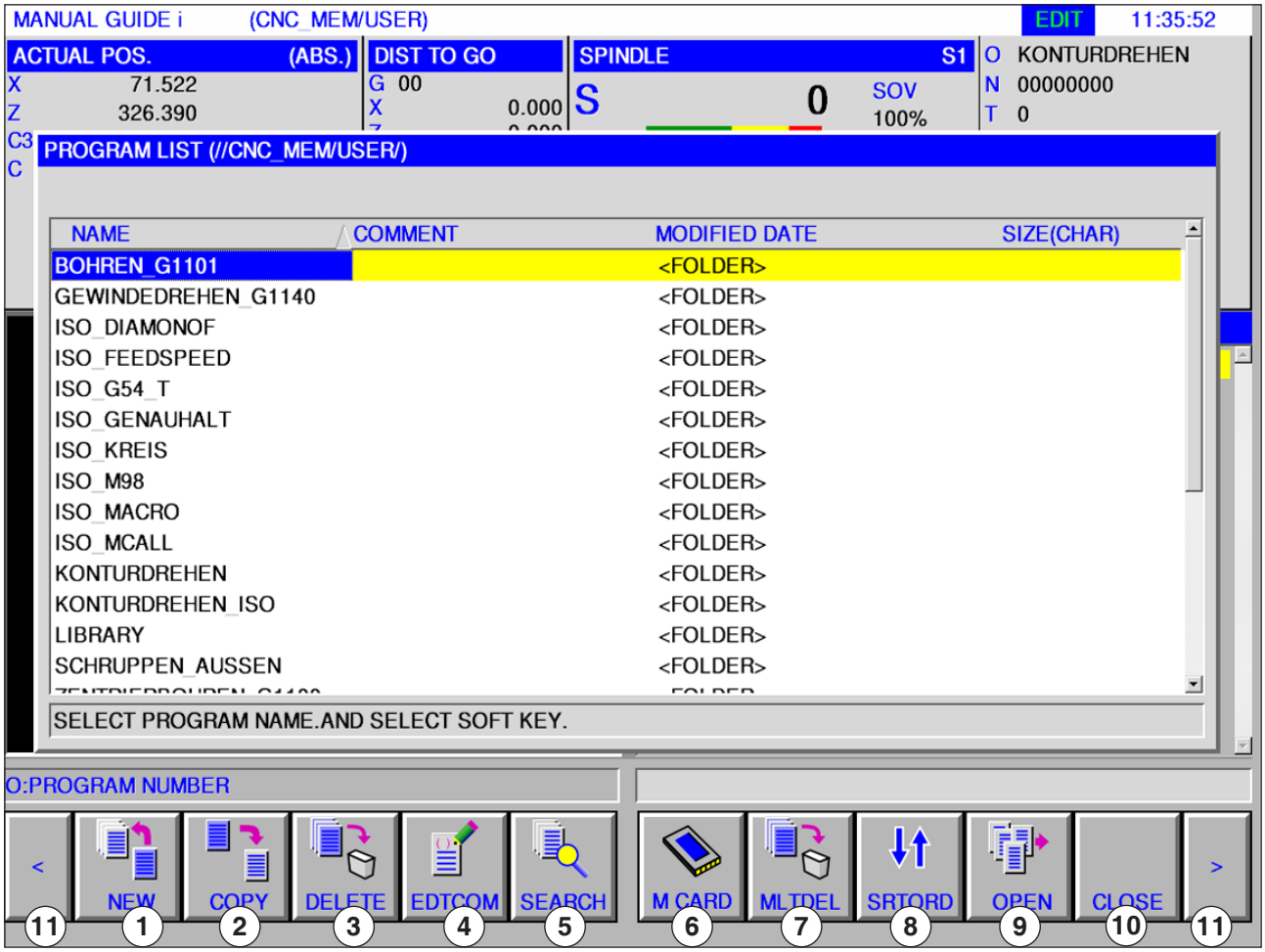
Programming list



1 Select "Edit" mode.



2 Press softkey.



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

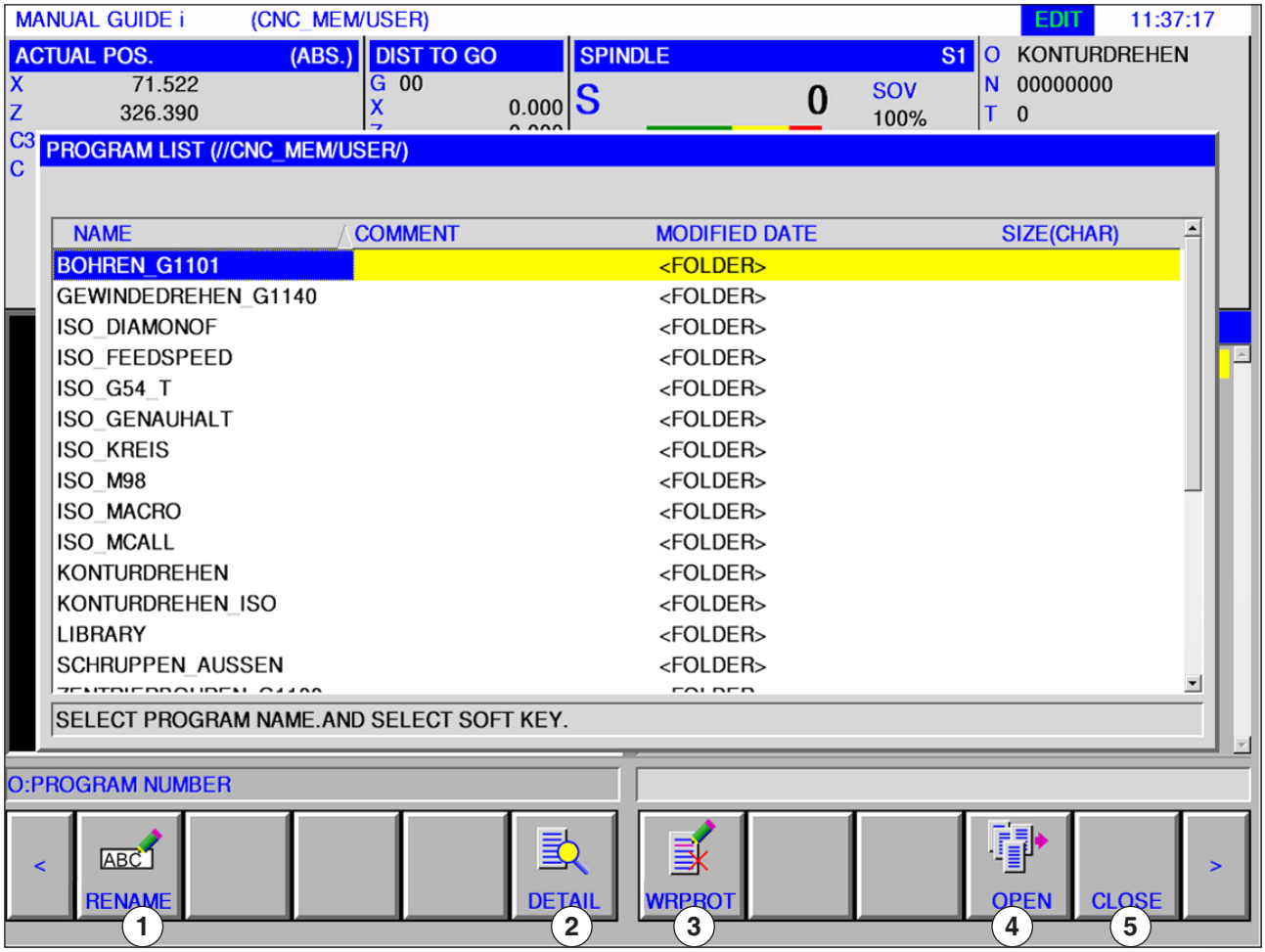
8 Change the sorting sequence

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

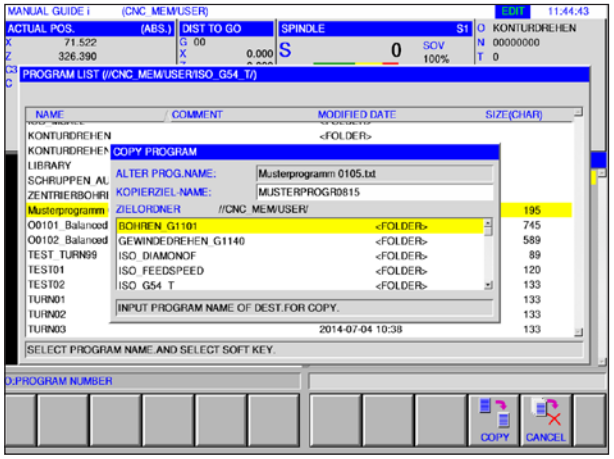


1 Select "Edit" mode.



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

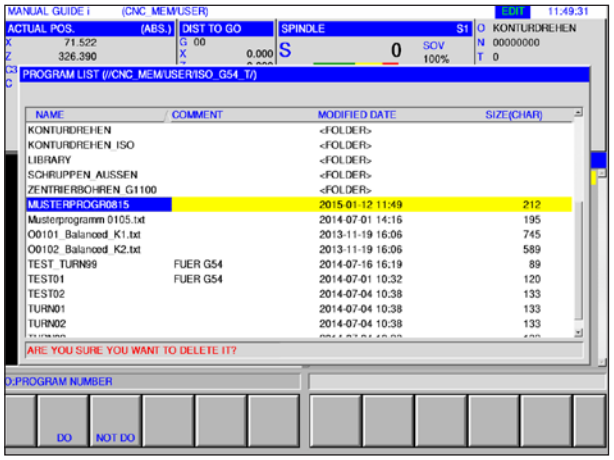


1 Select "Edit" mode.

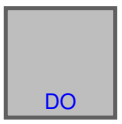


2 Move mark to the desired program.

3 Delete program.



4 This softkey calls the window for deleting programs. "DO" deletes the given program, "NOT DO" interrupts the delete process.



4 Press the softkey to delete.

Enter comment

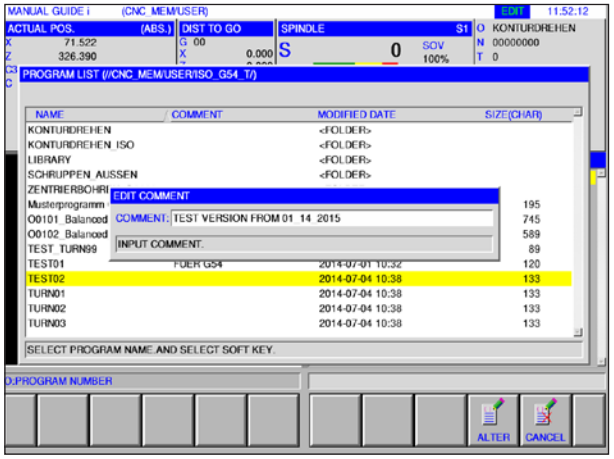


1 Select "Edit" mode.



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.

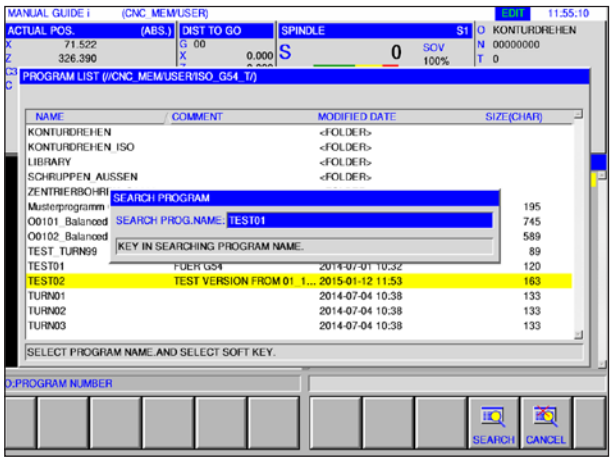
Search program



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 search for the given program.

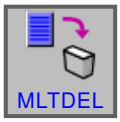


4 Press the softkey to search.

Delete several programs simultaneously



1 Select "Edit" mode.



2 Delete a number of programs.



3 This softkey calls the window for deleting several 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.



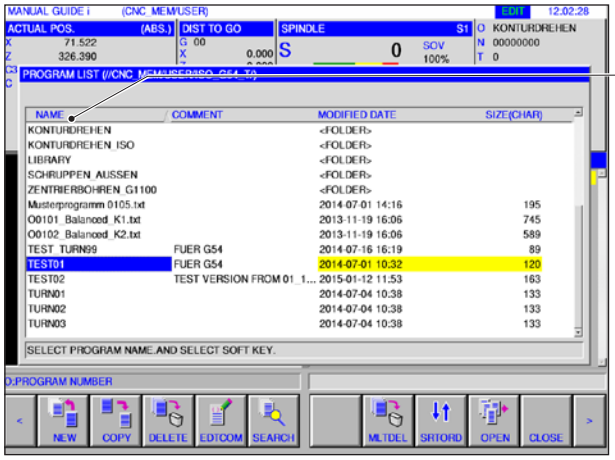
8 Press the softkey to delete.

Change the sorting sequence

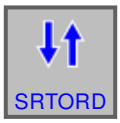
1 Select "Edit" mode.



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.



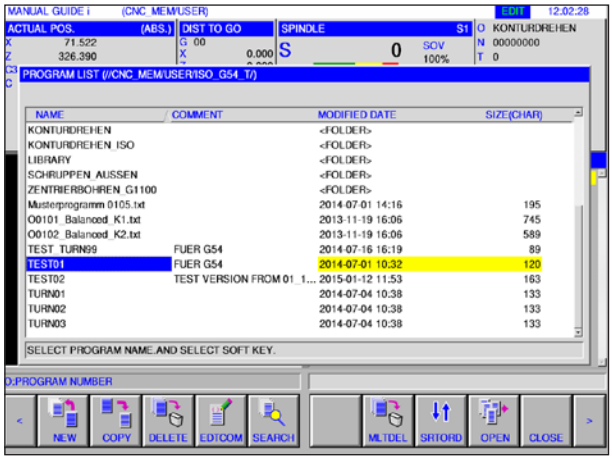
4 Press the softkey to change.

Open program

1 Select "Edit" mode.



2 Open program.



3 This softkey opens the selected program.

Change program name

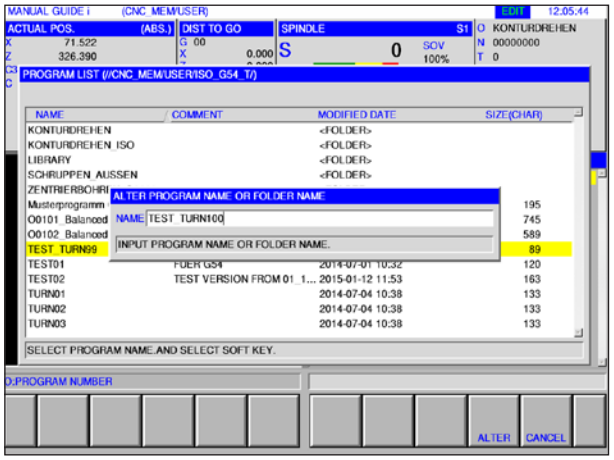


1 Select "Edit" mode.

2 Move mark to the desired program.



3 Change the program or folder name.



4 This softkey calls the window for renaming programs. After a new program name has been entered, press the "ALTER" softkey to rename the given program.



5 Change the program or folder name.

Program properties



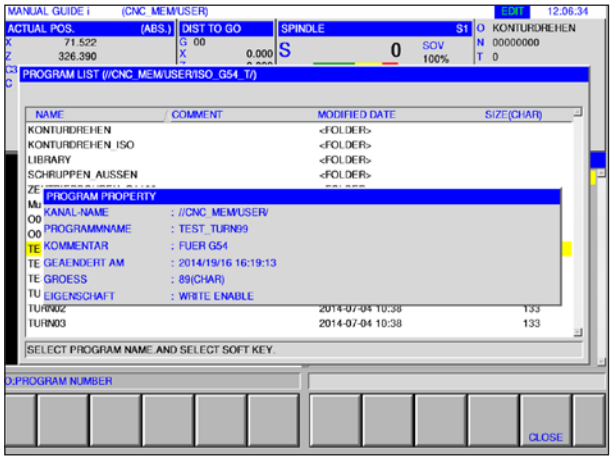
1 Select "Edit" mode.

2 Move mark to the desired program.

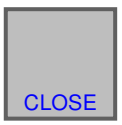


3 Show program properties.

This softkey calls the window for displaying program properties.



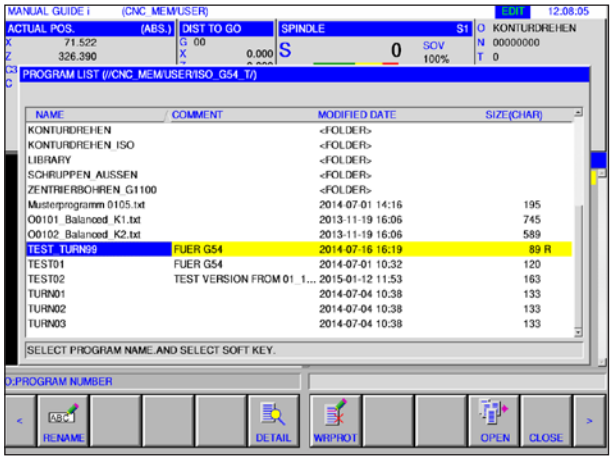
4 End program properties display.



Program protection







1 Select "Edit" mode.

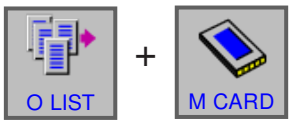
2 Move mark to the desired program.

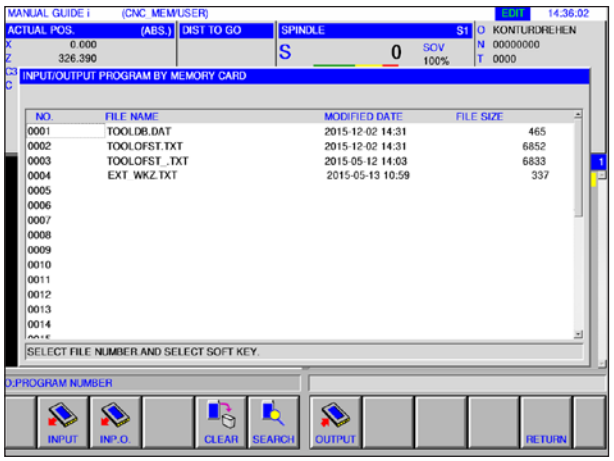
3 Enable write protection for the program.

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.

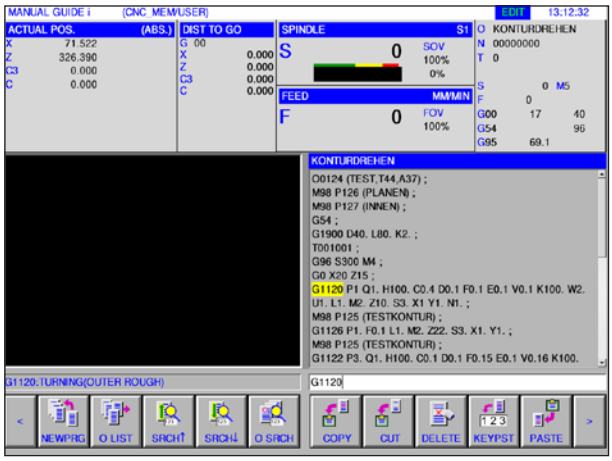
3 The following softkeys are available:

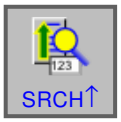
- "INPUT": Upload file from memory card
- "INP.O": Upload file and change O- number
- "CLEAR": Delete file
- "SEARCH" Search for file
- "OUTPUT": Save file
- "RETURN": End dialogue

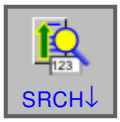
Note:
The path of the directory from where the control software was installed is set as default. The path can subsequently be changed in EMConfig.

Search for text from the input line in the program









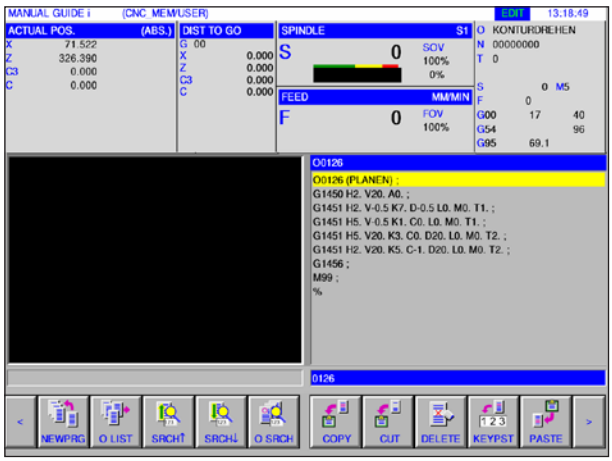
1 Select "Edit" mode.

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 highlighted in yellow.

Search for and open the program







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



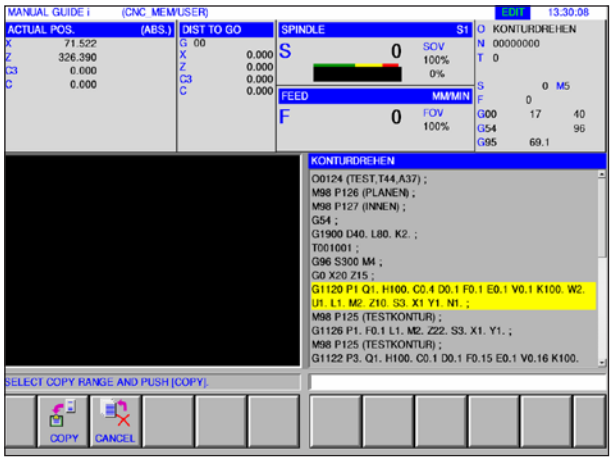
Instructions:
If "O SRCH" is pressed with empty entry line, the next program of the actual folder is opened.



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

Copy highlighted text to clip-board





1 Select "Edit" mode.

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



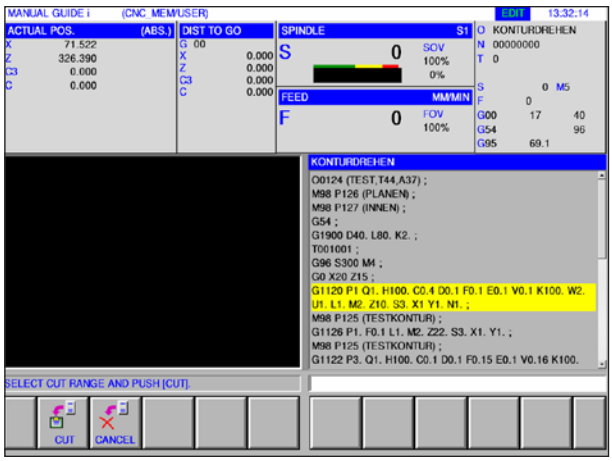
3 Press the "COPY" softkey to save the text in the Clipboard.



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

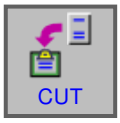
Move highlighted text to clip-board





1 Select "Edit" mode.

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

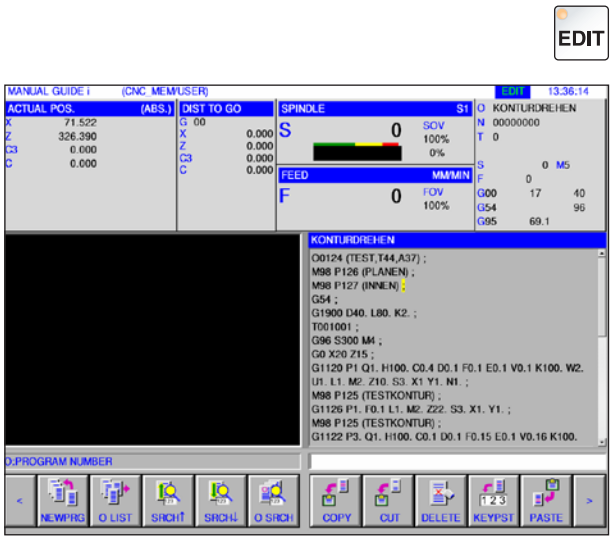


3 Press the "CUT" softkey to move the text from the program into the Clipboard.



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

Insert text



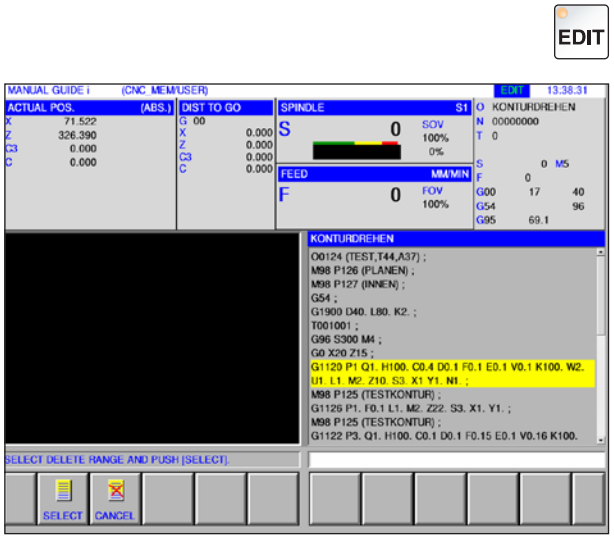


1 Select "Edit" mode.

2 Move the cursor to the point where the text from the Clipboard is to be inserted.

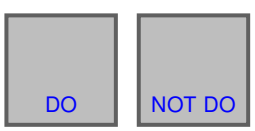
3 This softkey inserst the content of the clipboard at the point after the cursor.

Delete highlighted text









1 Select "Edit" mode.

2 Set the area for deletion with the cursor keys (the highlighted area will be yellow).

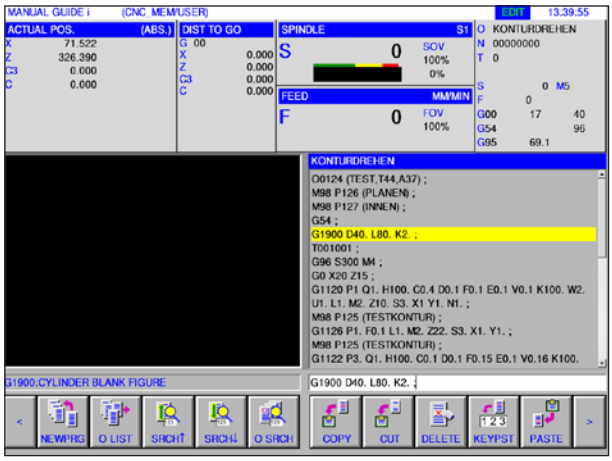
3 This softkey calls the window for deleting highlighted text.

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

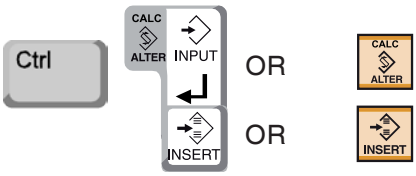
5 Confirm the deletion process with "DO" or cancel it with "NOT DO".

Insert highlighted text into an input line









1 Select "Edit" mode.

2 Set the area for insertion with the cursor keys (the highlighted 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.

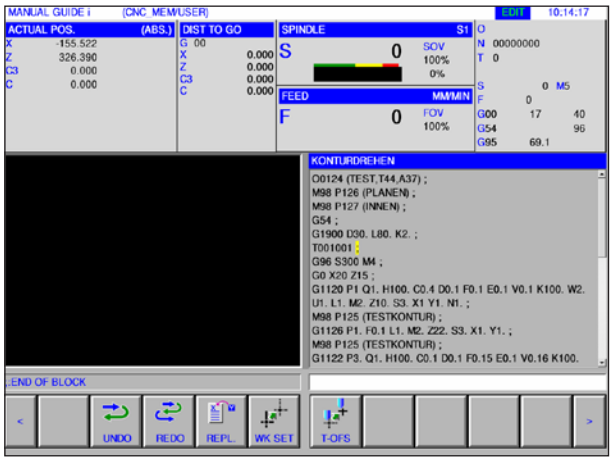


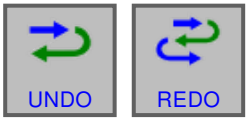
Instructions:
Use the "KEYPST" keyboard input to change very long program comments and parts of customer macro program blocks with ease.

Revocation and withdrawal



1 Select "Edit" mode.



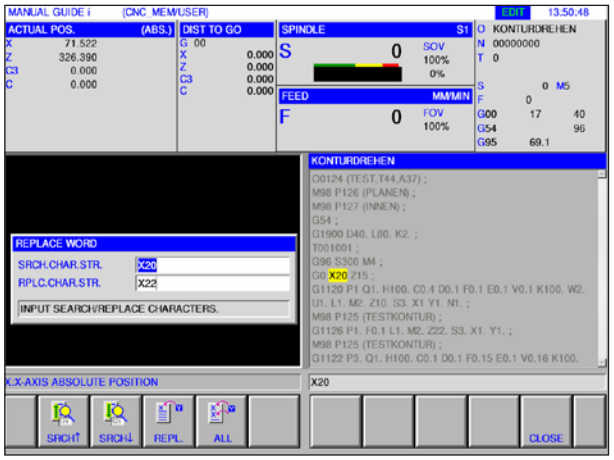


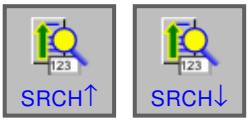
2 Press the "UNDO" softkey to revert to an editing process.
Press the "REDO" softkey to revoke a change retrospectively (redo).

Search and replace

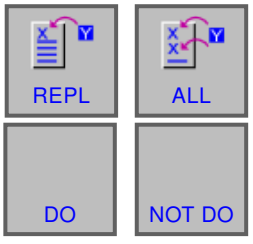


1 Select "Edit" mode.





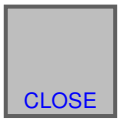
2 Press the "SRCH↑" or "SRCH↓" softkey to search for the string to be replaced.
Enter the replacement string value and press "REPL" or "ALL".



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

End program listing





1 Select "Edit" mode.

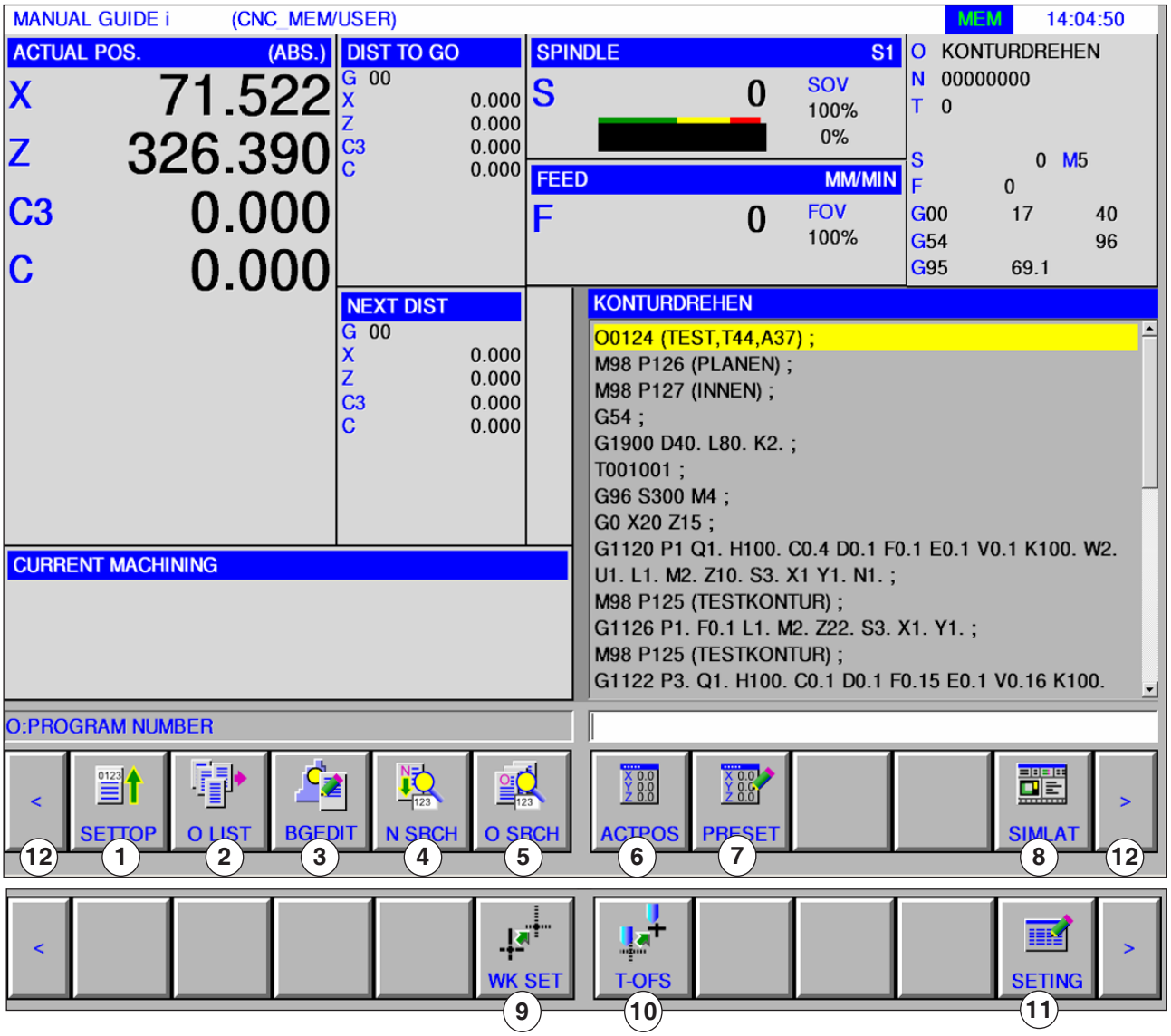
2 Press the "CLOSE" softkey to end the program listing.

Program mode

Auto

Select "Auto" mode.

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



1 Go to the start of the selected program

2 Call 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 Default relative position

8 Simulation

9 Workpiece coordinates system

10 Tool settings

11 Forward and Back (Extension keys)

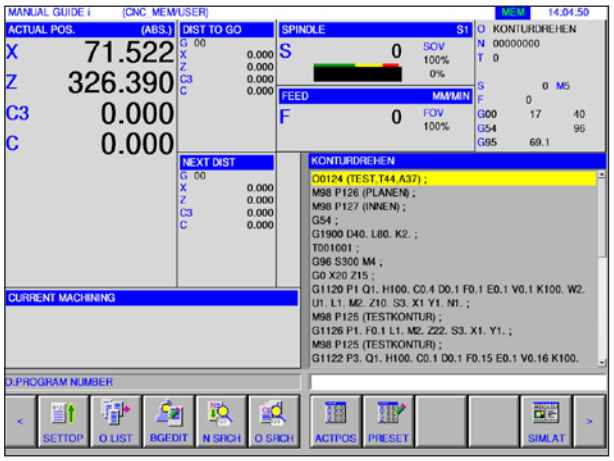
12 Open editor for fixforms

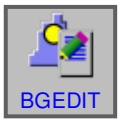
Instructions:
See chapter F for more information about tool settings: Tool management

Background Editing



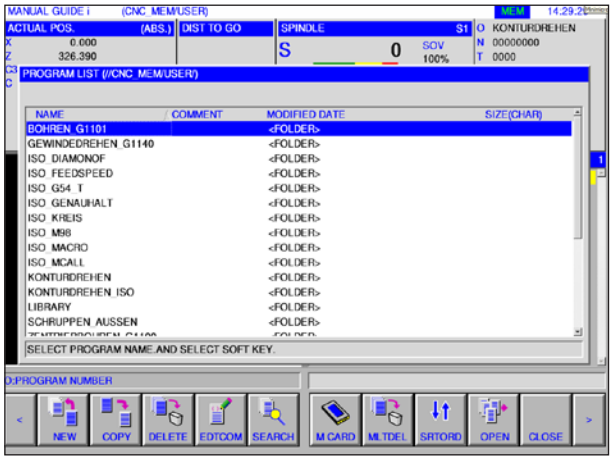
1 Select "Auto" mode.





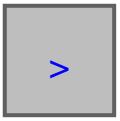
2 Press "BGEDIT" softkey.
While a part is being machined, contents of other parts programs can be edited.

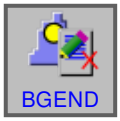
By clicking the "BGEDIT" softkey in MEM mode, the display switches to the screen for background editing. The program directory for choosing programs is displayed.





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



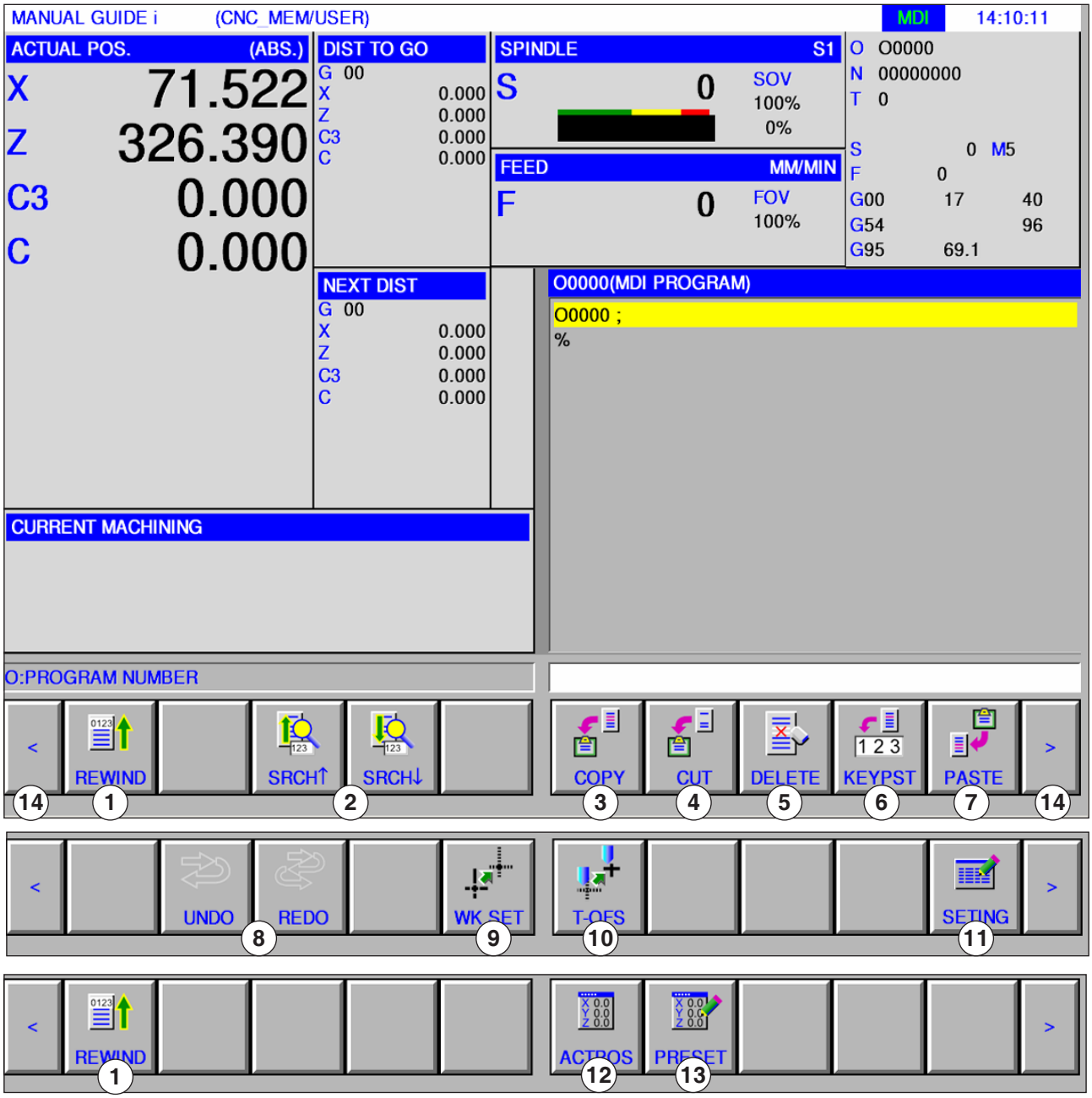
Note:
Switching to another operating mode during editing will end the background edit mode.

Semi-automatic mode



Select "MDA" mode.

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





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 marked text into a buffer file

4 Move marked text into a buffer file

5 Delete marked text

6 Insert marked text into an input line

7 Insert text from a buffer file

8 Revoke/Withdraw

9 Workpiece coordinates system

10 Tool settings

11 Open editor for fixforms

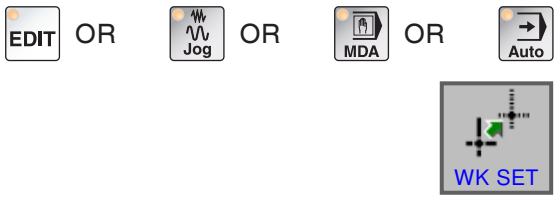
12 Display Actual position ABSOLUTE, RELATIVE and MACHINE

13 Default relative position

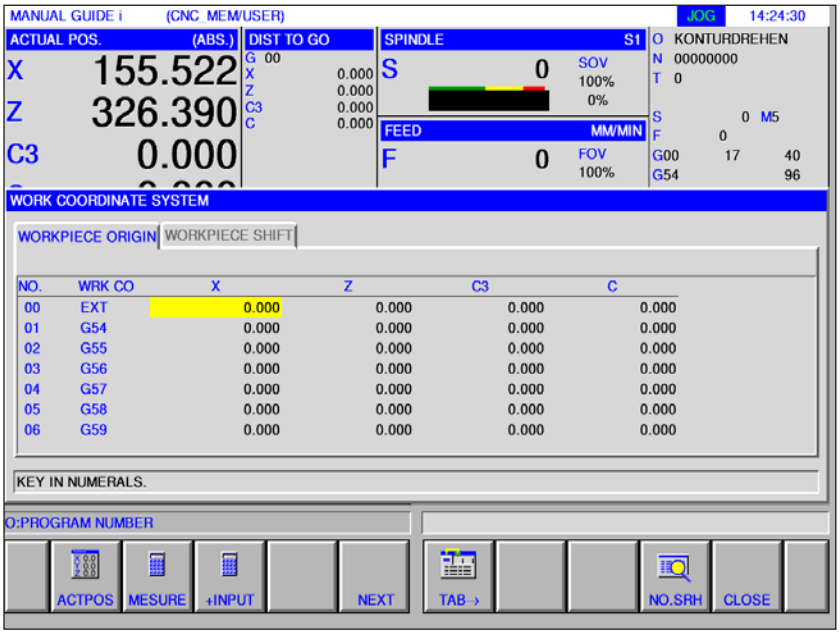
14 Forward and Back (Extension keys)

Zero point table

Setting up workpiece coordinates data

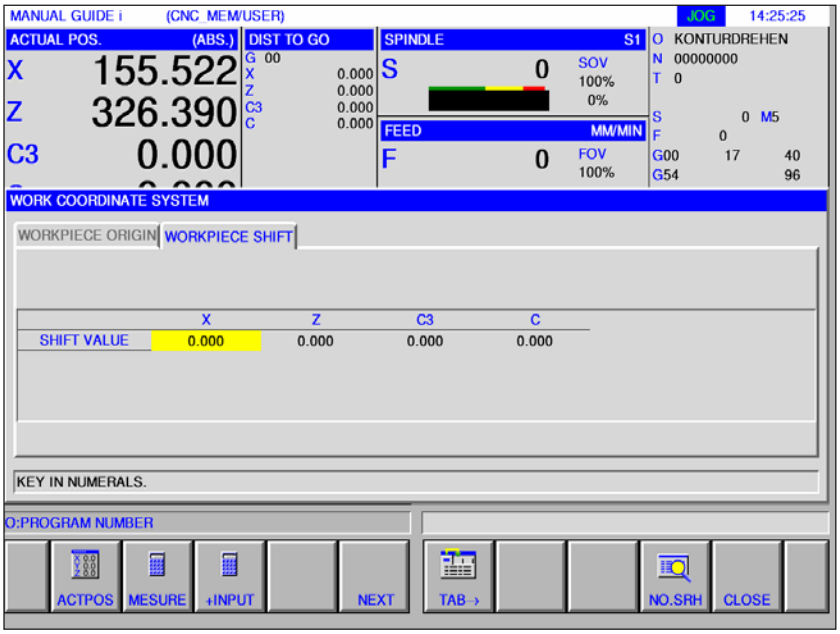


- Select "Edit", "JOG", "MDA" or "AUTO" mode.
- Press softkey.
- "WK SET" opens the workpiece coordinates window in all modes, such as EDIT, JOG, MDA and AUTO.



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

Workpiece zero point offset

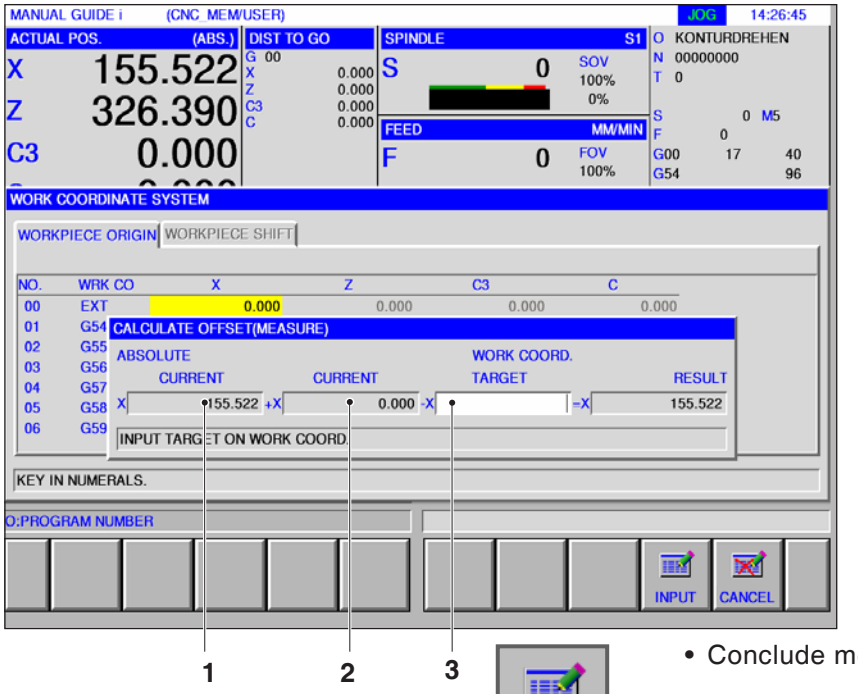


Workpiece coordinates system offset data

Measuring: workpiece zero point offset



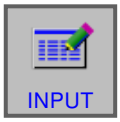
- Press softkey.



"MEASURE" performs subsequent calculations.

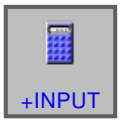
Current value (1) of the absolute coordinates + current setting (2) - target value for the working coordinates (3).

1 2 3

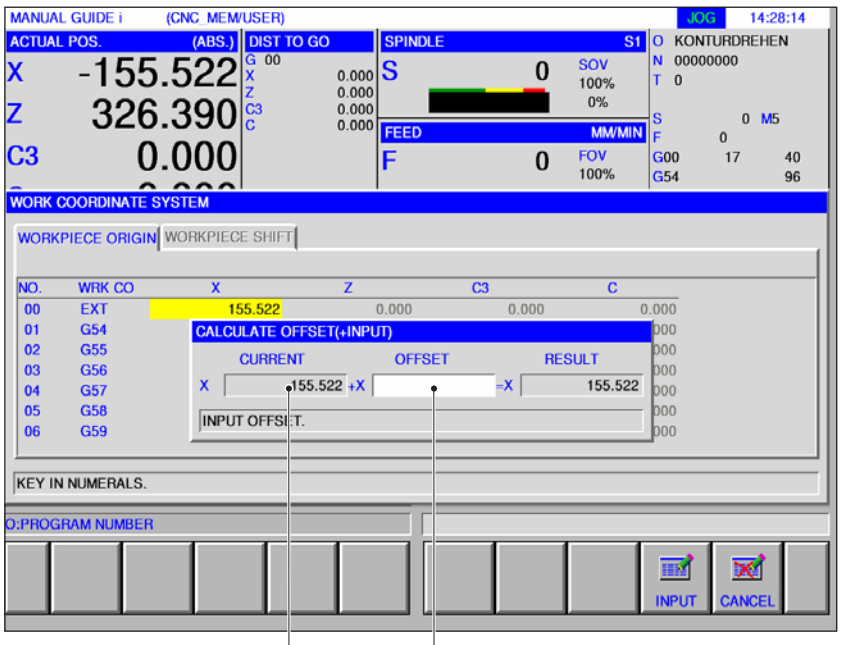


- Conclude measurement with "INPUT".

Compute: workpiece zero point offset



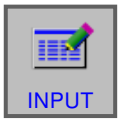
- Press softkey.



- "+INPUT" performs the following calculation.

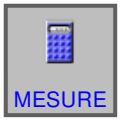
Current value (1) + offset (2).

1 2

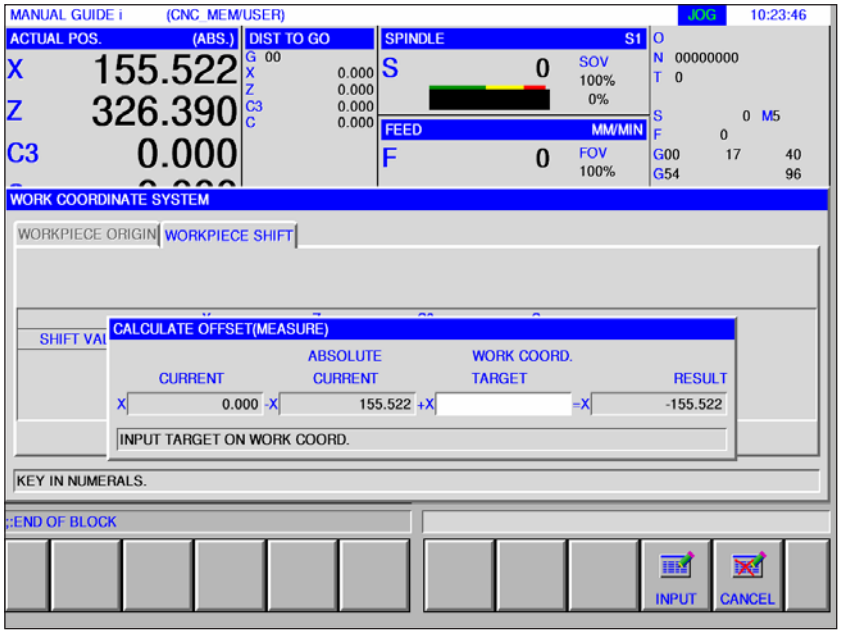


- End computation with "INPUT".

Measuring: workpiece coordinates system offset data

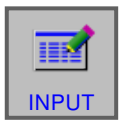


- Press softkey.



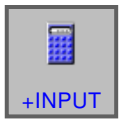
"MESURE" performs subsequent calculations.

Current value of the absolute coordinates + current setting - target value for the working coordinates.

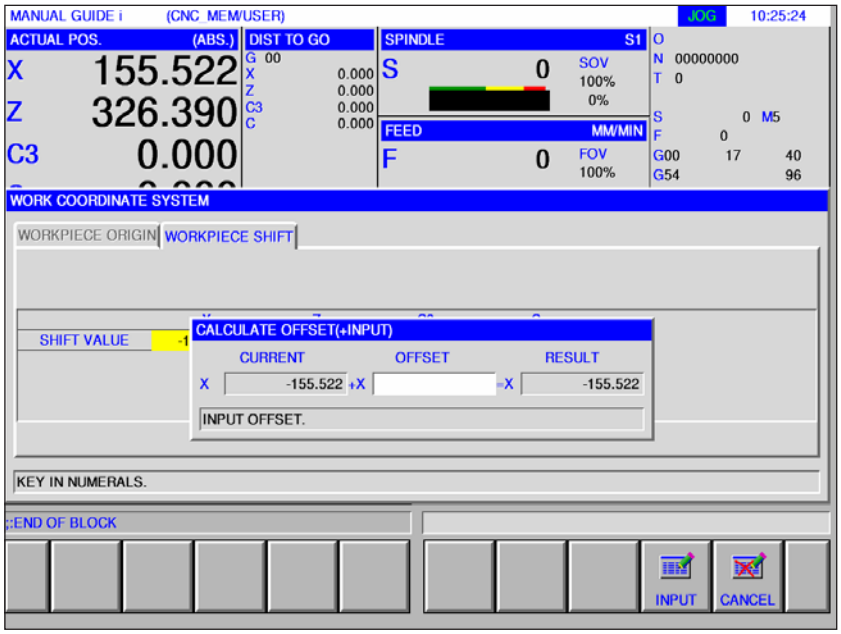


- Conclude measurement with "INPUT".

Calculating: workpiece coordinates system offset data

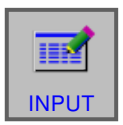


- Press softkey.



- "+INPUT" performs the following calculation.

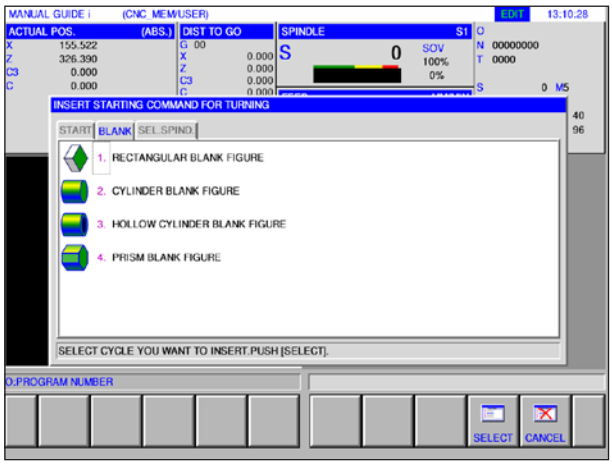
Current value + offset.



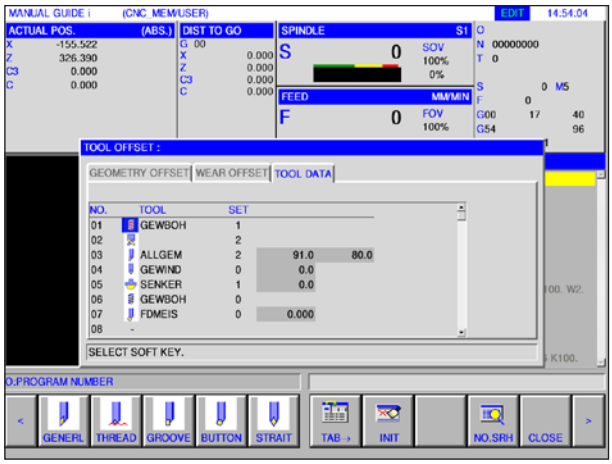
- End computation with "INPUT".

Graphical simulation

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



Raw part definition



Tool definition

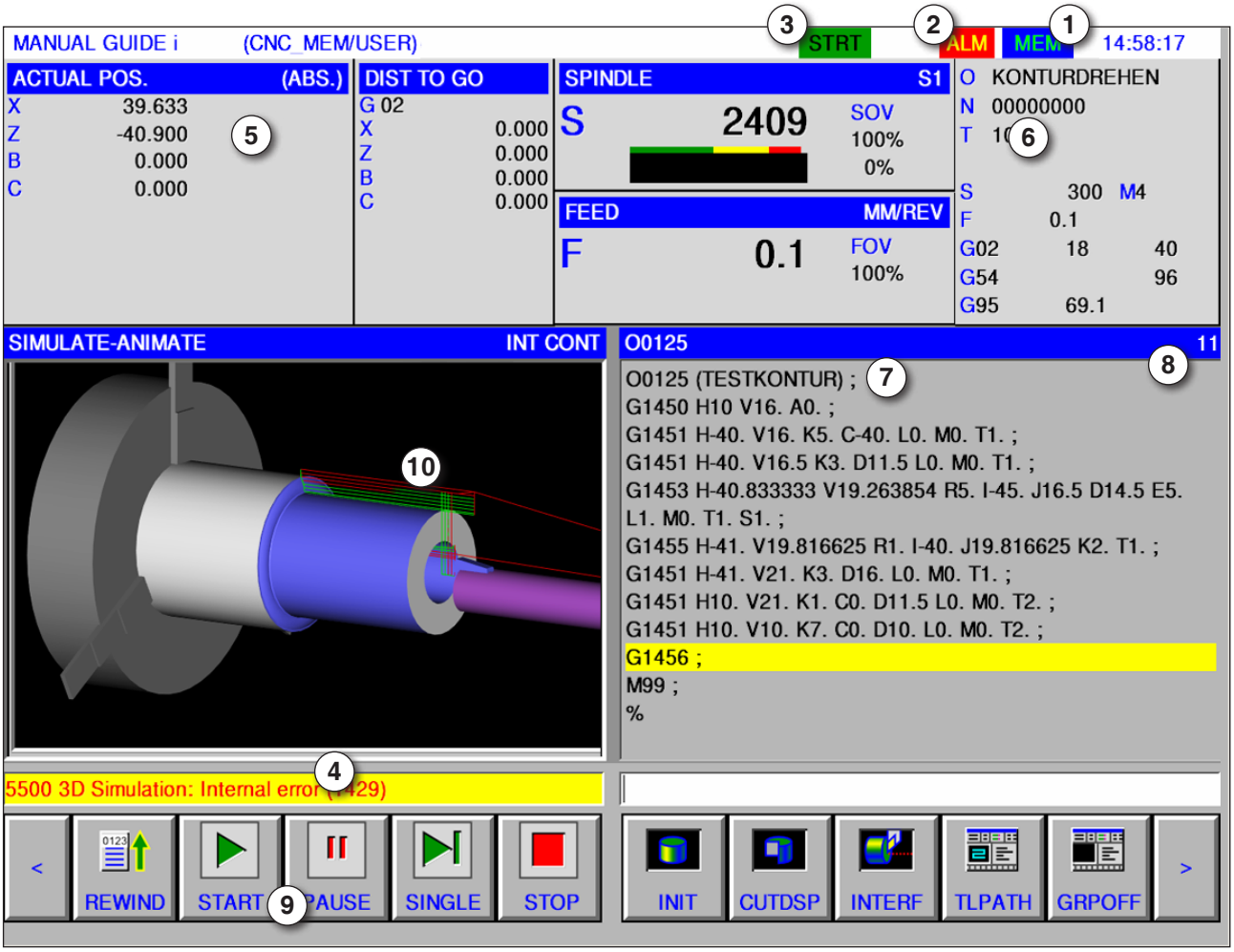
Unmachined part and tool definition

- 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 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-ANIMATE screen.

softkey functions



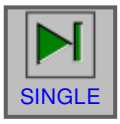
To start the simulation

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



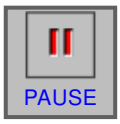
Start simulation

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



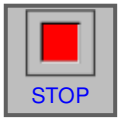
Single block

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



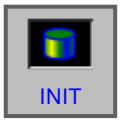
Stop simulation

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



Cancel simulation

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



Initialize unmachined part

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



Cut view of the unmachined part

The "CUTDSP" softkey switches between displaying the sectional view and the entire depiction of cylindrical unmachined parts and drilled cylindrical unmachined parts. Before beginning the simulation, switch accordingly if need be.



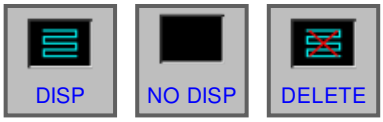
Workpiece collision

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.



Toolpath

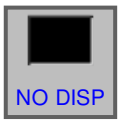
Press the "TLPATH" softkey to reveal the SIMULATE TOOL PATH toolbar with the following softkeys:



"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 "NO 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 "ANIMAT" softkey selects the mode for the machining simulation (animated graphics).

Graphics OFF



The "GRPOFF" softkey ends the machining simulation.

Scaling and moving



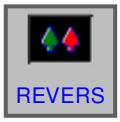
The "LARGE" softkey enlarges the drawing.



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



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



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



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



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



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

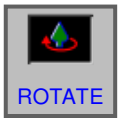


The "↓MOVE" softkey moves the viewing point down. Accordingly, the marked tool path is moved up.

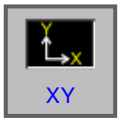


The "CENTER" softkey places the centre of the toolpath in the centre of the window.

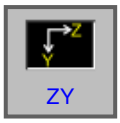
Selecting a graphics coordinates system



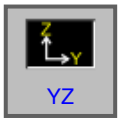
The "ROTATE" softkey selects the graphics co-ordinates system.



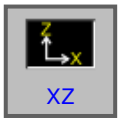
This softkey selects the XY plane.



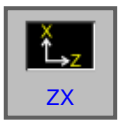
This softkey selects the ZY plane.



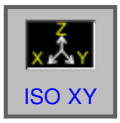
This softkey selects the ZY plane.



This softkey selects the ZY plane.



This softkey selects the ZY plane.



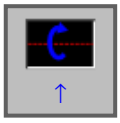
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis.



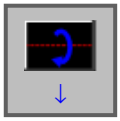
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis. The viewing point is opposite to that mentioned above.



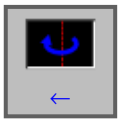
This softkey selects an equiangular coordinates system with the plus direction the upwards Z-axis.



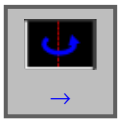
This softkey executes a counterclockwise rotation to the right with the screen as the central axis.



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



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



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



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



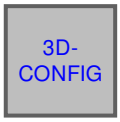
This softkey executes a counterclockwise rotation with the screen as the central axis.



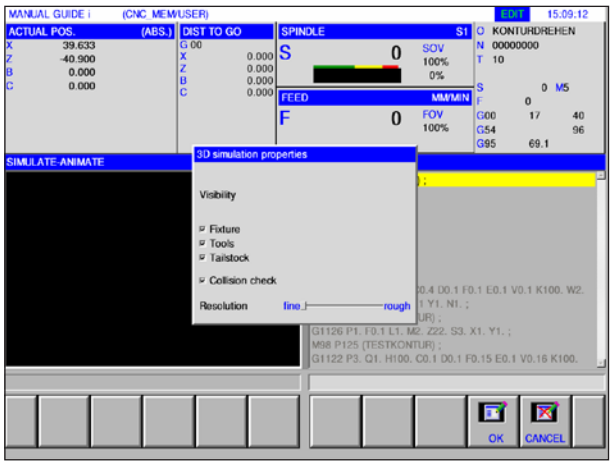


This softkey selects the simulation playback speed. There are 5 steps available for the speed.

3D configuration

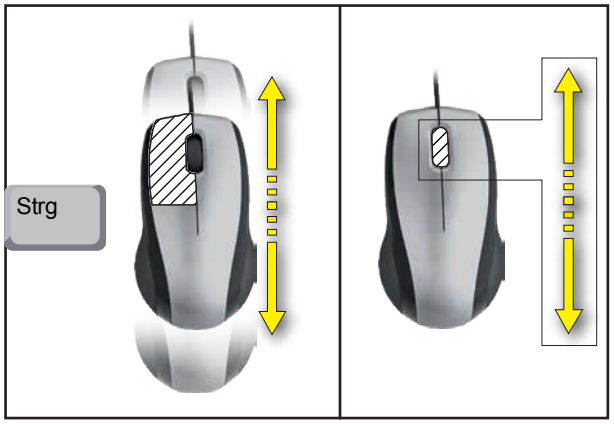


This softkey starts 3D configuration.

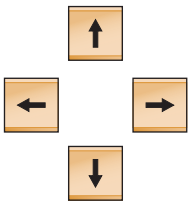


3D configuration

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

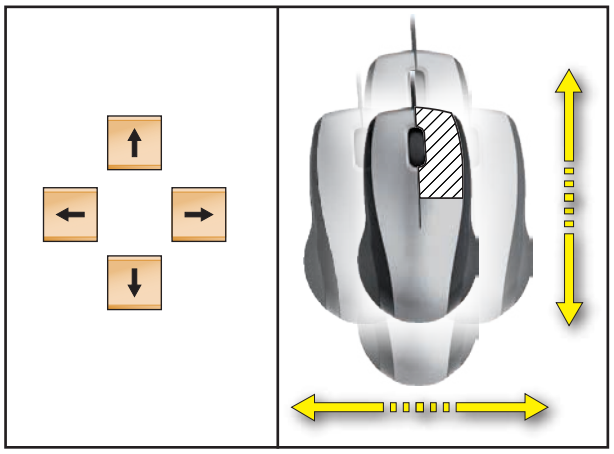


Zooming with the mouse



Shift graphics

Press cursor button to shift the graphics.



Shift

D: Programming with MANUAL GUIDE *i*



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

Example:
The Concept TURN 60 turning machine has no position-controlled main spindle, so neither is it possible to program spindle positions.

M- commands overview

M00	Programmed stop
M01	Optional stop
M02	End main program
M03	Main spindle ON, clockwise
M04	Main spindle ON, counterclockwise
M05	Main spindle OFF

Commands for the spindles of power tools in the different M-code systems: (Switched in EmConfig)	FANUC standard *)	EMCO Training	EMCO Industry
AWZ spindle ON clockwise	M103	M13	M303
AWZ spindle ON counterclockwise	M104	M14	M304
AWZ spindle OFF	M105	M15	M305

M07	Minimal lubrication ON
M08	Coolant ON
M09	Coolant OFF, minimal lubrication OFF

Commands for C-axis mode in the different M-code systems	FANUC standard *)	EMCO Training	EMCO Industry
C-axis mode activation	M14	M52	M52
C-axis mode deactivation	M15	M53	M53

M20	Quill BACK	M71	Purge ON
M21	Quill ADVANCE	M72	Purge OFF
M25	OPEN clamping device	M90	Manual chuck
M26	CLOSE clamping device	M91	Draw clamp
M29	Rigid tapping without length compensation	M92	Pressure clamp
M30	End main program	M98	Call sub-routine
M32	End main program with restart	M99	Jump back to calling program

*) FANUC standard is the default setting.

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 in this way.
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



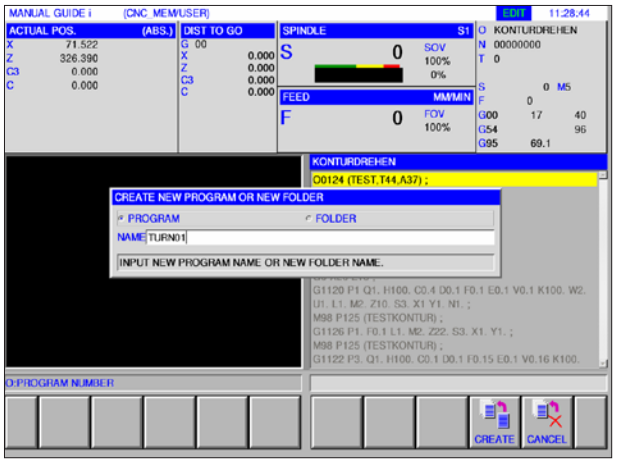
1 Select "Edit" mode.



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.



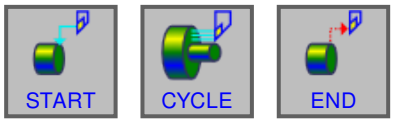
The Fanuc standard format for program names is as follows:
O1234. If fewer than 4 numbers are entered, those missing are populated with zeroes.

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



Turning cycles



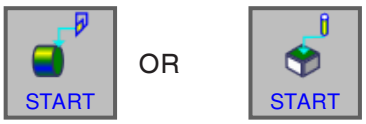
Milling cycles

A cycle is divided into 2 parts:

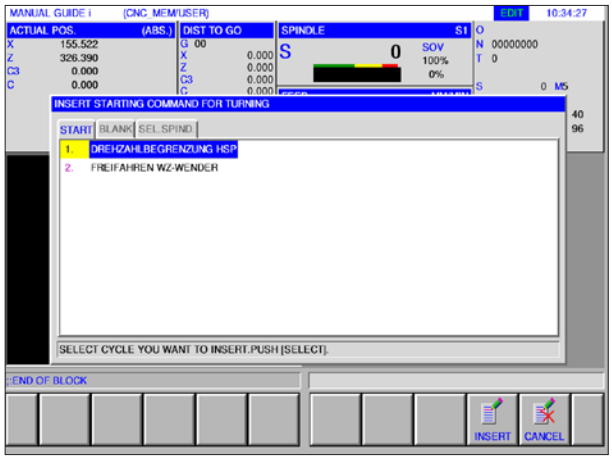
- Machining conditions
- Geometric data

Note:
Milling cycles are available only for machines with driven tools.

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".

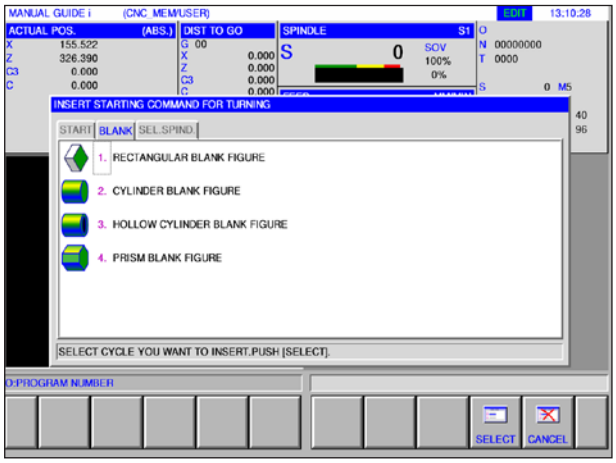
Available:

- Zero point offset
- Speed limitation
- Freewheel tool turret



Select unmachined part

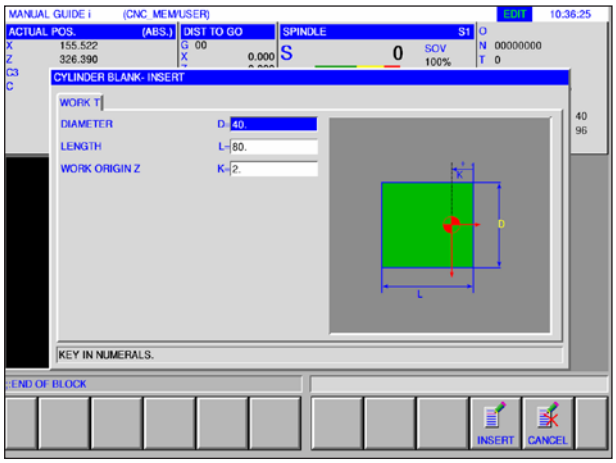
1 Press "START" softkey.



2 Select the register card for an unmachined part.



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





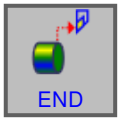
4 Enter the data for the unmachined part and close the entry with "INSERT".



Cycle programming

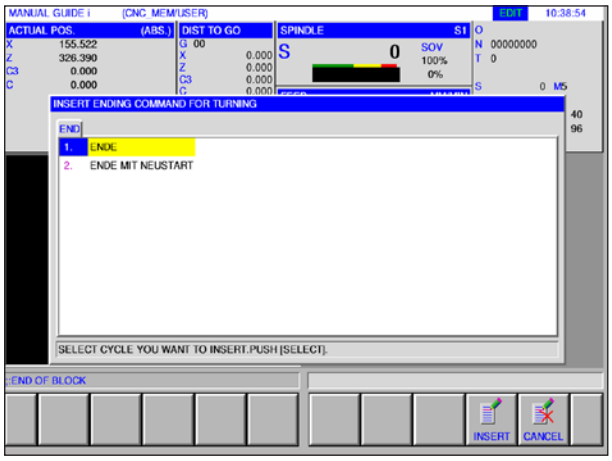
1 Press "CYCLE" softkey.

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



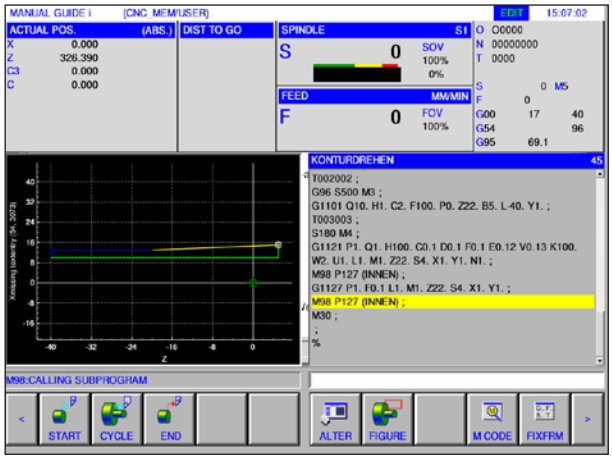
Program end END

1 Press "END" softkey.





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

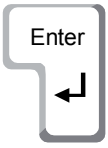


Contour editing

A contour that was created as a sub-program can be edited in the ISO window by pressing the "ALTER" softkey, or by pressing the "Enter" or "Input" keys.
By pressing these keys at the marked position in the program, the contour editor opens.



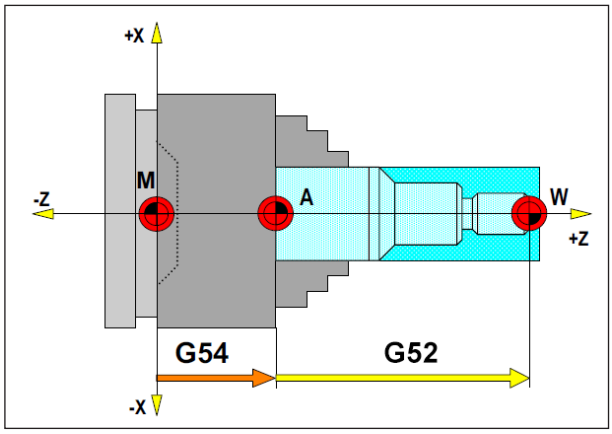
OR



OR



Mark the position in the program and press softkey or keys.



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

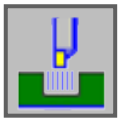
The cycle groups with the Fanuc Manual Guide *i* cycles defined within them are listed here.

Turning



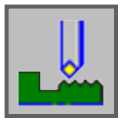
- Outer Rough G1120
- Inner Rough G1121
- Face Rough G1122
- Outer Semi-finish G1123
- Inner Semi-finish G1124
- Face Semi-finish G1125
- Outer Finish G1126
- Inner Finish G1127
- Face Finish G1128
- Outer Rough & Finish G1150
- Inner Rough & Finish G1151
- Face Rough & Finish G1152

Turning grooving



- Outside Roughing G1130
- Inside Roughing G1131
- Plane Roughing G1132
- Outside Coarse + Fine G1133
- Inside Coarse + Fine G1134
- Inside Coarse + Fine G1135
- Outside Roughing G1136
- Inside Fine G1137
- Plane Fine G1138

Threading



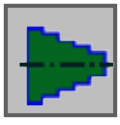
- Outer G1140
- Inner G1141

Hole machining



- Centre Drilling G1100
- Drilling G1101
- Tapping G1102
- Reaming G1103
- Boring G1104

Figure

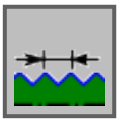


- Plane Turning Figure

Groove Contour



- Standard Outside Groove G1470
- Trapezoidal Outside Groove G1471
- Standard Inside Groove G1472
- Trapezoidal Inside Groove G1473
- Standard Groove Front Surface G1474
- Trapezoidal Groove Face G1475
- Contour Groove G1456



Thread Contour

- Thread General G1460
- Thread Metric G1461
- Thread Imperial G1462
- Thread Pipe G G1463
- Thread Pipe R G1464

Milling



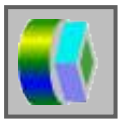
Drilling

- Centre Drilling G1110
- Drilling G1111
- Tapping G1112
- Grinding G1113
- Reaming G1114



Plane Machining

- Face Milling Roughing G1020
- Face Milling Finishing G1021



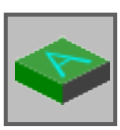
Contour Machining

- Outer Wall Roughing G1060
- Outer Wall Finishing G1061
- Outer Wall Side Finishing G1062
- Outer Wall Chamfer G1063
- Inner Wall Roughing G1064
- Inner Wall Finishing G1065
- Inner Wall Side Finishing G1066
- Inner Wall Chamfer G1067
- Partial Machining Roughing G1068
- Partial Machining Finishing G1069
- Partial Machining Side Finishing G1070
- Partial Machining Chamfer G1071



Pocket Milling

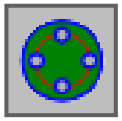
- Pocket Milling Roughing G1040
- Pocket Milling Finishing G1041
- Pocket Milling Side Finishing G1042
- Pocket Milling Chamfer G1043



Special

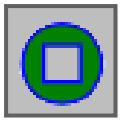
- Engraving G1025

Figure



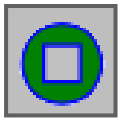
Hole Positions

- C-axis Hole on Face Arc Points G1572
- C-axis Hole on Face Circle G1573
- C-axis Hole on Face Coordinates G1574
- C-axis Hole on Cylindrical Surface Arc Points G1672
- C-axis Hole on Cylindrical Surface Free Choice G1673



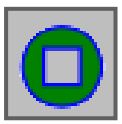
Front Surface Contour

- XC Square Front Surface G1520



Contouring Figure

- XC Square Convex G1520
- XC Circle Convex G1521
- XC Oval Convex G1522
- XC Polygon Convex G1525
- XC Free Contour Convex G1500
- ZC Free Contour Convex on Cylindrical Surface G1600
- XC Square Concave G1520
- XC Circle Concave G1521
- XC Oval Concave G1522
- XC Polygon Concave G1525
- XC Free Contour Concave G1500
- ZC Free Contour Concave on Cylindrical Surface G1600
- XC Free Contour Open G1500
- ZC Free Contour Open on Cylindrical Surface G1600



Pocket Contour

- XC Square Concave G1520
- XC Circle Concave G1521
- XC Oval Concave G1522
- XC Polygon Concave G1525
- XC Free Contour Concave G1500
- XC Free Contour Concave on Cylindrical Surface G1600

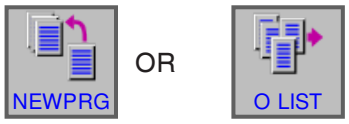
Working with cycles

Processes that repeat often, and which include several processing steps, are saved in the control system as cycles. Some special functions are also available as cycles.

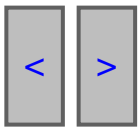
Define cycle



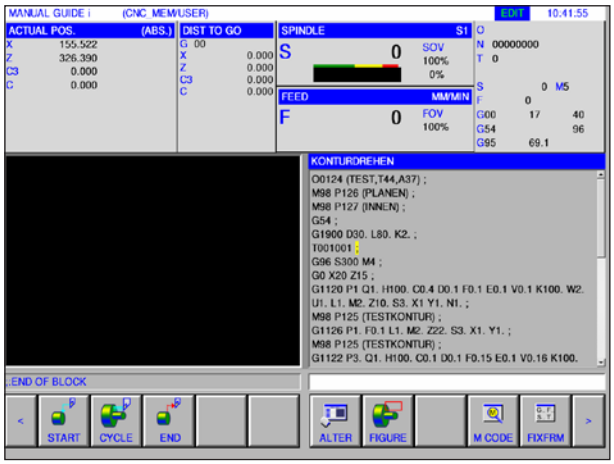
- Select "Edit" mode.

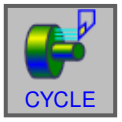


- Create a new program or open an existing program.



- Select the softkey toolbar for turning or milling cycles with the extension buttons.





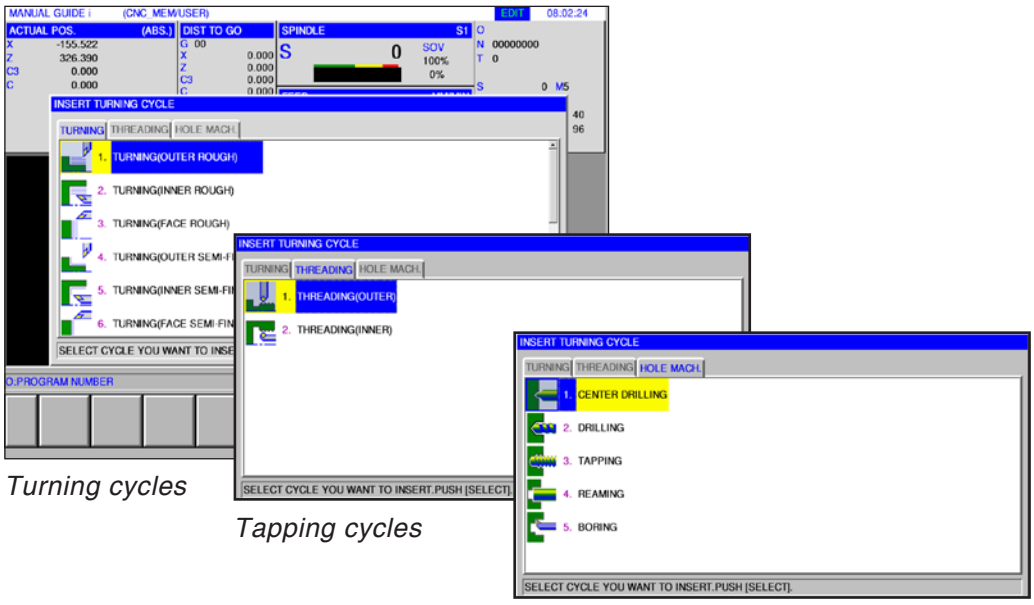
- Press softkey.

Instructions:
Depending on the machine you are operating with WinNC for Fanuc 31i, not all milling cycles may be available.

Example: a machine, with which no power tools can be operated, cannot execute any milling cycles.

The dropdown window shows the different cycle groups, on the example the turning cycles:

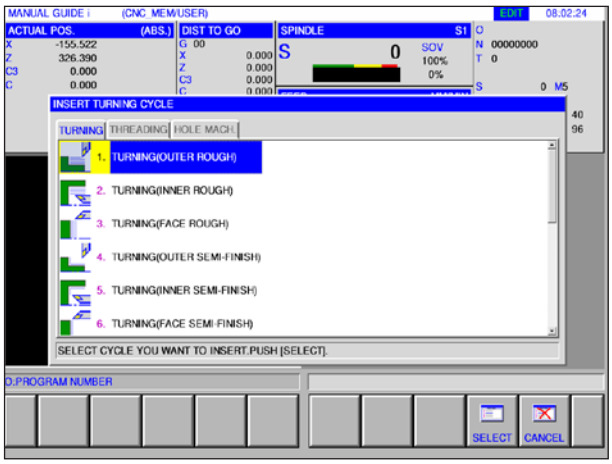
- Turning
- Threading
- Hole machining



Turning cycles

Tapping cycles

Drilling cycles



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

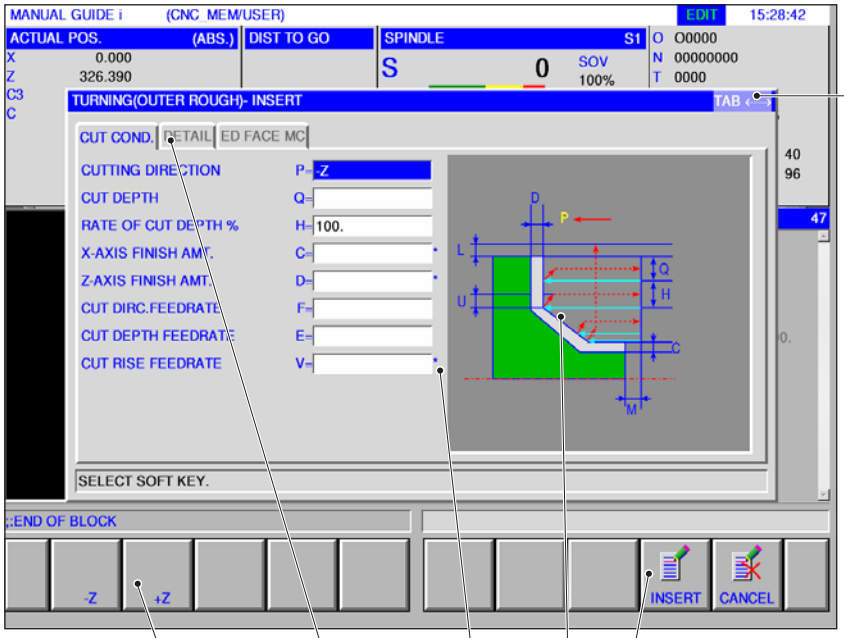


- "SELECT" softkey to confirm.



- "CANCEL" softkey to cancel.

Data Entry for Machining Cycles



For switching tabs with the cursor keys, "TAB ←→" is displayed in the upper right corner of the window.

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

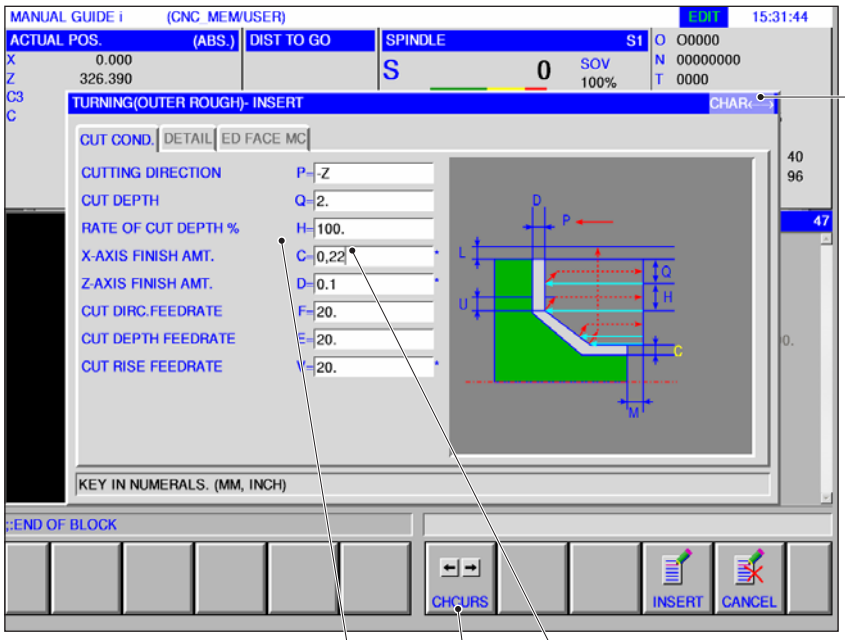
Coloured positioning movements:

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

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

Tab for entering machining conditions, tool distance and other details.
With the cursor keys <←> or <→> you can switch from one tab to another. The chosen active tab is depicted in blue. In the upper right corner of the window "TAB" is displayed.

Options menu for data entry via softkey.



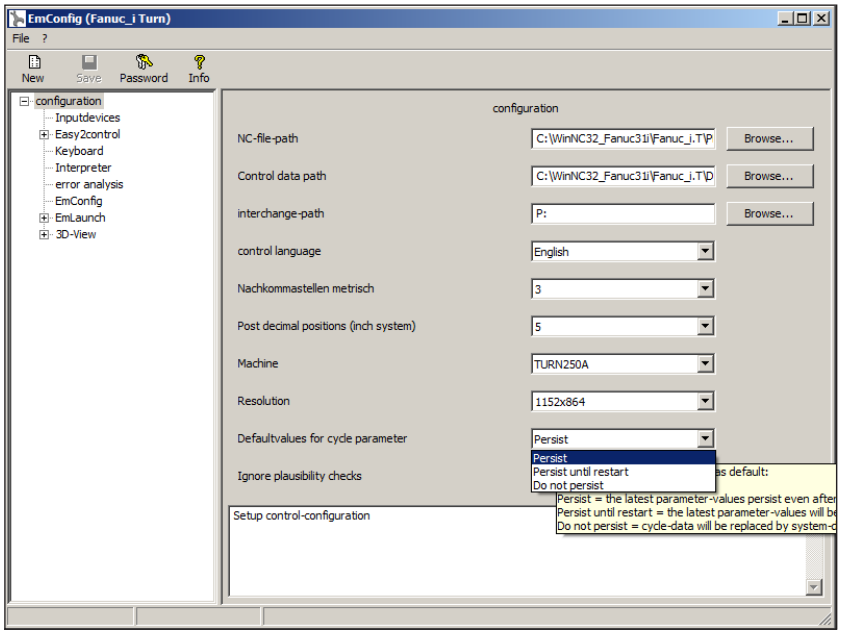
For cursor movement inside an entry field "CHAR ←→" is displayed.

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

- **Data entry**
  By actuating the cursor keys <↑> or <↓>, you can set the cursor on the desired data entry field.
- There are 2 options for data entry:
  1. Entering data as numerals.
  In the lower part of the window, the message "Enter data" is displayed.

  2. Entering data by using softkeys. For these fields an options menu is provided in the softkey bar.
  The message "Choose a softkey" is displayed.

Default values for cycle parameters



Default values for cycle parameters

EMConfig is an auxiliary software to WinNC.
WinNC settings can be changed with EMConfig.

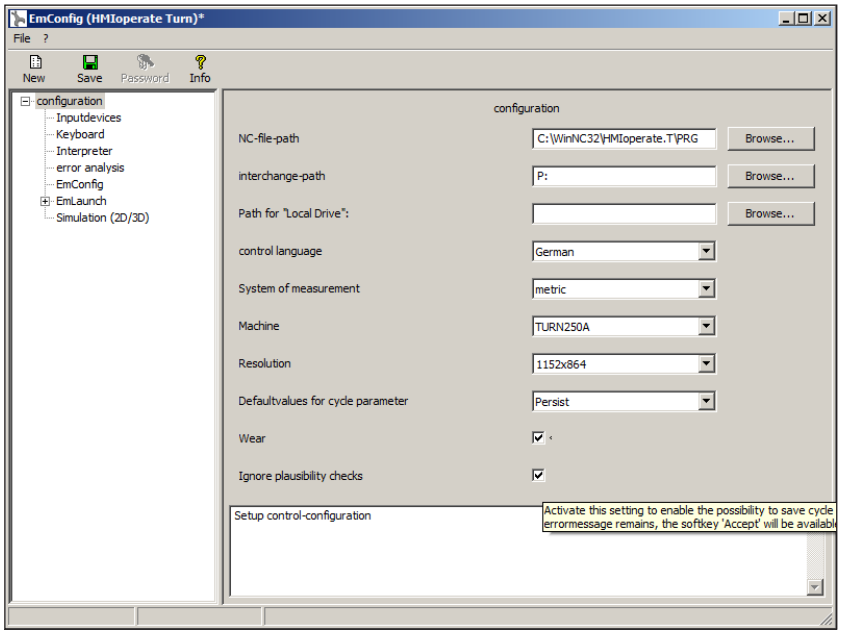
Open EMConfig and select the configuration point:

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

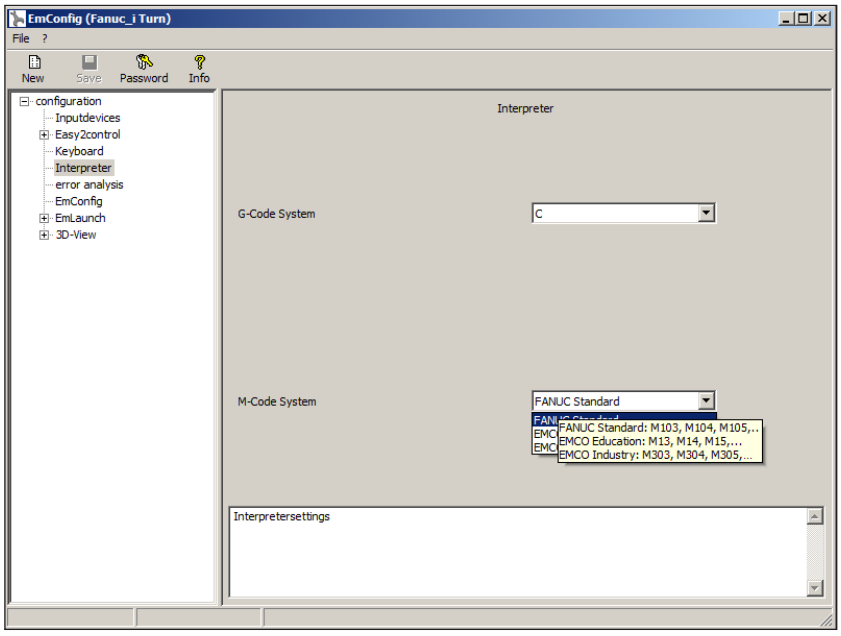


Set plausibility check for saving

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.

Set M-code system

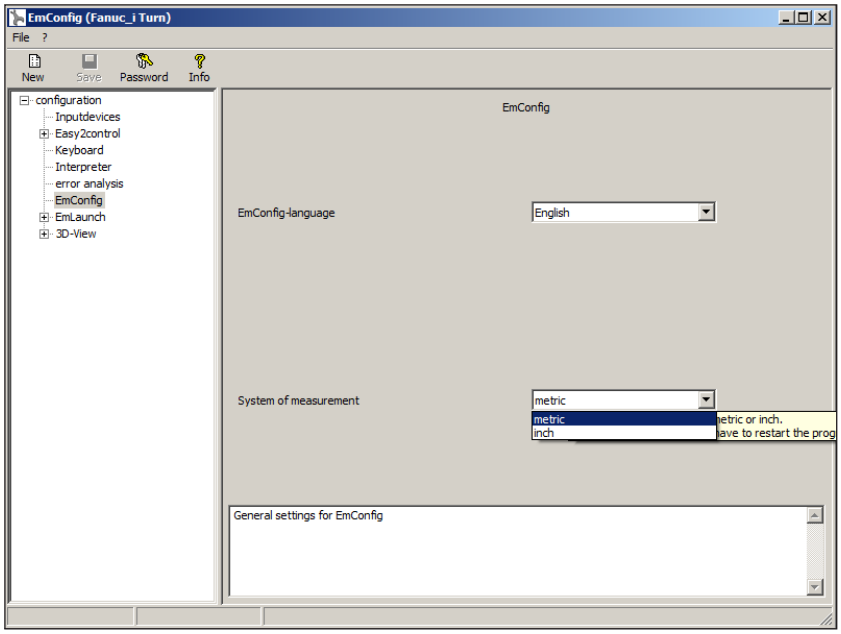


Select M-code system

3 different M-code systems are available for selection in the Interpreter:

- FANUC Standard - Default
- EMCO Education
- EMCO Industry

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

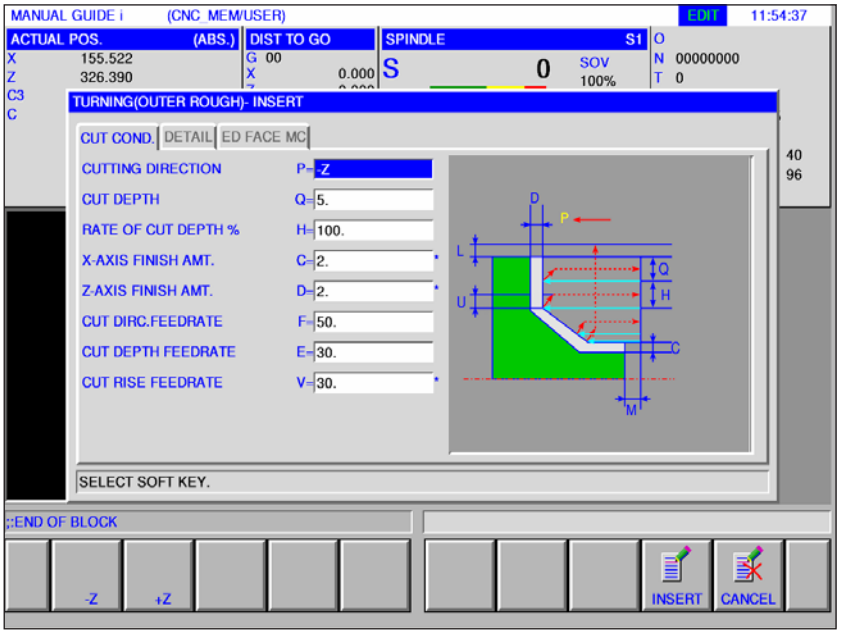


Turning

- Outer Rough G1120
- Inner Rough G1121
- Face Rough G1122
- Outer Semi-Finish G1123
- Inner Semi-Finish G1124
- Face Semi-Finish G1125
- Outer Finish G1126
- Inner Finish G1127
- Face Finish G1128
- Outer Rough & Finish G1150
- Inner Rough & Finish G1151
- Face Rough & Finish G1152



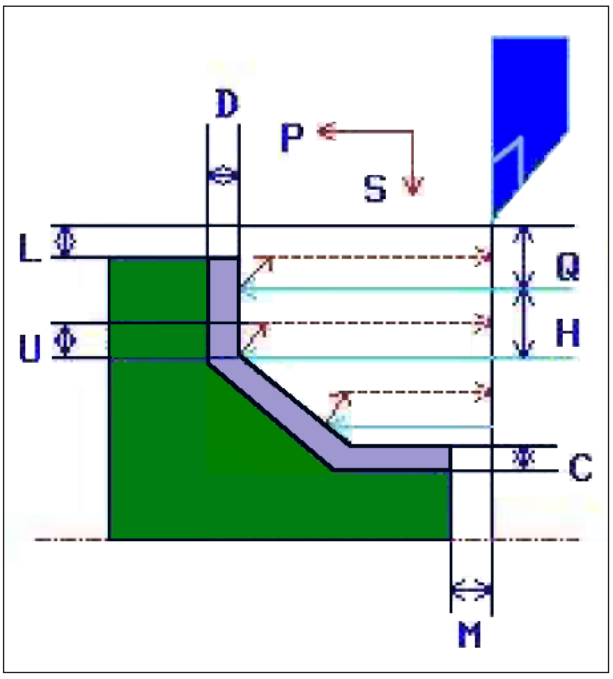
Outer Rough G1120



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
Q	Cut depth	Depth of each cut (radius value, positive value)
H	Rate of cut depth %	The delivery depth is stipulated as 100% fixed (standard value) and cannot be changed.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value)
E	Cut depth feedrate	Feed speed for cutting in Z-axis direction (positive value)
V*	Cut rise feedrate	Feed speed when cutting in the direction of retraction from the workpiece (positive value). Feed extract is fixed with the populated value of feed plunge.

Detail		
Data item		**Meaning**
K	1st feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cut rise method	• [CUT]: The tool returns after cutting along a contour about a "retraction distance".
U	Escape amount	Distance by which the tool returns from the machined surface after each cut (radius value, positive value).
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.
N	With / without end face allowance	Specifies whether a simultaneous plane machining is executed. • [UNAVIL]: Simultaneous plane machining is not executed. Frontal machining is stipulated as UNAVAILABLE and cannot be changed.



Tool track

Cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 After cutting in the X-axis direction with feed speed (F), which is stipulated for this cut direction, the tool cuts with feed speed (E), which is stipulated for the cutting in direction, in the given contour position, for which a finishing dimension is still outstanding in the Z-axis direction.

3 If [VORSCH] is preselected, the tool first cuts along the contour and then retracts by the retraction amount (U) in the X-and Z- axial dimensions.

4 The tool is moved to the cutting in start point in the Z axis direction.

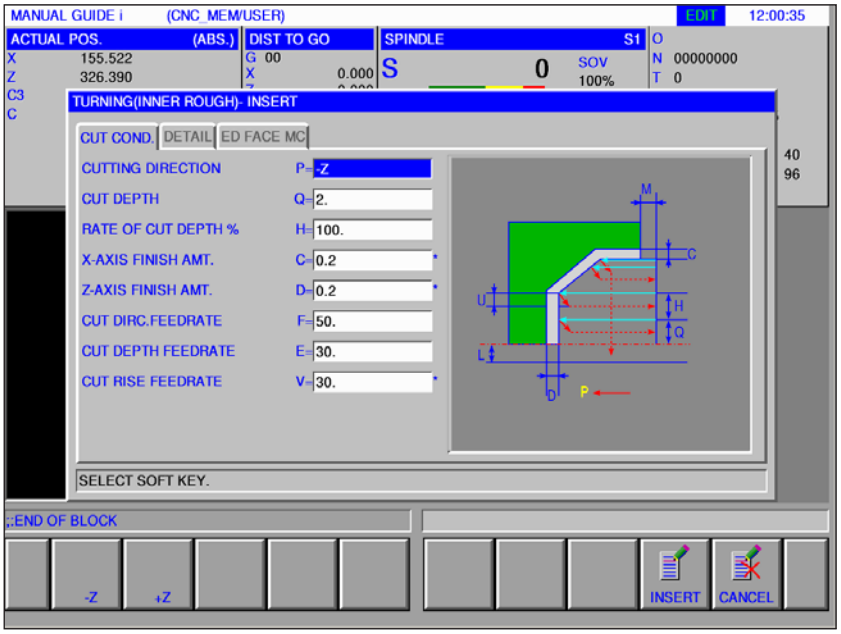
5 Steps <2>to <4> are repeated until the lowest part has been reached in the X-axis direction.

6 If a further pocket is still available, the tool is positioned on the pocket and steps <2> to <4> are repeated.

7 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" in rapid traverse.



Inner Rough G1121

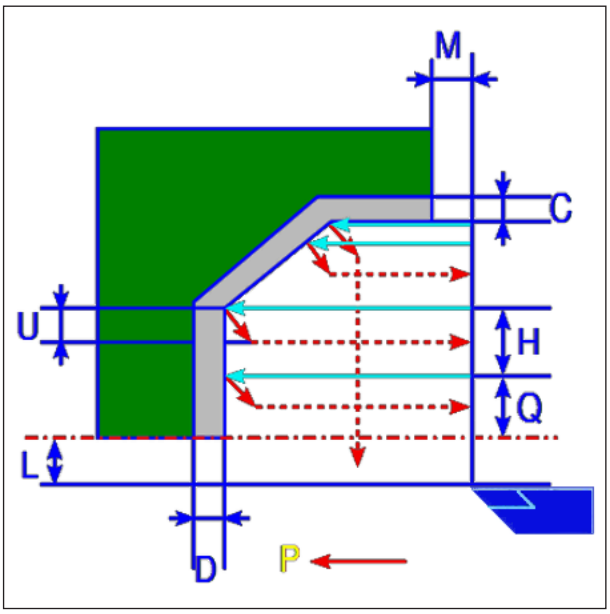


Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
Q	Cut depth	Depth of each cut (radius value, positive value)
H	Rate of cut depth %	The delivery depth is stipulated as 100% fixed (standard value) and cannot be changed.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value)
E	Cut depth feedrate	Feed speed for cutting in Z-axis direction (positive value)
V*	Cut rise feedrate	Feed speed when cutting in the direction of retraction from the workpiece (positive value). Feed extract is fixed with the populated value of feed plunge.

Detail		
Data item		**Meaning**
K	1st feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cut rise method	• [CUT]: The tool returns after cutting along a contour about a "retraction distance".
U	Escape amount	Distance by which the tool returns from the machined surface after each cut (radius value, positive value).
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.
N	With / without end face allowance	Specifies whether a simultaneous plane machining is executed. • [UNAVIL]: Simultaneous plane machining is not executed. Frontal machining is stipulated as UNAVAILABLE and cannot be changed.

Instructions:
When machining an inside surface, the input element for plane machining dimensions is not displayed. Therefore, specify the unmachined part contour for the plane machining dimension part during the unmachined part contour production.



Tool track

Cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 After cutting in the X-axis direction with feed speed (F), the tool cuts with feed speed (E) in the given contour position, for which a finishing dimension is still outstanding in the Z-axis direction.

3 If [VORSCH] is preselected, the tool first cuts along the contour and then retracts by the retraction amount (U) in the X-and Z- axial dimensions.

4 The tool is moved to the cutting in start point in the Z axis direction.

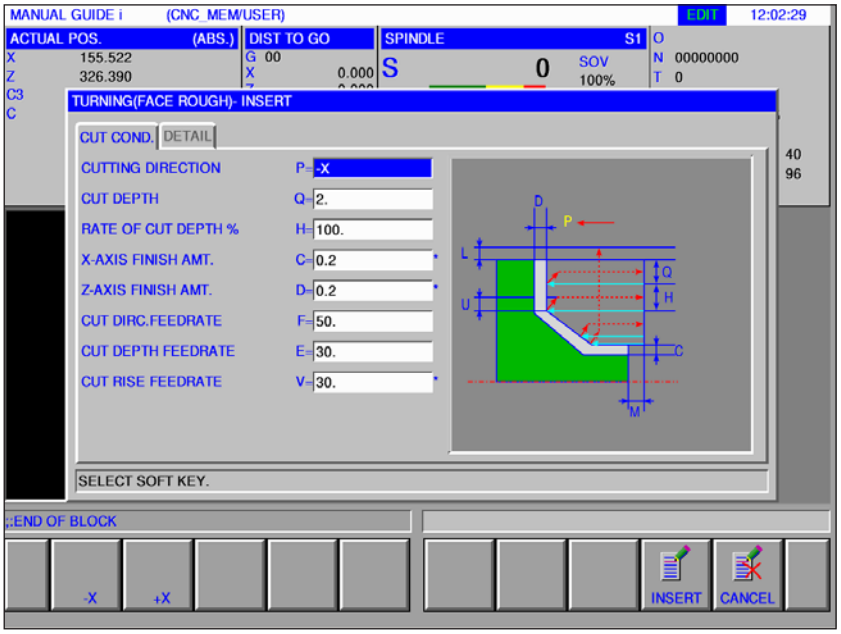
5 Steps <2>to <4> are repeated until the lowest part has been reached in the X-axis direction.

6 If a further pocket is still available, the tool is positioned on the pocket and steps <2> to <4> are repeated.

7 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" in rapid traverse.

CYCLE | TURNING | 3. TURNING (FACE ROUGH)

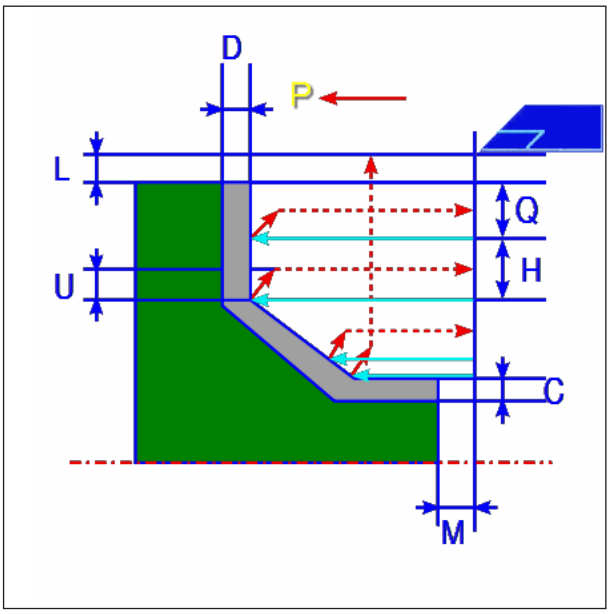
Face Rough G1122



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
Q	Cut depth	Depth of each cut (radius value, positive value)
H	Rate of cut depth %	The delivery depth is stipulated as 100% fixed (standard value) and cannot be changed.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value)
E	Cut depth feedrate	Feed speed for cutting in Z-axis direction (positive value)
V*	Cut rise feedrate	Feed speed when cutting in the direction of retraction from the workpiece (positive value). Feed extract is fixed with the populated value of feed plunge.

Detail		
Data item		**Meaning**
K	1st feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cut rise method	• [CUT]: The tool returns after cutting along a contour about a "retraction distance".
U	Escape amount	Distance by which the tool returns from the machined surface after each cut (radius value, positive value).
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 After cutting in the Z-axis direction with feed speed (F), the tool cuts with feed speed (E) in the entered contour position, for which a finishing dimension is still outstanding in the X-axis direction.

3 If [VORSCH] is preselected, the tool first cuts along the contour and then retracts by the retraction amount (U) in the X- and Z- axial dimensions.

4 The tool is moved to the cutting in start point in the Z axis direction.

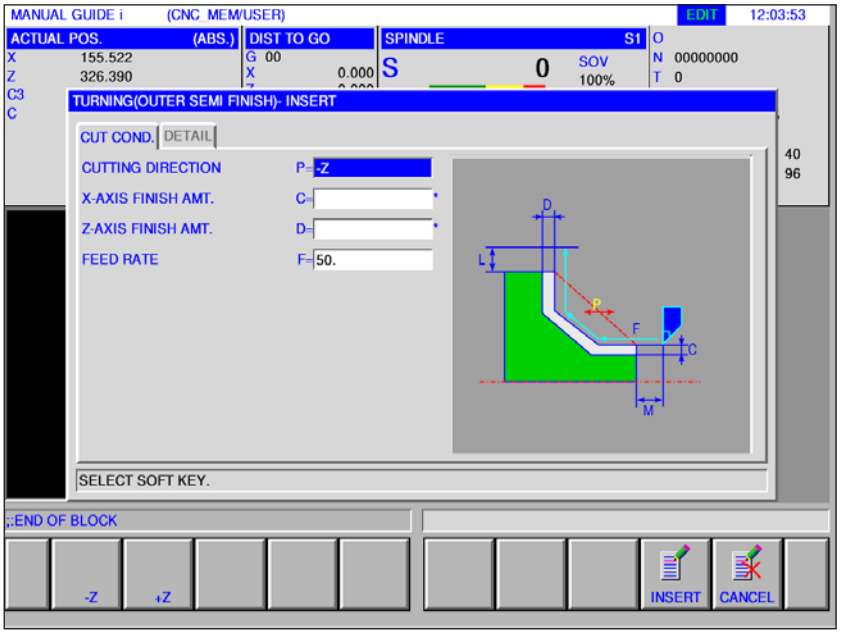
5 Steps <2>to <4> are repeated until the lowest part has been reached in the Z-axis direction.

6 If a further pocket is still available, the tool is positioned on the pocket and steps <2> to <4> are repeated.

7 After all areas of the part are cut, the tool retracts in the Z-axis direction into the cutting in start point + separation (M)" in rapid traverse.

CYCLE | TURNING | 4. TURNING (OUTER SEMI-FINISH)

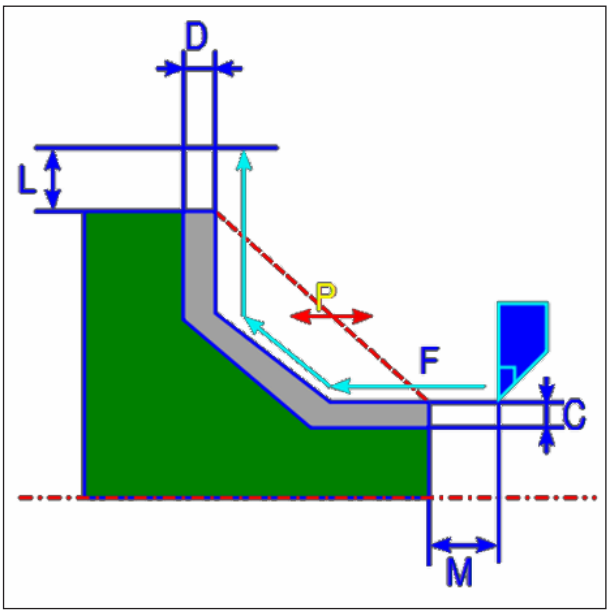
Outer Semi-Finish G1123



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Cut feed speed for intermediate machining (positive value)

Detail		
Data item		**Meaning**
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

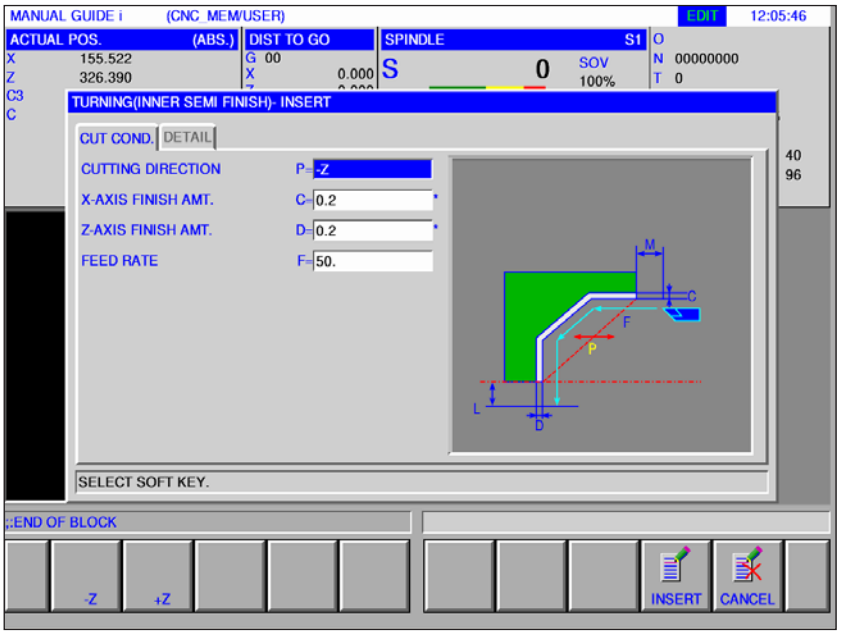
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for intermediate machining along the entered contour. The section which corresponds to the finished dimension remains unmachined until the final contour is reached. If "FEED SPEED INTERMEDIATE MACHINING" is set as contour element, the intermediate machining feed speed for the element is used for machining.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" in rapid traverse.



Inner Semi-Finish G1124

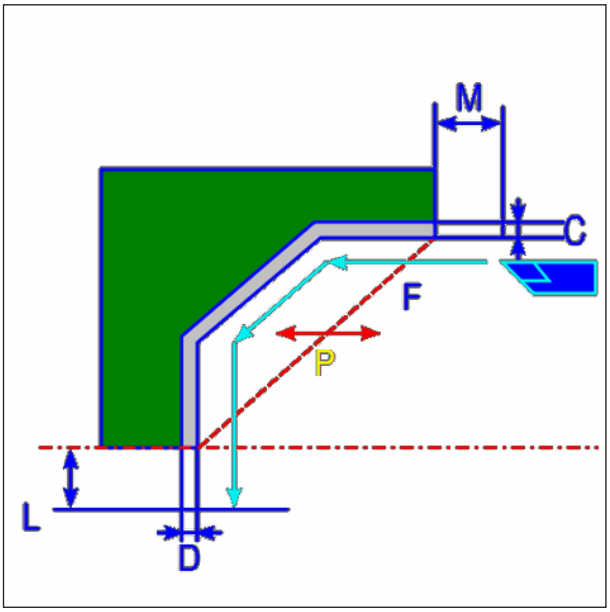


Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Cut feed speed for intermediate machining (positive value)

Detail		
Data item		**Meaning**
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).

Detail		
Data item		**Meaning**
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

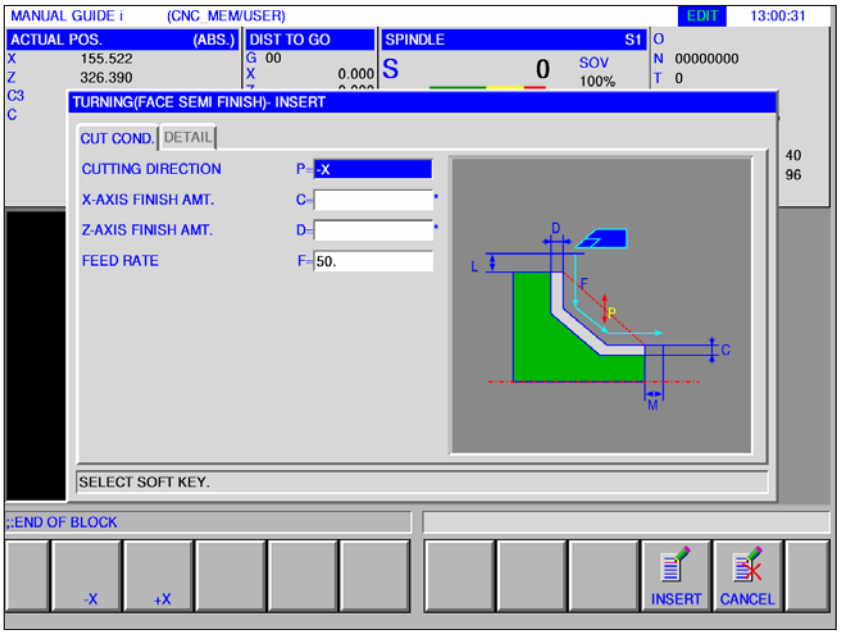
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for intermediate machining along the entered contour. The section which corresponds to the finished dimension remains unmachined until the final contour is reached. If "FEED SPEED INTERMEDIATE MACHINING" is set as contour element, the intermediate machining feed speed is used for machining.

3 After all areas of the part are cut, the tool retracts in the Z-axis direction into the cutting in start point + separation (M)" in rapid traverse.

CYCLE | TURNING | 6. TURNING (FACE SEMI-FINISH)

Face Semi-Finish G1125

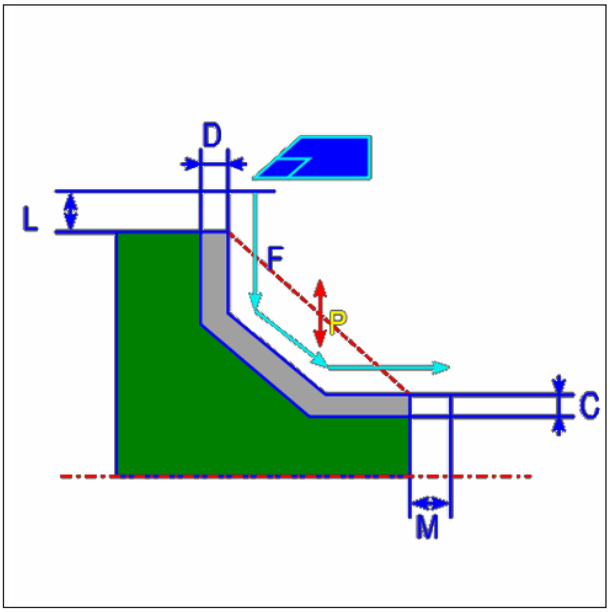


Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Cut feed speed for intermediate machining (positive value)

Detail		
Data item		**Meaning**
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).

Detail		
Data item		**Meaning**
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

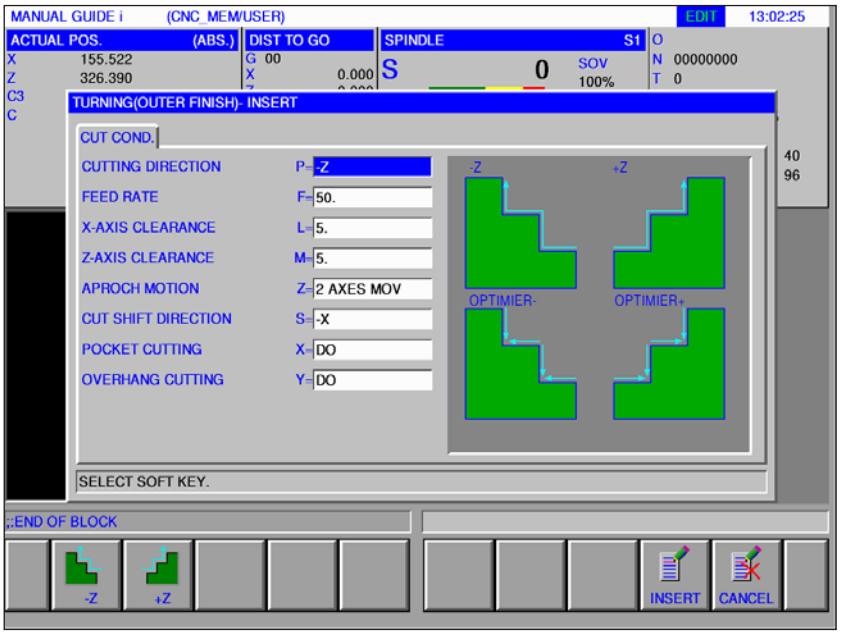
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for intermediate machining along the entered contour. The section which corresponds to the finished dimension remains unmachined until the final contour is reached. If "FEED SPEED INTERMEDIATE MACHINING" is set as contour element, the intermediate machining feed speed is used for machining.

3 After all areas of the part are cut, the tool retracts in the Z-axis direction into the cutting in start point + separation (M)" in rapid traverse.

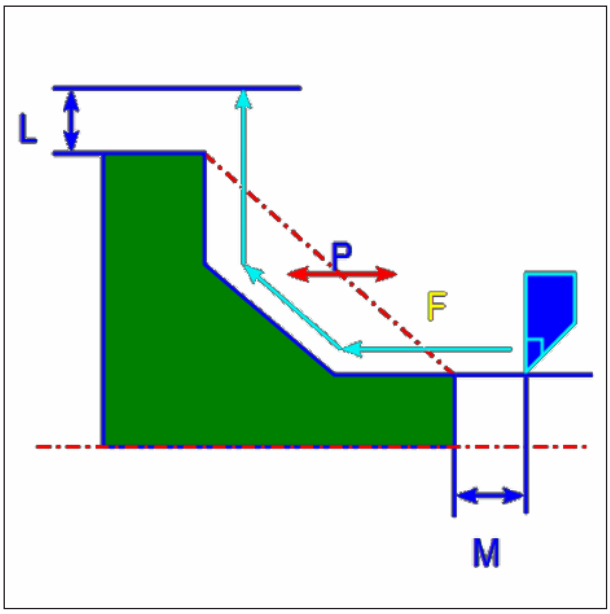
CYCLE | TURNING | 7. TURNING (OUTER FINISH)

Outer Finish G1126



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value)
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

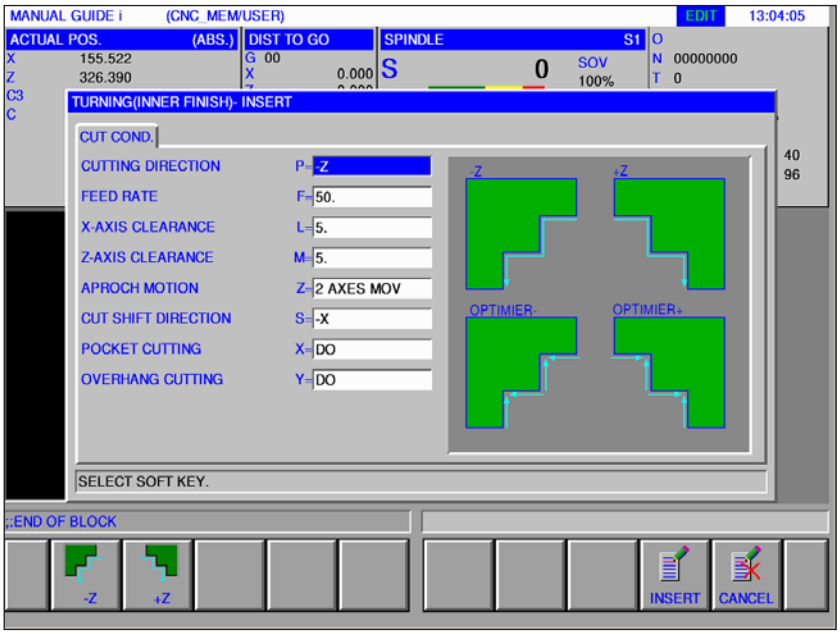
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for finishing along the entered contour, until the finished contour is achieved.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" position in rapid traverse.

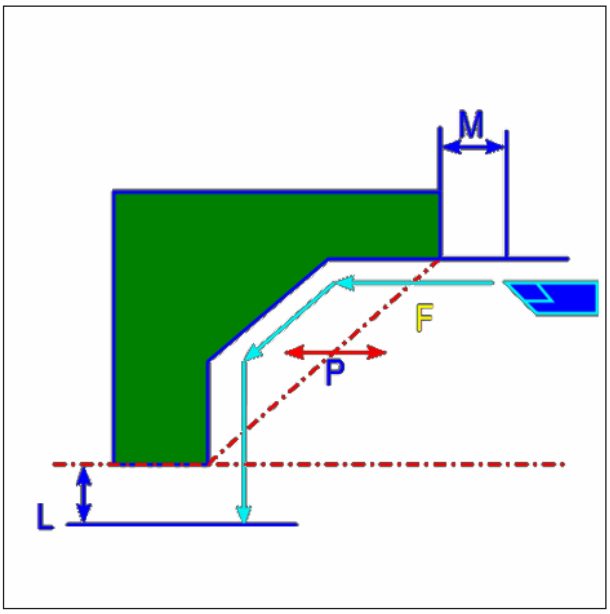


Inner Finish G1127



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
F	Cut direction feedrate	Cut feed speed for finishing (positive value)
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

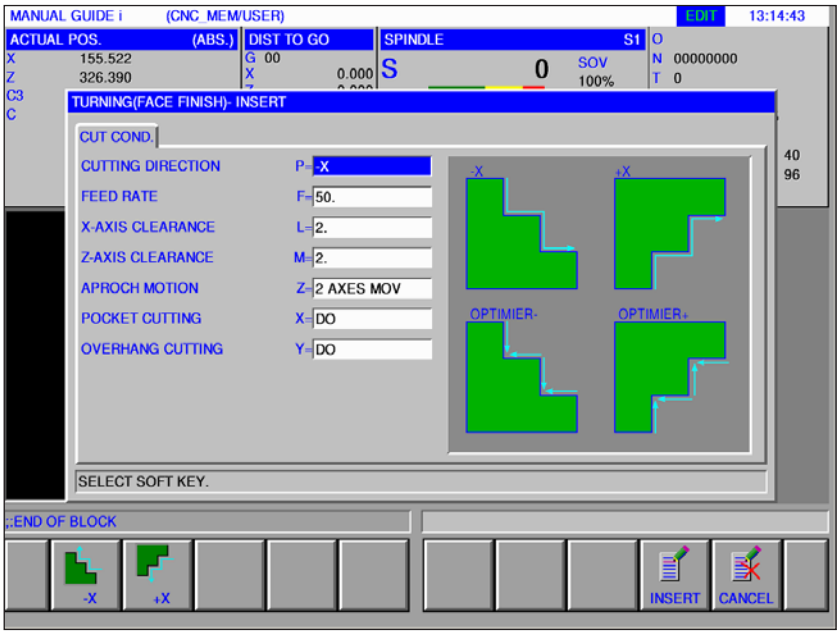
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for finishing along the entered contour, until the finished contour is achieved.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" position in rapid traverse.

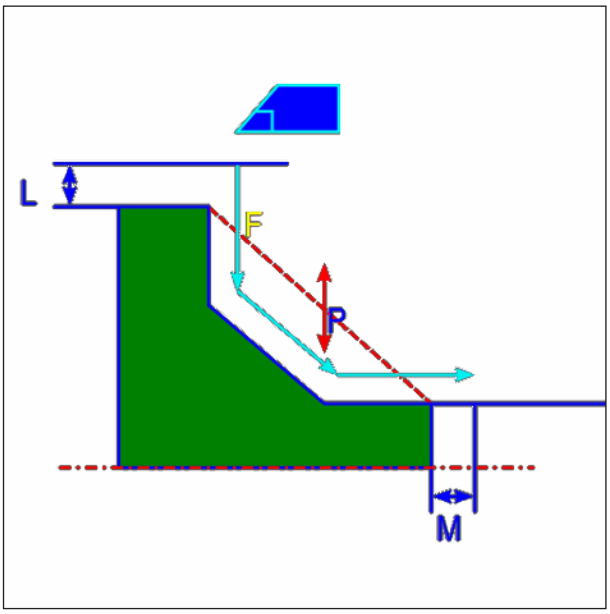


Face Finish G1128



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
F	Cut direction feedrate	Cut feed speed for finishing. (positive value)
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Cycle description

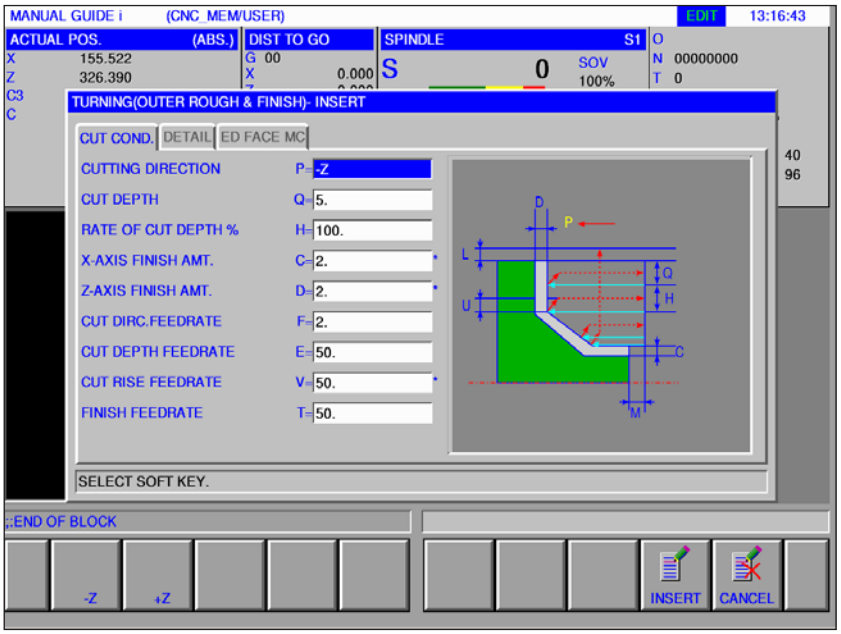
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for finishing along the entered contour, until the finished contour is achieved.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" position in rapid traverse.

CYCLE | TURNING | 10. TURNING (OUTER ROUGH & FINISH)

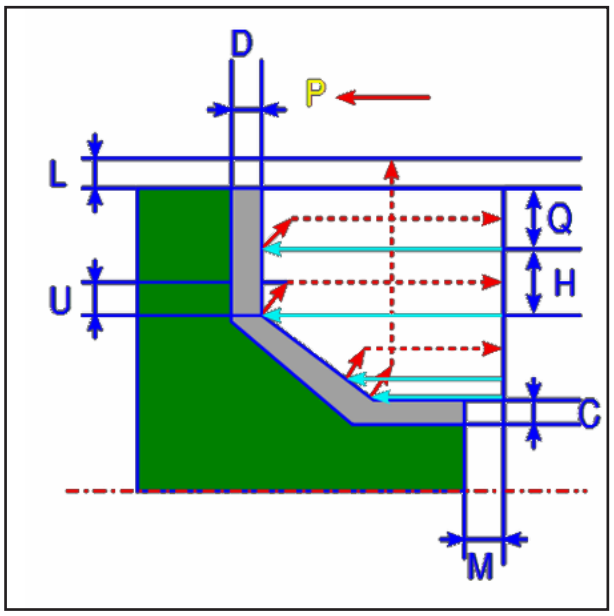
Outer Rough + Finish G1150



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
Q	Cut depth	Depth of each cut (radius value, positive value)
H	Rate of cut depth %	The delivery depth is stipulated as 100% fixed (standard value) and cannot be changed.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value).
E	Cut depth feedrate	Feed speed for cutting in Z-axis direction (positive value)
V*	Cut rise feedrate	Feed speed when cutting in the direction of retraction from the workpiece (positive value). Feed extract is fixed with the populated value of feed plunge.
T	Finish feedrate	Feed speed for finishing in workpiece radius direction (positive value).

Detail		
Data item		**Meaning**
K	1st feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cut rise method	• [VORSCH]: The tool returns after cutting along a contour about a "retraction distance".
U	Escape amount	Distance by which the tool returns from the machined surface after each cut (radius value, positive value).
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.
N	With / without end face allowance	Specifies whether a simultaneous plane machining is executed. • [UNAVIL]: Simultaneous plane machining is not executed. Frontal machining is stipulated as UNAVAILABLE and cannot be changed.



Tool track

Roughing cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 After cutting in the X-axis direction with feed speed (F), which is stipulated for this cut direction, the tool cuts with feed speed (E), which is stipulated for the cutting in direction, in the given contour position, for which a finishing dimension is still outstanding in the Z-axis direction.

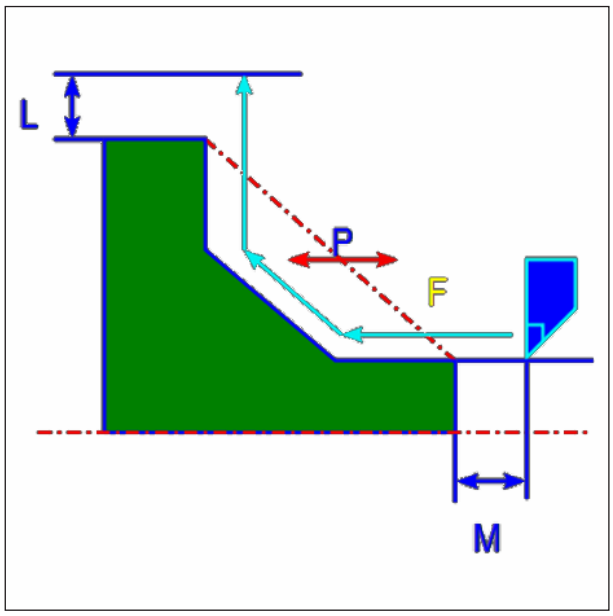
3 If [VORSCH] is preselected, the tool first cuts along the contour and then retracts by the retraction amount (U) in the X-and Z- axial dimensions.

4 The tool is moved to the cutting in start point in the Z axis direction.

5 Steps <2>to <4> are repeated until the lowest part has been reached in the X-axis direction.

6 If a further pocket is still available, the tool is positioned on the pocket and steps <2> to <4> are repeated.

7 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" in rapid traverse.



Tool track

Finishing cycle description

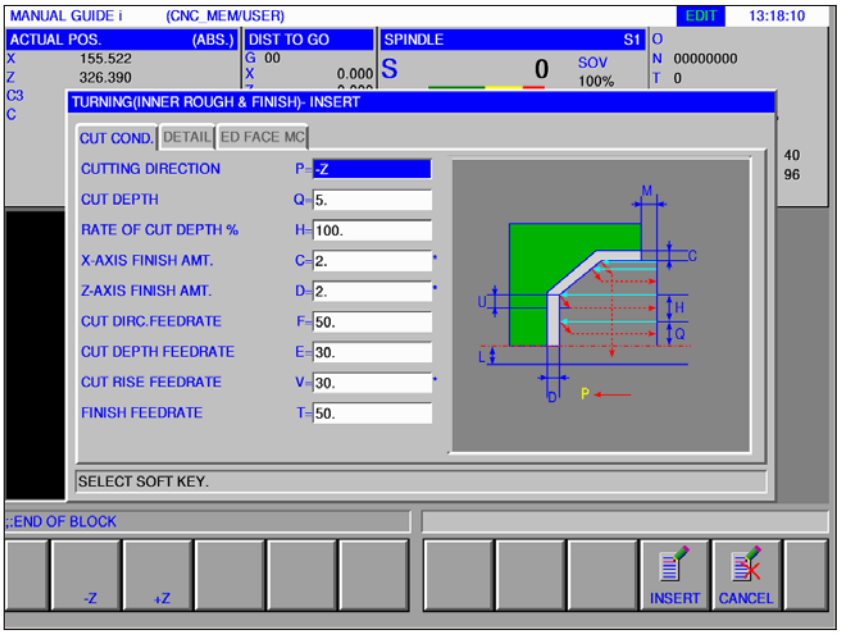
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for finishing along the entered contour, until the finished contour is achieved.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" position in rapid traverse.



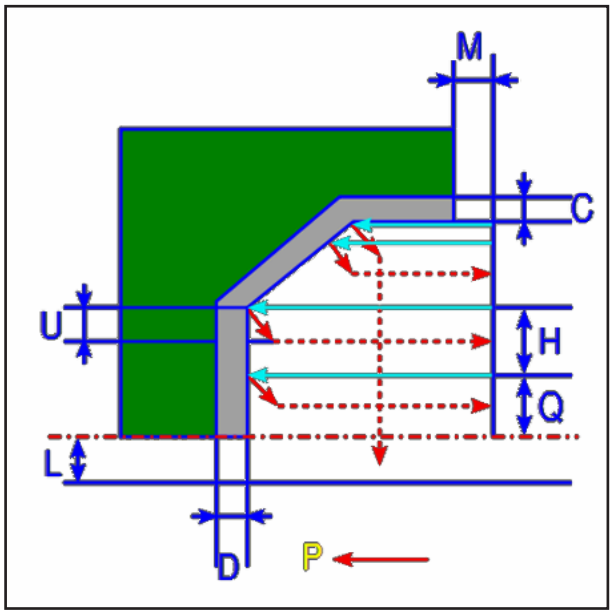
Inner Rough + Finish G1151



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
Q	Cut depth	Depth of each cut (radius value, positive value)
H	Rate of cut depth %	The delivery depth is stipulated as 100% fixed (standard value) and cannot be changed.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value)
E	Cut depth feedrate	Feed speed for cutting in Z-axis direction (positive value)
V*	Cut rise feedrate	Feed speed when cutting in the direction of retraction from the workpiece (positive value). Feed extract is fixed with the populated value of feed plunge.
T	Finish feedrate	Feed speed for finishing in workpiece radius direction (positive value)

Detail		
Data item		**Meaning**
K	1st feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cut rise method	• [CUT]: The tool returns after cutting along a contour about a "retraction distance".
U	Escape amount	Distance by which the tool returns from the machined surface after each cut (radius value, positive value).
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.
N	With / without end face allowance	Specifies whether a simultaneous plane machining is executed. • [UNAVIL]: Simultaneous plane machining is not executed. Frontal machining is stipulated as UNAVAILABLE and cannot be changed.



Tool track

Roughing cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 After cutting in the X-axis direction with feed speed (F), the tool cuts with feed speed (E) in the given contour position, for which a finishing dimension is still outstanding in the Z-axis direction.

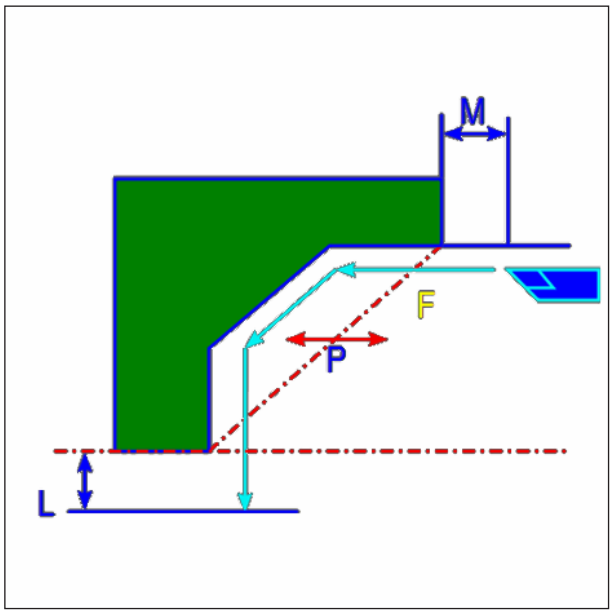
3 If [RAPID TRAVERSE] is preselected, the tool retracts directly by the retraction amount (U) in the X- and Z-axis directions. If [VORSCH] is preselected, the tool first cuts along the contour and then retracts by the retraction amount (U) in the X- and Z- axial dimensions.

4 The tool is moved to the cutting in start point in the Z axis direction.

5 Steps <2>to <4> are repeated until the lowest part has been reached in the X-axis direction.

6 If a further pocket is still available, the tool is positioned on the pocket and steps <2> to <4> are repeated.

7 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" in rapid traverse.



Tool track

Finishing cycle description

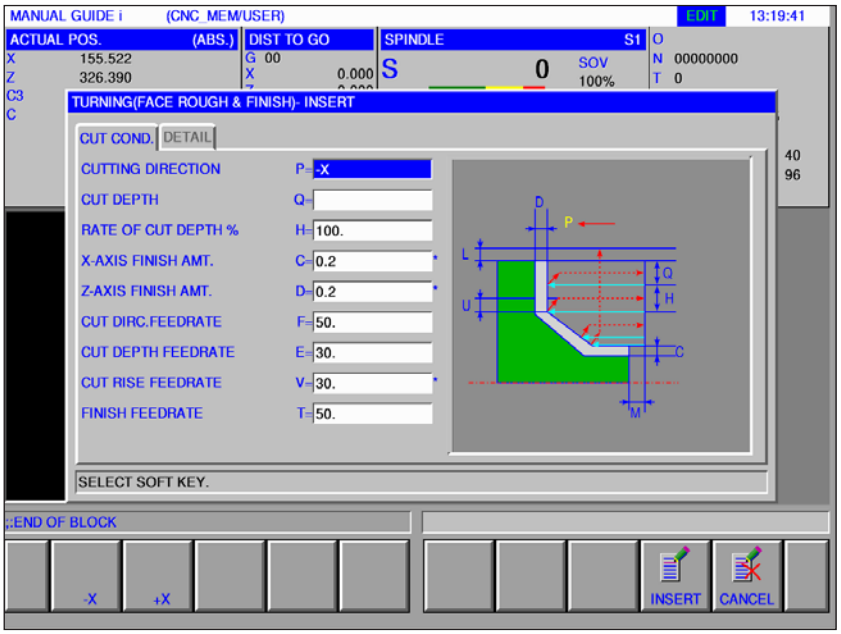
1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 The tool cuts with the feed speed for finishing along the entered contour, until the finished contour is achieved.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" position in rapid traverse.



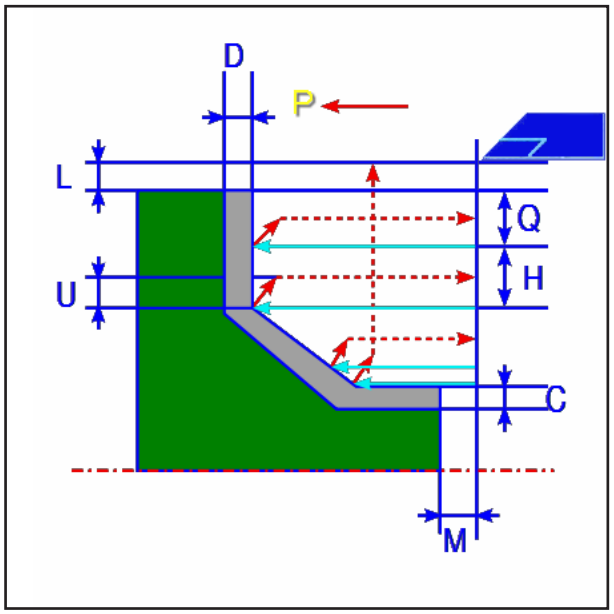
Face Rough + Finish G1152



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

Cutting condition		
Data item		**Meaning**
P	Cutting direction	• [-Z]: Cutting takes place in the -Z-direction. • [-Z]: Cutting takes place in the +Z-direction.
Q	Cut depth	Depth of each cut (radius value, positive value)
H	Rate of cut depth %	The delivery depth is stipulated as 100% fixed (standard value) and cannot be changed.
C*	X-axis finish amount	Finishing dimension in X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Z-axis finish amount	Finishing dimension in Z-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Cut direction feedrate	Feed speed for cutting in workpiece radius direction (positive value)
E	Cut depth feedrate	Feed speed for cutting in Z-axis direction (positive value)
V*	Cut rise feedrate	Feed speed when cutting in the direction of retraction from the workpiece (positive value). Feed extract is fixed with the populated value of feed plunge.
T	Finish feedrate	Feed speed for finishing in workpiece radius direction (positive value)

Detail		
Data item		**Meaning**
K	1st feed override %	The feed override value for the first cut is stipulated as 100% fixed and cannot be changed.
W	Cut rise method	• [CUT]: The tool returns after cutting along a contour about a "retraction distance".
U	Escape amount	Distance by which the tool returns from the machined surface after each cut (radius value, positive value).
L	X-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the X-axis direction (radius value, positive value).
M	Z-axis clearance	Distance between the unmachined part and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
S	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.
X	Pocket cutting	• [CUT]: Cuts a pocket (Initial value) • [NOTHIN]: Does not cut a pocket
Y	Overhang cutting	• [CUT]: Cuts an overhang (Initial value) Y is stipulated as fixed and cannot be changed.



Tool track

Roughing cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

2 After cutting in the Z-axis direction with feed speed (F), the tool cuts with feed speed (E) in the entered contour position, for which a finishing dimension is still outstanding in the X-axis direction.

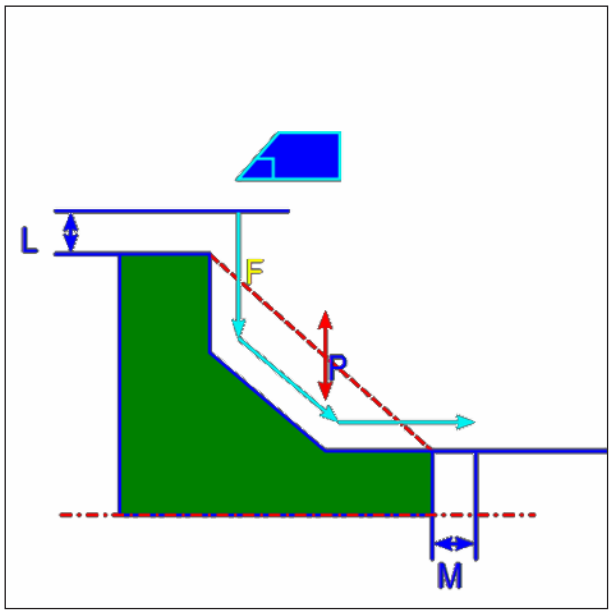
3 If [VORSCH] is preselected, the tool first cuts along the contour and then retracts by the retraction amount (U) in the X- and Z- axial dimensions.

4 The tool is moved to the cutting in start point in the Z axis direction.

5 Steps <2>to <4> are repeated until the lowest part has been reached in the Z-axis direction.

6 If a further pocket is still available, the tool is positioned on the pocket and steps <2> to <4> are repeated.

7 After all areas of the part are cut, the tool retracts in the Z-axis direction into the cutting in start point + separation (M)" in rapid traverse.



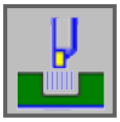
Tool track

Finishing cycle description

1 The tool traverses to the "Cut-in start point + distance (L, M)" in rapid traverse.

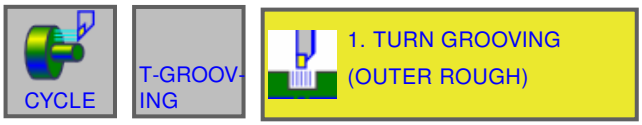
2 The tool cuts with the feed speed for finishing along the entered contour, until the finished contour is achieved.

3 After all areas of the part are cut, the tool retracts in the X-axis direction into the cutting in start point + separation (L)" position in rapid traverse.

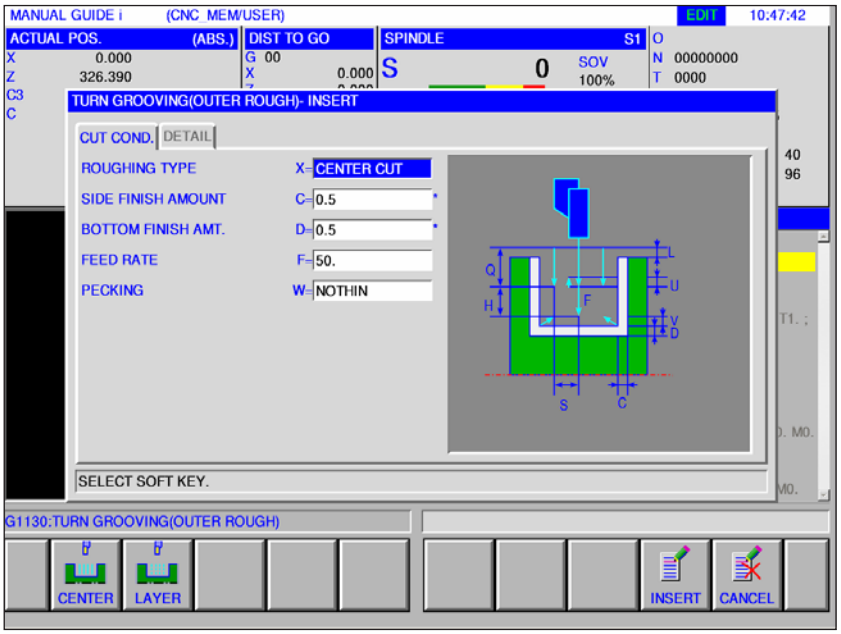


Grooving

- Outside Roughing G1130
- Inside Roughing G1131
- Face Roughing G1132
- Outside Coarse + Fine G1133
- Inside Coarse + Fine G1134
- Face Coarse + Fine G1135
- Outside Finishing G1136
- Inside Finishing G1137
- Face Finishing G1138



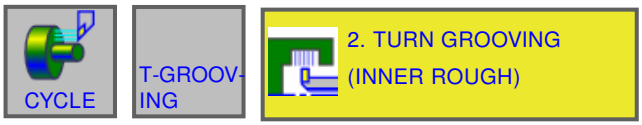
Turn grooving outside roughing G1130



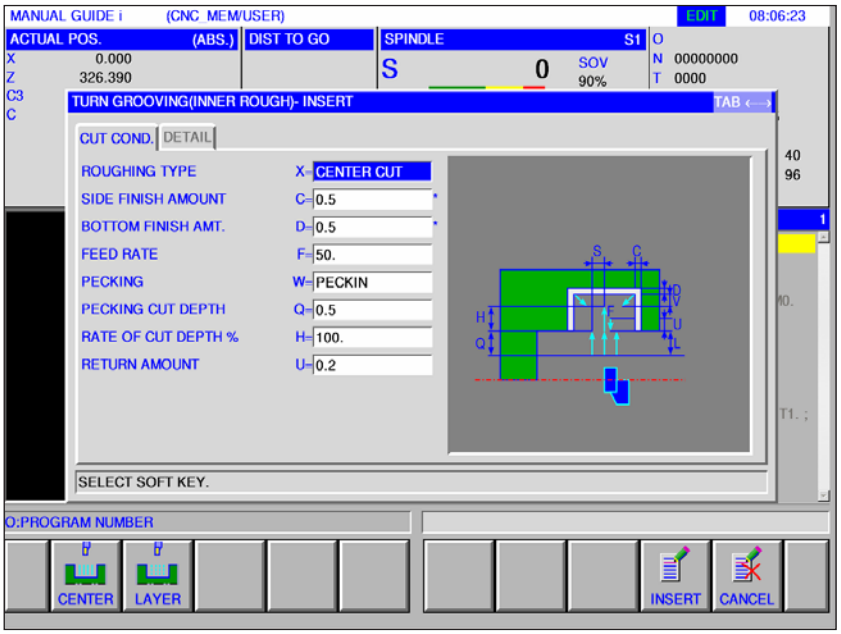
Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		**Meaning**
P	Roughing Type	• [CENTER]: Runs roughing for one side and then the centre point for the other side. • [LAYER]: Runs roughing for each position.
C*	Side finish amount	Finishing allowance for the side surfaces of a groove. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
D*	Bottom finish amount	Finishing allowance at the base of the groove. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
F	Feed rate	Feed speed for cutting in the tool axis direction (positive value)
W	Pecking	• [NO]: There is no cutting in the deep boring process (initial value) W is stipulated as fixed and cannot be changed.

Cutting condition		
Data item		**Meaning**
L	Clearance	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (Radius value, positive value).
P	Dwell time	Dwell time, if the tool has reached the base of the groove. (in seconds, positive value)
Z	Approach motion	• [2 AXES]: The tool is moved from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
V	Escape amount	Distance by which the tool retracts from the machining surface after each cut. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
A	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.



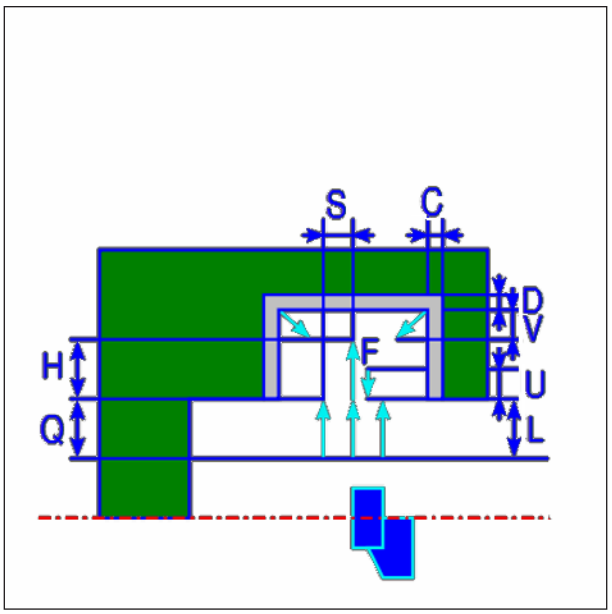
Grooving Inside Roughing G1131



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

Machining Operation		
Data element		**Meaning**
X	roughing type	• [CENTER] : Runs roughing for one side and then for the other side of the centre point. • [LAYER]: Runs roughing for each position.
C*	Finishing allowance side	Finishing allowance for the side surfaces of a groove. (radius value, positive value)
D*	Finishing allowance bottom	Finishing allowance at the base of the groove. (radius value, positive value)
F	Feed speed	Feed speed for grooving in the tool axis direction (positive value)
W	Chip crushing	• [NO]: Grooving is not performed via deep boring process (initial value). • [YES] : Grooving is performed via deep boring process.
Q	Stroke depth (INCR+)	Cut depth in tool axis direction per deep boring process (radius value, positive value). Is only provided, if for chip crushing [YES] has been selected
H	Infeed rate %	The position is pre-set as 100% fixed and cannot be changed.
U	Liftoff amount	Distance by which the tool retracts from the machining surface after each deep boring process. (radius value, positive value)

Detail		
Data element		**Meaning**
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
V	Retraction amount	Distance by which the tool retracts from the machining surface after each cut. (radius value, positive value)
A	Machining offset direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.



Tool track

Cycle Description Inside Roughing

1 The tool traverses to the "cut-in start point + clearance (L, M)" position in rapid traverse.

2 After the tool has moved in rapid traverse to the centre of the groove (in Z-axis direction), it cuts in feed speed (F) in X-axis direction that is predetermined for this machining direction. If deep boring has been set, the tool cuts a specified deep boring amount, while retracting in +X-axis direction between the deep boring cycles.

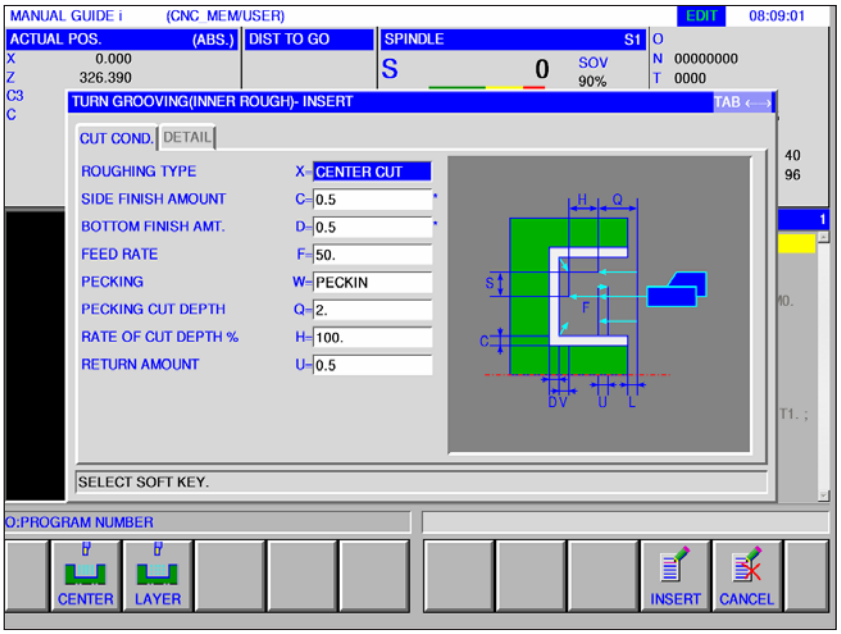
3 When the tool has reached the base of the groove in X-axis direction (up to the finishing allowance), it retracts in +X-axis direction, feeds by a cutting amount through the width in Z-axis direction and subsequently cuts in X-axis direction.

4 After step <3> has been repeated until the wall of the groove has been reached, the tool proceeds to work on another wall.

5 After all areas of the part have been cut, the tool moves in X-axis direction back to the "cut-in start point + clearance (L)" position in rapid traverse.

CYCLE | T-GROOV-ING | 3. TURN GROOVING (FACE ROUGH)

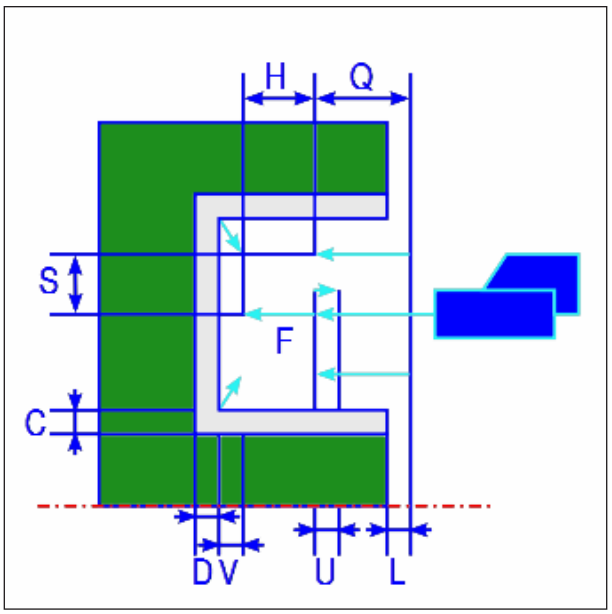
Grooving Face Roughing G1132



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

Machining Operation		
Data element		**Meaning**
X	Roughing type	• [CENTER]: Runs roughing for one side and then for the other side of the centre point. • [LAYER]: Runs roughing for each position.
C*	Finishing allowance side	Finishing allowance for the side surfaces of a groove. (radius value, positive value)
D*	Finishing allowance bottom	Finishing allowance at the base of the groove. (radius value, positive value)
F	Feed speed	Feed speed for grooving in the tool axis direction (positive value)
W	Chip crushing	• [NO]: Grooving is not performed via deep boring process (initial value). • [YES] : Grooving is performed via deep boring process.
Q	Stroke depth (INCR+)	Cut depth in tool axis direction per deep boring process (radius value, positive value). Is only provided, if for chip crushing [YES] has been selected
H	Infeed rate %	The position is pre-set as 100% fixed and cannot be changed.
U	Liftoff amount	Distance by which the tool retracts from the machining surface after each deep boring process. (radius value, positive value)

Detail		
Data element		**Meaning**
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
V	Retraction amount	Distance by which the tool retracts from the machining surface after each cut. (radius value, positive value)



Tool track

Cycle Description Face Roughing

1 The tool traverses to the "cut-in start point + clearance (L, M)" position in rapid traverse.

2 After the tool has moved in rapid traverse to the centre of the groove (in X-axis direction), it cuts in feed speed (F) in X-axis direction that is predetermined for this machining direction. If deep boring has been set, the tool cuts a specified deep boring amount, while retracting in +Z-axis direction between the deep boring cycles.

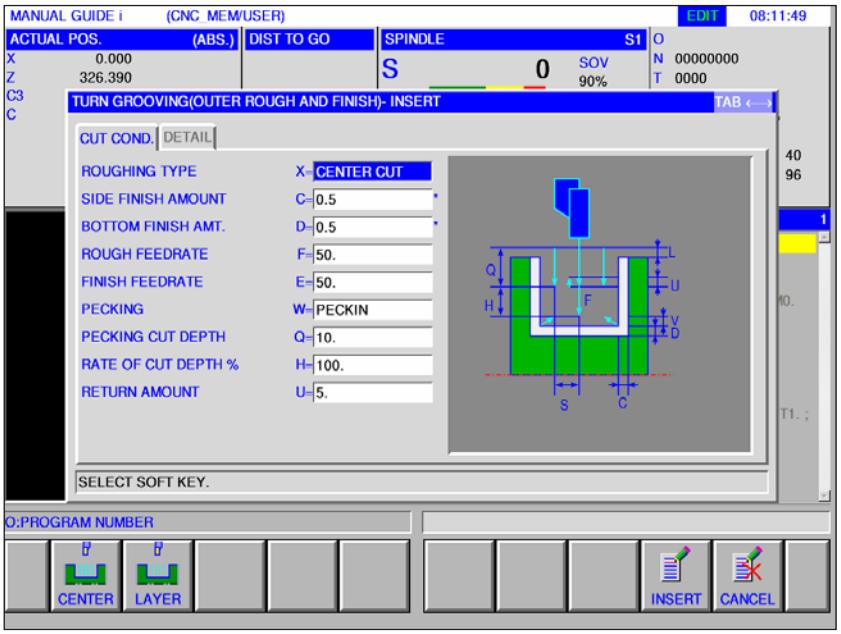
3 When the tool has reached the base of the groove in Z-axis direction (up to the finishing allowance), it retracts in +Z-axis direction, feeds by a cutting amount through the width in X-axis direction and subsequently cuts in Z-axis direction.

4 After step <3> has been repeated until the wall of the groove has been reached, the tool proceeds to work on another wall.

5 After all areas of the part have been cut, the tool moves in the Z-axis direction back to the "cut-in start point + clearance (L)" position in rapid traverse.



Turn Grooving Outside Coarse + Fine G1133



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

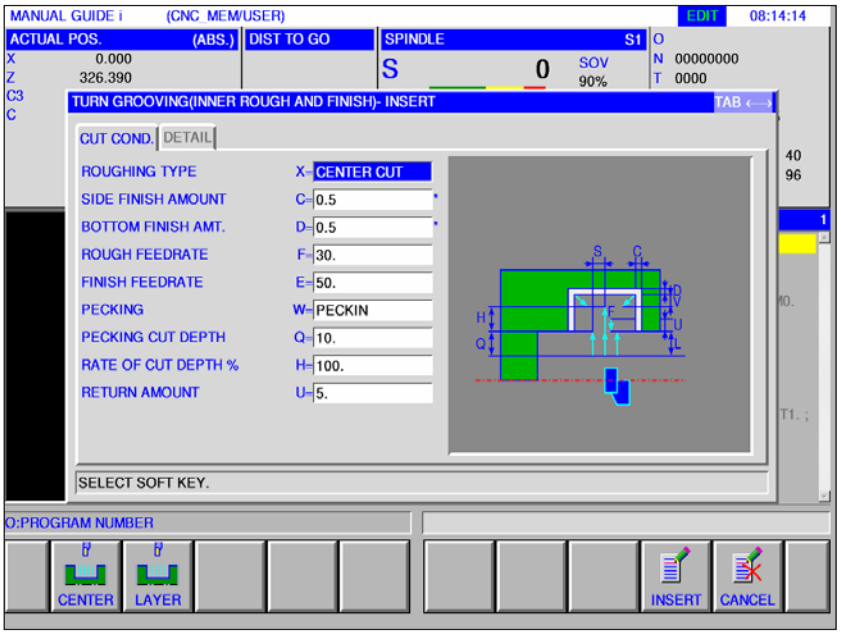
Machining Operation		
Data element		**Meaning**
X	Roughing type	• [CENTER]: Runs roughing for one side and then for the other side of the centre point. • [LAYER]: Runs roughing for each position.
C*	Finishing allowance side	Finishing allowance for the side surfaces of a groove. (radius value, positive value)
D*	Finishing allowance bottom	Finishing allowance at the base of the groove. (radius value, positive value)
F	Feed speed	Feed speed for grooving in the tool axis direction (positive value)
E	Finishing feed	Feed speed for finishing (positive value)
W	Chip crushing	• [NO]: Grooving is not performed via deep boring process (initial value). • [YES] : Grooving is performed via deep boring process.
Q	Stroke depth (INCR+)	Cut depth in tool axis direction per deep boring process (radius value, positive value). Is only provided, if for chip crushing [YES] has been selected
H	Infeed rate %	The position is pre-set as 100% fixed and cannot be changed.
U	Liftoff amount	Distance by which the tool retracts from the machining surface after each deep boring process. (radius value, positive value)

Detail		
Data element		**Meaning**
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
K	Machining end position	• [CENTER]: When finishing, machines both sides of the groove left and right evenly. • [CORNER] : Machines first one side surface and then the entire bottom of the groove and then proceeds to machining the other side surface only when finishing.
A	Infeed direction	Cut direction in the X-axis • [-X DIR] : Cutting takes place in the -X-direction. • [+X DIR] : Cutting takes place in the +X-direction.

Cycle Description Turn Grooving Roughing and Finishing
Details regarding the tool track can be found in the respective descriptions of roughing and finishing.



Turn Grooving Inside Coarse + Fine G1134



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

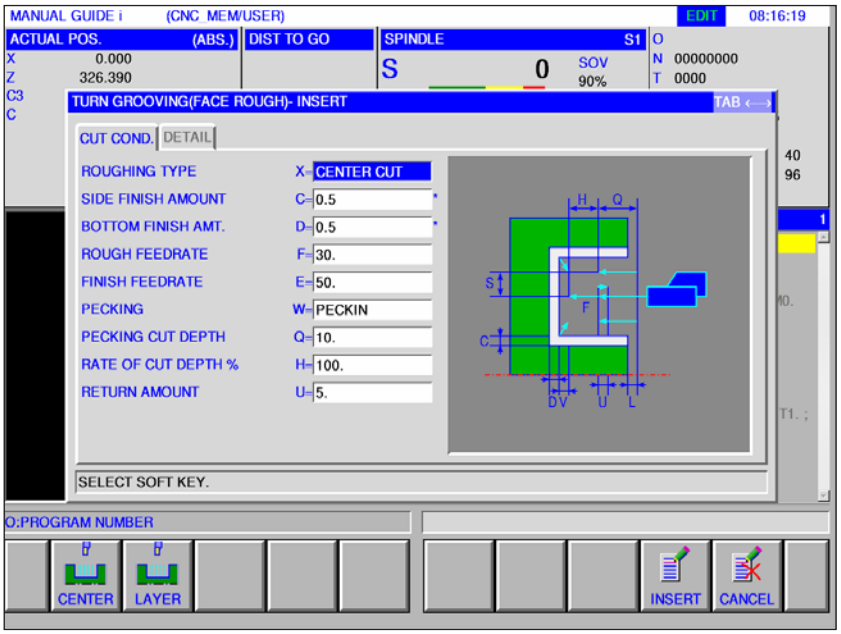
Machining Operation		
Data element		**Meaning**
X	Roughing type	• [CENTER]: Runs roughing for one side and then for the other side of the centre point. • [LAYER]: Runs roughing for each position.
C*	Finishing allowance side	Finishing allowance for the side surfaces of a groove. (radius value, positive value)
D*	Finishing allowance bottom	Finishing allowance at the base of the groove. (radius value, positive value)
F	Feed speed	Feed speed for grooving in the tool axis direction (positive value)
E	Finishing feed	Feed speed for finishing (positive value)
W	Chip crushing	• [NO] : Grooving is not performed via deep boring process (initial value). • [YES] : Grooving is performed via deep boring process.
Q	Stroke depth (INCR+)	Cut depth in tool axis direction per deep boring process (radius value, positive value). Is only provided, if for chip crushing [YES] has been selected
H	Infeed rate %	The position is pre-set as 100% fixed and cannot be changed.
U	Liftoff amount	Distance by which the tool retracts from the machining surface after each deep boring process. (radius value, positive value)

Detail		
Data element		**Meaning**
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
K	Machining end position	• [CENTER]: When finishing, machines both sides of the groove left and right evenly. • [CORNER] : Machines first one side surface and then the entire bottom of the groove and then proceeds to machining the other side surface only when finishing.
A	Machining offset direction	Cut direction in the X-axis • [-X DIR] : Cutting takes place in the -X-direction. • [+X DIR] : Cutting takes place in the +X-direction.

Cycle Description Turn Grooving Roughing and Finishing
Details regarding the tool track can be found in the respective descriptions of roughing and finishing.



Turn Grooving Face Coarse + Fine G1135



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

Machining Operation		
Data element		**Meaning**
X	Roughing type	• [CENTER]: Runs roughing for one side and then for the other side of the centre point. • [LAYER]: Runs roughing for each position.
C*	Finishing allowance side	Finishing allowance for the side surfaces of a groove. (radius value, positive value)
D*	Finishing allowance bottom	Finishing allowance at the base of the groove. (radius value, positive value)
F	Roughing feed	Feed speed for roughing (positive value)
E	Finishing feed	Feed speed for finishing (positive value)
W	Chip crushing	• [NO]: Grooving is not performed via deep boring process (initial value). • [YES] : Grooving is performed via deep boring process.
Q	Stroke depth (INCR+)	Cut depth in tool axis direction per deep boring process (radius value, positive value). Is only provided, if for chip crushing [YES] has been selected
H	Infeed rate %	The position is pre-set as 100% fixed and cannot be changed.
U	Liftoff amount	Distance by which the tool retracts from the machining surface after each deep boring process. (radius value, positive value)

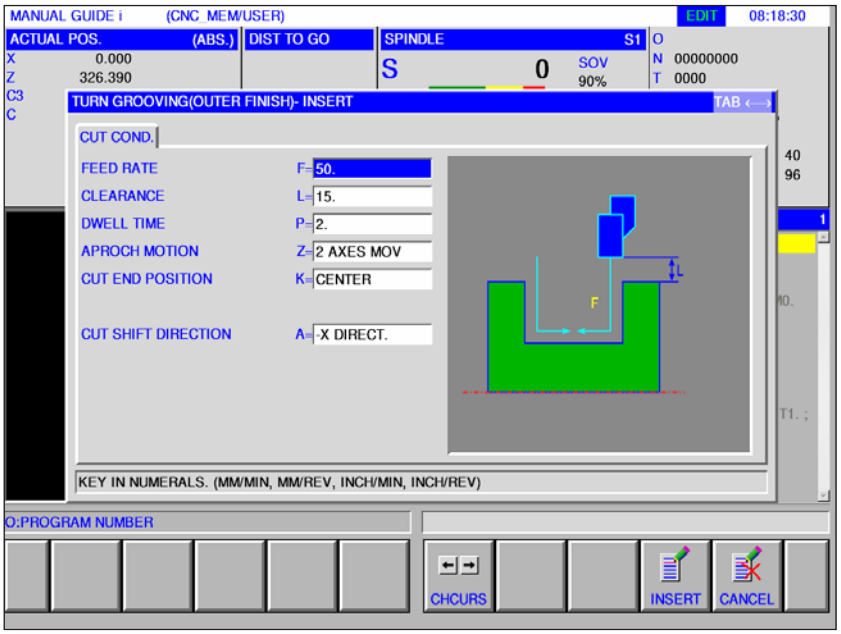
Detail		
Data element		**Meaning**
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
K	Machining end position	• [CENTER]: When finishing, machines both sides of the groove left and right evenly. • [CORNER] : Machines first one side surface and then the entire bottom of the groove and then proceeds to machining the other side surface only when finishing.

Cycle Description Turn Grooving Roughing and Finishing

Details regarding the tool track can be found in the respective descriptions of roughing and finishing.

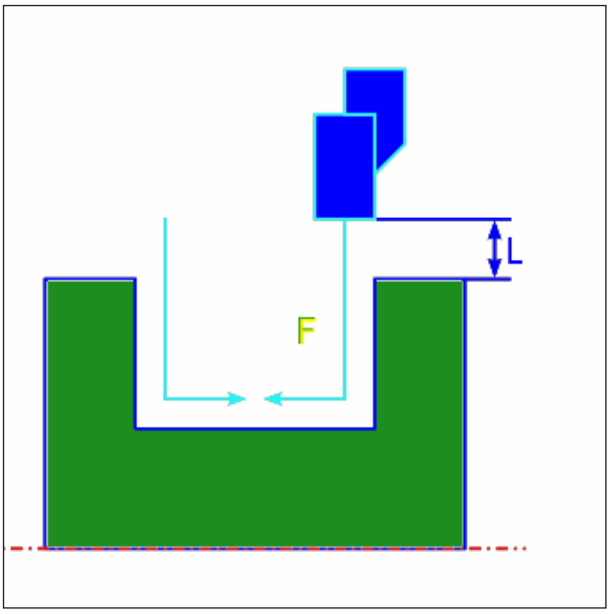


Turn Grooving Outside Finishing G1136



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

Machining Operation		
Data element		**Meaning**
F	Feed speed	Feed speed for finishing in the direction of the tool axis (positive value)
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
K	Machining end position	• [CENTER]: When finishing, machines both sides of the groove left and right evenly. The position is pre-set as fixed and cannot be changed.
A	Machining offset direction	Cut direction in the X-axis • [-X DIR] : Cutting takes place in the -X-direction. • [+X DIR]: Cutting takes place in the +X-direction.



Tool track

Cycle Description Outside Finishing

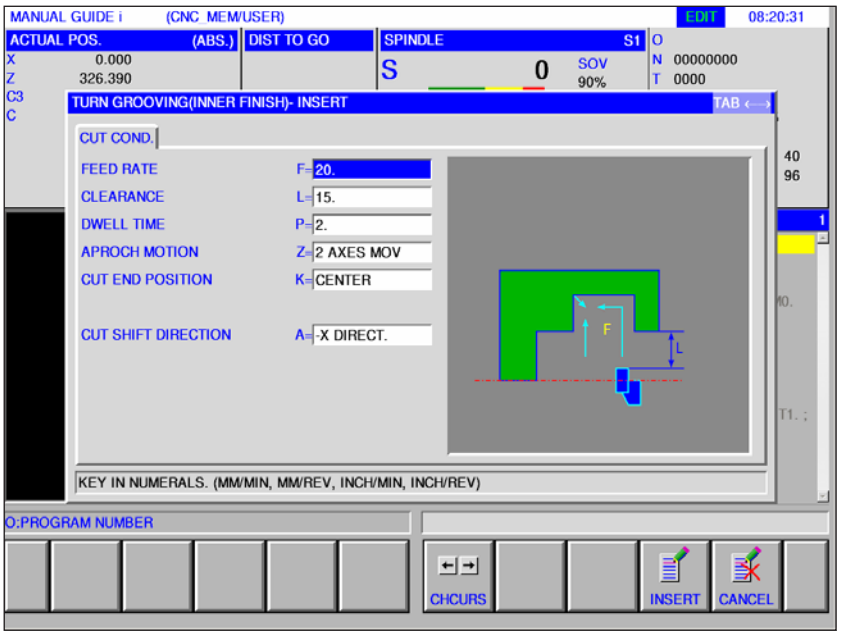
1 The tool traverses to the "cut-in start point + clearance (L, M)" position in rapid traverse.

2 If [CENTER] has been specified as machining end position, the tool cuts into one of the groove walls with finishing feed speed and continues to cut in the direction of the groove centre (in Z-axis direction). After retracting in X-axis direction, the tool performs the same cutting process at the other wall of the groove.

3 The tool moves in X-axis direction to the position "cut-in start point + distance (L)" in rapid traverse.

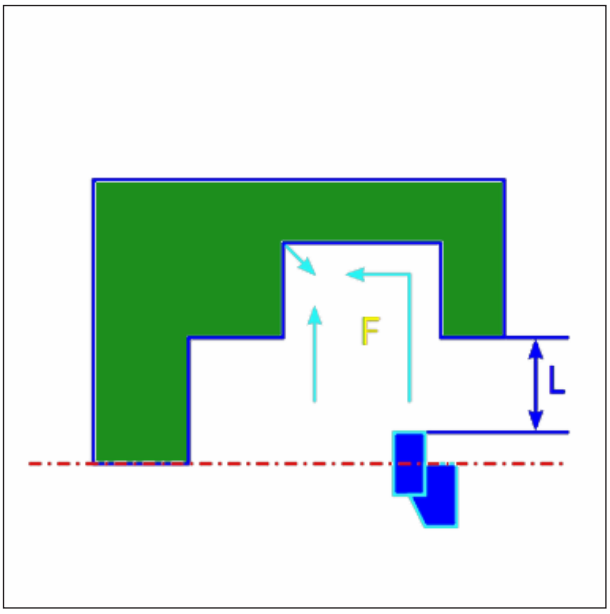


Turn Grooving Inside Finishing G1137

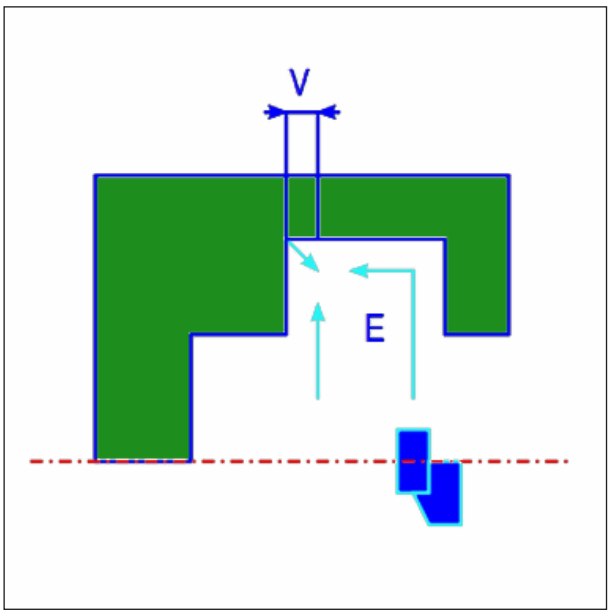


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
F	Feed speed	Feed speed for finishing in the direction of the tool axis (positive value)
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
K	Machining end position	• [CENTER]: When finishing, machines both sides of the groove left and right evenly. • [CORNER] : Machines first one side surface and then the entire bottom of the groove and then proceeds to machining the other side surface only when finishing.
A	Machining offset direction	Cut direction in the X-axis • [-X DIR] : Cutting takes place in the -X-direction. • [+X DIR]: Cutting takes place in the +X-direction.



Tool track centre



Tool track corner

Cycle Description Inside Finishing

1 The tool traverses to the "cut-in start point + clearance (L, M)" position in rapid traverse.

2 If [CENTER] has been specified as machining end position, the tool cuts into one of the groove walls with finishing feed speed and continues to cut in the direction of the groove centre (in Z-axis direction). After retracting in X-axis direction, the tool performs the same cutting process at the other wall of the groove.

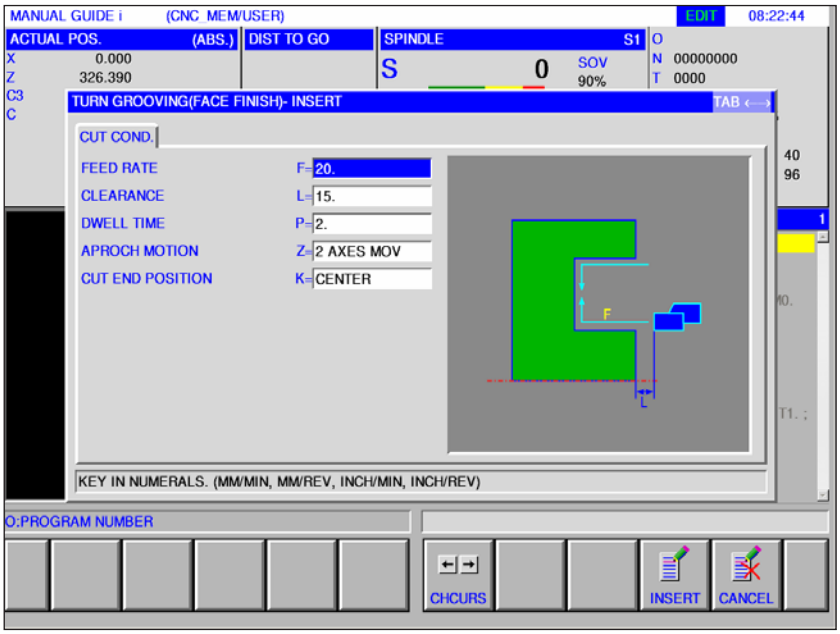
3 If [CORNER] has been specified as machining end position, the tool cuts in from the wall towards the bottom in Z-axis direction and retracts by the retraction distance in X- and Z-axis direction and subsequently in X-axis direction. After that, the tool starts the machining process at the other wall of the groove and continues the cut toward the corner at the bottom of the groove and then retracts by the retraction distance in X- and Z-axis direction.

4 The tool moves back in X-axis direction to the position "cut-in start point + clearance (L)" in rapid traverse.

5 The tool traverses in rapid traverse to the position "cut-in start point + clearance (M)" in Z-axis direction and then retracts out of the work piece.

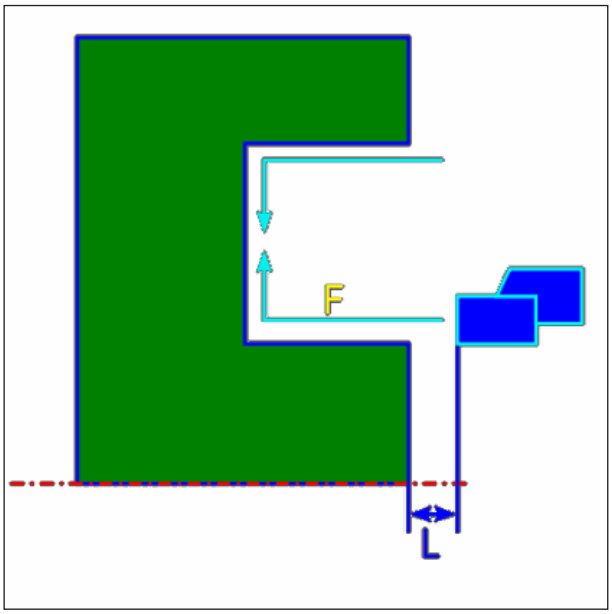


Face Finishing G1138

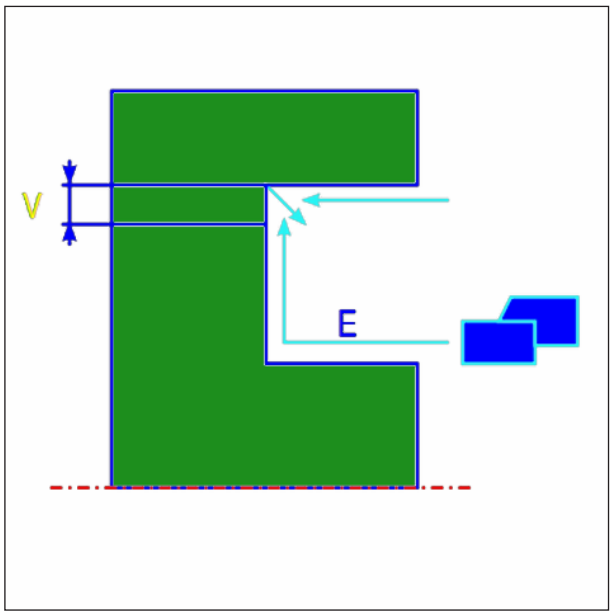


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
F	Feed speed	Feed speed for finishing in the direction of the tool axis (positive value)
L	Safety distance (Reference base)	Distance between the outer surface of the groove and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
P	Dwell time	Dwell time, when the tool has reached the base of the groove. (in seconds, positive value)
Z	Approaching movement	• [2 AXIS]: The tool is moved from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
K	Machining end position	• [CENTER]: When finishing, machines both sides of the groove left and right evenly. • [CORNER] : Machines first one side surface and then the entire bottom of the groove and then proceeds to machining the other side surface only when finishing.



Tool track centre



Tool track corner

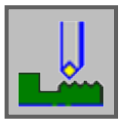
Cycle Description Face Finishing

1 The tool traverses to the "cut-in start point + clearance (L, M)" position in rapid traverse.

2 If [CENTER] has been specified as machining end position, the tool cuts into one of the groove walls with finishing feed speed and continues to cut in the direction of the groove centre (in X-axis direction). After retracting in Z-axis direction, the tool performs the same cutting process at the other wall of the groove.

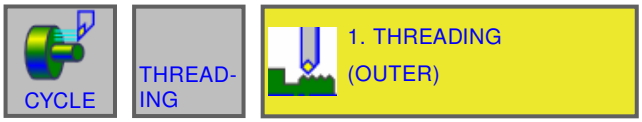
3 If [CORNER] has been specified as machining end position, the tool cuts in from the wall towards the bottom in X-axis direction and retracts by the retraction distance in X- and Z-axis direction and subsequently in X-axis direction. After that, the tool starts the machining process at the other wall of the groove and continues the cut toward the corner at the bottom of the groove and then retracts by the retraction distance in X- and Z-axis direction.

4 The tool moves back in Z-axis direction to the position "cut-in start point + clearance (L)" in rapid traverse.

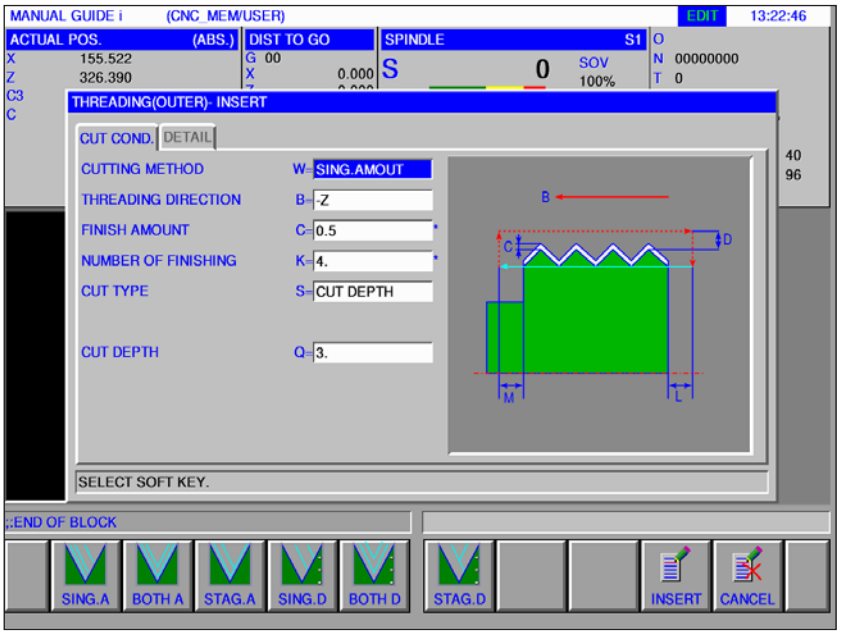


Threading

- Outer G1140
- Inner G1141



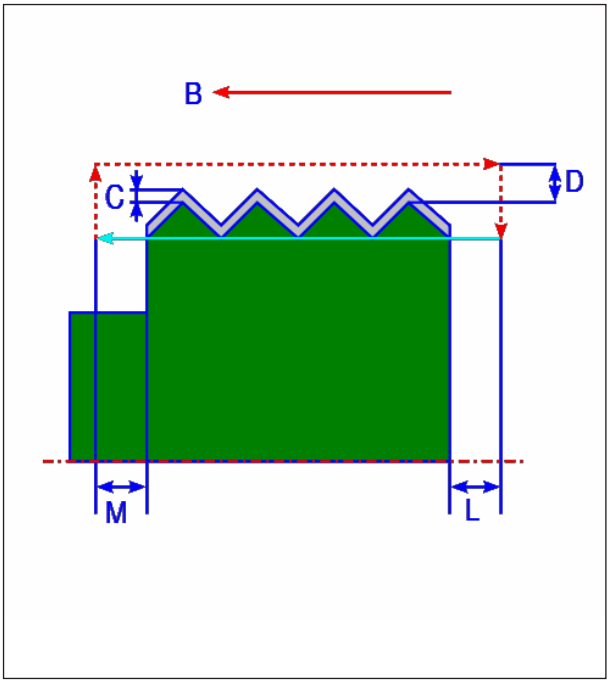
Threading Outer G1140



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

Cutting condition		
Data item		**Meaning**
W	Cutting method	• [SING.A]: Constant cut amount, single-sided cut. • [BOTH.A]: Constant cut amount, machining both sides. • [STAG.A]: Constant cut amount, zigzag tapping on both sides. • [SING.D]: Constant cut depth, machining on one side. • [BOTH.D]: Constant cut depth, machining on both sides. • [STAG.D]: Constant cut depth, zigzag tapping on both sides.
B	Threading direction	• [-Z]: Cutting in the -Z-direction. • [+Z]: Cutting in the +Z-direction.
C*	Finish amount	Finishing dimension for tapping in the X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
K*	Number of finishing	Number of finishing machining processes (positive value).
S	Cut type	• [NUMBER]: Number of tappings. • [DEPTH]: Tapping by means of the first cut amount.
P	Cut depth number	Number of cuts for thread roughing (max. 999, positive value). The number of finishing machining cuts is included. With zigzag dual-sided thread cutting an even number of cuts must be given. With an uneven number of cuts, only one additional dual sided thread cutting process is executed in zigzag. This parameter is only available for [NUMBER].
Q	Cut depth	Cut depth per thread roughing process (radius value, positive value). The number of cuts is determined by the cut depth and by the machining method.

Detail		
Data item		**Meaning**
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
D	Surface clearance	Distance between the thread comb and the machining start point (approach point) in the X-axis direction (radius value, positive value).
L	Entrance clearance	Distance between the thread comb and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
M	Exit clearance	Distance between the thread end point and the end point of the tapping process in the Z-axis direction (radius value, positive value).
Y	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.



Cut outside thread

Cycle description

1 The tool traverses into the "Thread start point + surface clearance (D in X-axis direction)" position and thread entry separation (L in Z-axis direction) in rapid traverse.

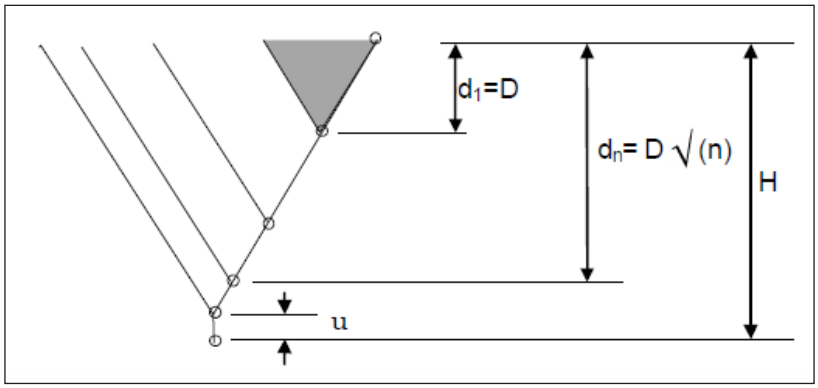
2 Threads are cut with the given machining method. The end point of the thread cutting process in the Z-axis direction is remote from the thread end point by the thread exit clearance.

3 **If** [NUMBER] is given as the cut type, the thread cutting process takes place as often as stated by the [NUMBER] value. If [DEPTH] is stipulated, this amount is delivered in the first cut and the thread cutting process is executed until the stipulated thread contour is reached.

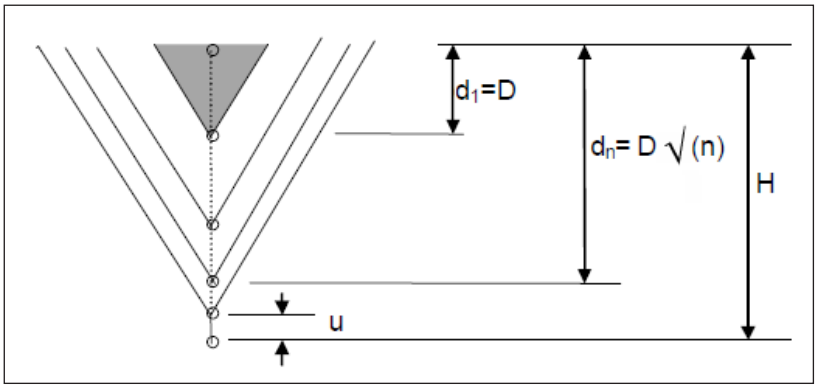
4 After all areas of the part are cut, the tool moves in the X-axis direction back into the "cutting in start point + separation (L)" position in rapid traverse.

Detail views of the machining method

1 [SING.A]: Constant cut amount, single-sided cut.

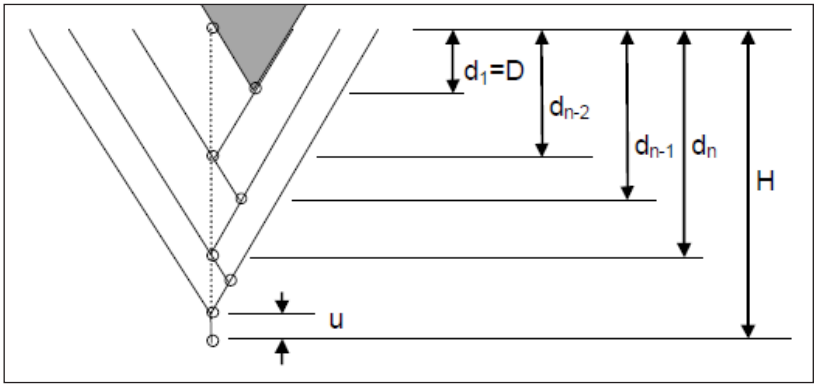


2 [BOTH.A]: Constant cut amount, machining both sides.



H = Thread tip height, D = Cut amount, u = Finishing dimension

3 [STAG.A]: Constant cut amount, zigzag tapping on both sides.

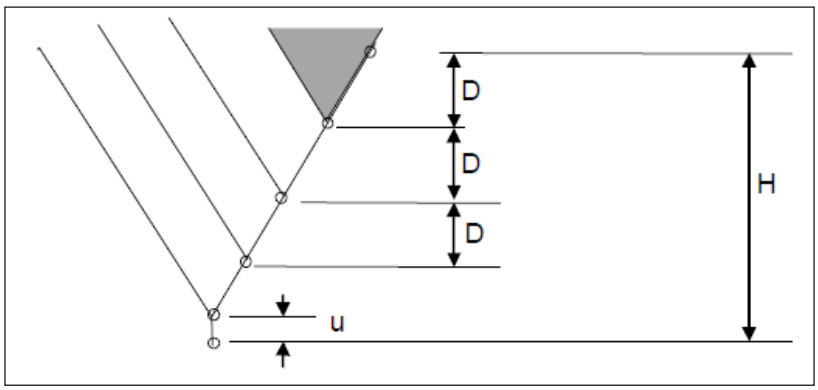


$d_{n-2} = D \sqrt{(n-2)}$
$d_{n-1} = (D(\sqrt{(n-2)} + \sqrt{(n)}))/2$
$d_n = D \sqrt{(n)}$

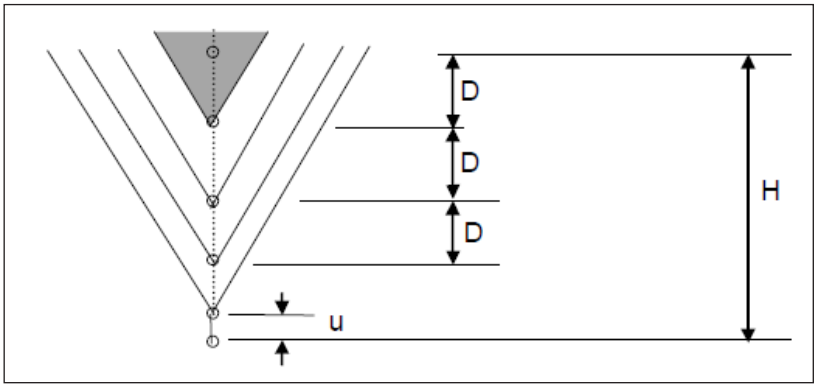


Instructions:
Depending on the minimum cut, the given number of cuts can be excessively large. In this case, the thread is cut with a lower number of cuts than that given.
The number of cuts actually made can differ from the given number of cuts owing to a computer error.

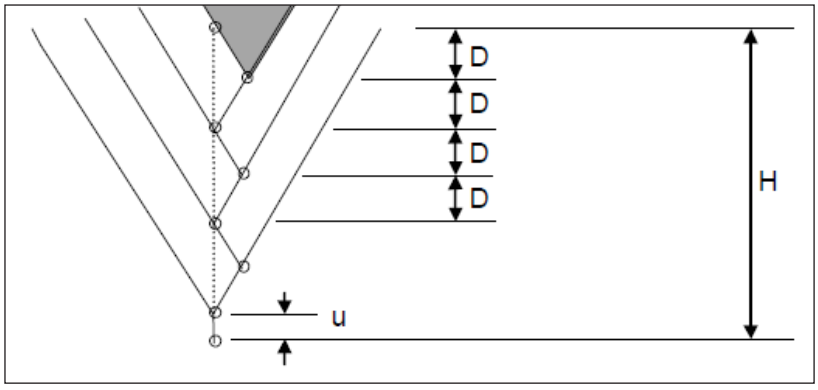
4 [SING.D]: Constant cut depth, machining on one side



5 [BOTH.D]: Constant cut depth, machining on both sides

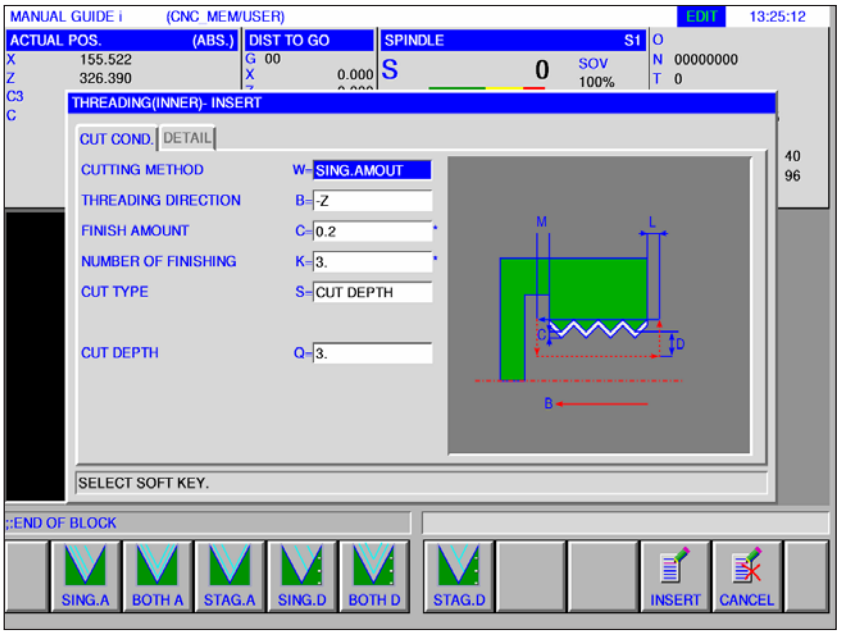


6 [STAG.D]: Constant cut depth, zigzag tapping on both sides





Threading Inner G1141



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

Cutting condition		
Data item		**Meaning**
W	Cutting method	• [SING.A]: Constant cut amount, single-sided cut. • [BOTH.A]: Constant cut amount, machining both sides. • [STAG.A]: Constant cut amount, zigzag tapping on both sides. • [SING.D]: Constant cut depth, machining on one side. • [BOTH.D]: Constant cut depth, machining on both sides. • [STAG.D]: Constant cut depth, zigzag tapping on both sides.
B	Threading direction	• [-Z]: Cutting in the -Z-direction. • [+Z]: Cutting in the +Z-direction.
C*	Finish amount	Finishing dimension for tapping in the X-axis direction. The unmachined part is viewed as 0 degrees. (Radius value, positive value)
K*	Number of finishing	Number of finishing machining processes (positive value).
S	Cut type	• [NUMBER]: Number of tappings. • [DEPTH]: Tapping by means of the first cut amount.
P	Cut depth number	Number of cuts for thread roughing (max. 999, positive value). The number of finishing machining cuts is included. With zigzag dual-sided thread cutting an even number of cuts must be given. With an uneven number of cuts, only one additional dual sided thread cutting process is executed in zigzag. This parameter is only available for [NUMBER].
Q	Cut depth	Cut depth per thread roughing process (radius value, positive value). The number of cuts is determined by the cut depth and by the machining method.

Detail		
Data item		**Meaning**
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.
D	Surface clearance	Distance between the thread comb and the machining start point (approach point) in the X-axis direction (radius value, positive value).
L	Entrance clearance	Distance between the thread comb and the machining start point (approach point) in the Z-axis direction (radius value, positive value).
M	Exit clearance	Distance between the thread end point and the end point of the tapping process in the Z-axis direction (radius value, positive value).
Y	Cut shift direction	Cut direction in the X-axis • [-X]: Cutting takes place in the -X-direction. • [+X]: Cutting takes place in the +X-direction.

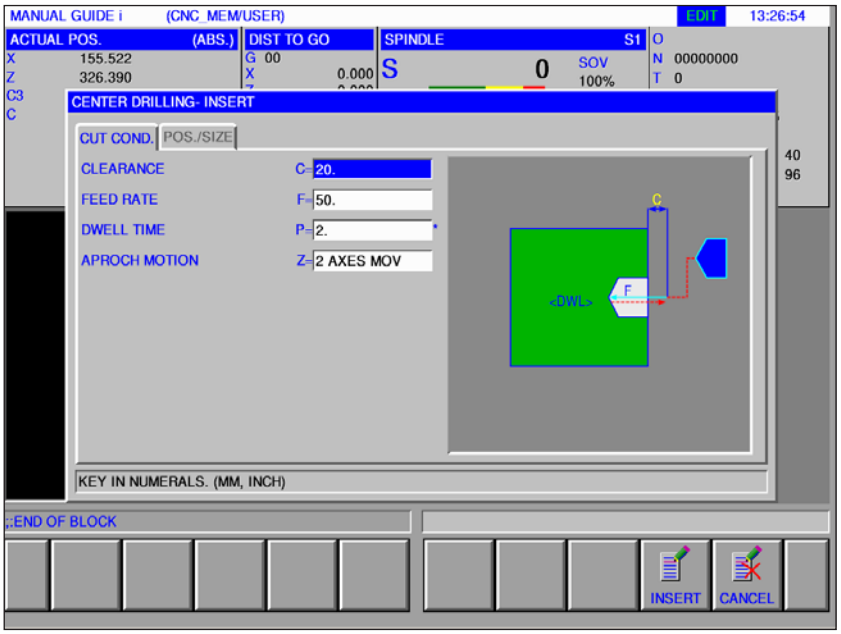


Hole machining

- Centre Drilling G1100
- Drilling G1101
- Tapping G1102
- Reaming G1103
- Boring G1104



Centre Drilling G1100



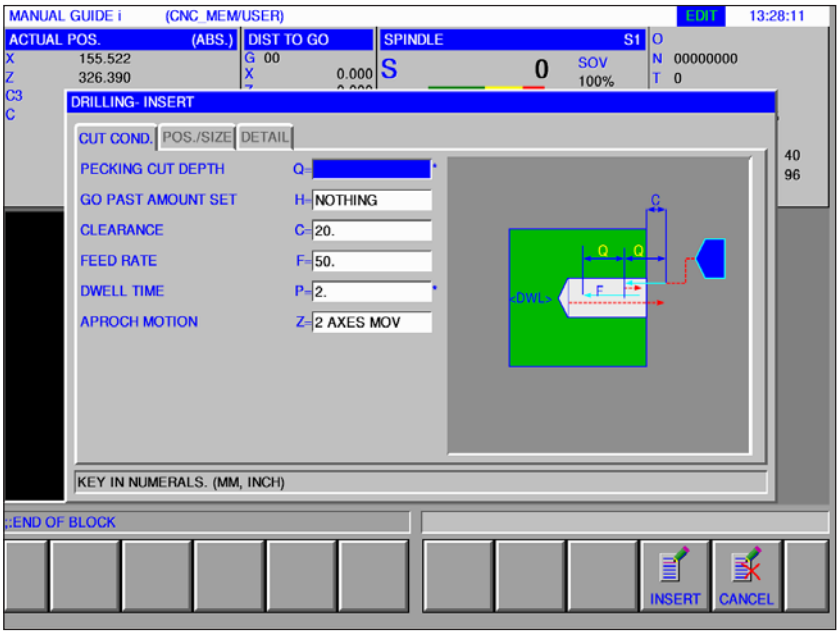
Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		**Meaning**
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 boring (in seconds, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.

Position / Size		
Data item		**Meaning**
B	Base position	Z-coordinates of the workpiece surface.
L	Cut depth	Boring depth (radius value, negative value)



Drilling G1101



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

Cutting condition		
Data item		**Meaning**
Q*	Pecking cut depth	Cut depth per drilling process (radius value, positive value).
H	Go past amount set	Boring depth settings: • [NOTHIN]: Drilling depth relative to the tool shaft • [SETING]: Drilling depth relative to the tool tip For the setting [SETING] the parameters U, V, K and the softkey [CALC] are available under Details.
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 boring (in seconds, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.

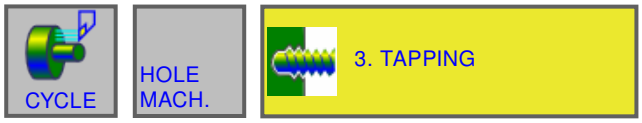
Position / Size		
Data item		**Meaning**
B	Base position	Z-coordinates of the workpiece surface.
L	Cut depth	Boring depth (radius value, negative value)

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

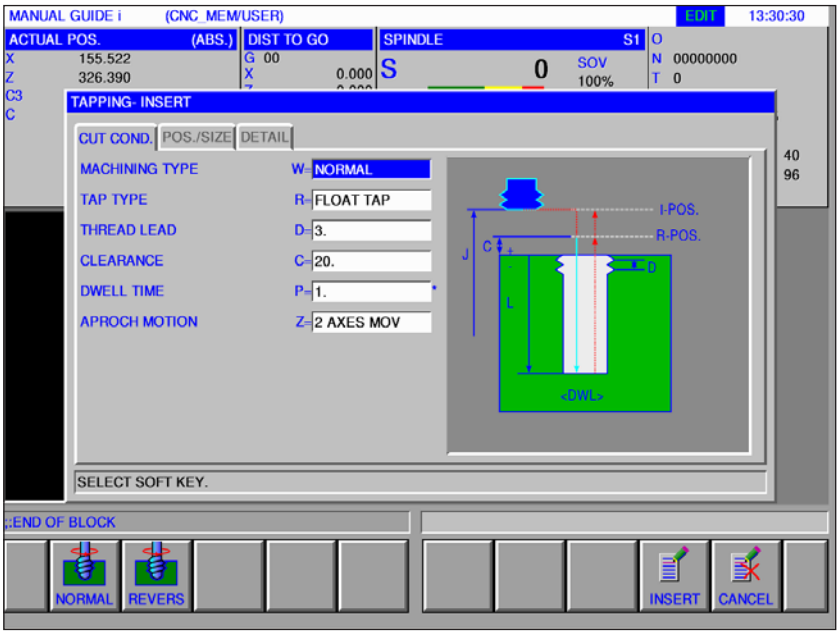


Instructions:
For the parameters A,S,D and E in "Detail", for all boring cycles:

All 4 parameters must be entered as soon as only one of these fields is populated.



Tapping G1102

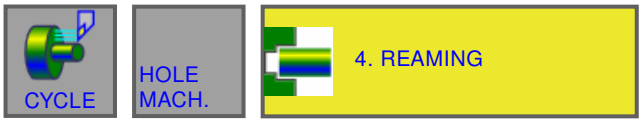


Entry fields marked with * are optional and should not be populated.

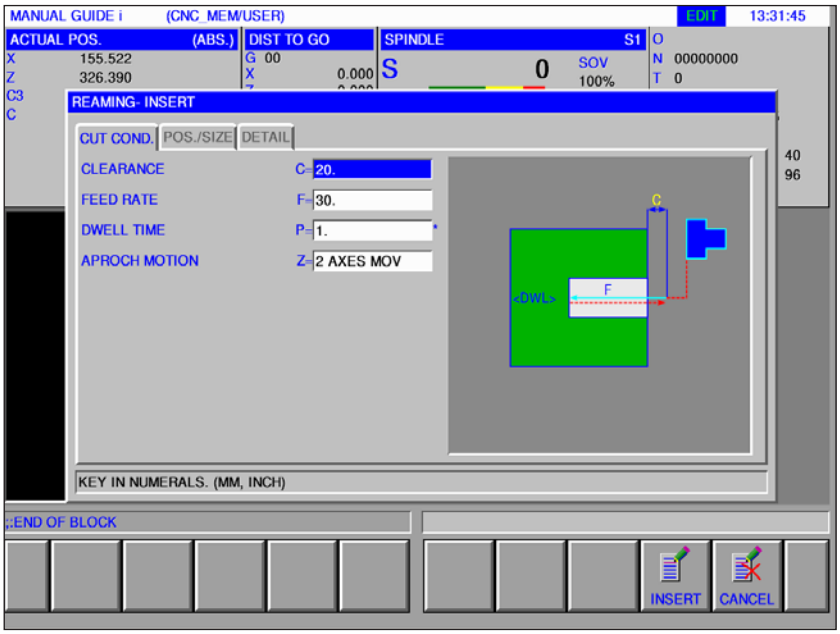
Cutting condition		
Data item		**Meaning**
W	Machining type	• [NORMAL]: Normal tapping (right hand thread) • [REVERS]: Left hand thread tapping
R	Tap type	• [FLOAT]: Thread tapping with compensating chuck. • [RIGID]: Thread tapping without compensating chuck.
D	Thread lead	Tap pitch (radius value, positive value).
C	Clearance	Distance between workpiece surface and R-position (Radius value, positive value)
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.

Position / Size		
Data item		**Meaning**
B	Base position	Z-coordinates of the workpiece surface.
L	Cut depth	Boring depth (radius value, negative value)

Detail		
Data item		**Meaning**
Q	Spindle number	• [SPNDL1]: Main spindle • [SPNDL2]: Counterspindle



Reaming G1103



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

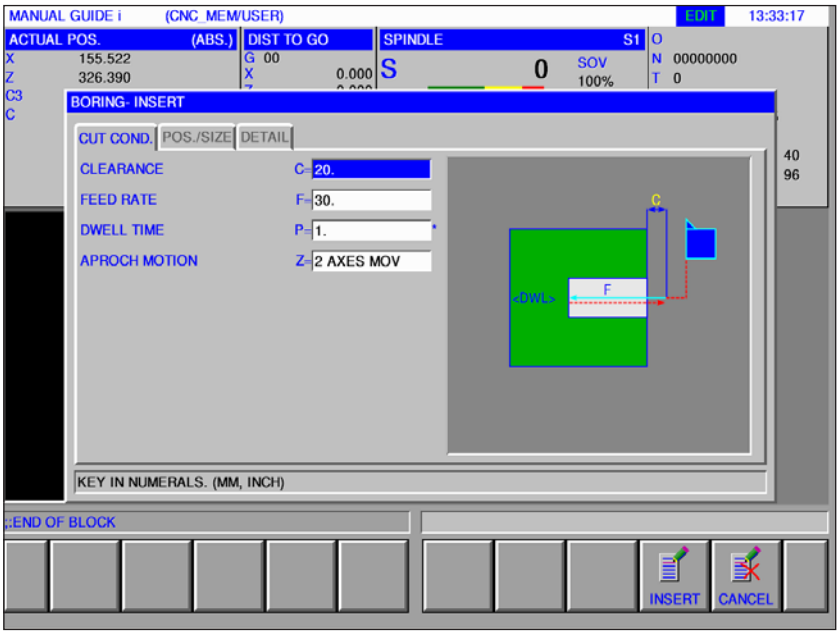
Cutting condition		
Data item		**Meaning**
C	Clearance	Distance between workpiece surface and R-position (Radius value, positive value)
F	Feed rate	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.

Position / Size		
Data item		**Meaning**
B	Base position	Z-coordinates of the workpiece surface.
L	Cut depth	Boring depth (radius value, negative value)

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



Boring G1104

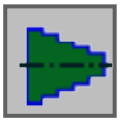


Entry fields marked with * are optional and should not be populated.

Cutting condition		
Data item		**Meaning**
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 boring (in seconds, positive value).
Z	Approach motion	• [2 AXES MOV]: The tool traverses from the instantaneous position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is stipulated as fixed and cannot be changed.

Position / size		
Data item		**Meaning**
B	Base position	Z-coordinates of the workpiece surface.
L	Cut depth	Boring depth (radius value, negative value)

Detail		
Data item		**Meaning**
D*	End cut depth	The end feed E applies for the end cut depth D
E*	End feed rate	

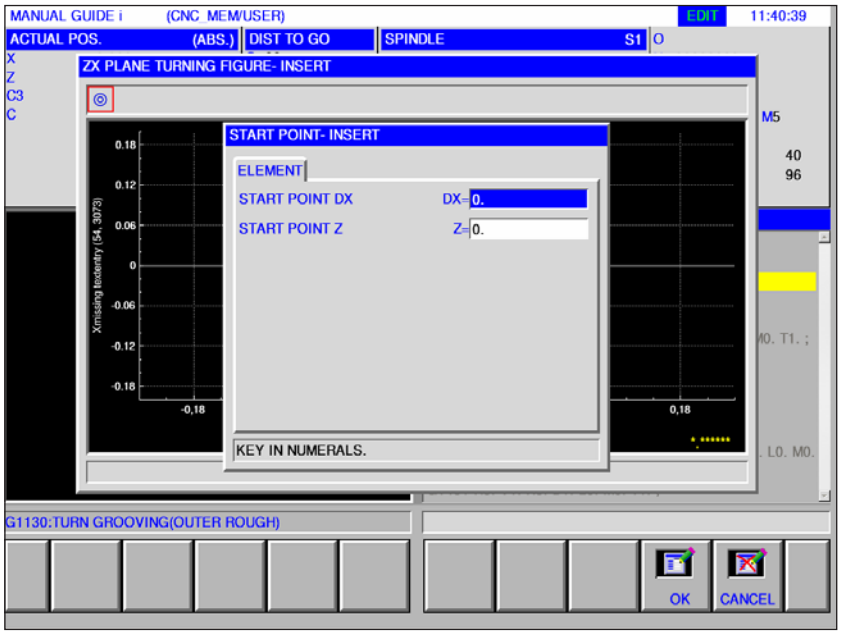


Figure

- Plane Turning Figure



Plane Turning Figure G1450

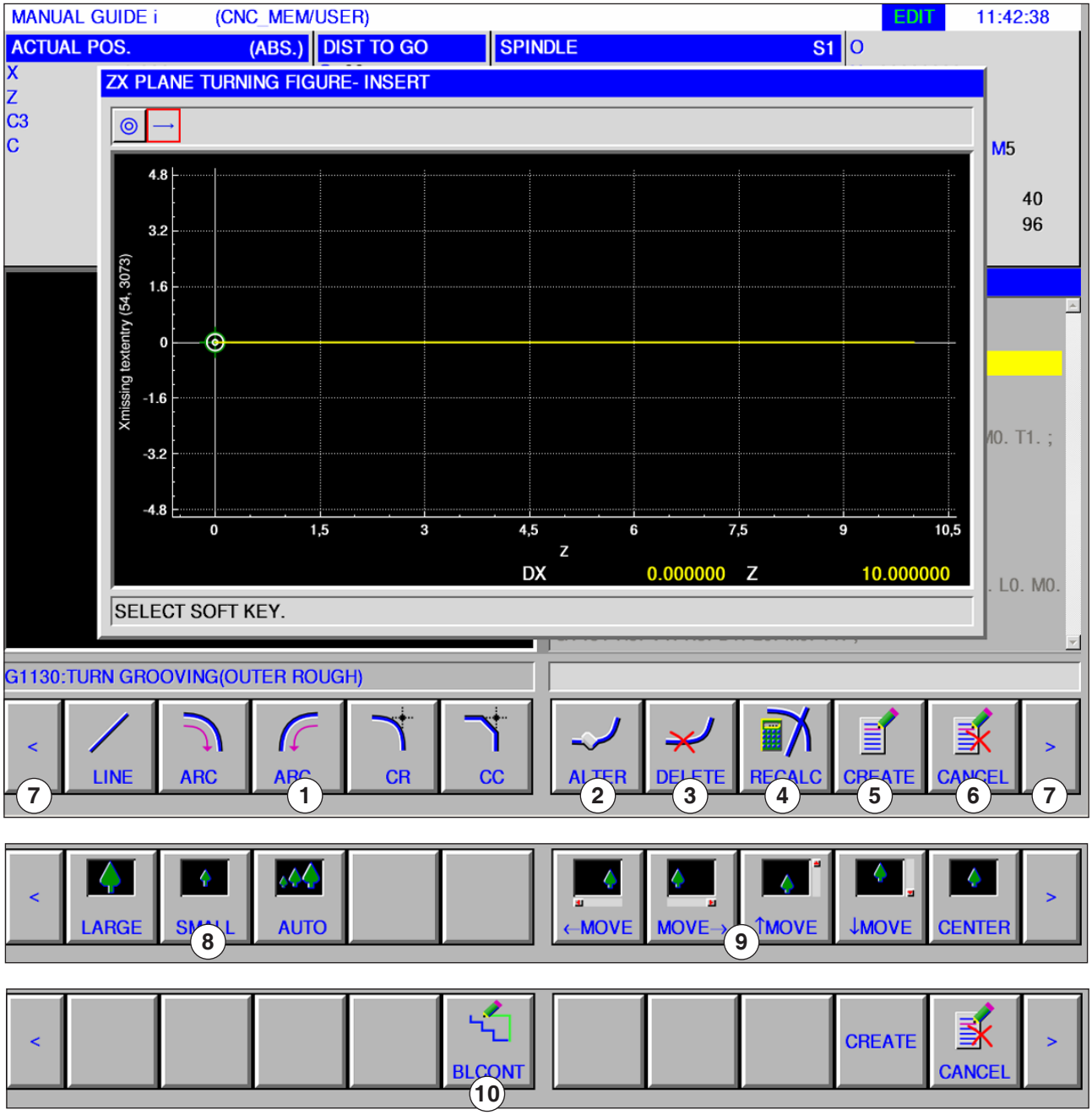


Start point insert		
Data item		**Meaning**
DX	Start point DX	X-coordinates of the contour startpoint.
Z	Start point Z	Z-coordinates of the contour startpoint.

Instructions:
INPUT DATA are elements that are displayed when changing or editing in the data input window.
This applies for cycles G1451 to G1456.

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 G1451 to G1456.

Overview of the input elements for free contour programming



1 Input elements: Line, arc, radius, chamfer

2 Change contour element

3 Delete contour element

4 Recalculate contour data

5 Generate contour

6 Cancel input

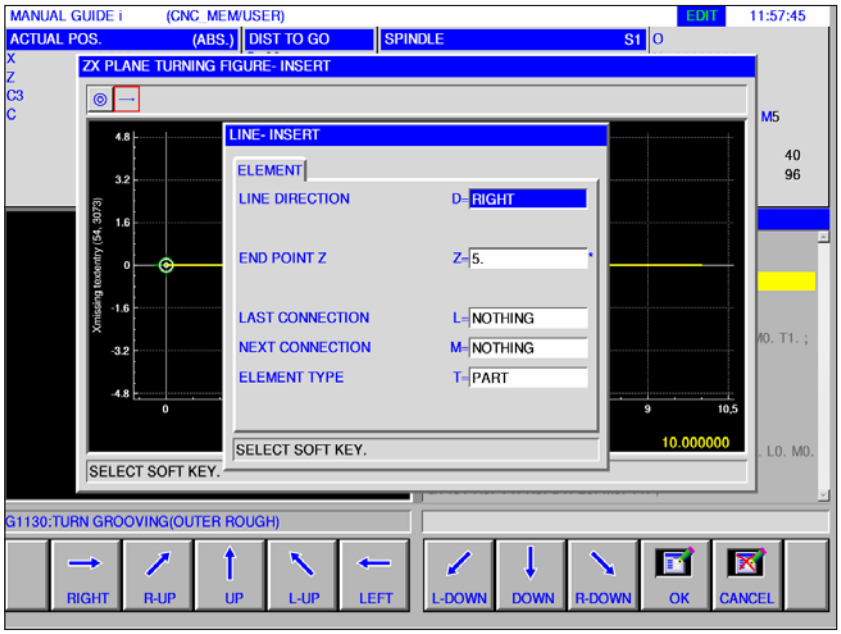
7 Expansion buttons

8 Zoom graphic display in and out

9 Move graphic display

10 Unmachined element connection (closing contours)

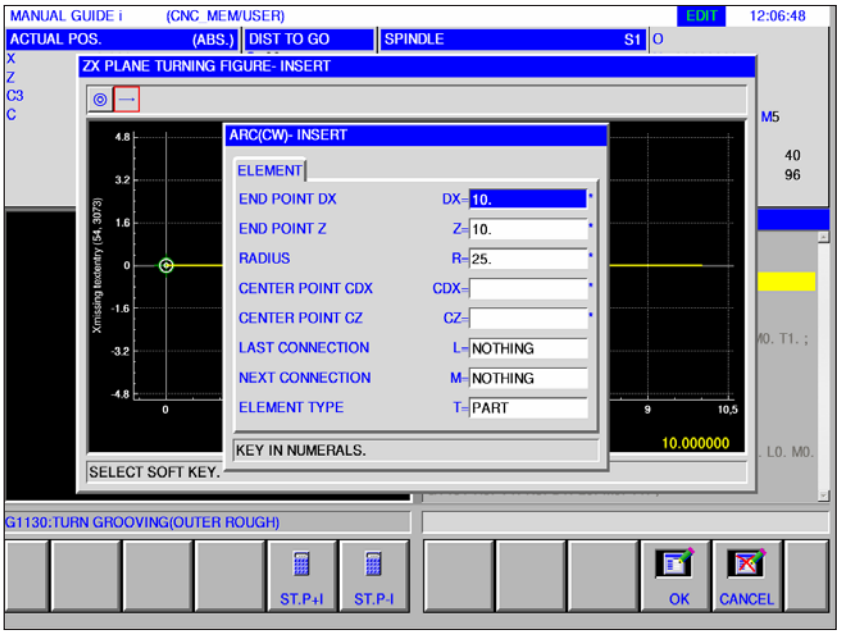
Input items of line (ZX plane) G1451



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

Line insert		
Element		**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]:
DX*	End point DX	X-coordinates of the line end point.
Z*	End point Z	Z-coordinates of the line end point.
A*	Angle	Angle of the straight line
L	Last connection	• [TANGNT]: Contact with the immediately preceding contour. • [N SET]: No contact with the immediately preceding contour (initial value).
M	Next connection	• [N SET]:No contact with the immediately following contour (initial value).
T	Element type	• [PART]: Machines the contour on the workpiece. • [BLANK]: Elements that are required so that the contour can be closed.

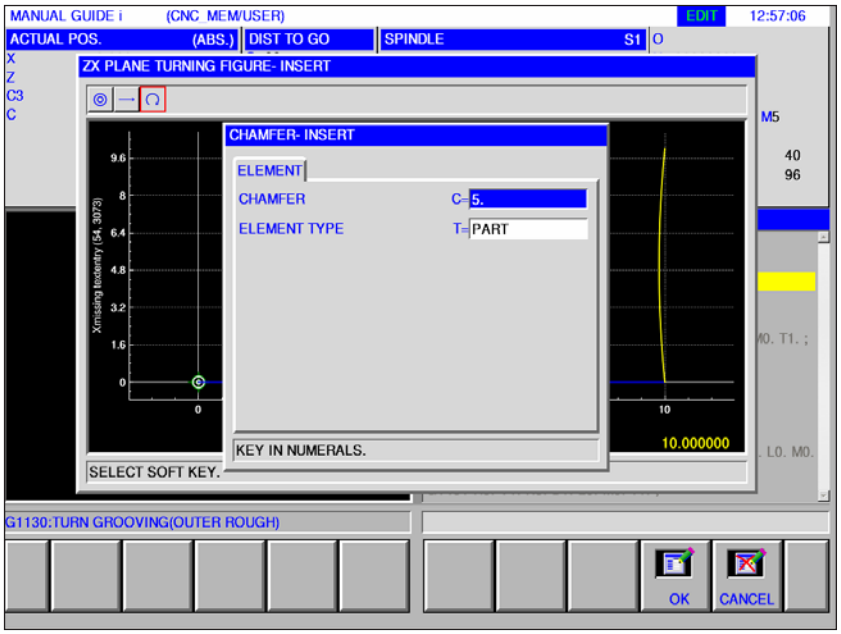
Input items of arc (ZX plane) G1452, 1453



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

Arc insert		
Element		**Meaning**
DX*	End point DX	X-coordinates of an arc endpoint Incremental dimension programming is possible.
Z*	End point Z	Z-coordinates of an arc endpoint Incremental dimension programming is possible.
R*	Radius	Radius of curvature
CDX*	Centre point CDX	X-coordinates of the centre of the arc
CZ*	Centre point CZ	Z-coordinates of the centre of the arc
L	Last connection	• [TANGNT]: Contact with the immediately preceding contour. • [N SET]: No contact with the immediately preceding contour.
M	Next connection	• [N SET]:No contact with the immediately following contour (initial value).
T	Element type	• [PART]: Machines the contour on the workpiece. • [BLANK]: Elements that are required so that the contour can be closed.

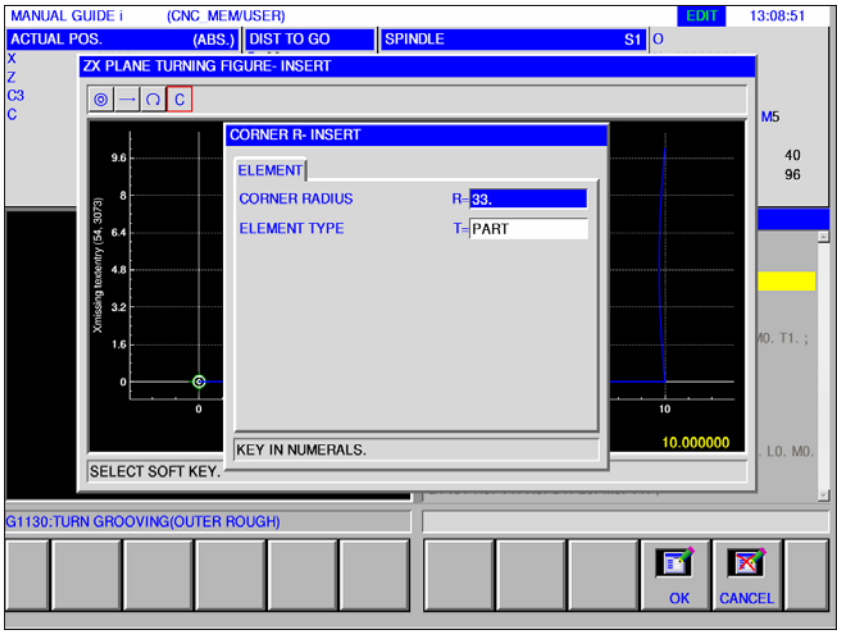
Input items of chamfer (ZX plane) G1454



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

Element		Meaning
C	Chamfer	Chamfer, radius value, positive value.
T	Element type	• [PART]: Machines the contour on the workpiece. • [BLANK]: Elements that are required so that the contour can be closed.

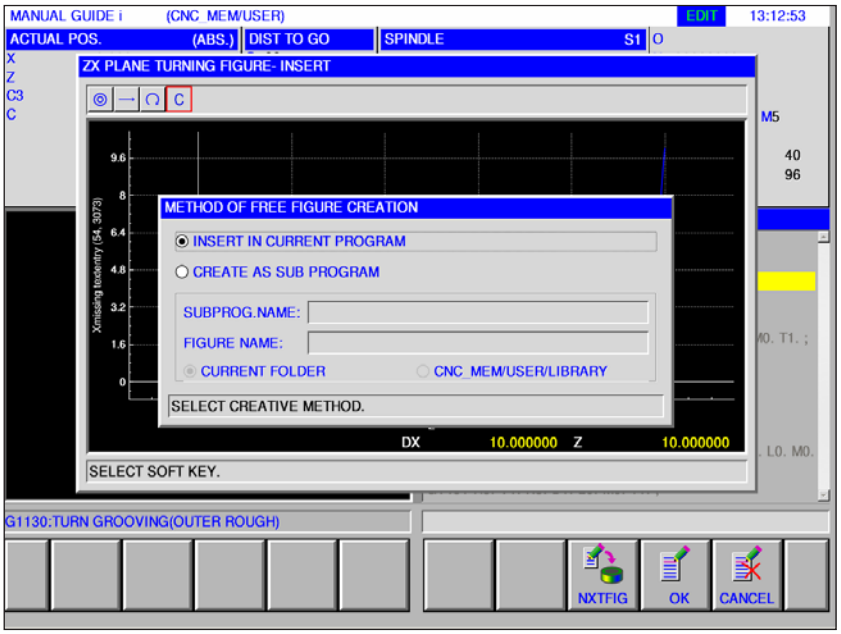
Input items of corner (ZX plane) G1455



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

Element		Meaning
R	Corner radius	Corner rounding, radius value, positive value.
T	Element type	• [PART]: Machines the contour on the workpiece. • [BLANK]: Elements that are required so that the contour can be closed.

Input items of end of arbitrary figures (ZX plane) G1456





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.

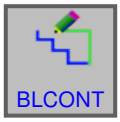


- Enter the next figure
  This softkey opens the contour editor input mask. Other free contours can be created.



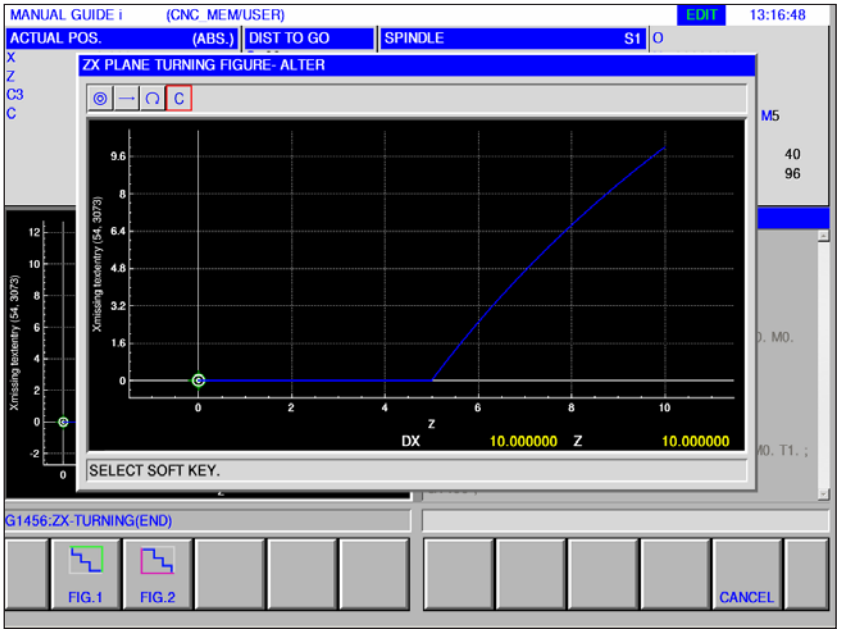
- Close input with "OK".

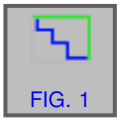
Unmachined part element connection: Closing a contour



Close contour

- Press softkey.

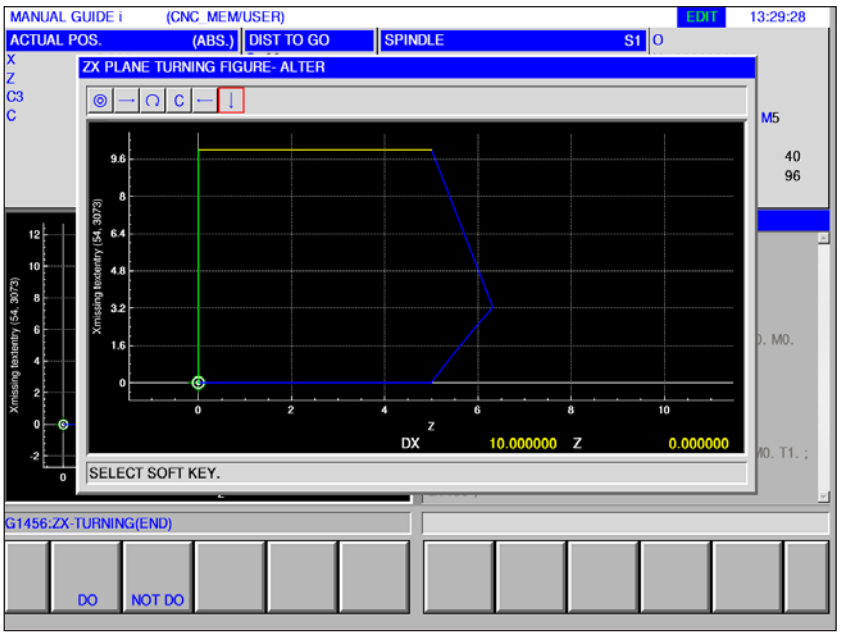


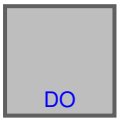


- "FIG 1": Outer surface machining

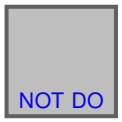


- "FIG 2": Inner surface machining





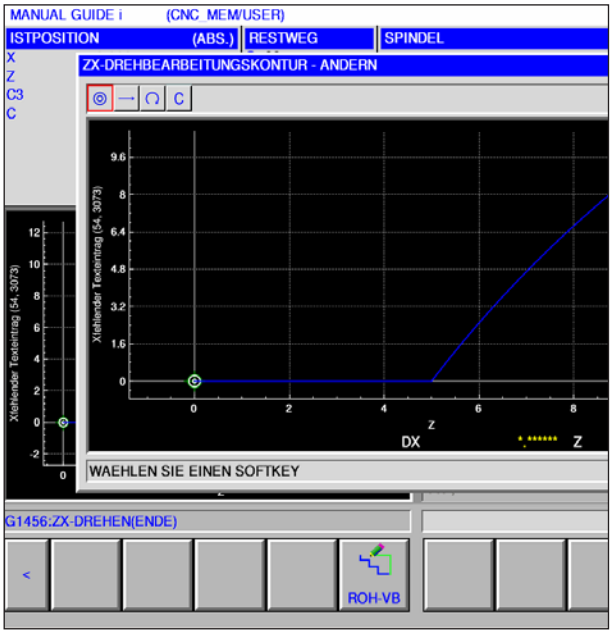
- "DO" connects the unmachined part contour and the screen display returns to the input screen for free contours.

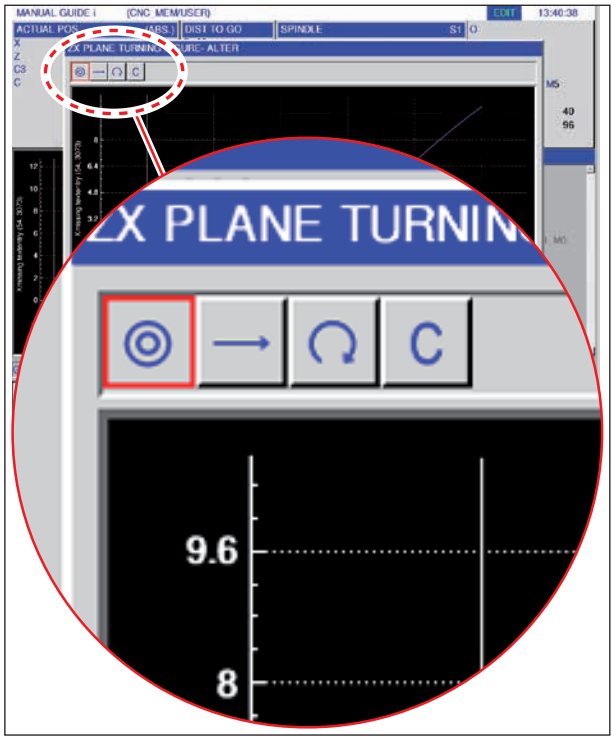


- Cancel and return to the input screen for free contours.

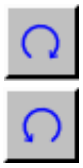
Information:

- If a free contour is already closed and the [BLCONT] softkey is pressed, the message "CONTOUR IS ALREADY CLOSED" is displayed and the input screen for free contours is displayed again.
- If only one startpoint is defined, the message "ELEMENT DEFINITION DATA MISSING" is displayed. If the endpoint is not determined, the message "END POINT OF THE SELECTED CONTOUR UNDEFINED" is displayed.
- One of the unmachined part contour lines for the connection connects the endpoint with the startpoint of a parts contour parallel to the Z or X axis.
  If one of the lines overlaps with a defined element, the defined contour is displayed in front.





Symbol representation of the contour elements

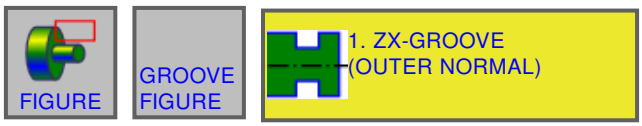
Contour element	Symbol	Meaning
Start point		Start point of the figure
Straight up Straight down		Straight line in 90° grid
Straight left Straight right		Straight line in 90° grid
Straight line anywhere		Straight line with arbitrary pitch
Curve right Curve left		Arc
Radius		
Chamfer		

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

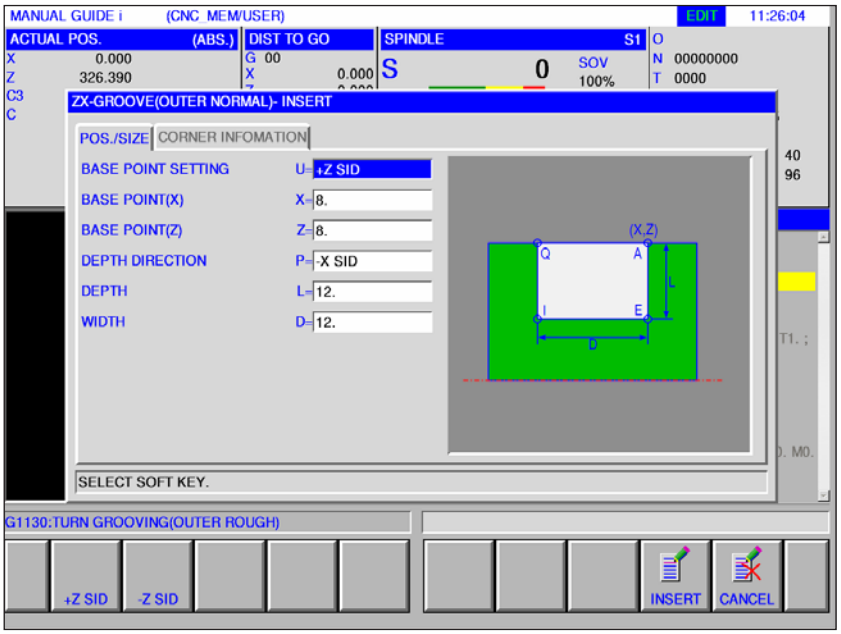


Groove Contour

- Standard Outside Groove G1470
- Trapezoidal Outside Groove G1471
- Standard Inside Groove G1472
- Trapezoidal Inside Groove G1473
- Standard Groove Front Surface G1474
- Trapezoidal Groove Face G1475
- Contour Groove G1456



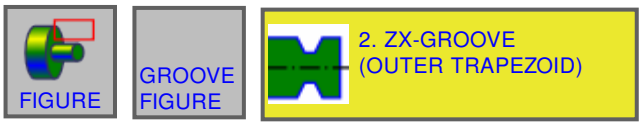
ZX-Groove Outer Normal G1470



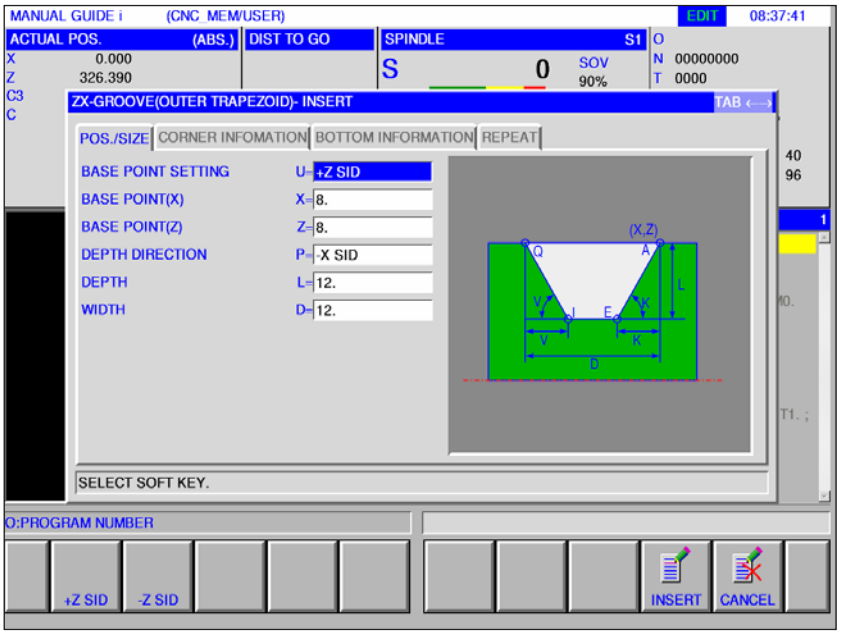
Entry fields marked with * are optional and are not required to be populated.

Cutting condition		
Data item		**Meaning**
U	Base point setting	• [+Z SID]: Sets the base point in the +Z direction. (initial value). • [-Z SID]: Sets the base point in the -Z direction.
X	Base point X	X-coordinate of the groove reference point.
Z	Base point Z	Z-coordinate of the groove reference point.
P	Depth direction	Cut depth in the X-axis • [-X]: Delivery takes place in the -X direction. • [+X]: Delivery takes place in the +X direction.
L	Depth	Groove depth (radius value, positive value).
D	Width	Groove width (radius value, positive value).

Corner information		
Data item		**Meaning**
A	Corner type-1	For reference point corner (1) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
B	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-1.
E	Corner type-2	For corner (2) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
F	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-2.
I	Corner type-3	For corner (3) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
J	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-3.
Q	Corner type-4	For corner (4) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
R	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-4.



ZX-Trapezoidal Outside Groove G1471



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

Position / Size		
Data element		**Meaning**
U	Initial position	• [+Z SID]: Sets the base point in the +Z direction. (Initial value). • [-Z SID]: Sets the base point in the -Z direction.
X	Base point X	X-coordinate of the groove reference point.
Z	Base point Z	Z-coordinate of the groove reference point.
P	Infeed direction	Cut depth in the X-axis • [-X]: Infeed takes place in the -X direction. • [+X]: Infeed takes place in the +X direction.
L	Depth (Reference base)	Groove depth (radius value, positive value).
D	Width	Groove width (radius value, positive value).

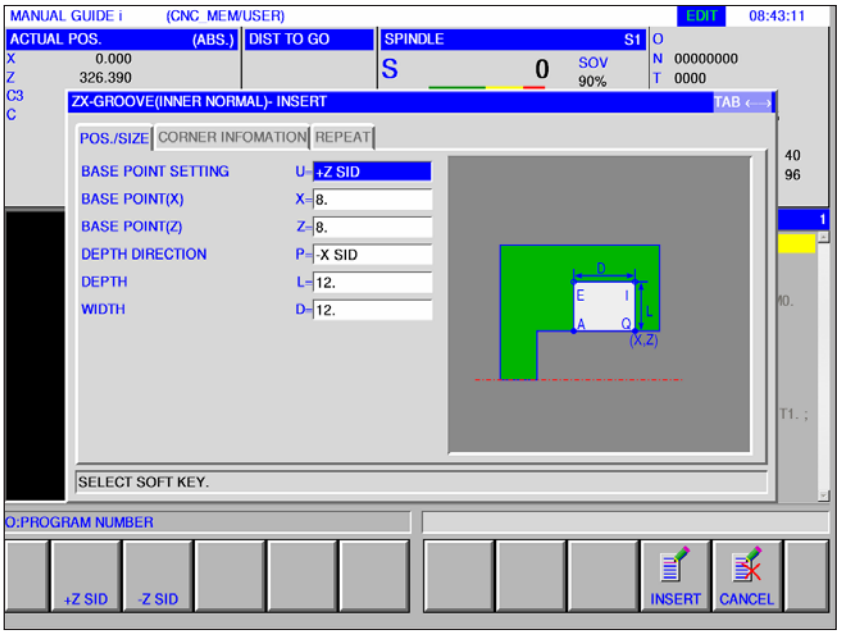
Corner shape		
Data element		**Meaning**
A	Corner type-1	For reference point corner (1) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
B	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-1.
E	Corner type-2	For corner (2) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
F	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-2.
I	Corner type-3	For corner (3) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
J	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-3.
Q	Corner type-4	For corner (4) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
R	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-4.

Bottom shape		
Data element		**Meaning**
H	Bottom type	• [WIDTH] : Specifies the difference between the starting width between the side surfaces of the groove and the width of the groove at the bottom (initial value, if type 1 is selected). • [ANGLE] : Specifies the side surface angle of the groove (initial value, if type 2 is selected).
K*	Bottom size / angle	Difference between the beginning of the side surface of the groove on the reference point side and the width of the groove bottom, if [WIDTH] is specified (radius value, positive value). If [ANGLE] is specified, this element specifies the inclination angle of the side surface on the reference point side toward the X-axis direction (positive value). If for the entry type [C] and the base type [H] [TYP1] and [WIDTH], [TYP2] and [ANGLE] or [TYP2] and [WIDTH] is set respectively, the standard width is 0, if WIDTH is specified, or the standard angle is 90, if ANGLE is specified. If [TYP1] and [ANGLE] are specified, the standard angle is 90.
V*	Bottom size / angle	Difference between the beginning of the side surface of the groove on the side opposite to the reference point and the width of the groove bottom, if [WIDTH] is specified (radius value, positive value). Inclination angle of the side surface opposite to the reference point, if [ANGLE] is specified (positive value). If for the entry type [C] and the base type [H] [TYP1] and [WIDTH], [TYP2] and [ANGLE] or [TYP2] and [WIDTH] is set respectively, the standard width is 0, if WIDTH is specified. If [TYP1] and [ANGLE] are specified, the standard angle is 90.

Repetition		
Data element		**Meaning**
M	Number of grooves	Number of grooves to be machined that have the same contour. The unmachined part is viewed as 1 degree. (positive value)
S*	Division	Distance between the reference points of two consecutive grooves (radius value, positive value)
W	Division direction	• [-Z] : Positions the second and the following grooves in -Z-direction (initial value) • [+Z] : Positions the second and the following grooves in +Z-direction.



ZX-Standard Inside Groove G1472



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

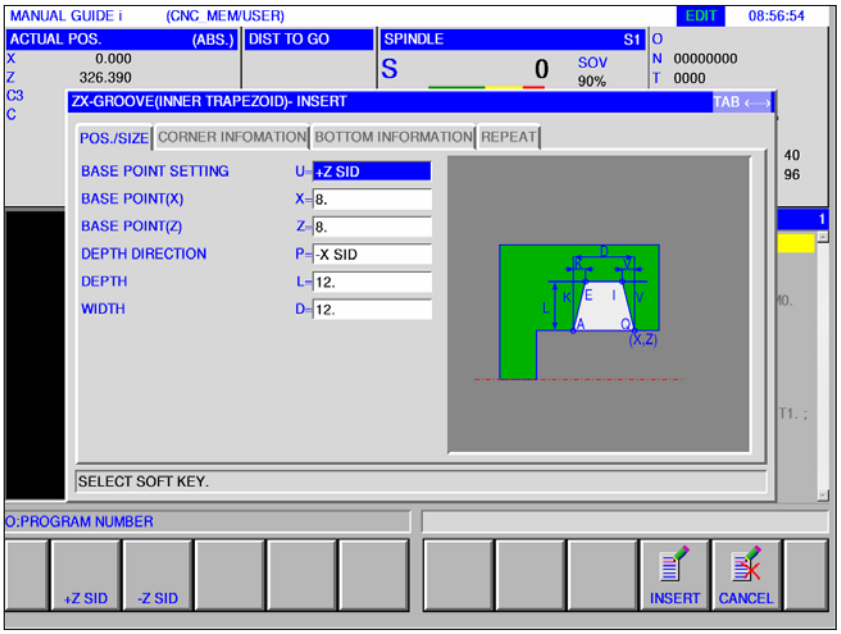
Position / Size		
Data element		**Meaning**
U	Initial position	• [+Z SID]: Sets the base point in the +Z direction. (Initial value). • [-Z SID]: Sets the base point in the -Z direction.
X	Base point X	X-coordinate of the groove reference point.
Z	Base point Z	Z-coordinate of the groove reference point.
P	Infeed direction	Cut depth in the X-axis • [-X]: Infeed takes place in the -X direction. • [+X]: Infeed takes place in the +X direction.
L	Depth (Reference base)	Groove depth (radius value, positive value).
D	Width	Groove width (radius value, positive value).

Corner shape		
Data element		**Meaning**
A	Corner type-1	For reference point corner (1) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
B	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-1.
E	Corner type-2	For corner (2) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
F	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-2.
I	Corner type-3	For corner (3) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
J	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-3.
Q	Corner type-4	For corner (4) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
R	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-4.

Repetition		
Data element		**Meaning**
M	Number of grooves	Number of grooves to be machined that have the same contour. The unmachined part is viewed as 1 degree. (positive value)
S*	Division	Distance between the reference points of two consecutive grooves (radius value, positive value)
W	Division direction	• [-Z] : Positions the second and the following grooves in -Z-direction (initial value) • [+Z] : Positions the second and the following grooves in +Z-direction.



ZX-Trapezoidal Inside Groove G1473



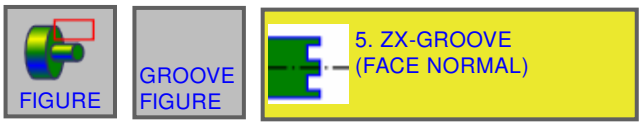
Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
U	Initial position	• [+Z SID]: Sets the base point in the +Z direction. (Initial value). • [-Z SID]: Sets the base point in the -Z direction.
X	Base point X	X-coordinate of the groove reference point.
Z	Base point Z	Z-coordinate of the groove reference point.
P	Infeed direction	Cut depth in the X-axis • [-X]: Infeed takes place in the -X direction. • [+X]: Infeed takes place in the +X direction.
L	Depth (Reference base)	Groove depth (radius value, positive value).
D	Width	Groove width (radius value, positive value).

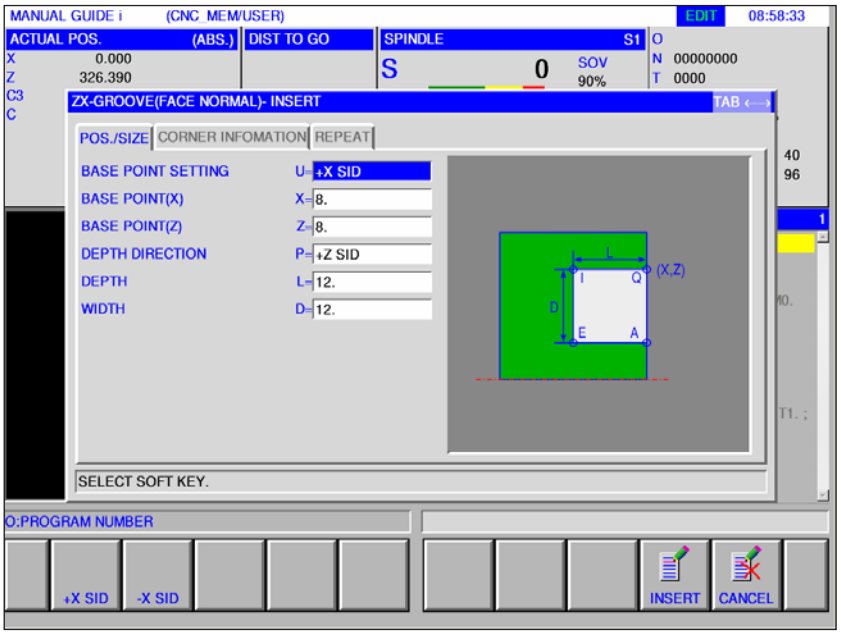
Corner shape		
Data element		**Meaning**
A	Corner type-1	For reference point corner (1) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
B	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-1.
E	Corner type-2	For corner (2) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
F	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-2.
I	Corner type-3	For corner (3) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
J	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-3.
Q	Corner type-4	For corner (4) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
R	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-4.

Bottom shape		
Data element		**Meaning**
H	Bottom type	• [WIDTH] : Specifies the difference between the starting width between the side surfaces of the groove and the width of the groove at the bottom (initial value, if type 1 is selected). • [ANGLE] : Specifies the side surface angle of the groove (initial value, if type 2 is selected).
K*	Bottom size / angle	Difference between the beginning of the side surface of the groove on the reference point side and the width of the groove bottom, if [WIDTH] is specified (radius value, positive value). If [ANGLE] is specified, this element specifies the inclination angle of the side surface on the reference point side toward the X-axis direction (positive value). If for the entry type [C] and the base type [H] [TYP1] and [WIDTH], [TYP2] and [ANGLE] or [TYP2] and [WIDTH] is set respectively, the standard width is 0, if WIDTH is specified, or the standard angle is 90, if ANGLE is specified. If [TYP1] and [ANGLE] are specified, the standard angle is 90.
V*	Bottom size / angle	Difference between the beginning of the side surface of the groove on the side opposite to the reference point and the width of the groove bottom, if [WIDTH] is specified (radius value, positive value). Inclination angle of the side surface opposite to the reference point, if [ANGLE] is specified (positive value). If for the entry type [C] and the base type [H] [TYP1] and [WIDTH], [TYP2] and [ANGLE] or [TYP2] and [WIDTH] is set respectively, the standard width is 0, if WIDTH is specified. If [TYP1] and [ANGLE] are specified, the standard angle is 90.

Repetition		
Data element		**Meaning**
M	Number of grooves	Number of grooves to be machined that have the same contour. The unmachined part is viewed as 1 degree. (positive value)
S*	Division	Distance between the reference points of two consecutive grooves (radius value, positive value)
W	Division direction	• [-Z] : Positions the second and the following grooves in -Z-direction (initial value) • [+Z] : Positions the second and the following grooves in +Z-direction.



ZX-Standard Face Groove G1474

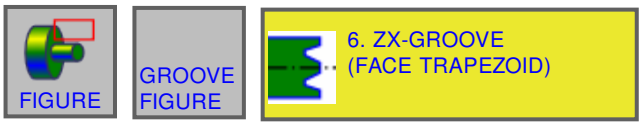


Entry fields marked with * are optional and are not required to be populated.

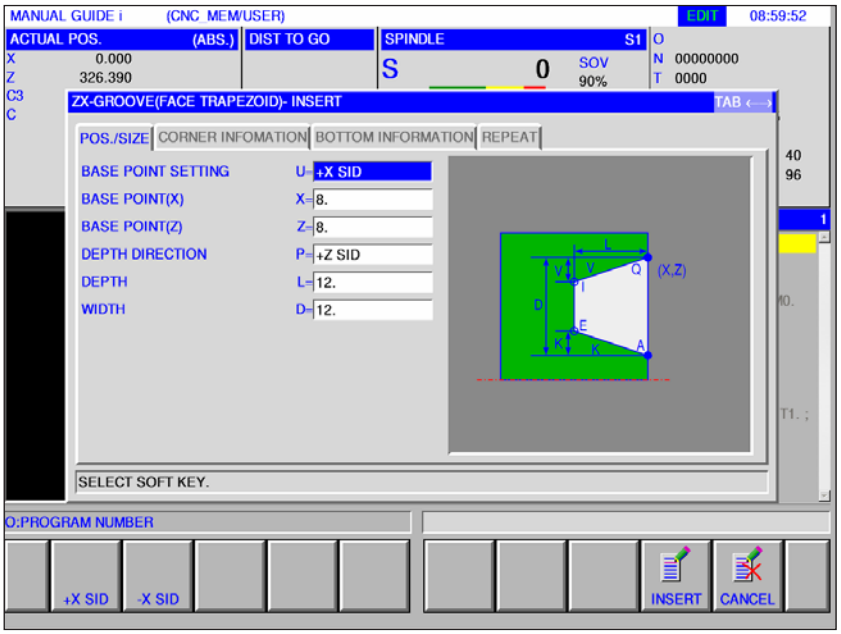
Position / Size		
Data element		**Meaning**
U	Initial position	• [+Z SID]: Sets the base point in the +Z direction. (Initial value). • [-Z SID]: Sets the base point in the -Z direction.
X	Base point X	X-coordinate of the groove reference point.
Z	Base point Z	Z-coordinate of the groove reference point.
P	Infeed direction	Cut depth in the X-axis • [-X]: Infeed takes place in the -X direction. • [+X]: Infeed takes place in the +X direction.
L	Depth (Reference base)	Groove depth (radius value, positive value).
D	Width	Groove width (radius value, positive value).

Corner shape		
Data element		**Meaning**
A	Corner type-1	For reference point corner (1) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
B	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-1.
E	Corner type-2	For corner (2) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
F	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-2.
I	Corner type-3	For corner (3) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
J	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-3.
Q	Corner type-4	For corner (4) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
R	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-4.

Repetition		
Data element		**Meaning**
M	Number of grooves	Number of grooves to be machined that have the same contour. The unmachined part is viewed as 1 degree. (positive value)
S*	Division	Distance between the reference points of two consecutive grooves (radius value, positive value)
W	Division direction	• [-Z] : Positions the second and the following grooves in -Z-direction (initial value) • [+Z] : Positions the second and the following grooves in +Z-direction.



ZX-Trapezoidal Groove Face G1475



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

Position / Size		
Data element		**Meaning**
U	Initial position	• [+Z SEI] : Sets the base point in the +Z direction. (Initial value). • [-Z SEI] : Sets the base point in the -Z direction.
X	Base point X	X-coordinate of the groove reference point.
Z	Base point Z	Z-coordinate of the groove reference point.
P	Infeed direction	Cut depth in the X-axis • [-X]: Infeed takes place in the -X direction. • [+X]: Infeed takes place in the +X direction.
L	Depth (Reference base)	Groove depth (radius value, positive value).
D	Width	Groove width (radius value, positive value).

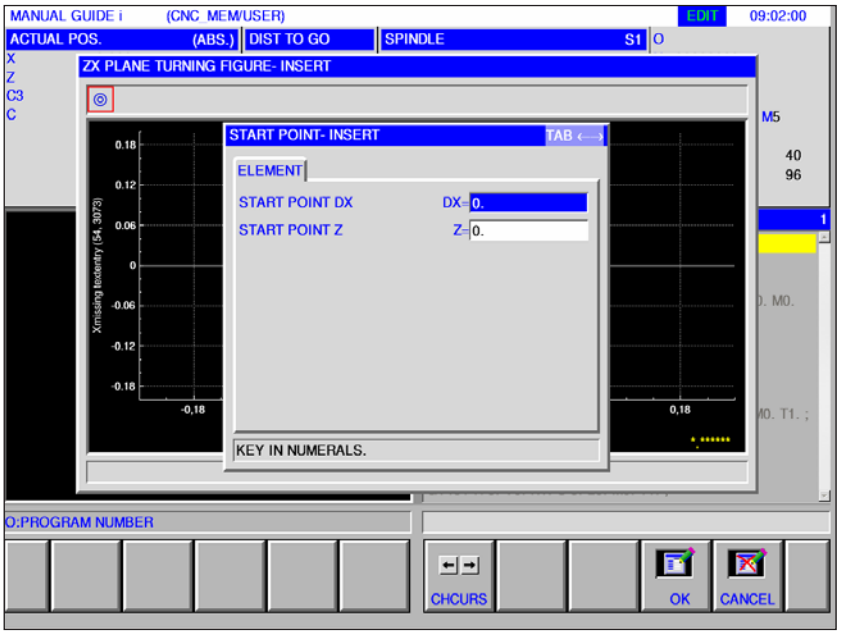
Corner shape		
Data element		**Meaning**
A	Corner type-1	For reference point corner (1) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
B	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-1.
E	Corner type-2	For corner (2) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
F	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-2.
I	Corner type-3	For corner (3) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
J	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-3.
Q	Corner type-4	For corner (4) • [NOTHIN]: Stipulates neither chamfer nor corner rounding (initial value). • [CHAMFER]: Chamfer • [ARC]: Corner rounding
R	Corner size	Chamfer amount or corner radius (radius value, positive value). This data element is given only if [CHAMFER] or [ARC] is stipulated for CORNER TYPE-4.

Bottom shape		
Data element		**Meaning**
H	Bottom type	• [WIDTH] : Specifies the difference between the starting width between the side surfaces of the groove and the width of the groove at the bottom (initial value, if type 1 is selected). • [ANGLE] : Specifies the side surface angle of the groove (initial value, if type 2 is selected).
K*	Bottom size / angle	Difference between the beginning of the side surface of the groove on the reference point side and the width of the groove bottom, if [WIDTH] is specified (radius value, positive value). If [ANGLE] is specified, this element specifies the inclination angle of the side surface on the reference point side toward the X-axis direction (positive value). If for the entry type [C] and the base type [H] [TYP1] and [WIDTH], [TYP2] and [ANGLE] or [TYP2] and [WIDTH] is set respectively, the standard width is 0, if WIDTH is specified, or the standard angle is 90, if ANGLE is specified. If [TYP1] and [ANGLE] are specified, the standard angle is 90.
V*	Bottom size / angle	Difference between the beginning of the side surface of the groove on the side opposite to the reference point and the width of the groove bottom, if [WIDTH] is specified (radius value, positive value). Inclination angle of the side surface opposite to the reference point, if [ANGLE] is specified (positive value). If for the entry type [C] and the base type [H] [TYP1] and [WIDTH], [TYP2] and [ANGLE] or [TYP2] and [WIDTH] is set respectively, the standard width is 0, if WIDTH is specified. If [TYP1] and [ANGLE] are specified, the standard angle is 90.

Repetition		
Data element		**Meaning**
M	Number of grooves	Number of grooves to be machined that have the same contour. The unmachined part is viewed as 1 degree. (positive value)
S*	Division	Distance between the reference points of two consecutive grooves (radius value, positive value)
W	Division direction	• [-Z] : Positions the second and the following grooves in -Z-direction (initial value) • [+Z] : Positions the second and the following grooves in +Z-direction.



ZX-Contour Groove G1456

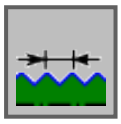


Entry fields marked with * are optional and are not required to be populated.

Insert startpoint		
Data element		**Meaning**
DX	Start point DX	X-coordinate of the contour start point.
Z	Start point Z	Z-coordinate of the contour start point.

Note:
INPUT DATA are elements that are displayed when changing or editing in the data input window.
This applies for cycles G1451 to G1456.

Note:
OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can activate these only for program display purposes.
This applies also for cycles G1451 to G1456.

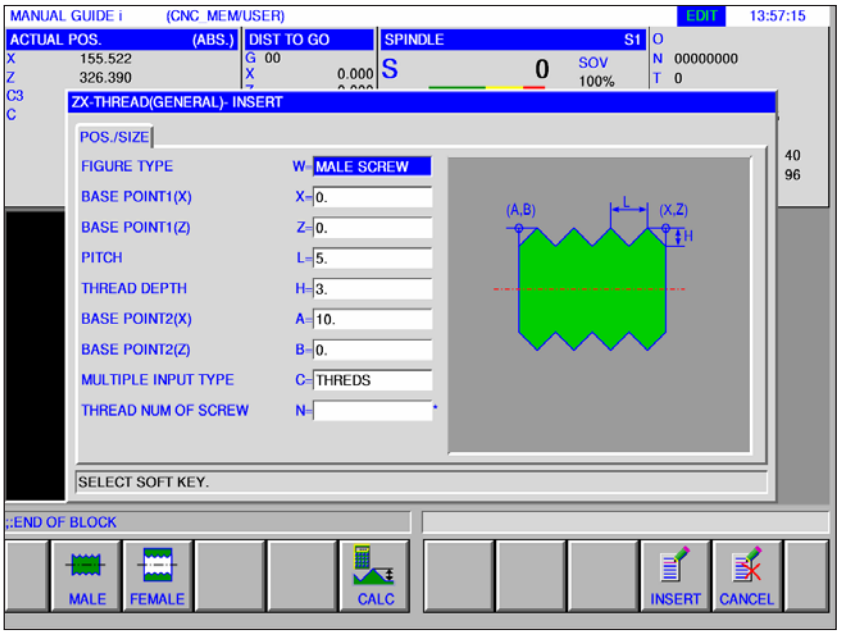


Thread Contour

- Thread General G1460
- Thread Metric G1461
- Thread Imperial G1462
- Thread Pipe G G1463
- Thread Pipe R G1464



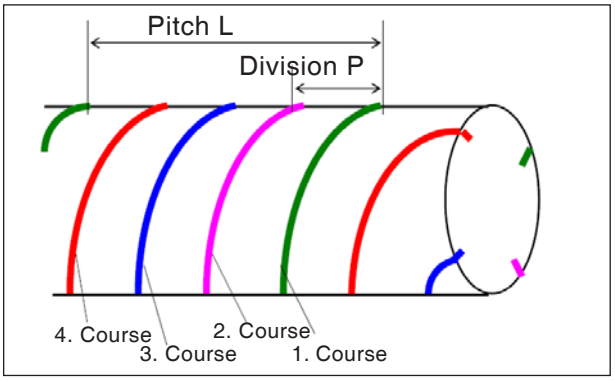
Thread General G1460



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

Position / Size		
Data item		**Meaning**
W	Figure type	• [MALE]: If "Outside thread" is stated for the thread type • [FEMALE] If "Inside thread" is stated for the thread type.
X	Base point 1 (X)	X-coordinates of reference point 1
Z	Base point 1 (Z)	Z-coordinates of reference point 1
L	Pitch	Thread pitch (radius value, positive value)
H	Thread depth	Thread depth (radius value, positive value). The thread depth is calculated automatically after entering the pitch by pressing the softkey [CALC].
A	Base point 2 (X)	X-coordinates of reference point 2
B	Base point 2 (Z)	Z-coordinates of reference point 2
C	Multiple input type	• [THRDS]: Specifies a multiple thread by the number of courses (start value). • [PITCH]: Specifies a multiple thread by the pitch.
N*	Thread num of screw	Number of courses that are to be cut per pitch. [N] can only be entered if [THRDS] is selected. The number of courses N, the pitch L and the division P is patterned as follows: N=L/P

Position / Size		
Data item		**Meaning**
P	Pitch	The number of courses N is calculated automatically from the entered pitch L and the pitch P: N=L/P. This parameter is available only if, for MEHRF-GEW. [PITCH] is stated.



Multiple thread

General for multiple thread

If "MEHRF-
GEW. C" and "GEW.-GÄNGE N" or "PITCH P" are not given for a thread contour block, the following default settings apply:

- If none of the parameters is entered:
  The number of courses is set to 1.
- If "N" or "P" is entered, but not "C":
  The alarm message "no necessary address" is output.

Cycle description: multiple thread

If a multiple thread is cut, the individual courses are cut behind each other, beginning with 1. Course

If finishing is executed, cutting and finishing for the first course ends and cutting and finishing are executed for the subsequent courses.

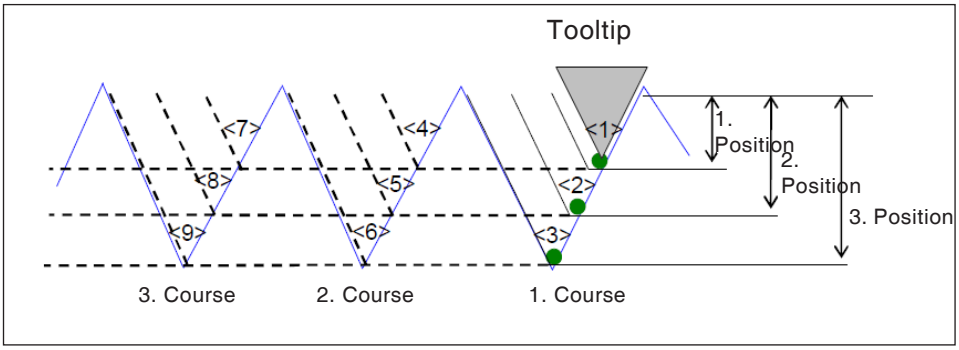
Example:

Constant cut and single-sided machining for 3 courses:

1 Cutting the first course from the first to the third position. <1><2><3>

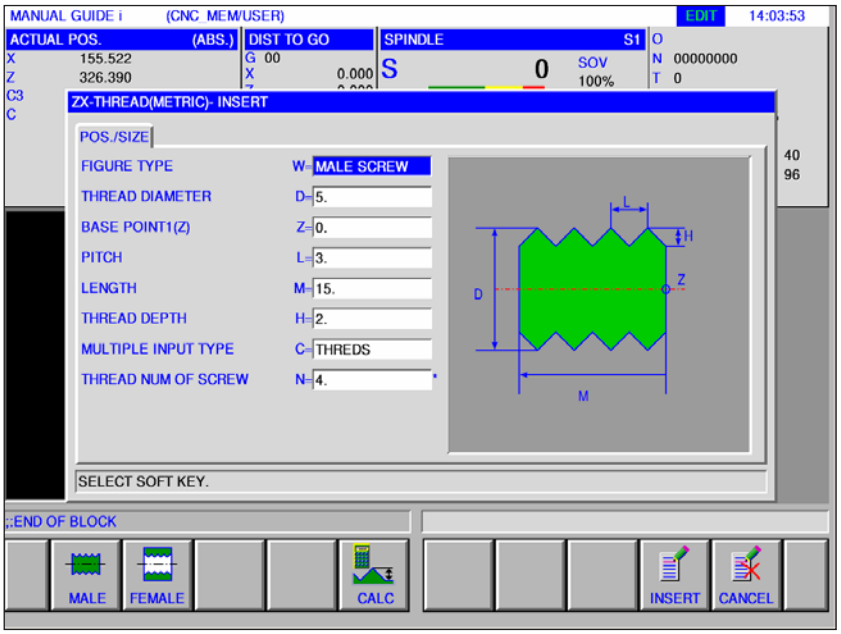
2 Cutting the first course from the first to the third position. <4><5><6>

3 Cutting the first course from the first to the third position <7><8><9>

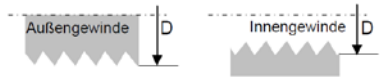




Thread Metric G1461



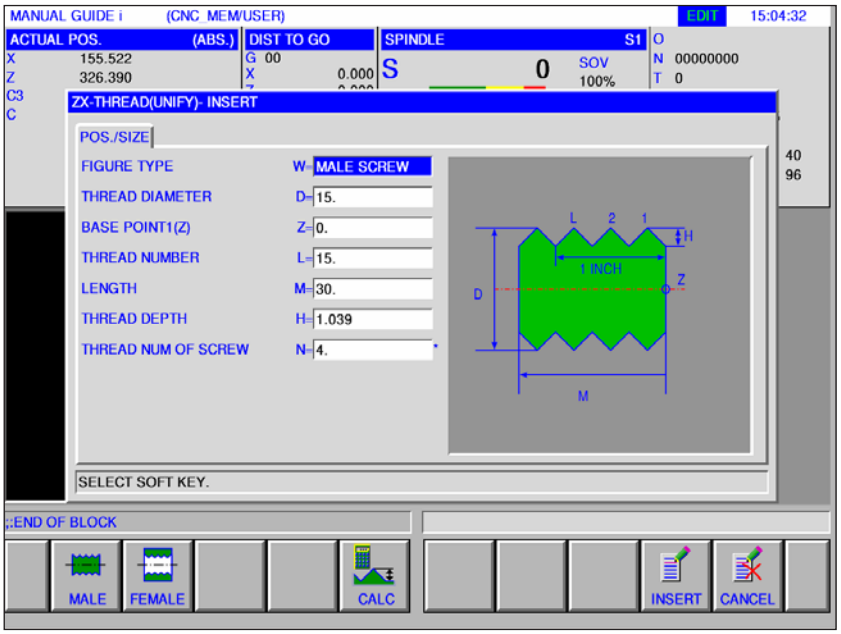
Entry fields marked with * are optional and should not be populated.

Position / Size		
Data item		**Meaning**
W	Figure type	• [MALE]: If "Outside thread" is stated for the thread type • [FEMALE]: If "Inside thread" is stated for the thread type.
D	Thread diameter	Thread diameter (positive value) 
Z	Base point 1 (Z)	Z-coordinates of reference point.
L	Pitch	Thread pitch (radius value, positive value)
M	Length	Thread length (radius value, positive value)
H	Thread depth	The thread depth is calculated automatically after entering the pitch by pressing [CALC].
C	Multiple input type	• [THRDS]: Specifies a multiple thread by the number of courses (start value). • [PITCH]: Specifies a multiple thread by the pitch.

Position / Size		
Data item		**Meaning**
N*	Thread num of screw	Number of courses that are to be cut per pitch. [N] can only be entered if [THRDS] is selected. The number of courses N, the pitch L and the division P is patterned as follows: N=L/P
P	Pitch	The number of courses N is calculated automatically from the entered pitch L and the pitch P: N=L/P. This parameter is available only if, for MEHRF-GEW. [PITCH] is stated.

FIGURE | THREAD FIGURE | 3. ZX- THREAD (UNIFY)

Thread Unify G1462



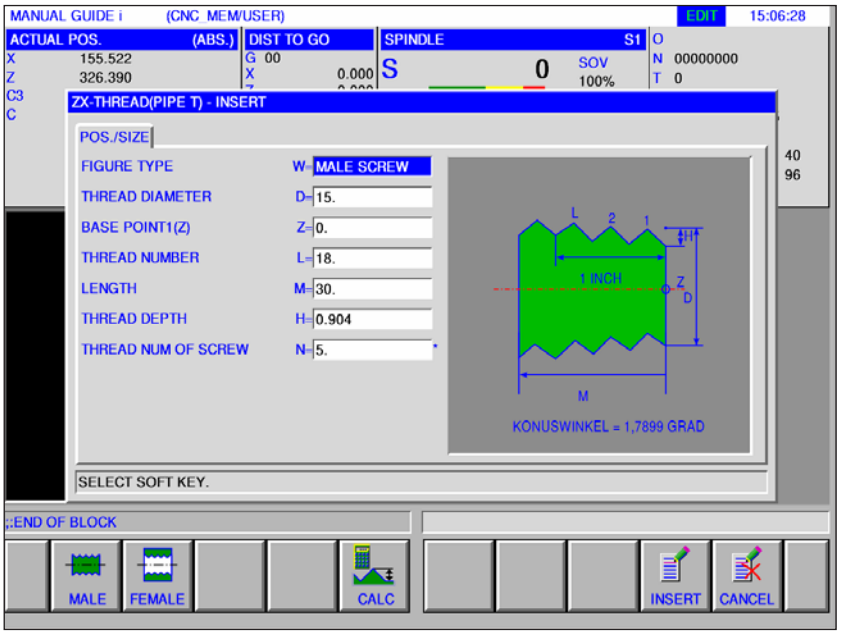
Entry fields marked with * are optional and should not be populated.

Position / Size		
Data item		**Meaning**
W	Figure type	• [MALE]: If "Outside thread" is stated for the thread type • [FEMALE]: If "Inside thread" is stated for the thread type.
D	Thread diameter	Thread diameter (positive value)
Z	Base point 1 (Z)	Z-coordinates of reference point.
L	Thread number	Number of thread courses per inch.
M	Length	Thread length (radius value, positive value)
H	Thread depth	The thread depth is calculated automatically after entering the pitch by pressing [CALC].
N*	Thread num of screw	Number of courses that are to be cut per pitch.

Instructions:
In a unify imperial thread, only one cylindrical thread is machined. With an imperial thread, the "Number of thread courses per inch" is to be given instead of the thread pitch.
The tool angle is to be set with 60 degrees.

FIGURE | THREAD FIGURE | 4. ZX- THREAD (PIPE T)

Thread Pipe T G1463



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

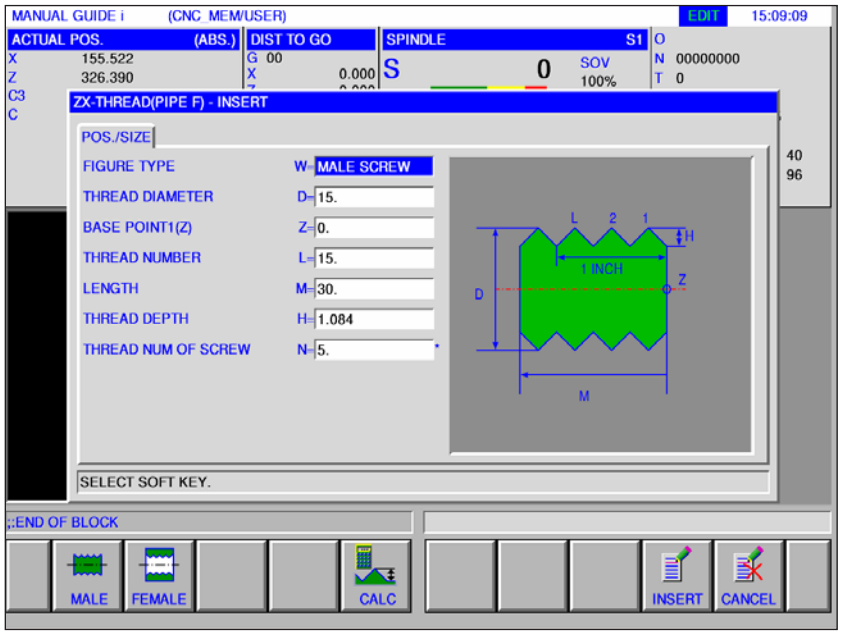
Position / Size		
Data item		**Meaning**
W	Figure type	• [MALE]: If "Outside thread" is stated for the thread type • [FEMALE]: If "Inside thread" is stated for the thread type.
D	Thread diameter	Thread diameter (positive value)
Z	Base point 1 (Z)	Z-coordinates of reference point.
L	Thread number	Number of thread courses per inch.
M	Length	Thread length (radius value, positive value)
H	Thread depth	The thread depth is calculated automatically after entering the pitch by pressing [CALC].
N*	Thread num of screw	Number of courses that are to be cut per pitch.

Notes

- With a conical pipe thread (conical thread for pipes), only one conical thread (1.7899 degrees conically) is machined. The tool angle is to be set with 55 degrees.
- The conical form of an outside thread (screw thread) must be created so that the plane surface of the unmachined part coincides with the section with the smallest diameter.
- The conical form of an inside thread (nut thread) must be created so that the plane surface of the unmachined part coincides with the section with the largest diameter.



Thread Pipe F G1464



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

Position / Size		
Data item		**Meaning**
W	Figure type	• [MALE]: If "Outside thread" is stated for the thread type • [FEMALE] If "Inside thread" is stated for the thread type.
D	Thread diameter	Thread diameter (positive value)
Z	Base point 1 (Z)	Z-coordinates of reference point.
L	Thread number	Number of thread courses per inch.
M	Length	Thread length (radius value, positive value)
H	Thread depth	The thread depth is calculated automatically after entering the pitch by pressing [CALC].
N*	Thread num of screw	Number of courses that are to be cut per pitch.

Notes
For a cylindrical pipe thread, the tool angle is given as 55 degrees.

Milling

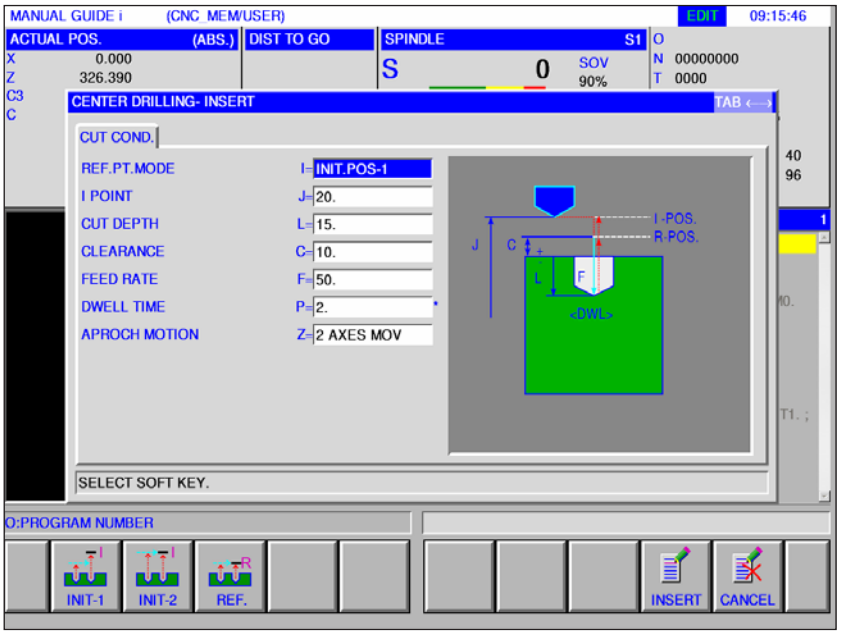
Drilling



- Centre Drilling G1110
- Drilling G1111
- Tapping G1112
- Grinding G1113
- Reaming G1114



Centre Drilling G1110

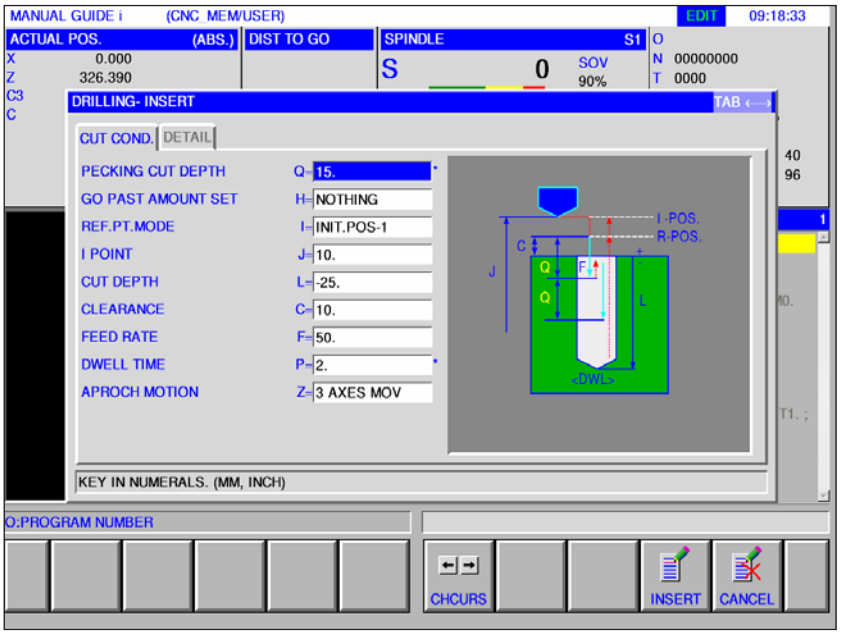


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
I	Retraction mode	• [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	Safety distance 1 NVP	Coordinate I-Point
L	Drilling depth (reference basis)	Drilling depth (radius value, negative value)
C	Safety distance (Reference base)	Distance between work piece surface and R-position (radius value, positive value).
F	Feed speed	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approaching movement	• [2 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.



Drilling G1111



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

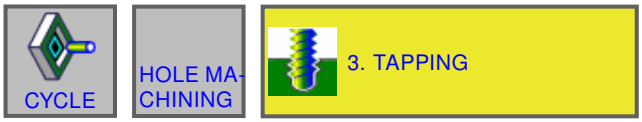
Machining Operation		
Data element		**Meaning**
Q*	Stroke depth (INCR+)	Cut depth executed per cut (radius value, positive value)
H	Path amount setting	Drilling depth settings: • [NOTHIN]: Drilling depth relative to the tool shaft • [SETTIN] : Drilling depth relative to the tool tip
I	Retraction mode	• [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	Safety distance 1 NVP	Coordinate I-Point
L	Drilling depth (reference basis)	Drilling depth (radius value, negative value)
C	Safety distance (Reference base)	Distance between work piece surface and R-position (radius value, positive value).
F	Feed speed	Feed speed (positive value).
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approaching movement	• [3 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.

Detail		
Data element		**Meaning**
A*	Start cut depth	The start feed S applies to the start cut depth A
S*	Start feed	
D*	End cut depth	The end feed E applies to the end cut depth D
E*	End feed	

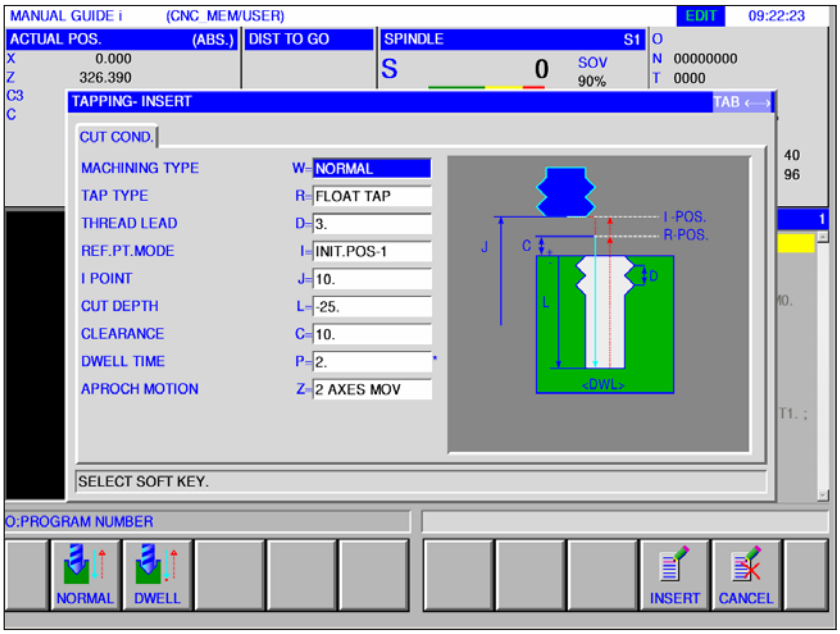


Note:
For the parameters A,S,D and E in "Detail", for all boring cycles:

All 4 parameters must be entered as soon as only one of these fields is populated.



Tapping G1112

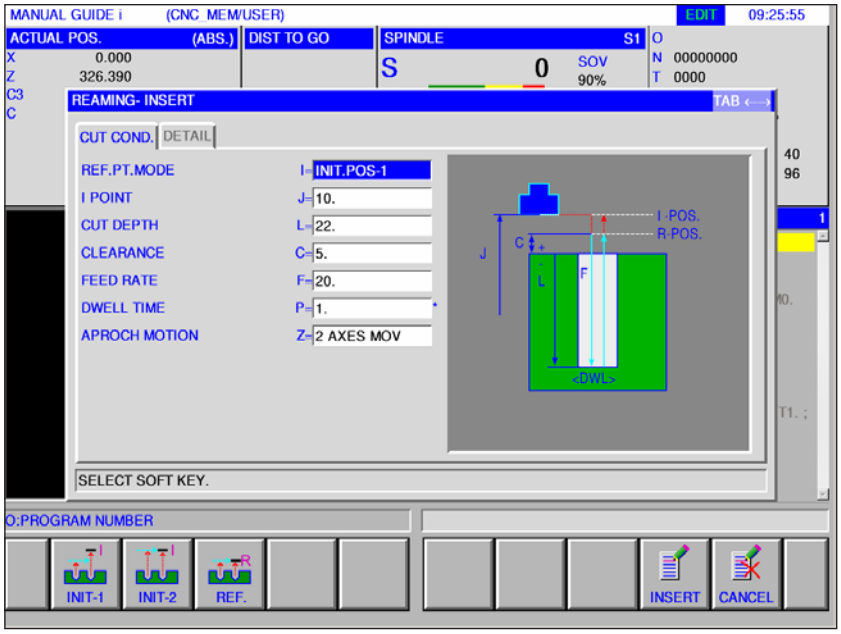


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
W	Machining type	• [NORMAL]: Normal tapping (right hand thread) • [DWELL] : Left hand thread tapping
R	Thread type	• [FLOAT]: Thread tapping with compensating chuck. • [RIGID]: Thread tapping without compensating chuck.
D	Thread pitch	Tap pitch (radius value, positive value).
I	Retraction mode	• [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	Safety distance 1 NVP	Coordinate I-Point
L	Drilling depth (reference basis)	Drilling depth (radius value, negative value)
C	Safety distance (Reference base)	Distance between work piece surface and R-position (radius value, positive value).
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approaching movement	• [2 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.

CYCLE | HOLE MACHINING | 4. REAMING

Grinding G1113

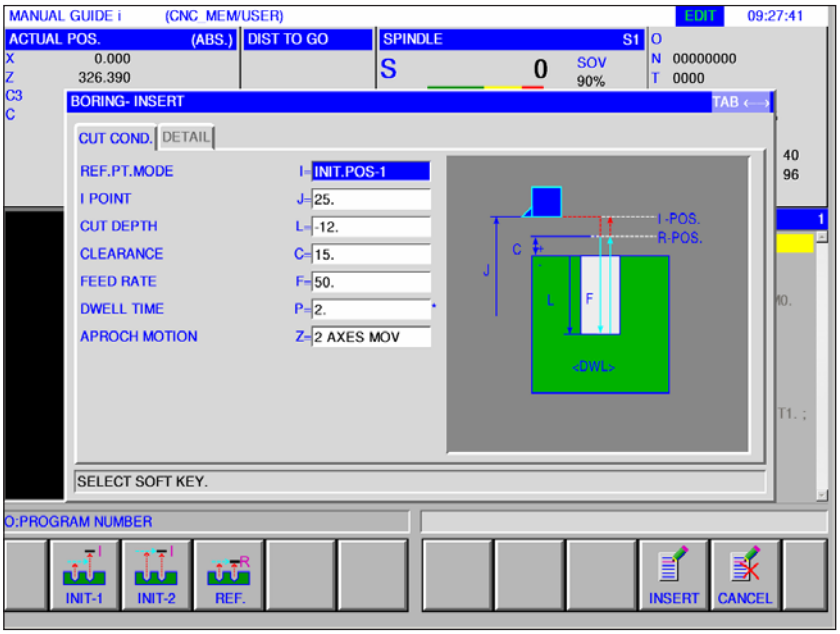


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
I	Retraction mode	• [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	Safety distance 1 NVP	Coordinate I-Point
L	Drilling depth (reference basis)	Drilling depth (radius value, negative value)
C	Safety distance	Distance between work piece surface and R-position (radius value, positive value).
F	Feed speed	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approaching movement	• [2 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be
Detail		
Data element		**Meaning**
A*	Start cut depth	The start feed S applies to the start cut depth A
S*	Start feed	
D*	End cut depth	The end feed E applies to the end cut depth D
E*	End feed	

CYCLE | HOLE MACHINING | 5. BORING

Reaming G1114



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

Machining Operation		
Data element		**Meaning**
I	Retraction mode	• [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	Safety distance 1 NVP	Coordinate I-Point
L	Drilling depth (reference basis)	Drilling depth (radius value, negative value)
C	Safety distance (Reference base)	Distance between work piece surface and R-position (radius value, positive value).
F	Feed speed	Feed speed (positive value)
P*	Dwell time	Dwell time at the bottom of the boring (in seconds, positive value).
Z	Approaching movement	• [2 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.
Detail		
Data element		**Meaning**
D*	End cut depth	The end feed E applies to the end cut depth D
E*	End feed	

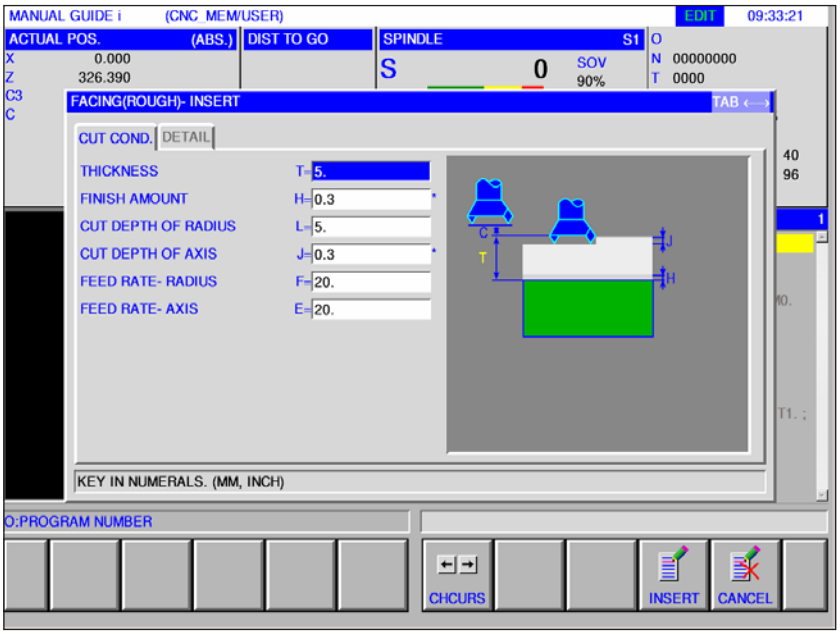


Plane Machining

- Face Milling Roughing G1020
- Face Milling Finishing G1021



Face Milling Roughing G1020



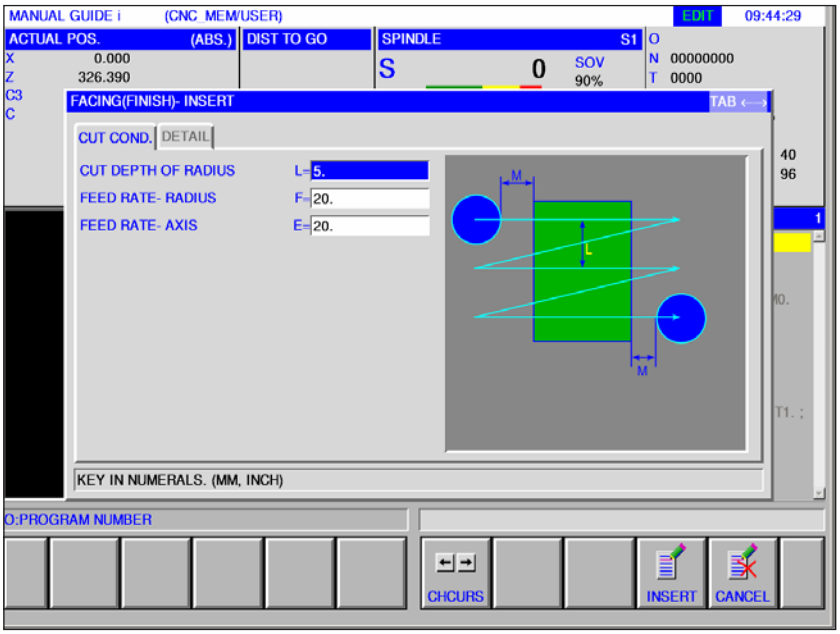
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
T	Roughing allowance	Face mill diameter
H*	Finishing allowance	Finishing allowance at face machining.
L	Infeed side	Cut depth in tool radius direction to the next cutting track.
J*	Infeed depth	Cut depth in tool axis direction per cutting process
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction

Detail		
Data element		**Meaning**
I	1. Chip removal	The feed override value for the first cut is pre-set as 100% fixed and cannot be changed.
W	Machining method	• [SINGLE]: Cutting in tool radius direction is always done in the same direction. • [ZIGZAG]: Direct approach to the start point of the next cutting track without returning to point R.
P	Track feed method	• [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	Outside feed	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	Safety distance Z	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	Safety distance XY	Distance between the end of the work piece and the end of the tool in the retraction position (radius value).
A	Machining direction	• [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. The actual cutting direction is determined by the coordinate axis, which is shown in the sketch.
B	Machining offset direction	• [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. The actual cutting direction is determined by the coordinate axis, which is shown in the sketch.
Z	Approaching movement	• [3 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.



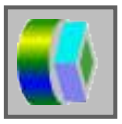
Face Milling Finishing G1021



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

Machining Operation		
Data element		**Meaning**
L	Infeed side	Cut depth in tool radius direction to the next cutting track.
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction

Detail		
Data element		**Meaning**
W	Machining method	• [SINGLE]: Cutting in tool radius direction is always done in the same direction. • [ZIGZAG]: Direct approach to the start point of the next cutting track without returning to point R.
P	Track feed method	• [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	Outside feed	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	Safety distance Z	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	Safety distance XY	Distance between the end of the work piece and the end of the tool in the retraction position (radius value).
A	Machining direction	• [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. The actual cutting direction is determined by the coordinate axis, which is shown in the sketch.
B	Machining offset direction	• [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. The actual cutting direction is determined by the coordinate axis, which is shown in the sketch.
Z	Approaching movement	• [3 AXIS]: The tool traverses from the current position to the machining start point simultaneously in the Z-axis direction and in the X-axis direction. The position is pre-set as fixed and cannot be changed.

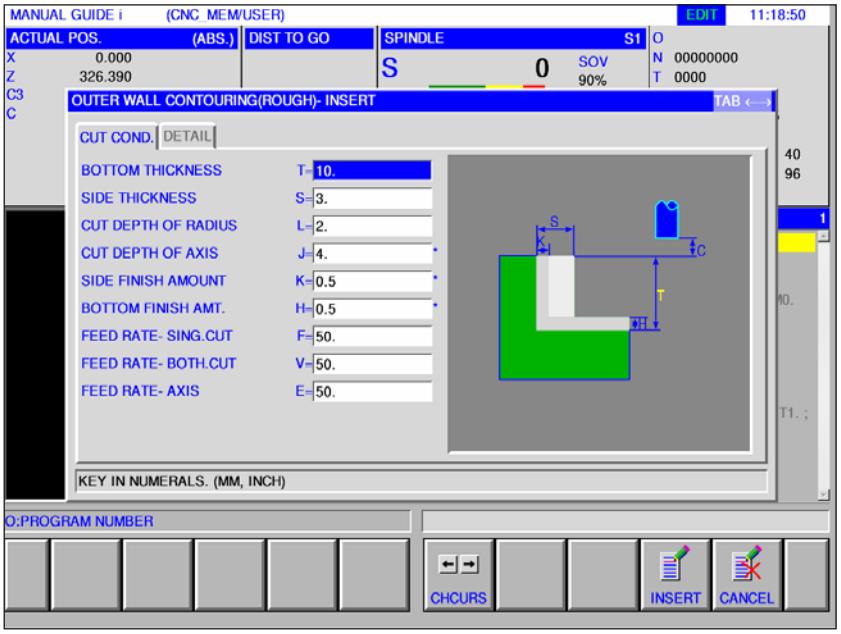


Contour Machining

- Outer Wall Roughing G1060
- Outer Wall Finishing G1061
- Outer Wall Side Finishing G1062
- Outer Wall Chamfer G1063
- Inner Wall Roughing G1064
- Inner Wall Finishing G1065
- Inner Wall Side Finishing G1066
- Inner Wall Chamfer G1067
- Partial Machining Roughing G1068
- Partial Machining Finishing G1069
- Partial Machining Side Finishing G1070
- Partial Machining Chamfer G1071



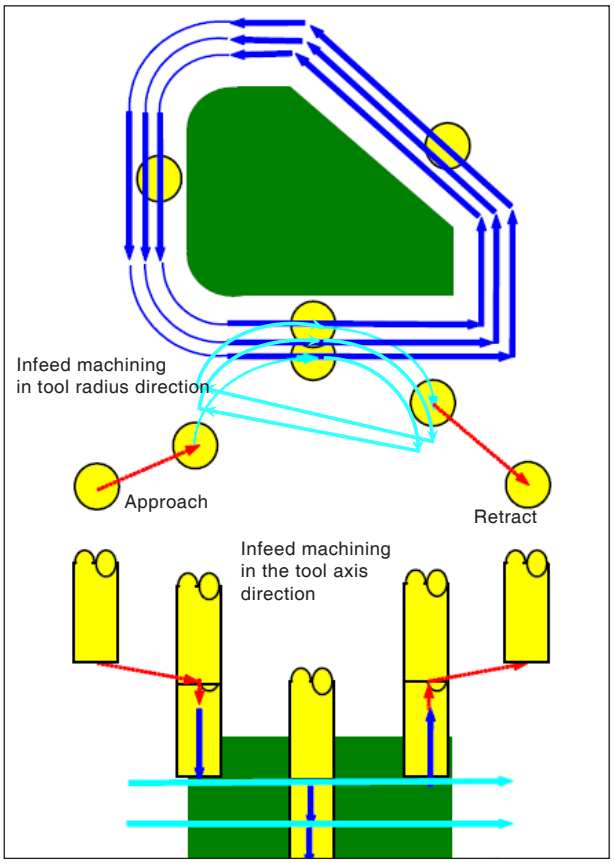
Outer Wall (Roughing) G1060



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

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
L	Infeed side	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
J*	Infeed depth	Cut depth in tool axis direction per cutting process (radius value, positive value). Standard is (Machining allowance at bottom - Finishing allowance at bottom).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
M	1. Chip removal	The feed override value for the first cut is pre-set as 100% fixed and cannot be changed.
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current 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 removed. The following tool track is generated.

1 The tool is moved to the upper approach start point.

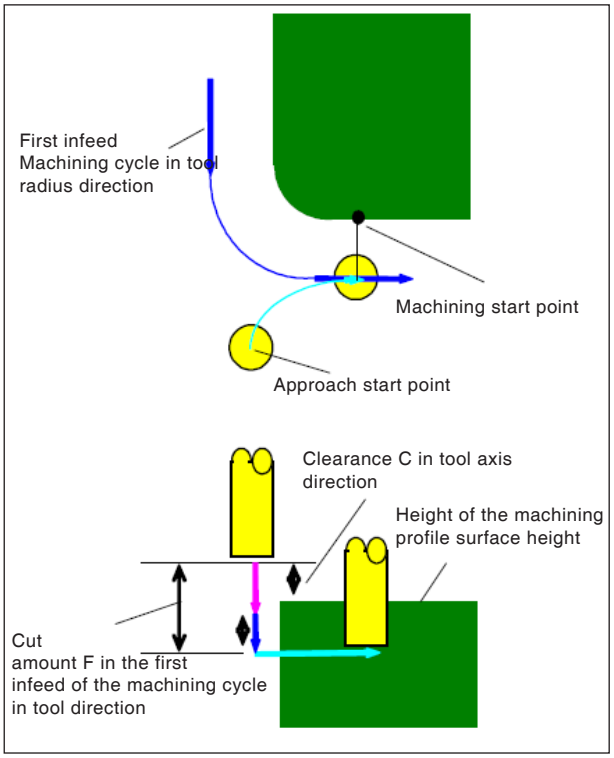
2 The tool is moved to the height of the machining surface.

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 removed.

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

5 The tool retracts.



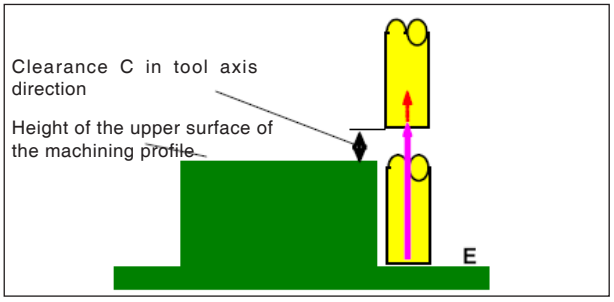
Approach G1060, G1064, G1068

Approach cycle description:

1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".

2 The tool traverses to 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.

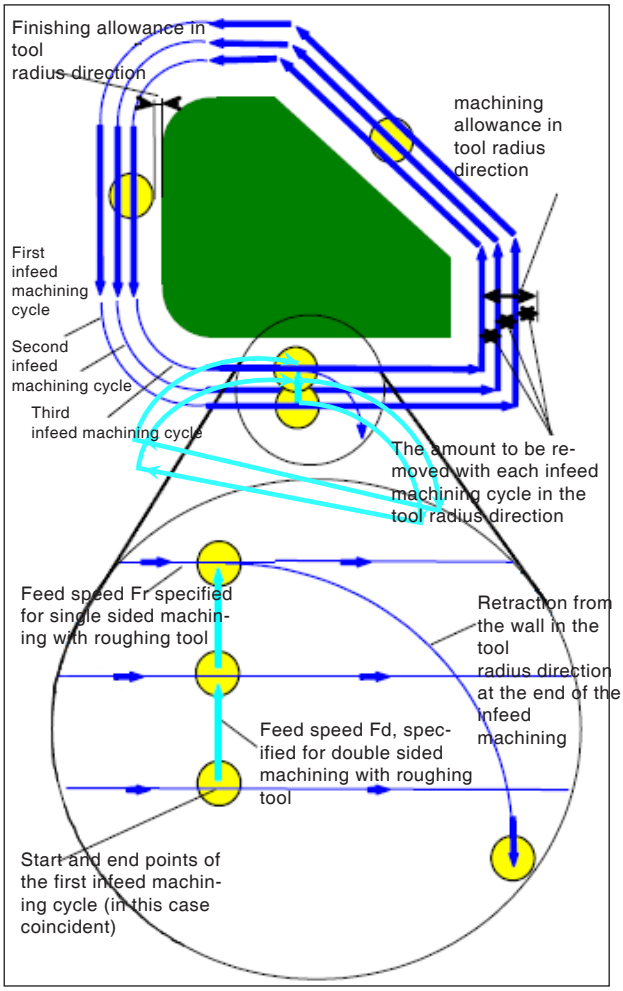
3 The tool approaches in the tool radius direction the start point for the first infeed machining cycle in the tool radius direction.



Retraction G1060, G1064, G1068

Retraction cycle description:

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



Infeed machining in 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.

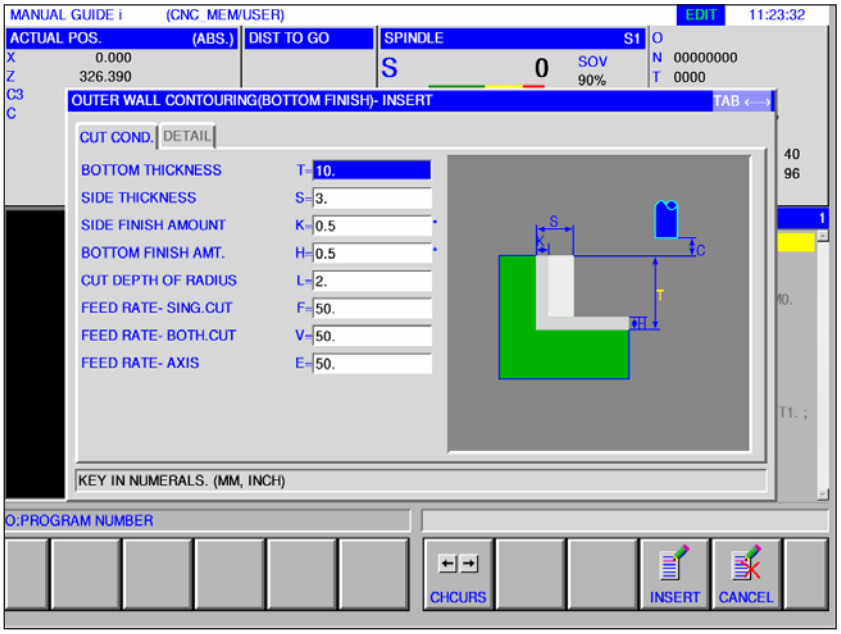
3 The tool is moved for cutting along the machining profile contour with the feed speed (F) specified for single side machining with roughing tool.

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

5 The tool retracts.



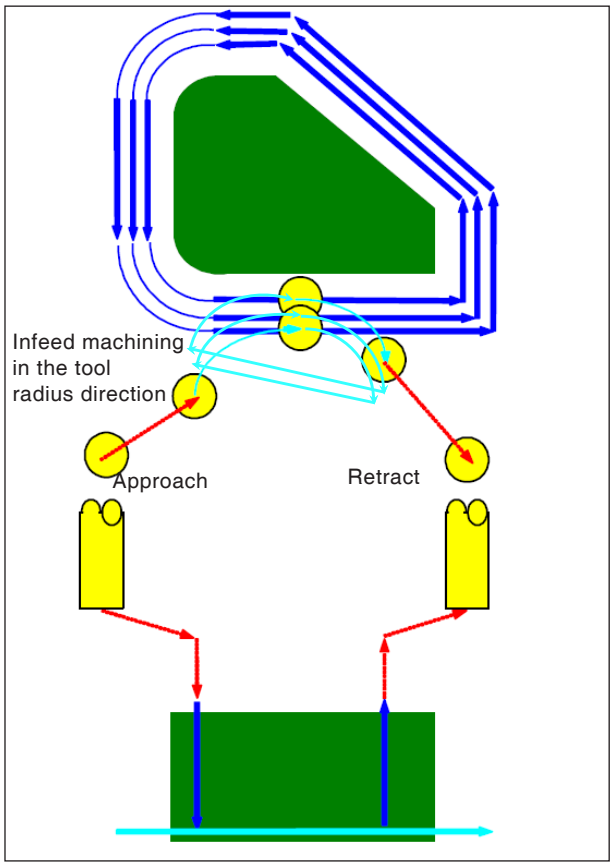
Outer Wall (Z-Finishing) G1061



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

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
L	Infeed side	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current 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.

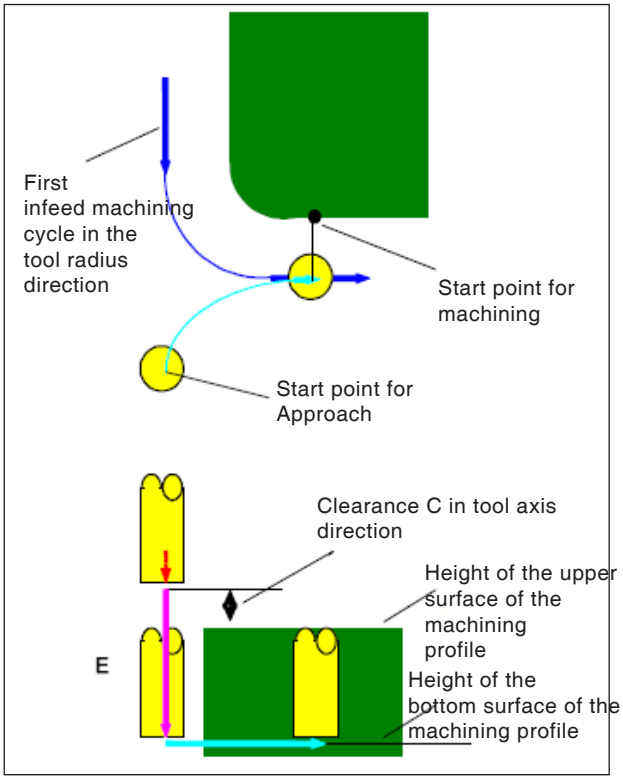
1 The tool is moved to the machining profile approach start point.

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 removed.

4 The tool retracts.



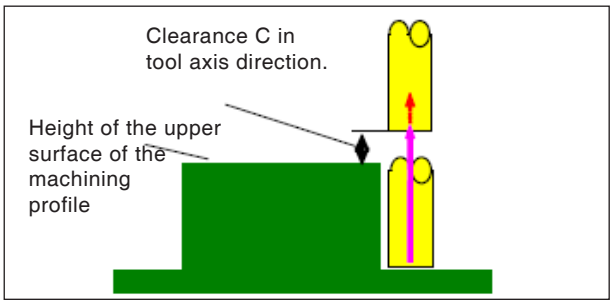
Approach G1061, G1065, G1069

Approach cycle description:

1 The tool moves in rapid traverse into the position "Height of the upper surface of the machining profile + clearance (C) in the tool axis direction".

2 The tool traverses into the position "Machining profile bottom surface + machining allowance (Vt) in the tool axis direction" with the feed speed (E) specified for movement in the tool axis direction.

3 The tool approaches in the tool radius direction the start point for the infeed machining in the tool radius direction.

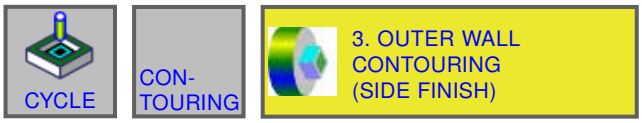


Retraction track G1061, G1065, G1069

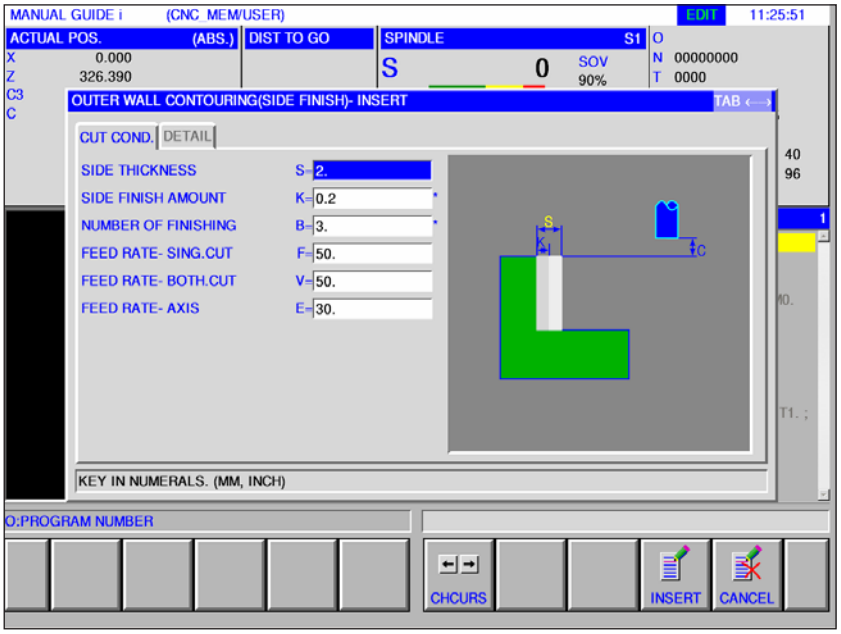
Retraction cycle description:

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

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



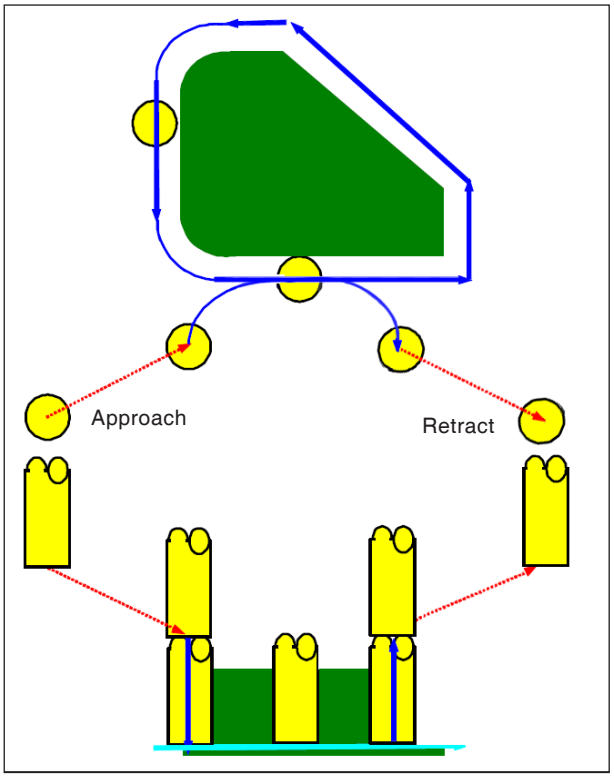
Outer Wall (Side Finishing) G1062



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

Machining Operation		
Data element		**Meaning**
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
B*	Number of finishing runs	Number of cuts during finishing (positive value) Depth per cut = (superfluous side thickness)/(number of finishing cuts)
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



Tool track G1062, G1066, G1070

Cycle description for G1062, G1066 and G1070:

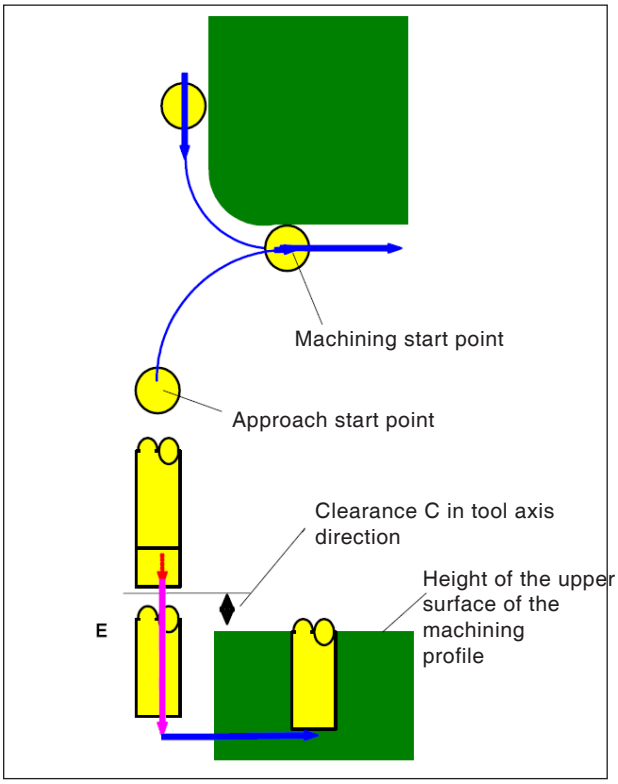
1 The tool is moved to the upper approach start point.

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 (Vt) 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

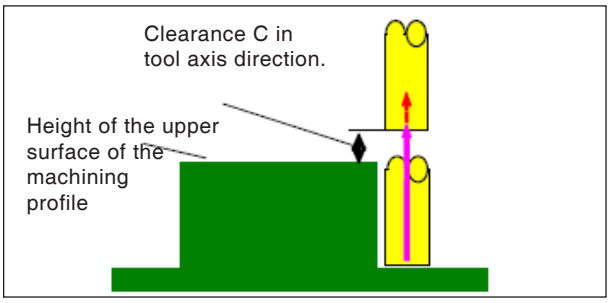
Approach cycle description:

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 allowance (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 allowance (Tt) in the tool axis direction" with the feed speed (Ft) specified for cutting the machining profile.

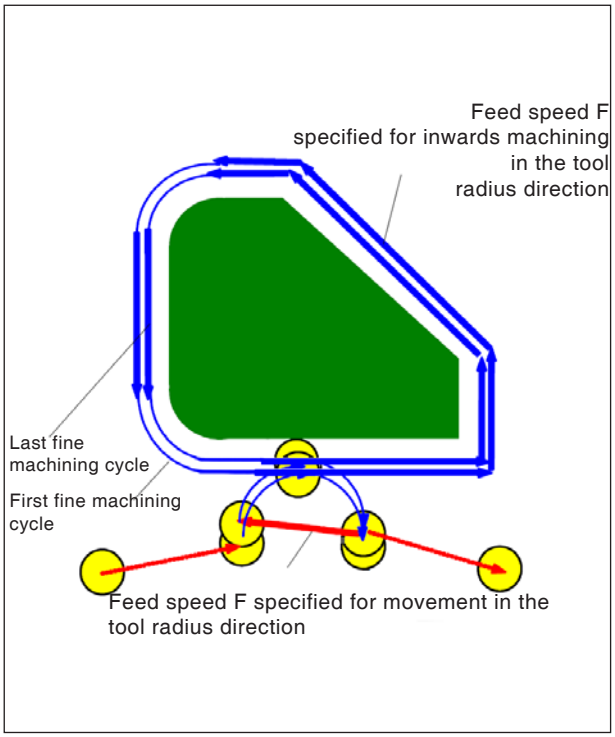
4 The tool approaches in the tool radius direction the start point for the infeed machining cycle in the tool radius direction.



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:

1 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.

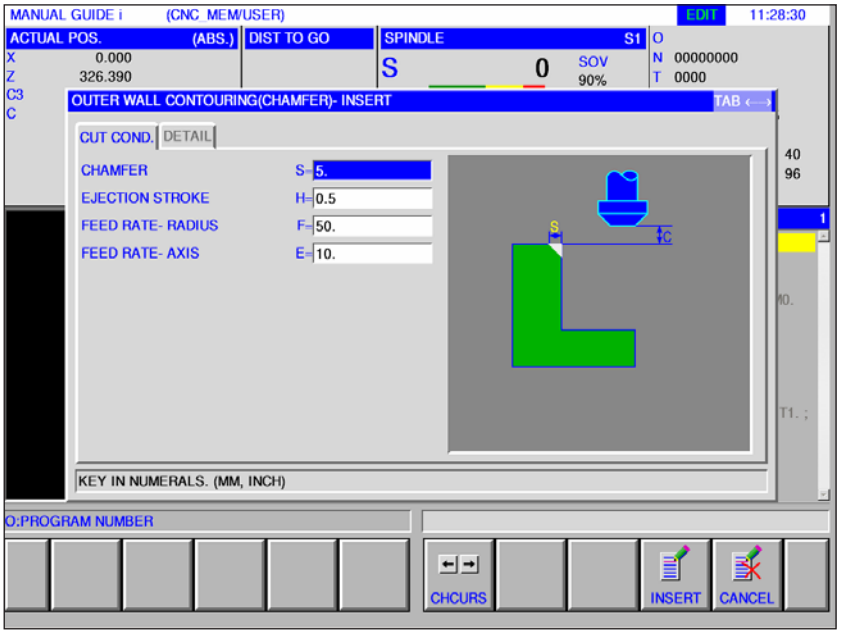
3 The tool retracts from the end point for machining in the tool radius direction with the feed speed (F) specified for cutting in the tool radius direction.

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.

CYCLE | CON-TOURING | 4. OUTER WALL CONTOURING (CHAMFER)

Outer Wall (Chamfer) G1063

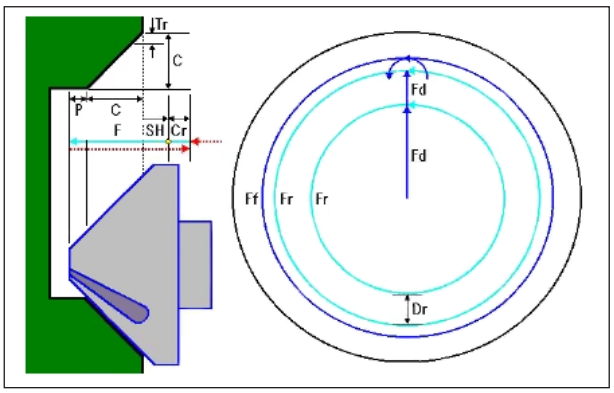


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
S	Chamfer width	Chamfer length (radius value, positive value)
H	Travelling track	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value)
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.

Detail		
Data element		**Meaning**
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current 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.

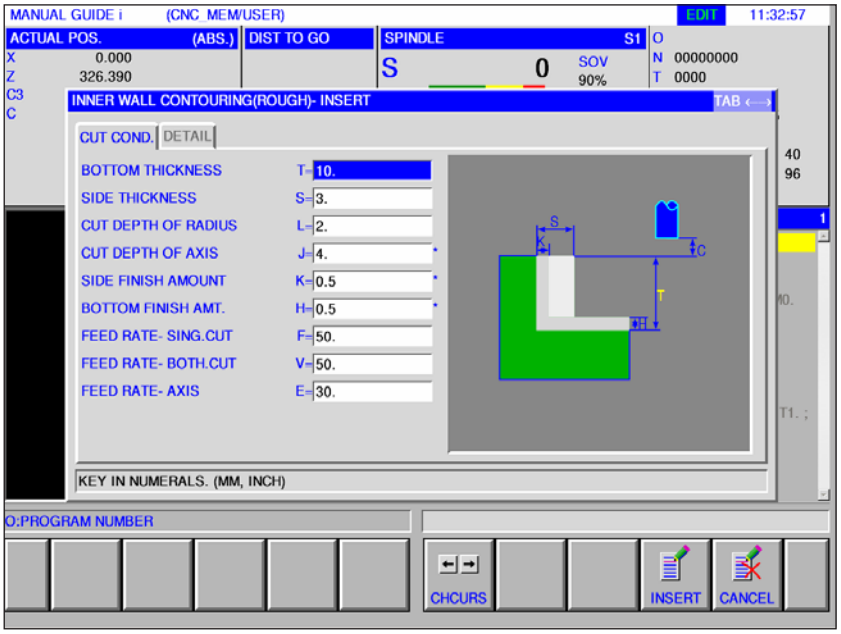
3 The tool cuts into the section to be chamfered to the cut depth (Dr) in the tool radius direction with feed speed (F).

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 (Roughing) G1064



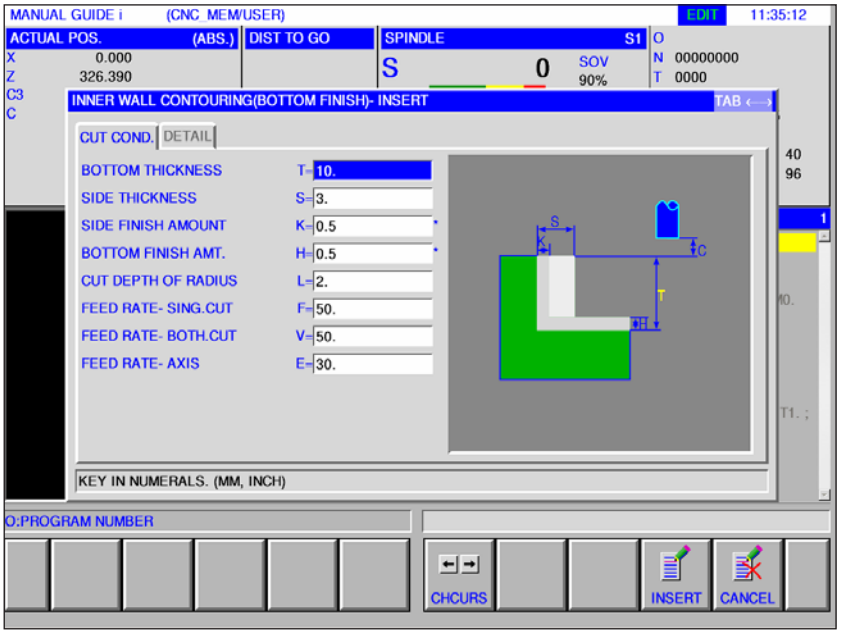
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
L	Infeed side	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
J*	Infeed depth	Cut depth in tool axis direction per cutting process (radius value, positive value). Standard is (Machining allowance at bottom - Finishing allowance at bottom).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
M	1. Chip removal	The feed override value for the first cut is pre-set as 100% fixed and cannot be changed.
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



Inner Wall (Finishing) G1065



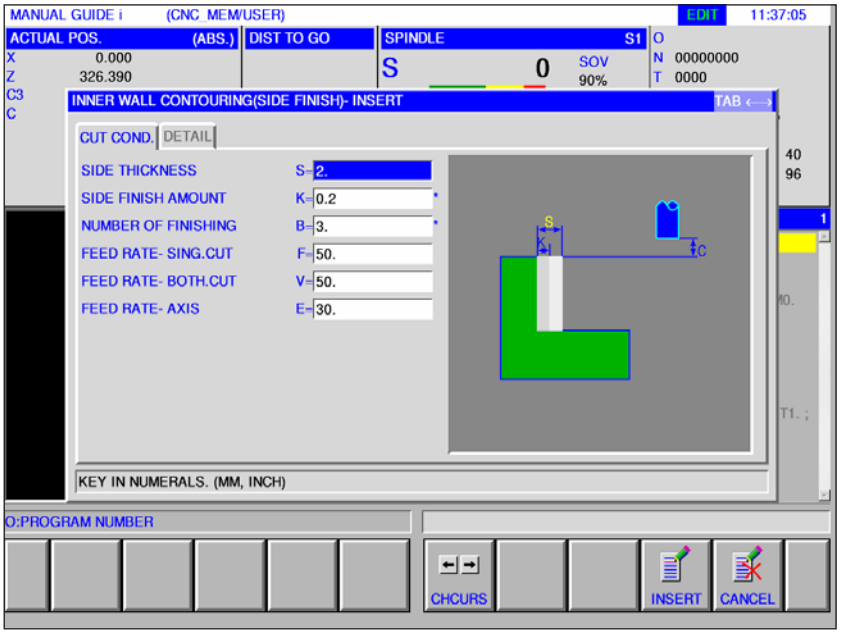
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
L	Infeed side	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



Inner Wall (Side Finishing) G1066



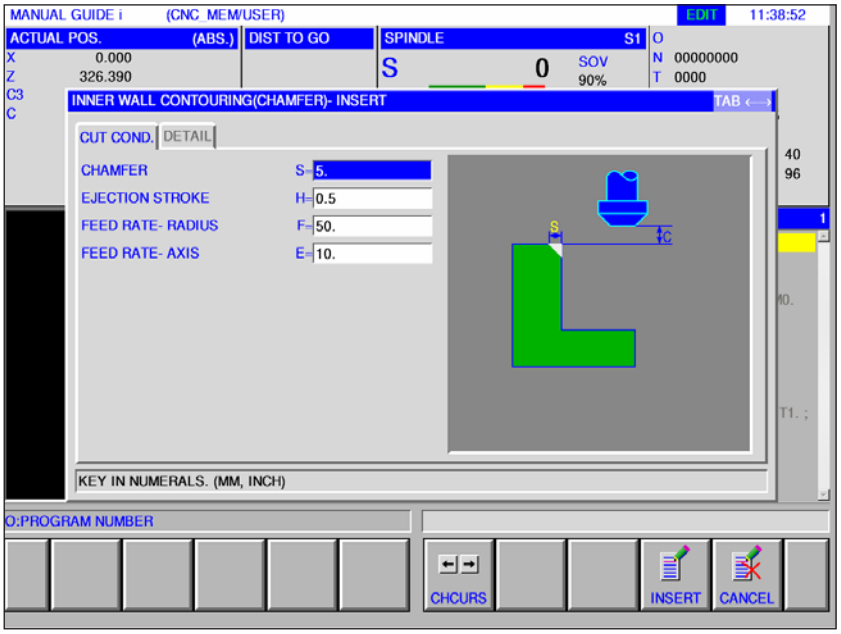
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
B*	Number of finishing runs	Number of cuts during finishing (positive value) Depth per cut = (superfluous side thickness)/(number of finishing cuts)
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



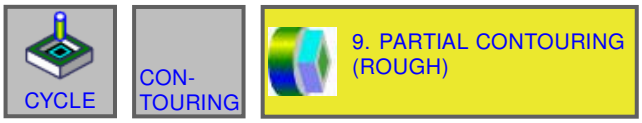
Inner Wall (Chamfer) G1067



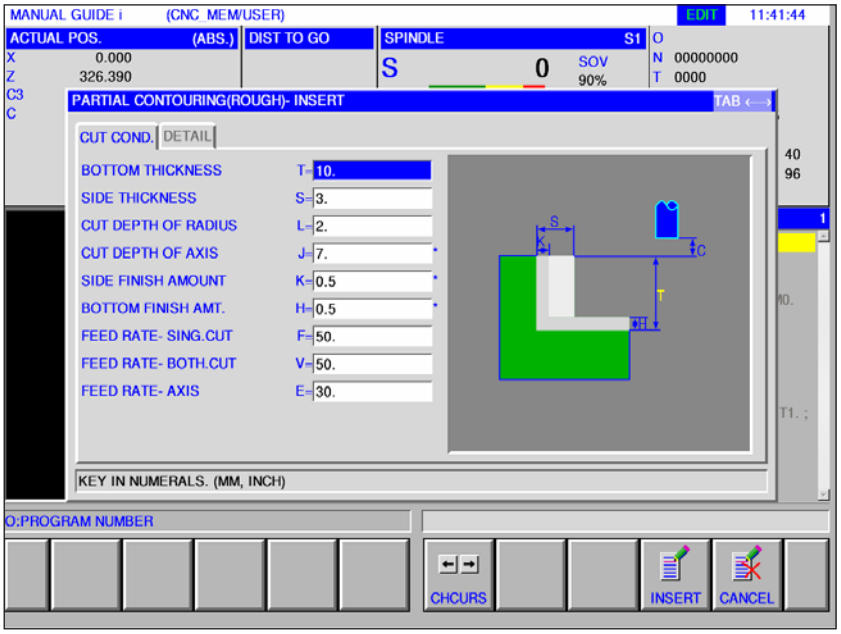
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
S	Chamfer width	Chamfer length (radius value, positive value)
H	Travelling track	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value)
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



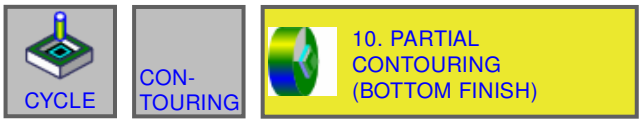
Partial Machining (Roughing) G1068



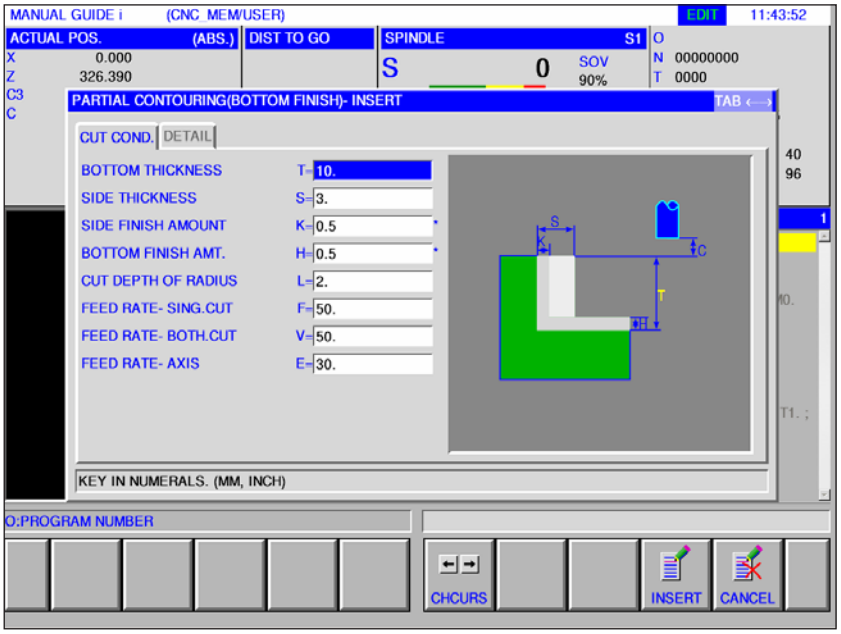
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
L	Infeed side	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
J*	Infeed depth	Cut depth in tool axis direction per cutting process (radius value, positive value). Standard is (Machining allowance at bottom - Finishing allowance at bottom).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
M	1. Chip removal	The feed override value for the first cut is pre-set as 100% fixed and cannot be changed.
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



Partial Machining (Finishing) G1069



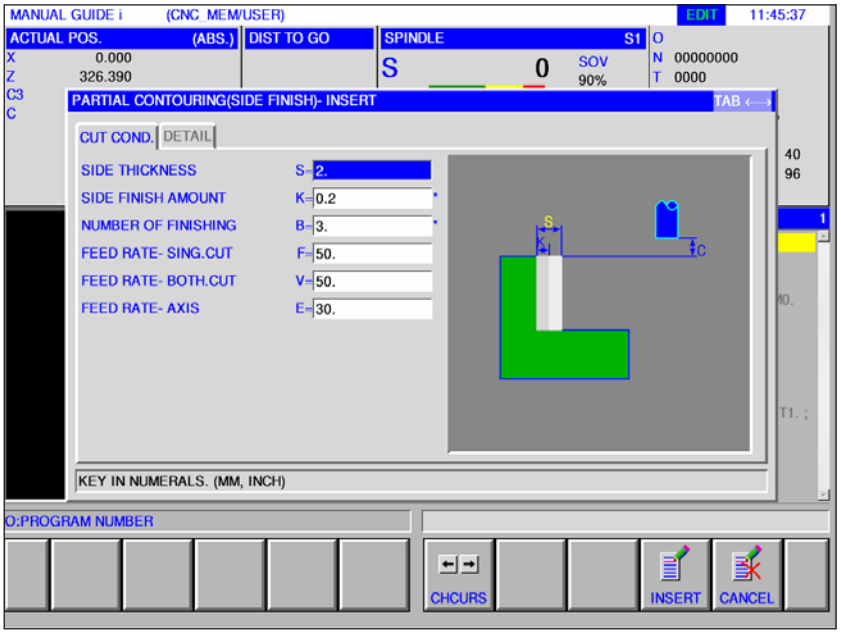
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
L	Infeed side	Cut depth per side surface machining process (in tool radius direction) (radius value, positive value)
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



Partial Machining (Side Finishing) G1070



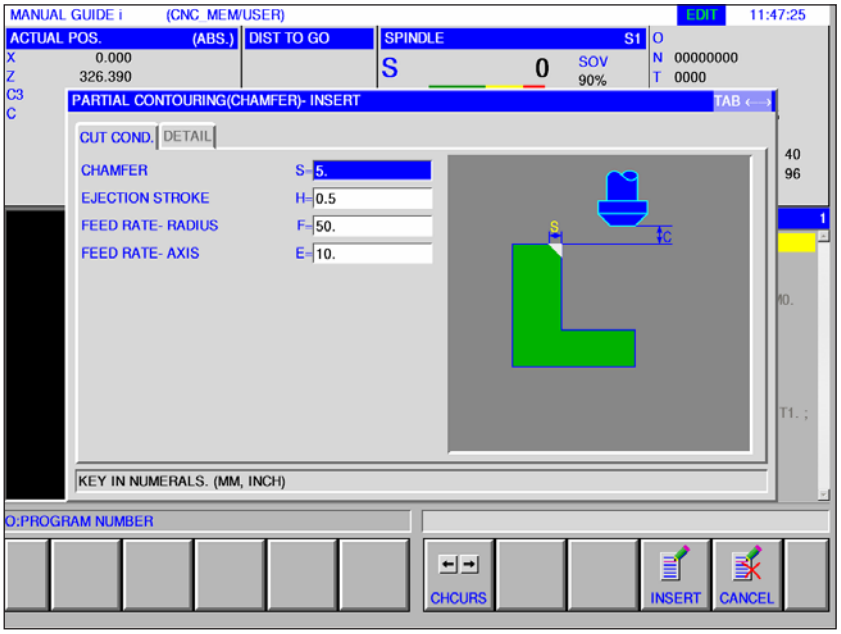
Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
S	Unmachined part allowance side	Side surface machining allowance (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
B*	Number of finishing runs	Number of cuts during finishing (positive value) Depth per cut = (superfluous side thickness)/(number of finishing cuts)
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.



Partial Machining (Chamfer) G1071



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

Machining Operation		
Data element		**Meaning**
S	Chamfer width	Chamfer length (radius value, positive value)
H	Travelling track	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value)
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 vertical to the first contour in side surface machining.
R	Approach radius / -track	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	Retraction type	• [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	Retraction radius / -track	Radius, only if [ARC] is specified. Length of a straight line, if [TANGEN] or [VERTIC] is specified. (radius value, positive value).
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXIS]: The tool is moved from the current position to the machining start point in 3-axis synchronous mode.

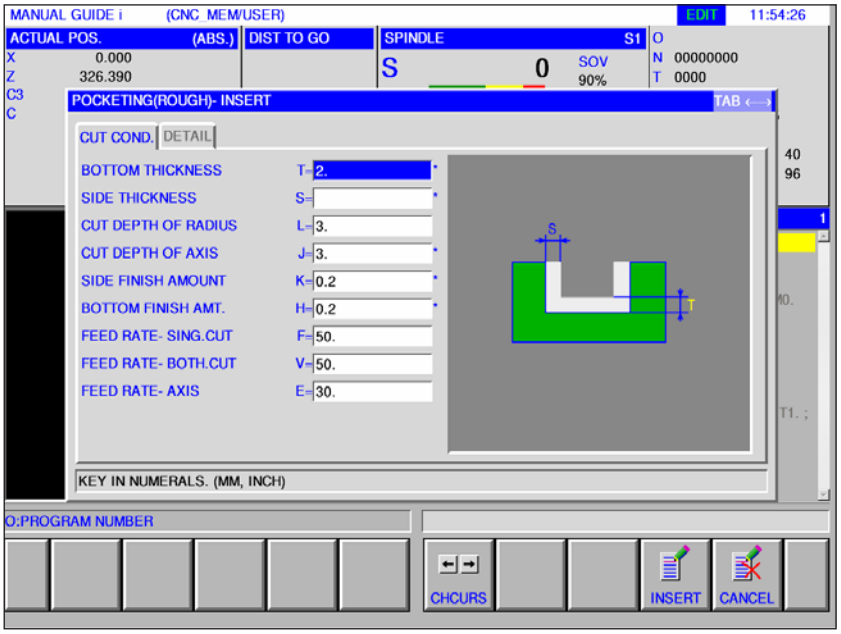


Pocket Milling

- Pocket Milling Roughing G1040
- Pocket Milling Finishing G1041
- Pocket Milling Side Finishing G1042
- Pocket Milling Chamfer G1043



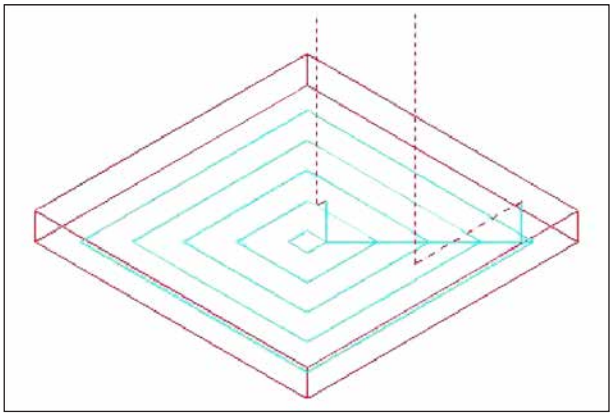
Pocket Milling (Roughing) G1040



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

Machining Operation		
Data element		**Meaning**
T*	Unmachined part allowance bottom	Machining allowance at the bottom in side surface machining (radius value, positive value).
S*	Unmachined part allowance side	Machining allowance at the side surface. (radius value, positive value) Remarks: If side finishing allowance and bottom finishing allowance are ignored, a complete pocket area is machined.
L	Infeed side	Cut depth at the side surface (in tool radius direction) per cutting process (radius value, positive value).
J*	Infeed depth	Cut depth in tool axis direction per cutting process (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
B	Safety distance XY	Distance between the pocket wall and the tool retraction position in tool radius direction (radius value, positive value).
C	Safety distance Z	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	Approaching movement	• [3 AXES] : The tool is moved from the current position to the machining start point in 3-axis synchronous mode.
X	Depth cutting method	• [STRAIT]: Plunging in the form of a straight line. • [HELICL]: Plunging on spiral path (helix).
A*	Plunging angle	Angle under which the tool cuts obliquely or spirally into the pocket. (In 1-degree increments, positive value).
R	Helical radius	Radius of the helical circle when plunging helically.
M	Helical feed	Feed speed when plunging helically.



Tool track

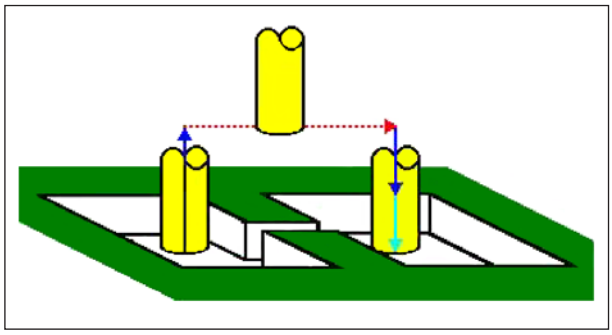
Cycle description
The interior of a pocket machining profile is cut out spirally.
The following tool track is generated.

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.

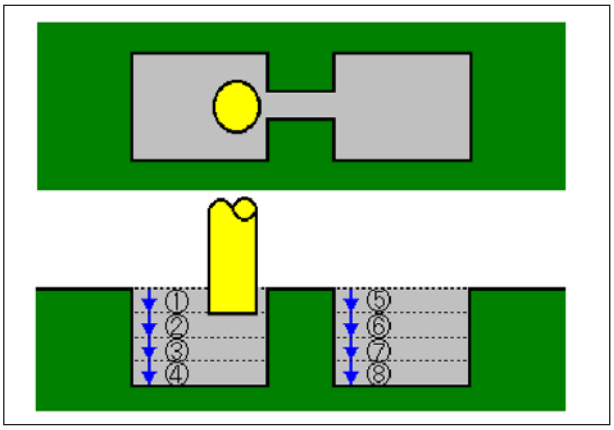
The tool track is produced in a manner such that a collision to be expected with pocket machining profiles or island machining profiles is avoided.

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 up-cut or for down-cut machining. The machining direction around the islands is controlled automatically.



Tool track



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.

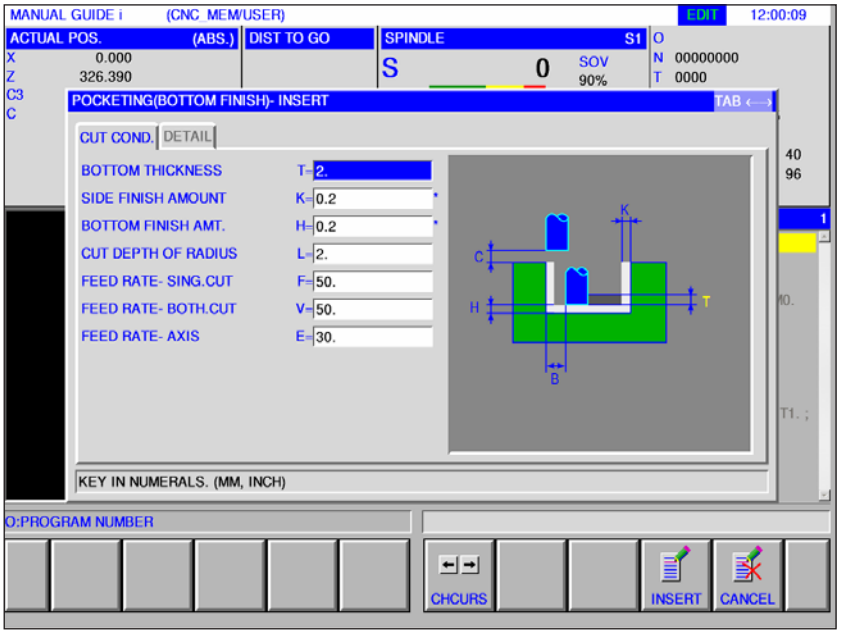
In the following pocket machining profile, where the tool can move through the pocket, the tool is lifted off automatically, so that only the pocket area that has to be processed is machined.

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

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



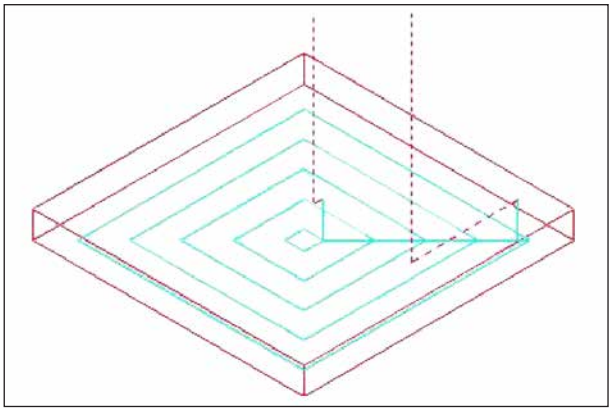
Pocket Milling (Finishing) G1041



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

Machining Operation		
Data element		**Meaning**
T	Unmachined part allowance bottom	Machining allowance at the bottom in side surface machining (radius value, positive value).
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
L	Infeed side	Cut depth at the side surface (in tool radius direction) per cutting process (radius value, positive value).
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
B	Safety distance XY	Distance between the pocket wall and the tool retraction position in tool radius direction (radius value, positive value).
C	Safety distance Z	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	Approaching movement	• [3 AXES] : The tool is moved from the current position to the machining start point in 3-axis synchronous mode.
X	Depth cutting method	• [STRAIT]: Plunging in the form of a straight line. • [HELICL]: Plunging on spiral path (helix).
A*	Plunging angle	Angle under which the tool cuts obliquely or spirally into the pocket. (In 1-degree increments, positive value).



Tool track

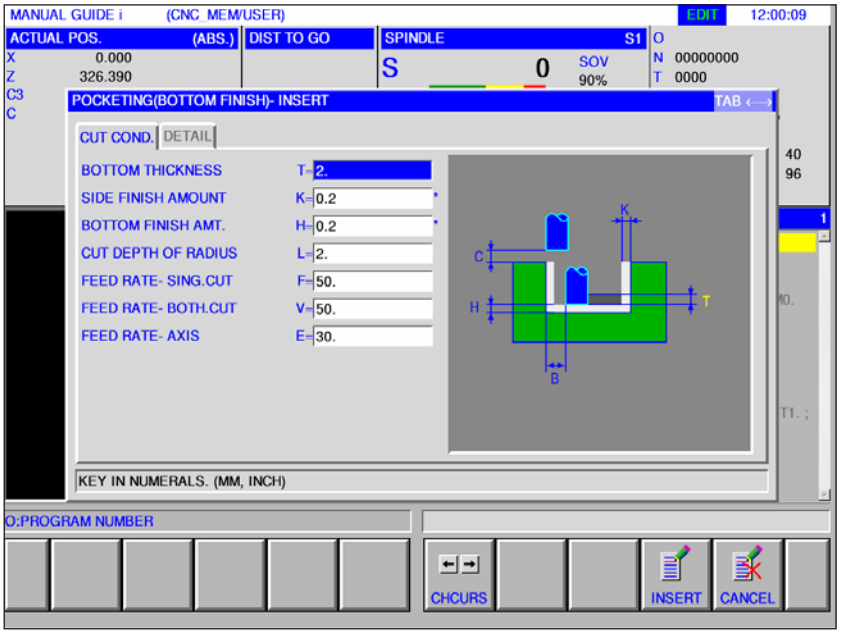
Cycle description

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.



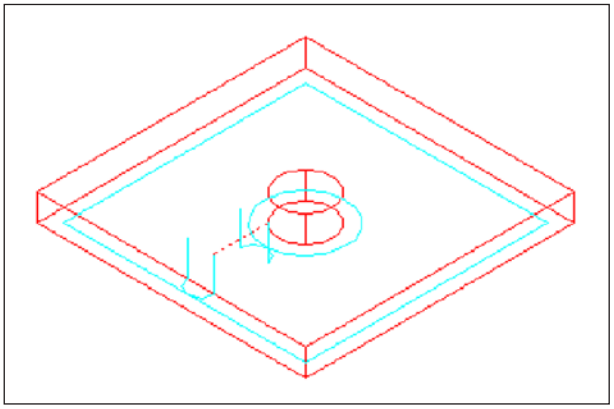
Pocket Milling (Side Finishing) G1042



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

Machining Operation		
Data element		**Meaning**
K*	Finishing allowance side	Finishing allowance at the side surface. (radius value, positive value).
H*	Finishing allowance bottom	Finishing allowance at the bottom in side surface machining (radius value, positive value).
F	Partial cut feed	Feed speed when cutting with only one side of the cutting edge of a shaft mill. With this feed speed, cutting takes place in the retraction process and on the side surface, except during start of cut.
V	Full cut speed	Feed speed when cutting with the entire face of a shaft mill. This feed speed is used for start of cutting.
E	Infeed speed depth	Feed speed when cutting in the tool axis direction to the bottom while machining side surfaces.

Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	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	• [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 / -track	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	Retraction type	• [ARC]: Retraction from the side surface takes the form of an arc. The setting is specified as fixed and cannot be changed.
X	Retraction radius / -track	Radius, only if [ARC] is specified.
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXES] : The tool is moved from the current 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.

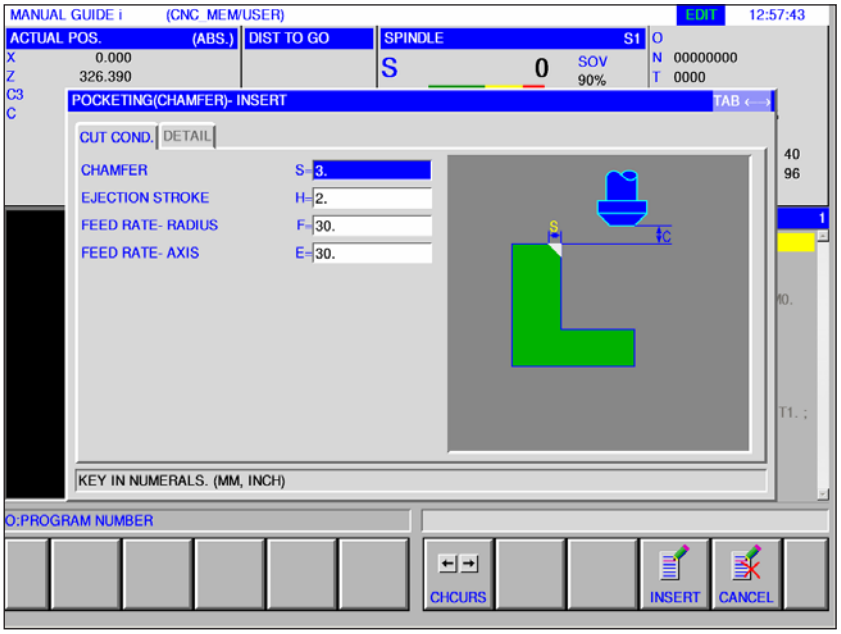
Although it can be assumed that the tool might collide with a pocket or island machining profile during finishing, no tool track is produced that can prevent this collision.







Pocket Milling (Chamfer) G1043

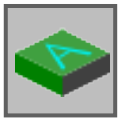


Entry fields marked with * are optional and are not required to be populated.

Machining Operation		
Data element		**Meaning**
S	Chamfer width	Chamfer length (radius value, positive value).
H	Travelling track	Distance between the tip of the chamfering tool and the actual cutting position in the tool axis direction (radius value, positive value).
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction

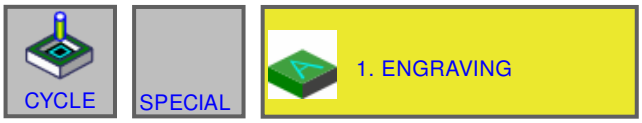
Detail		
Data element		**Meaning**
W	Up-cut / down-cut	• [UP CUT]: Machining is done by up-cut milling, in which the tool turns clockwise. • [DWNCUT]: Machining is done by down-cut milling, in which the tool turns clockwise.
C	Safety distance Z	Distance between the surface of an unmachined part to be machined and the machining start point (point R) in the tool axis direction (radius value, positive value).
P	Approach type	• [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 / -track	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	Retraction type	• [ARC]: Retraction from the side surface takes the form of an arc. The setting is specified as fixed and cannot be changed.
X	Retraction radius / -track	Radius, only if [ARC] is specified.
Y*	Retraction angle	Centre angle of the arc, only if [ARC] is specified. The standard value is 90 degrees. (positive value).
Z	Approaching movement	• [3 AXES] : The tool is moved from the current 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 contouring figure (chamfer).

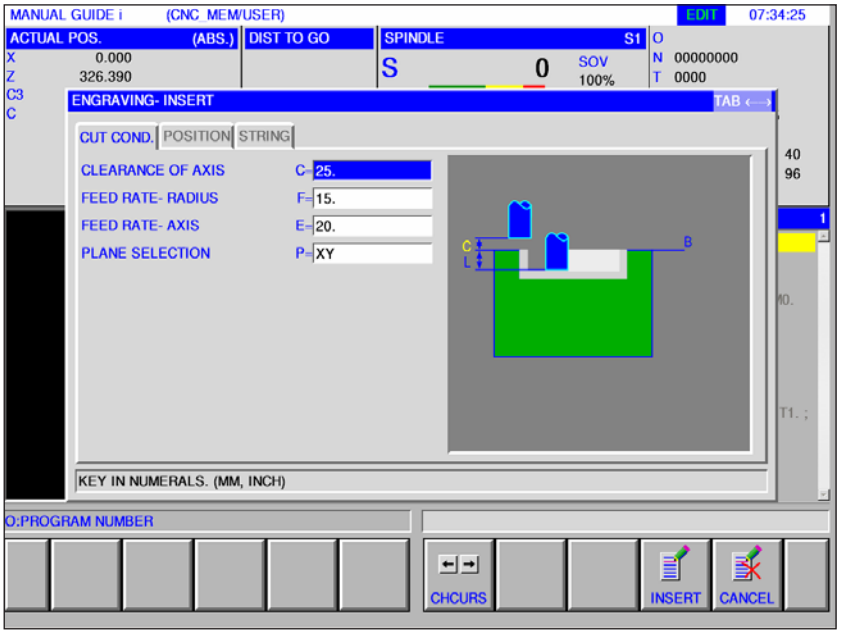


Special

- Engraving G1025



Engraving G1025



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

Machining Operation		
Data element		**Meaning**
C	Safety distance Z	Distance between work piece surface and cutting start point in tool axis direction.
F	Feed XY	Feed speed when cutting in tool radius direction
E	Infeed speed depth	Feed speed when cutting in the tool axis direction
P	Plane selection	1= XY Plane 2= YZ Plane 3 =XC Plane (polar coordinates interpolation plane) 4 =ZC Plane (cylinder plane)

Position		
Data element		**Meaning**
H	Start point 1. Axis	Position 1. Character at 1st Axis.
V	Start point 2. Axis	Position 1. Character at 2nd Axis.
B	Basis position	Coordinate on the work piece surface.
L	Character depth	Depth of the characters
U	Character size	Size of the characters
A	Rotation angle	Rotation angle of the characters

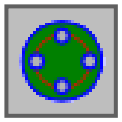
Character		
Data element		**Meaning**
*	Character	Characters to be engraved, maximum 80 characters.



Note:
- The following characters are allowed:
- Letters A .. Z, Numerals 0 .. 9
- allowed special characters:
  ? , @#=*+ / ‘_“ <>

Figure

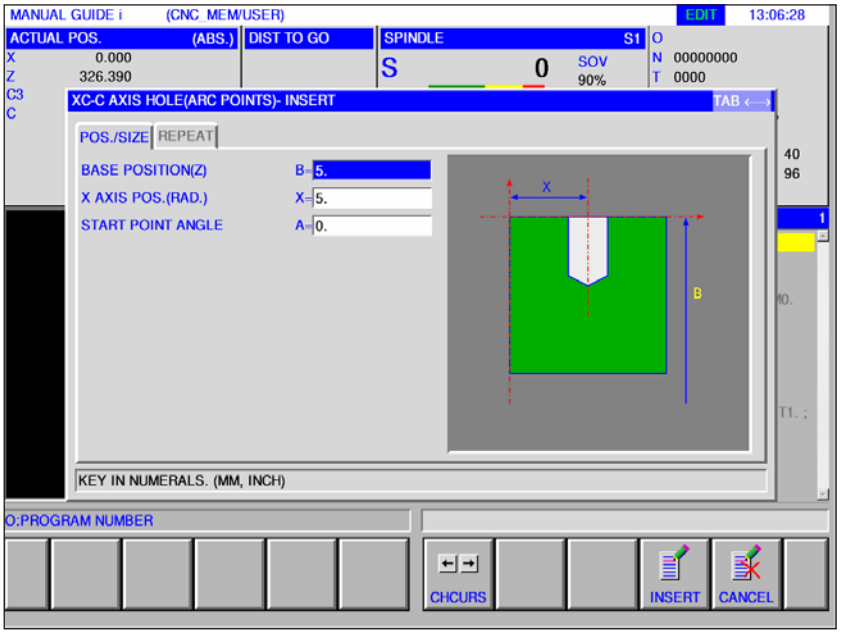
Hole Positions



- C-axis Hole on Face Arc Points G1572
- C-axis Hole on Face Circle G1573
- C-axis Hole on Face Coordinates G1574
- C-axis Hole on Cylindrical Surface Arc Points G1672
- C-axis Hole on Cylindrical Surface Free Choice G1673

FIGURE | HOLE POSITION | 1. C AXIS HOLE ON FACE (ARC POINTS)

C-axis Hole on Face (Arc Points) G1572



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

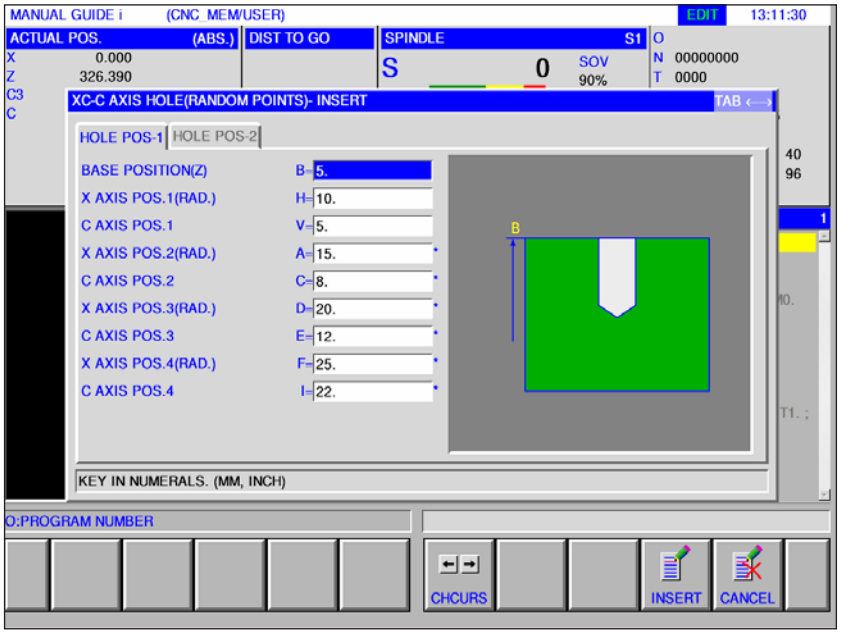
Position / Size		
Data element		**Meaning**
B	Basis position Z	Z-coordinate of the work piece surface.
X	X-axis position (radius)	X-coordinate of the drilling position (same for all holes).
A	Start point angle	Centre angle to the C-axis of the first hole (positive or negative value).

Repetition		
Data element		**Meaning**
C*	Pitch angle	Travel amount between two holes in the C-axis (positive or negative value).
M*	Number of holes	Number of holes (positive value).

FIGURE | HOLE POSITION | 2. C AXIS HOLE ON FACE (RANDOM POINTS)

C-axis Hole on Face (Circle), Polar Coordinates G1573

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

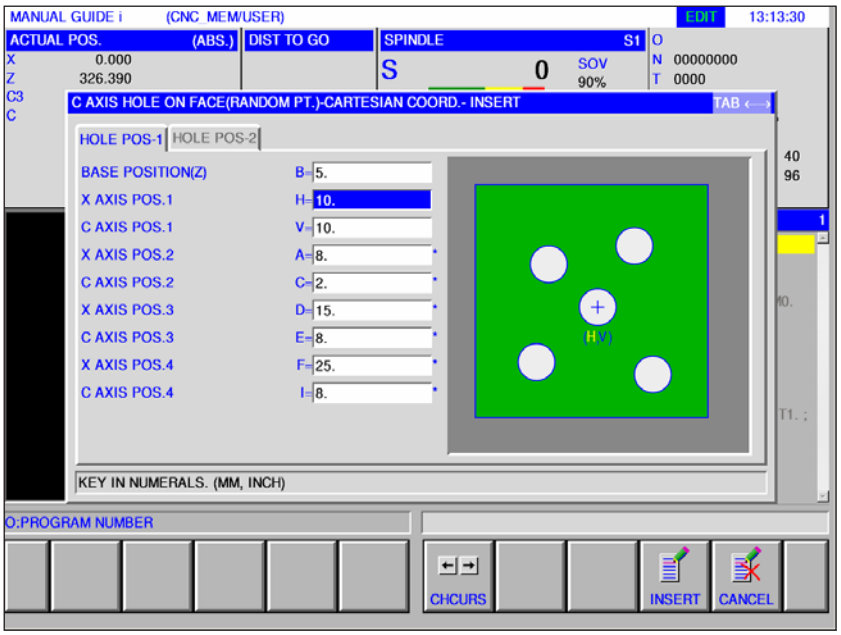


Drilling position 1 / size		
Data element		**Meaning**
B	Basis position Z	Z-coordinate of the work piece surface.
H	X-axis position 1 (radius)	X-coordinate of the 1st hole (radius value).
V	C-axis position 1	C-coordinate of the 1st hole.
A*	X-axis position 2 (radius)	X-coordinate of the 2nd hole (radius value).
C*	C-axis position 2	C-coordinate of the 2nd hole.
D*	X-axis position 3 (radius)	X-coordinate of the 3rd hole (radius value).
E*	C-axis position 3	C-coordinate of the 3rd hole.
F*	X-axis position 4 (radius)	X-coordinate of the 4th hole (radius value).
I*	C-axis position 4	C-coordinate of the 4th hole.

Drilling position 2 / Size		
Data element		**Meaning**
J*	X-axis position 5 (radius)	X-coordinate of the 5th hole (radius value).
K*	C-axis position 5	C-coordinate of the 5th hole.
M*	X-axis position 6 (radius)	X-coordinate of the 6th hole (radius value).
P*	C-axis position 6	C-coordinate of the 6th hole.
Q*	X-axis position 7 (radius)	X-coordinate of the 7th hole (radius value).
R*	C-axis position 7	C-coordinate of the 7th hole.
S*	X-axis position 8 (radius)	X-coordinate of the 8th hole (radius value).
T*	C-axis position 8	C-coordinate of the 8th hole.

FIGURE | HOLE POSITION | 3. C AXES HOLE ON FACE (RANDOM POINTS) CARTESIAN COORD.

C-axis Hole on Face (Cart. Coordinates) G1574



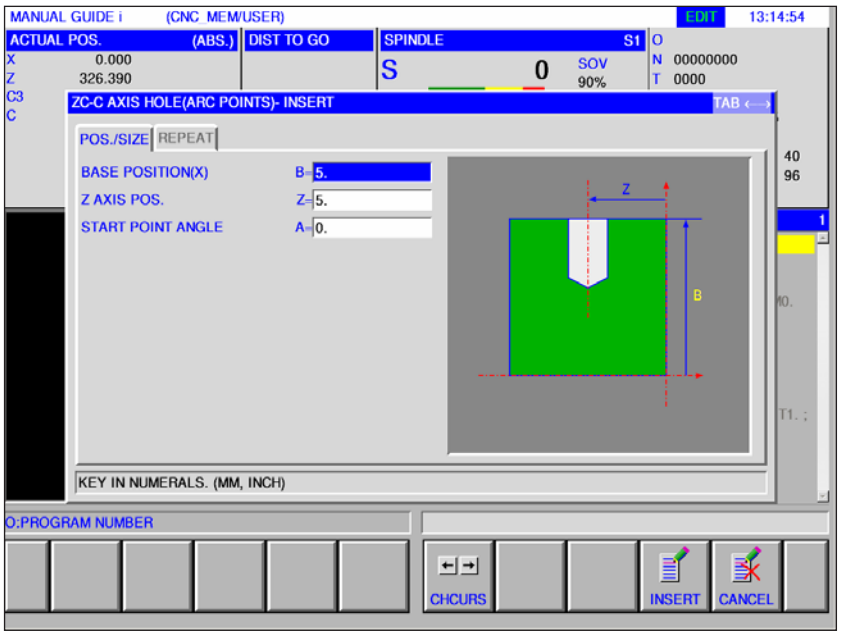
Entry fields marked with * are optional and are not required to be populated.

Drilling position 1 / Size		
Data element		**Meaning**
B	Basis position Z	Z-coordinate of the work piece surface, Cartesian coordinates.
H	X-axis position 1 (radius)	X-coordinate of the 1st hole (radius value).
V	C-axis position 1	C-coordinate of the 1st hole.
A*	X-axis position 2 (radius)	X-coordinate of the 2nd hole (radius value).
C*	C-axis position 2	C-coordinate of the 2nd hole.
D*	X-axis position 3 (radius)	X-coordinate of the 3rd hole (radius value).
E*	C-axis position 3	C-coordinate of the 3rd hole.
F*	X-axis position 4 (radius)	X-coordinate of the 4th hole (radius value).
I*	C-axis position 4	C-coordinate of the 4th hole.

Drilling position 2 / Size		
Data element		**Meaning**
J*	X-axis position 5 (radius)	X-coordinate of the 5th hole (radius value).
K*	C-axis position 5	C-coordinate of the 5th hole.
M*	X-axis position 6 (radius)	X-coordinate of the 6th hole (radius value).
P*	C-axis position 6	C-coordinate of the 6th hole.
Q*	X-axis position 7 (radius)	X-coordinate of the 7th hole (radius value).
R*	C-axis position 7	C-coordinate of the 7th hole.
S*	X-axis position 8 (radius)	X-coordinate of the 8th hole (radius value).
T*	C-axis position 8	C-coordinate of the 8th hole.

FIGURE | HOLE POSITION | 4. C AXIS HOLE ON FACE CYLINDER (ARC POINTS)

C-axis Hole on Cylindrical Surface (Arc Points) G1672



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

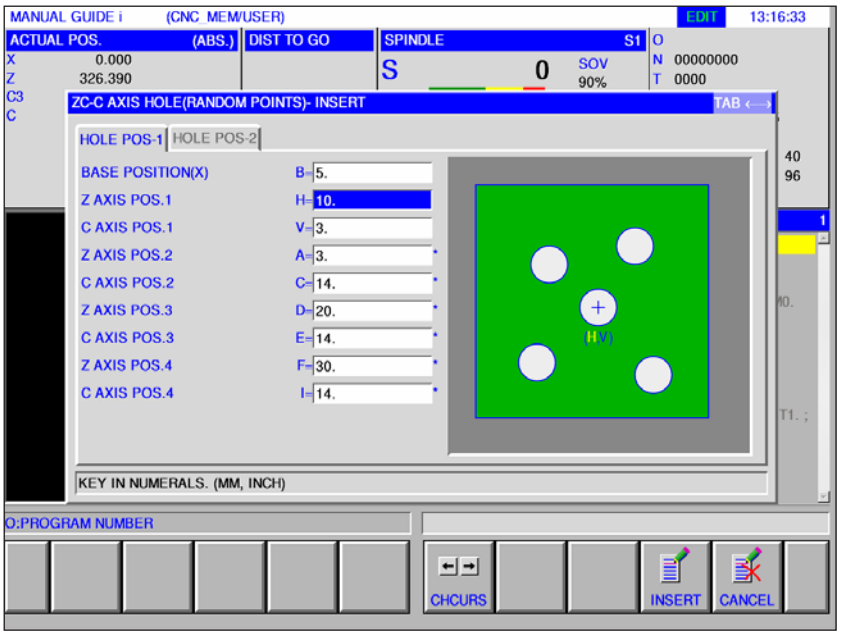
Position / Size		
Data element		**Meaning**
B	Basis position X	X-coordinate of the work piece surface.
X	Y-axis position (radius)	Z-coordinate of the drilling position (same for all holes).
A	Start point angle	Centre angle to the C-axis of the first hole (positive or negative value).

Repetition		
Data element		**Meaning**
C*	Pitch angle	Travel amount between two holes in the C-axis (positive or negative value).
M*	Number of holes	Number of holes (positive value).

FIGURE | HOLE POSITION | 5. C AXIS HOLE ON FACE CYLINDER (RANDOM POINTS)

C-axis Hole on Cylindrical Surface (Free Choice) G1673

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



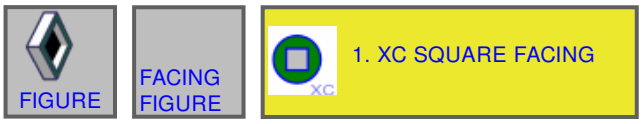
Drilling position 1 / Size		
Data element		**Meaning**
B	Basis position X	X-coordinate of the work piece surface
H	Z-axis position 1	Z-coordinate of the 1st hole.
V	C-axis position 1	C-coordinate of the 1st hole.
A*	Z-axis position 2	Z-coordinate of the 2nd hole.
C*	C-axis position 2	C-coordinate of the 2nd hole.
D*	Z-axis position 3	Z-coordinate of the 3rd hole.
E*	C-axis position 3	C-coordinate of the 3rd hole.
F*	Z-axis position 4	Z-coordinate of the 4th hole.
I*	C-axis position 4	C-coordinate of the 4th hole.

Drilling position 2 / Size		
Data element		**Meaning**
J*	Z-axis position 5	Z-coordinate of the 5th hole.
K*	C-axis position 5	C-coordinate of the 5th hole.
M*	Z-axis position 6	Z-coordinate of the 6th hole.
P*	C-axis position 6	C-coordinate of the 6th hole.
Q*	Z-axis position 7	Z-coordinate of the 7th hole.
R*	C-axis position 7	C-coordinate of the 7th hole.
S*	Z-axis position 8	Z-coordinate of the 8th hole.
T*	C-axis position 8	C-coordinate of the 8th hole.

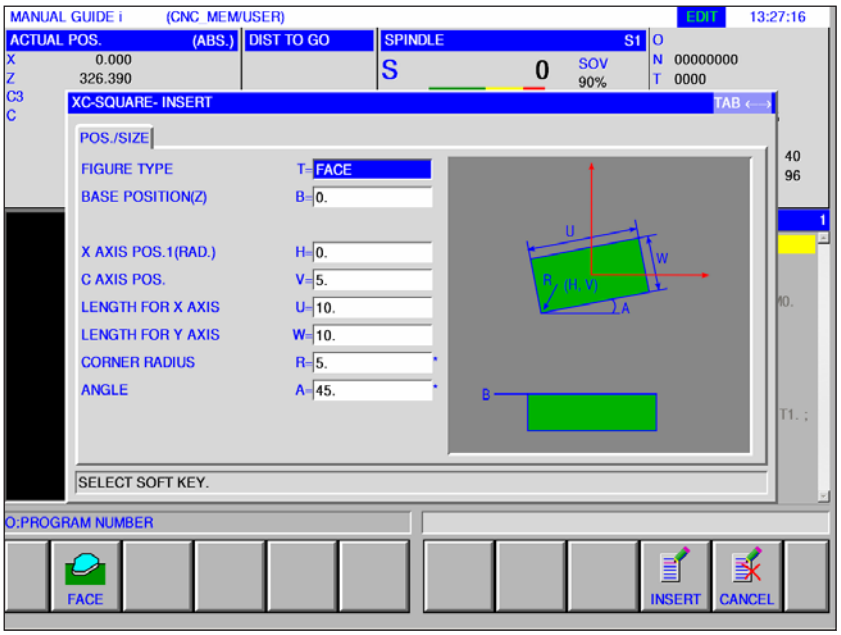


Front Surface Contour

- XC Square Front Surface G1520

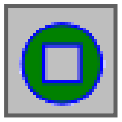


XC Square Face Contour G1520



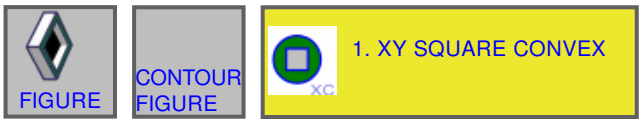
Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [FACE]: used as contour in plane machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
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 hole (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

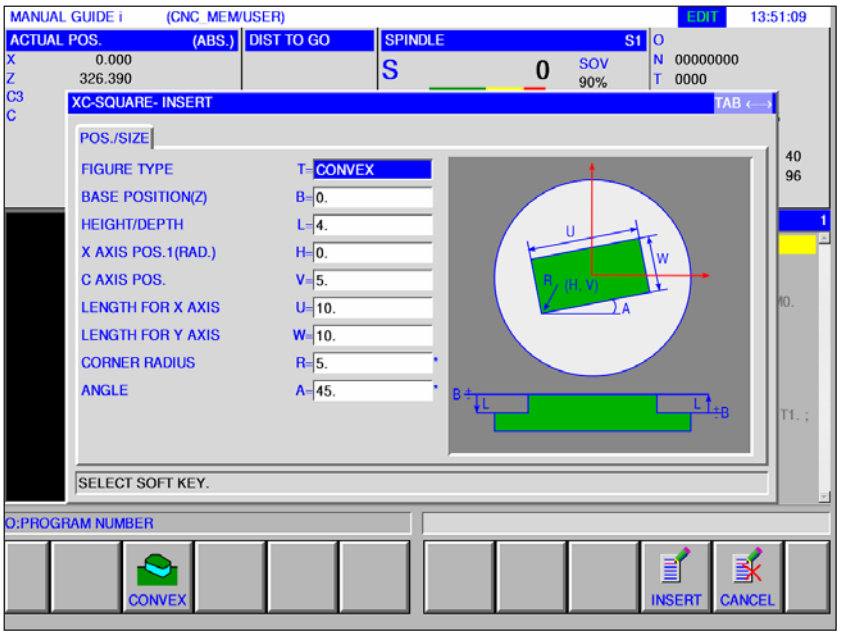


Contouring Figure

- XC Square Convex G1520
- XC Circle Convex G1521
- XC Oval Convex G1522
- XC Polygon Convex G1525
- XC Free Contour Convex G1500
- ZC Free Contour Convex on Cylindrical Surface G1600
- XC Square Concave G1520
- XC Circle Concave G1521
- XC Oval Concave G1522
- XC Polygon Concave G1525
- XC Free Contour Concave G1500
- ZC Free Contour Concave on Cylindrical Surface G1600
- XC Free Contour Open G1500
- ZC Free Contour Open on Cylindrical Surface G1600

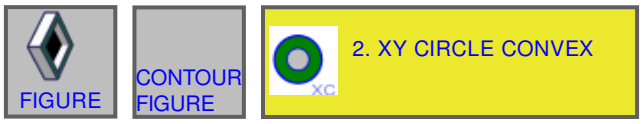


XC Square Convex G1520

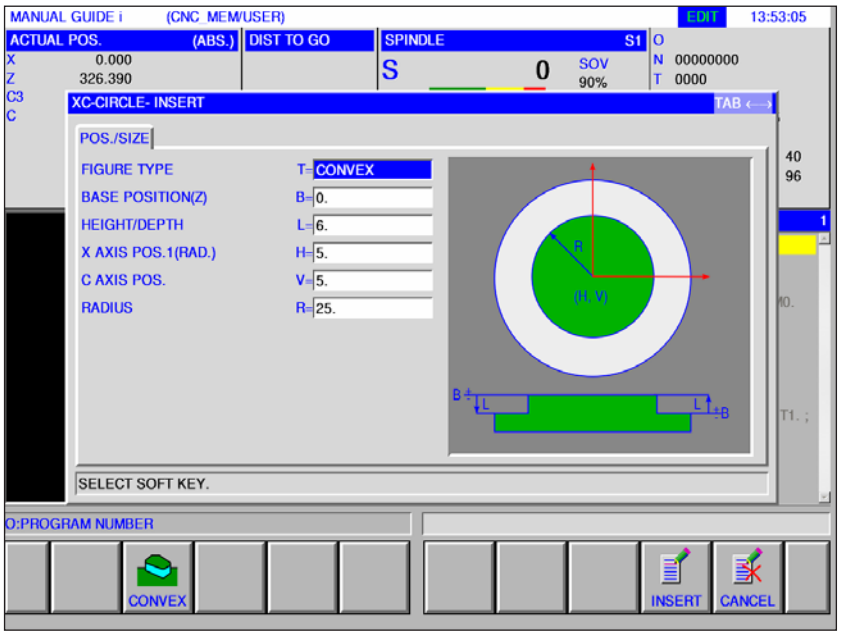


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONVEX]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
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 hole (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).



XC Circle Convex G1521

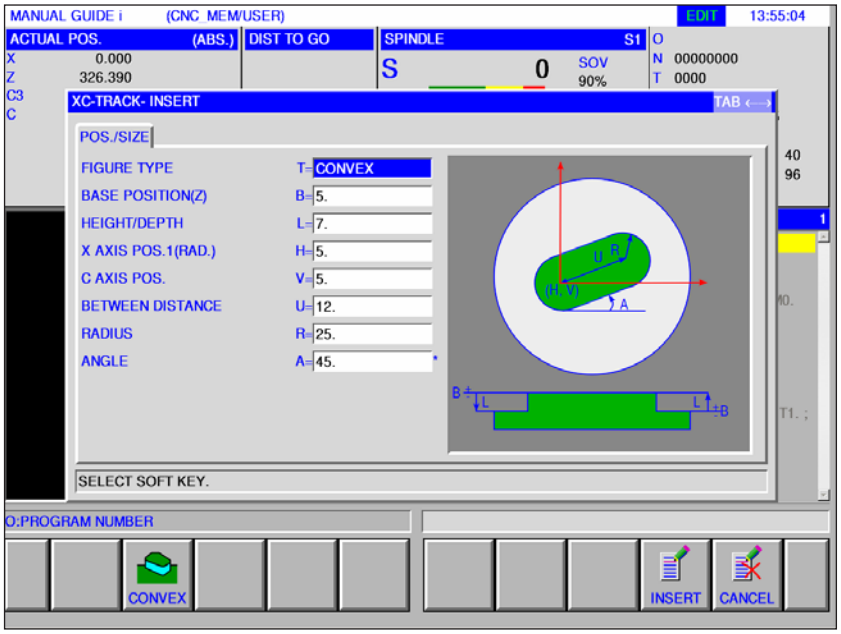


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONVEX]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
R	Corner radius	Radius for corner rounding (radius value, positive value).

FIGURE | CONTOUR FIGURE | 3. XY TRACK CONVEX

XC Oval Convex G1522

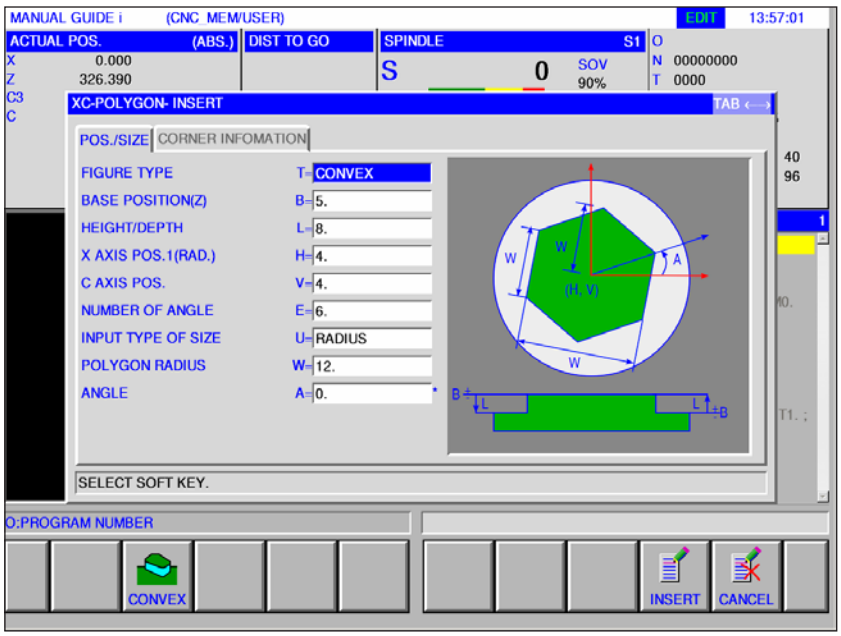


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONVEX]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
U	Centre distance	Distance between the centres of the right and the left semicircle (radius value, positive value).
R	Radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

FIGURE | CONTOUR FIGURE | 4. XY POLYGON CONVEX

XC Polygon Convex G1525



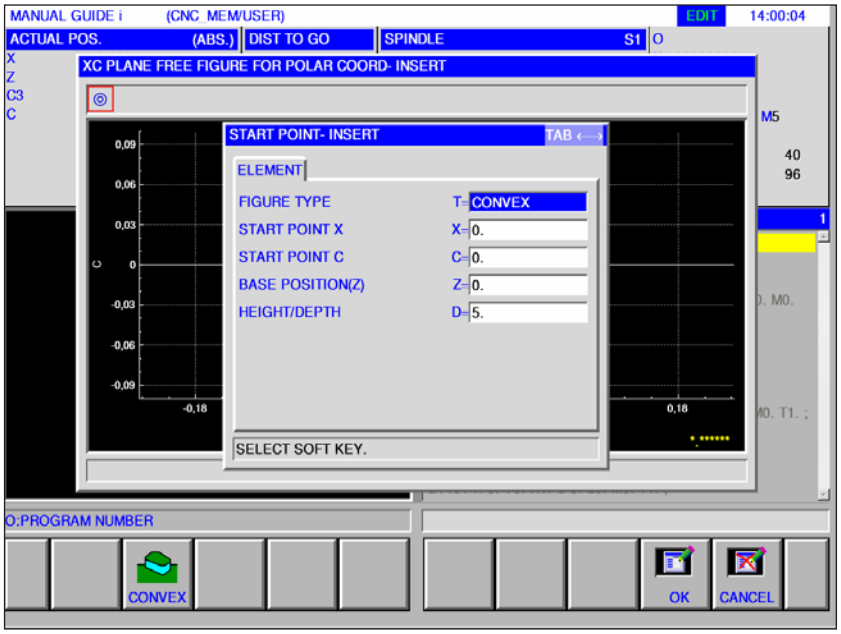
Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONVEX]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
E	Angle number	Number of angles (positive value) Integer value between 3 and 99.
U	Entry type size	• [RADIUS]: Polygon radius • [LENGTH] : Side length of polygon • [WIDTH] : Key width
W	radius / length / key	• Polygon radius at entry [RADIUS] • Polygon side length at entry [LENGTH] • Key at entry [WIDTH]
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

Corner shape		
Data element		**Meaning**
C	Corner type	• [NOTHIN] : no corner shape • [CHAMFER]: Chamfer • [ARC]: Corner radius
R	Corner size	Radius for corner rounding

FIGURE | CONTOUR FIGURE | 5. XC FREE CONVEX FIGURE

XC Free Contour Convex G1500



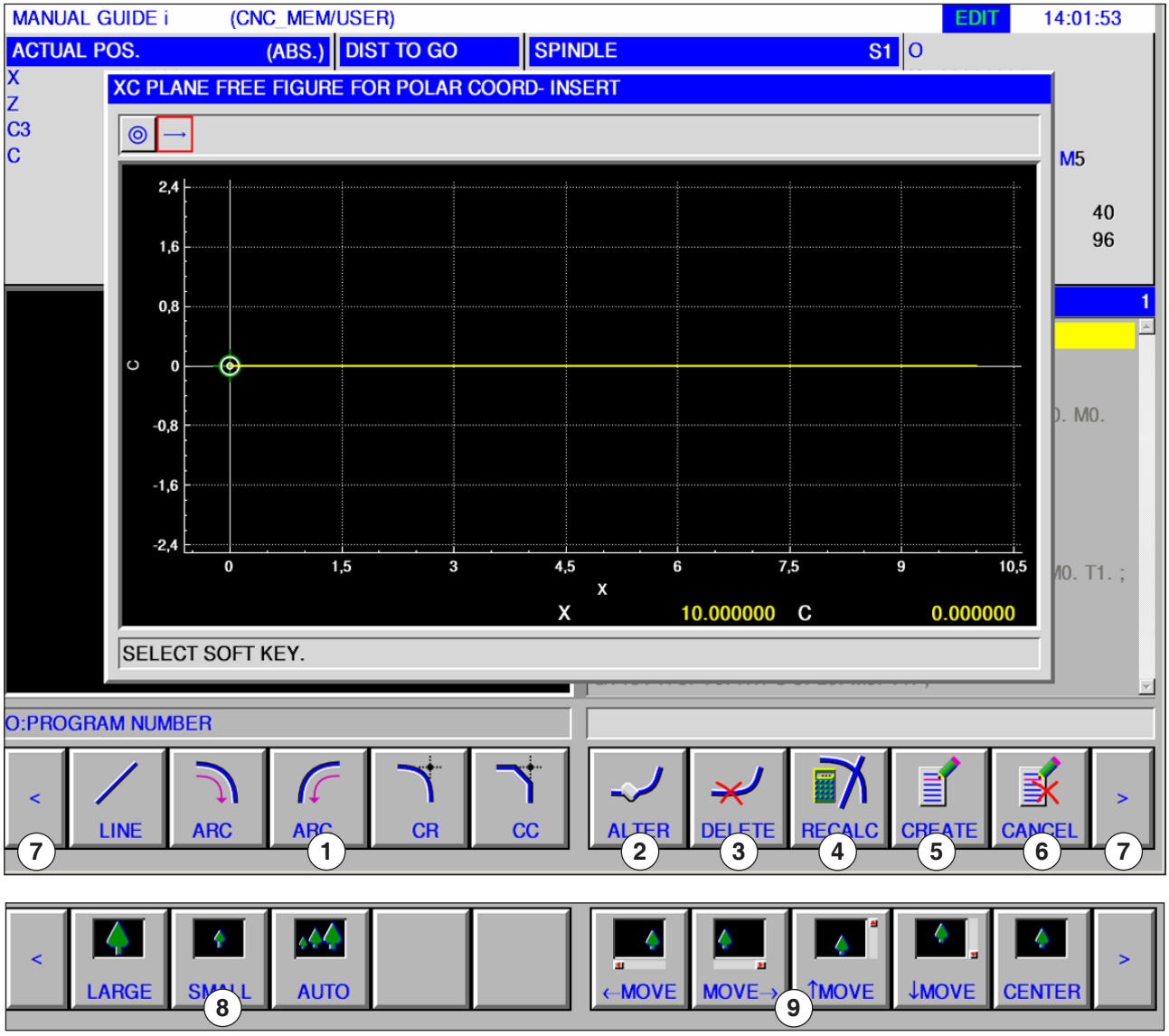
Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONVEX]: used as outside contour for contour machining.
X	Start point X	X-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
Z	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height

Note:
INPUT DATA are elements that are displayed when changing or editing in the data input window.
This applies for cycles G1451 to G1456.

Note:
OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can activate these only for program display purposes.
This applies also for cycles G1451 to G1456.

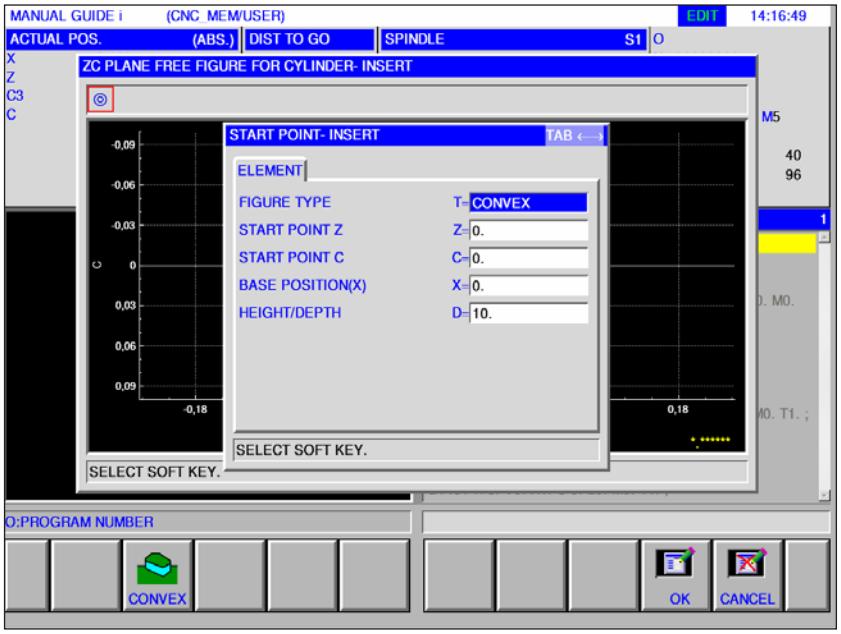
Overview of the input elements for free contour programming



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



ZC Free Contour Convex on Cylindrical Surface G1600

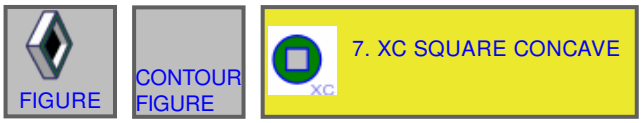


Entry fields marked with * are optional and are not required to be populated.

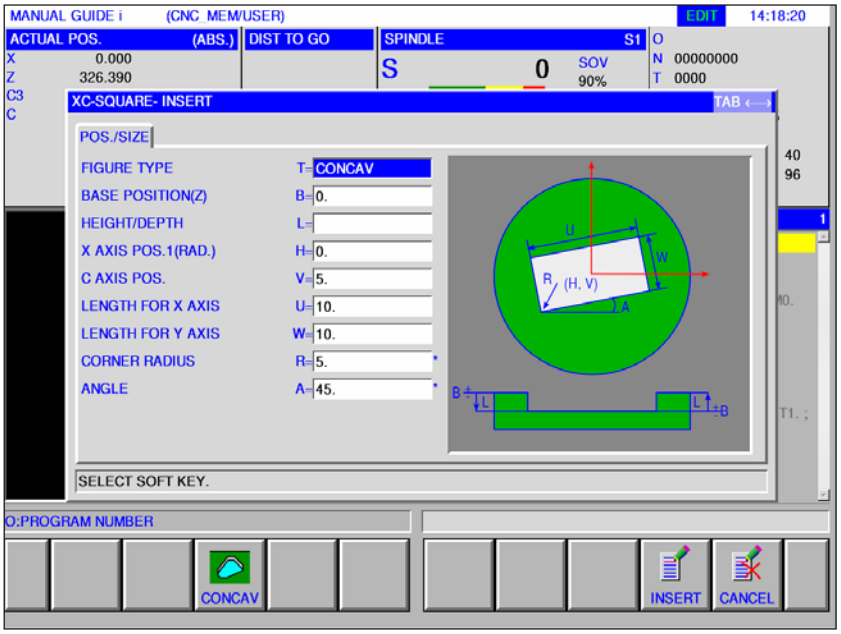
Position / Size		
Data element		**Meaning**
T	Contour type	• [CONVEX]: used as outside contour for contour machining.
Z	Start point Z	Z-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
X	Basis position X	X-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height

Note:
INPUT DATA are elements that are displayed when changing or editing in the data input window.

Note:
OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can activate these only for program display purposes.

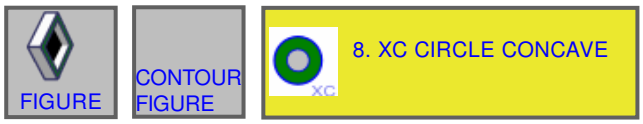


XC Square Concave G1520

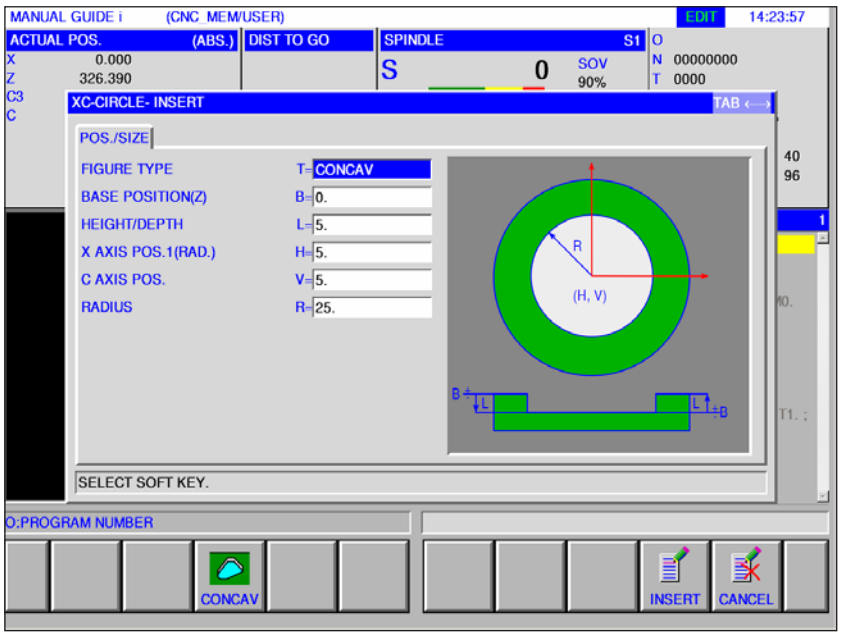


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
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 hole (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

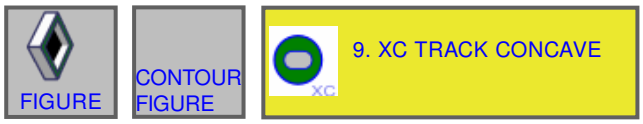


XC Circle Concave G1521

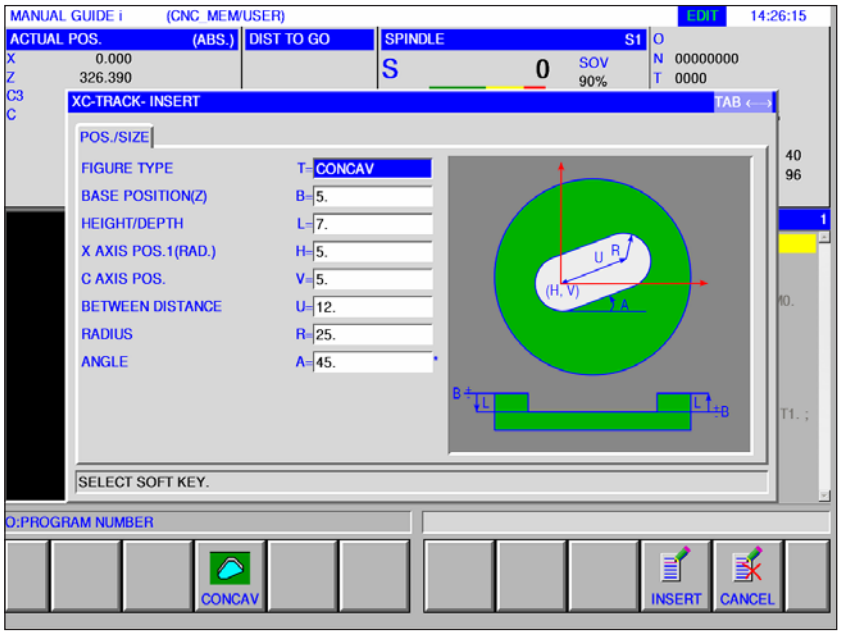


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
R	Corner radius	Radius for corner rounding (radius value, positive value).

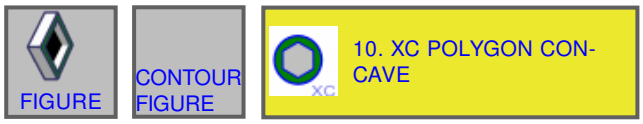


XC Oval Concave G1522

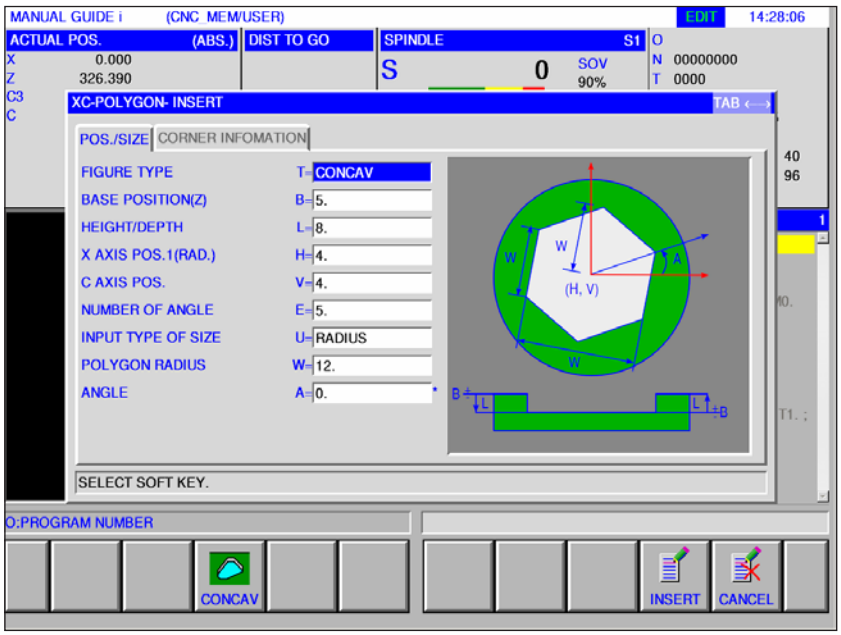


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
U	Centre distance	Distance between the centres of the right and the left semicircle (radius value, positive value).
R	Radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).



XC Polygon Concave G1525



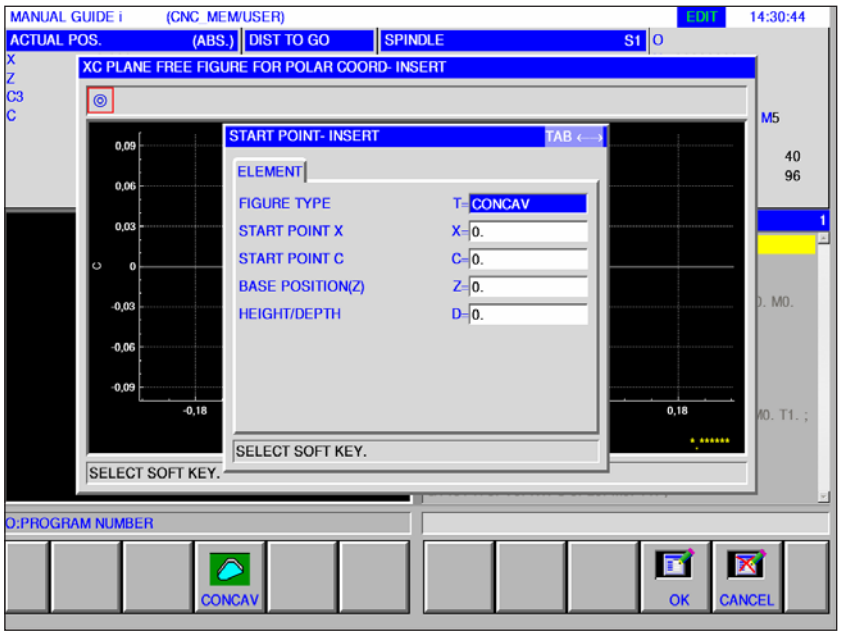
Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
E	Angle number	Number of angles (positive value) Integer value between 3 and 99.
U	Entry type size	• [RADIUS]: Polygon radius • [LENGTH] : Side length of polygon • [WIDTH] : Key width
W	radius / length / key	• Polygon radius at entry [RADIUS] • Polygon side length at entry [LENGTH] • Key at entry [WIDTH]
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

Corner shape		
Data element		**Meaning**
C	Corner type	• [NOTHIN] : no corner shape • [CHAMFER]: Chamfer • [ARC]: Corner radius
R	Corner size	Radius for corner rounding

FIGURE | CONTOUR FIGURE | 11. XC FREE CONCAVE FIGURE

XC Free Contour Concave G1500



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

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
X	Start point X	X-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
Z	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height

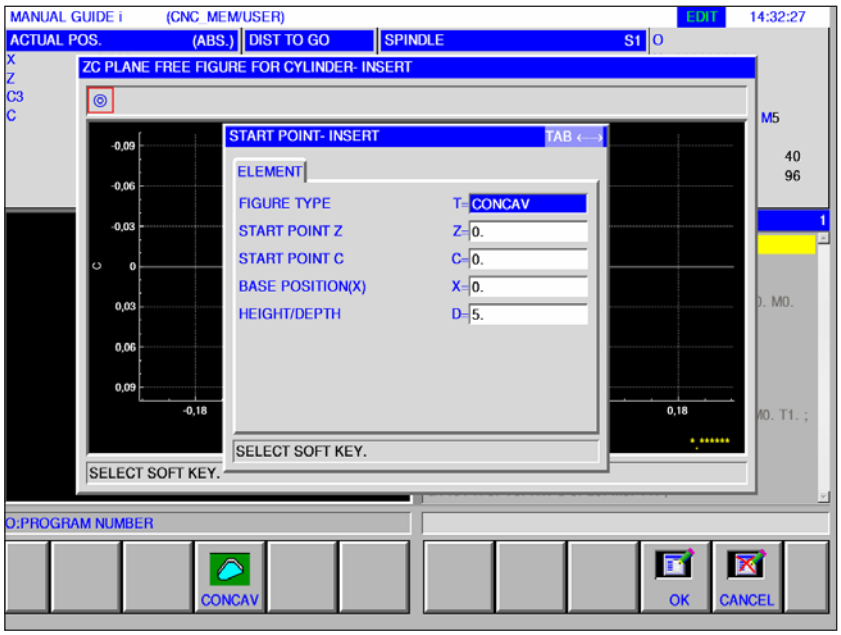
Note:
INPUT DATA are elements that are displayed when changing or editing in the data input window.
This applies for cycles G1451 to G1456.

Note:
OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can activate these only for program display purposes.
This applies also for cycles G1451 to G1456.



ZC Free Contour Concave on Cylindrical Surface G1600

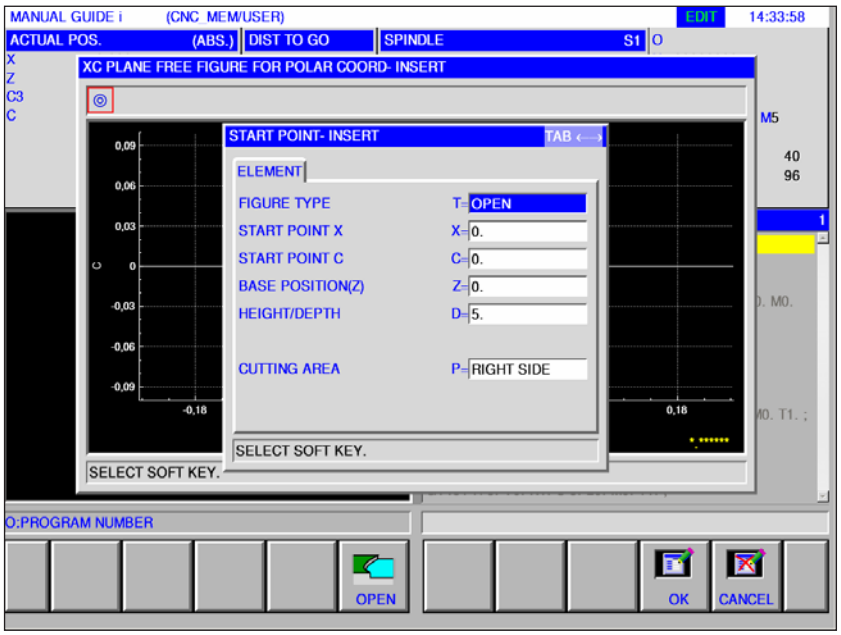
Entry fields marked with * are optional and are not required to be populated.



Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
Z	Start point Z	Z-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
X	Basis position X	X-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height



XC Free Contour Open G1500

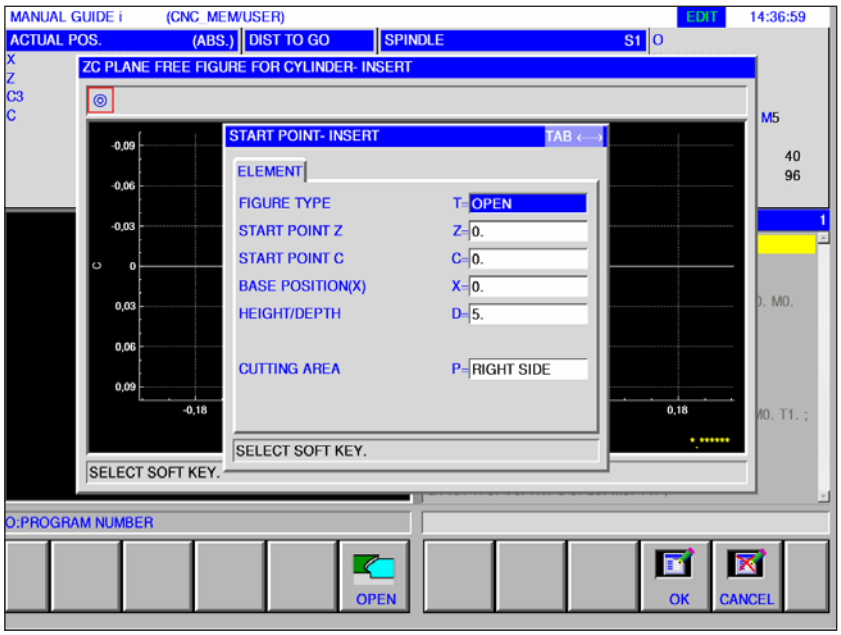


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [OPEN]: used as outside contour for contour machining.
X	Start point X	X-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
Z	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
P	Machining area	• [RIGHT]: Cutter radius compensation right side • [LEFT] : Cutter radius compensation left side See also chapter E, programming G41, G42

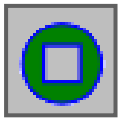


ZC Free Contour Open on Cylindrical Surface G1600



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

Position / Size		
Data element		**Meaning**
T	Contour type	• [OPEN]: used as outside contour for contour machining.
Z	Start point Z	Z-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
X	Basis position X	X-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
P	Machining area	• [RIGHT]: Cutter radius compensation right side • [LEFT] : Cutter radius compensation left side See also chapter E, programming G41, G42

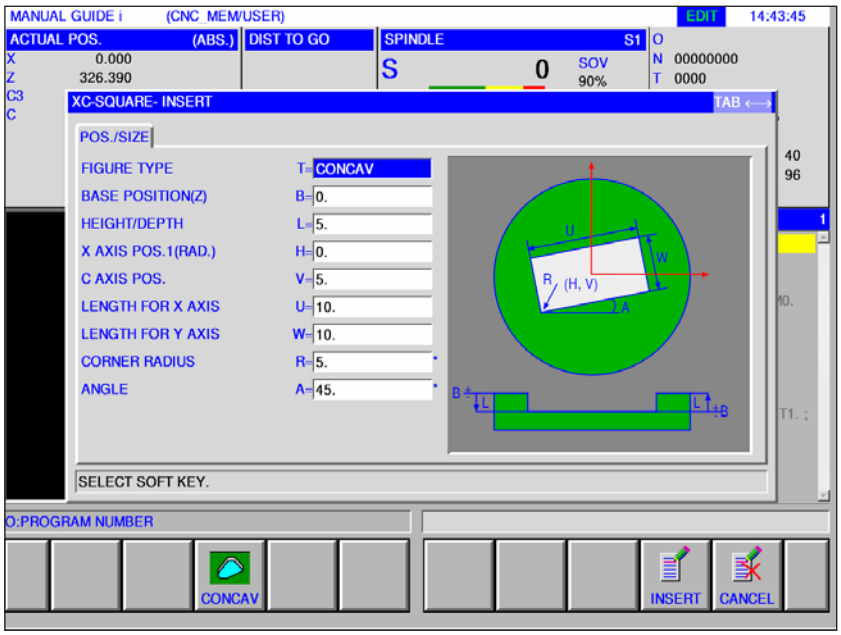


Pocket Contour

- XC Square Concave G1520
- XC Circle Concave G1521
- XC Oval Concave G1522
- XC Polygon Concave G1525
- XC Free Contour Concave G1500
- ZC Free Contour Concave on Cylindrical Surface G1600

FIGURE | POCKET-FIGURE | 1. XC SQUARE CONCAVE

XC Square Concave G1520

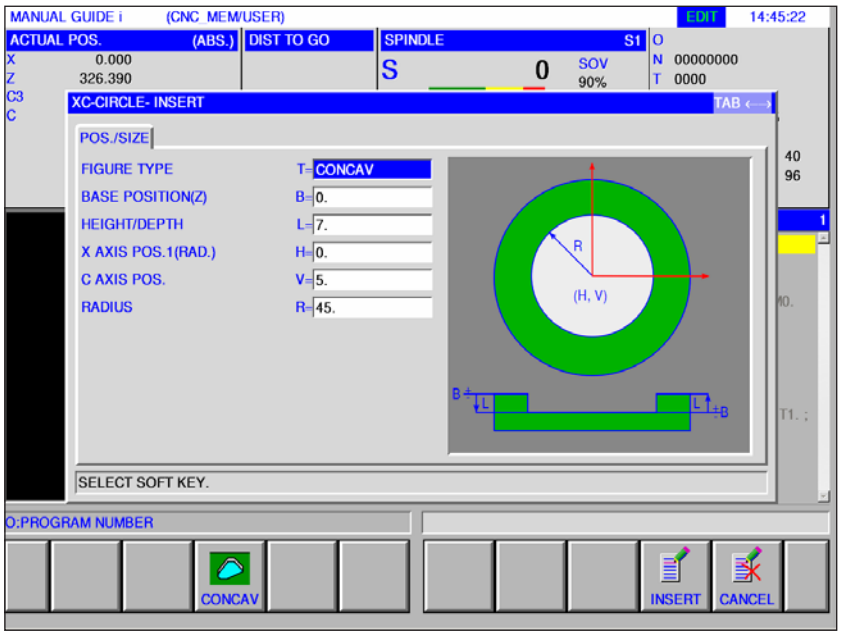


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
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 hole (positive or negative value) (initial value = 0).
R*	Corner radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

FIGURE | POCKET-FIGURE | 2. XC CIRCLE CONCAVE

XC Circle Concave G1521

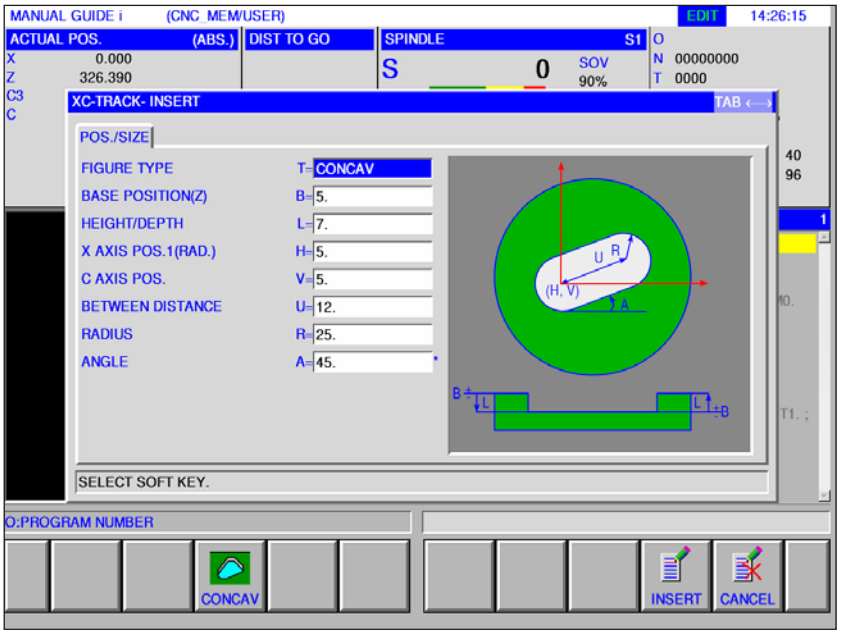


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
R	Corner radius	Radius for corner rounding (radius value, positive value).

FIGURE | POCKET-FIGURE | 3. XC TRACK CONCAVE

XC Oval Concave G1522

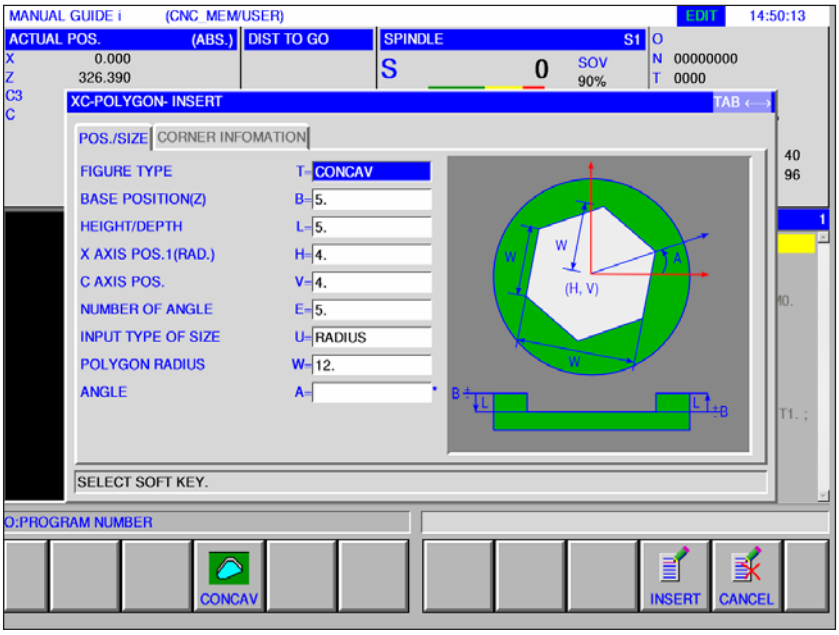


Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
U	Centre distance	Distance between the centres of the right and the left semicircle (radius value, positive value).
R	Radius	Radius for corner rounding (radius value, positive value).
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

FIGURE | POCKET-FIGURE | 4. XC POLYGON CONCAVE

XC Polygon Concave G1525



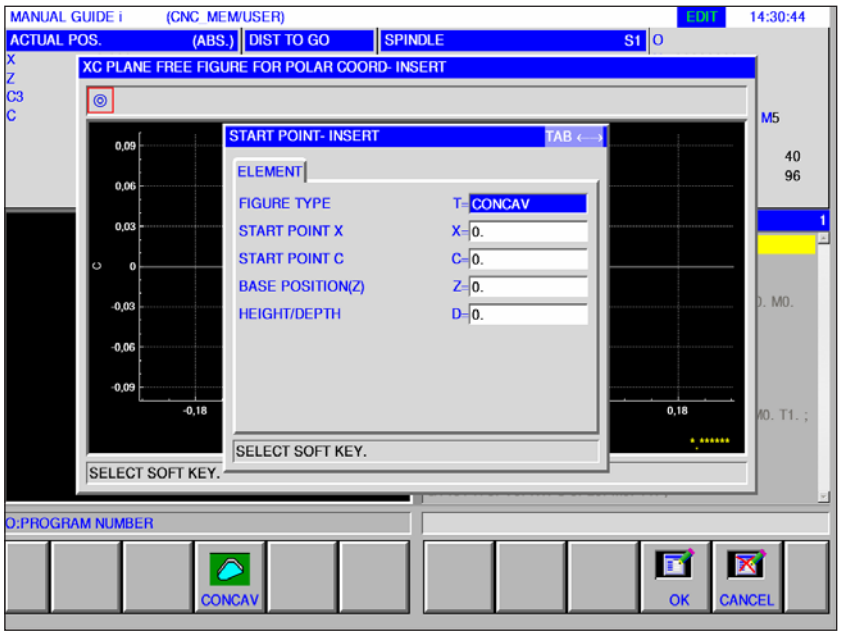
Entry fields marked with * are optional and are not required to be populated.

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
B	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
L	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height
H	X-axis position 1 (radius)	X-coordinate of the centre point of a rectangular contour. (radius value) virtual Cartesian coordinate.
V	C-axis position	C-coordinate of the centre point of a rectangular contour. Virtual Cartesian coordinate.
E	Angle number	Number of angles (positive value) Integer value between 3 and 99.
U	Entry type size	• [RADIUS]: Polygon radius • [LENGTH] : Side length of polygon • [WIDTH] : Key width
W	radius / length / key	• Polygon radius at entry [RADIUS] • Polygon side length at entry [LENGTH] • Key at entry [WIDTH]
A*	Rotation angle	Gradient of a rectangular contour to the X-axis (positive or negative value).

Corner shape		
Data element		**Meaning**
C	Corner type	• [NOTHIN] : no corner shape • [CHAMFER]: Chamfer • [ARC]: Corner radius
R	Corner size	Radius for corner rounding

FIGURE | POCKET-FIGURE | 5. XC FREE CONCAVE FIGURE

XC Free Contour Concave G1500



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

Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
X	Start point X	X-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
Z	Basis position Z	Z-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height

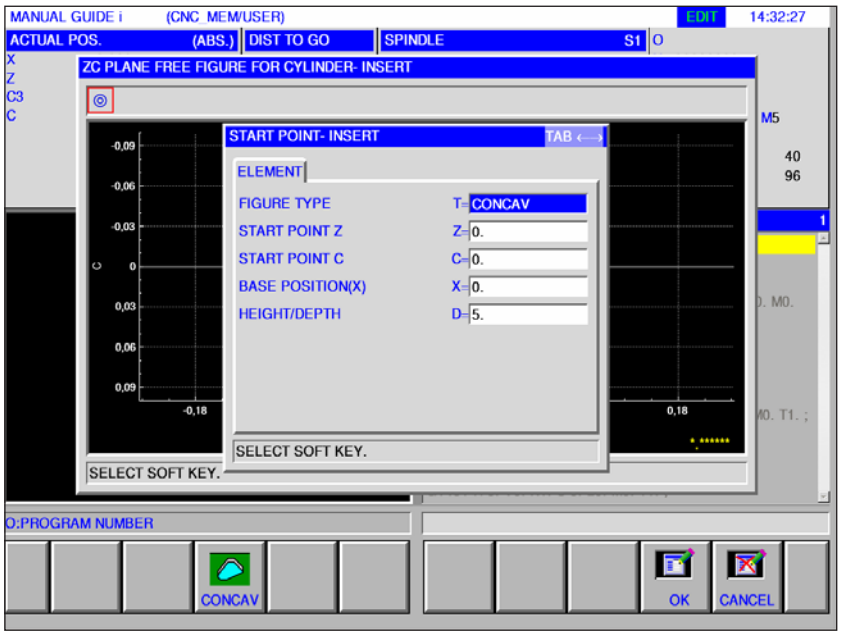
Note:
INPUT DATA are elements that are displayed when changing or editing in the data input window.
This applies for cycles G1451 to G1456.

Note:
OUTPUT DATA are elements that are displayed in ISO code format as the generated program in the program window. You can activate these only for program display purposes.
This applies also for cycles G1451 to G1456.

FIGURE | POCKET-FIGURE | 6. ZC PLANE FREE CONCAVE FIGURE FOR CYLINDER

ZC Free Contour Concave on Cylindrical Surface G1600

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



Position / Size		
Data element		**Meaning**
T	Contour type	• [CONCAVE]: used as outside contour for contour machining.
Z	Start point Z	Z-coordinate of the start point of any contour.
C	Start point C	C-coordinate of the start point of any contour.
X	Basis position X	X-coordinate of the upper surface or the bottom surface of the side surface of a part, on which the contour machining is being performed (in the tool axis direction).
D	Hight/depth	If the top surface of a work piece 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 work piece is selected as BASISPOSITION, the distance to the top surface of the side is given as a positive (radius value). → height



Subroutines

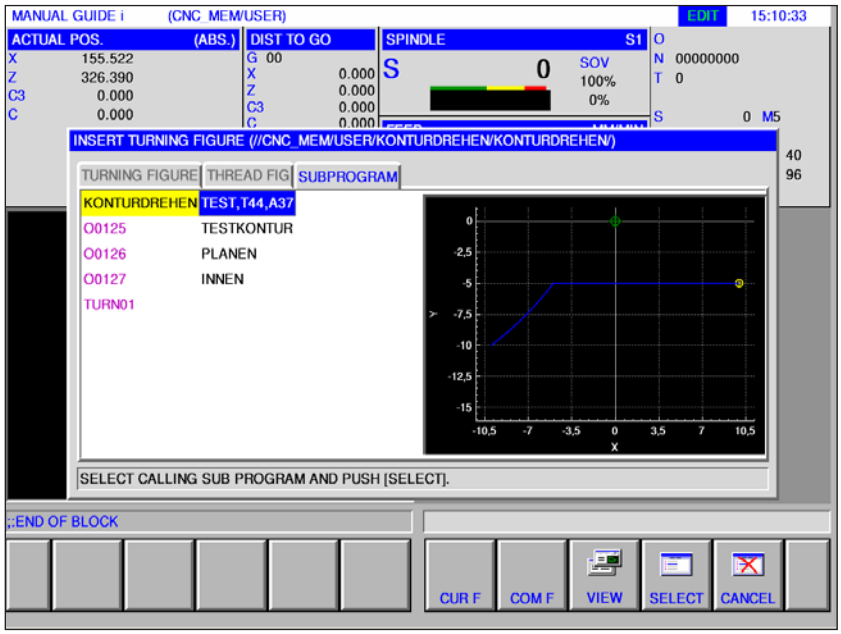


Fixforms



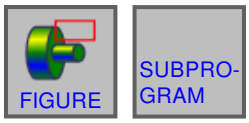
M-code menu

Sub-routines





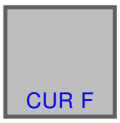
1 Select "Edit" mode.



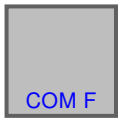
2 Press softkey and register card



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



4 The register card "CUR F" 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 softeky again to turn off the preview.

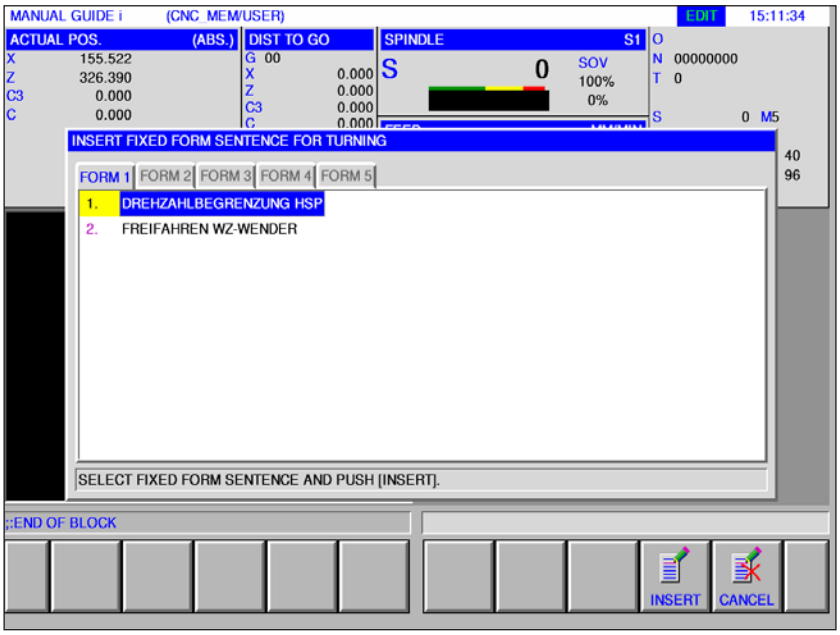
Programming

M98 Call subroutine
M99 Jump back to the calling program

Example

M98 P1234

Insert 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.

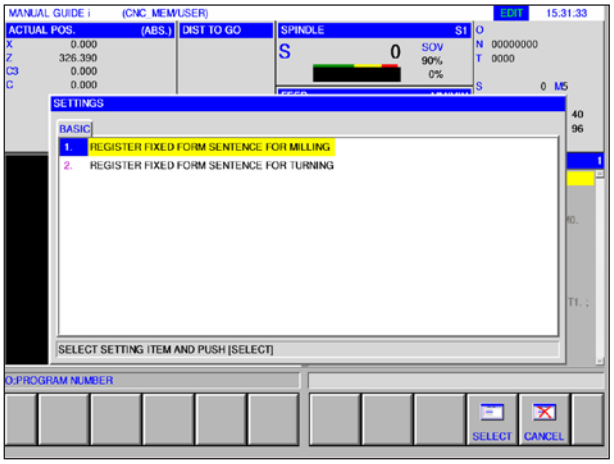


3 Select the desired Fixform with "INSERT" and insert it into the program.

Creating fixed forms



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



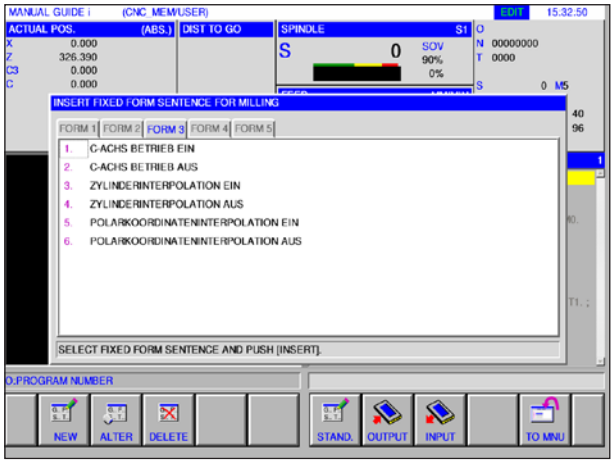
Note:
Fixed forms can be created in all operating modes.



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".

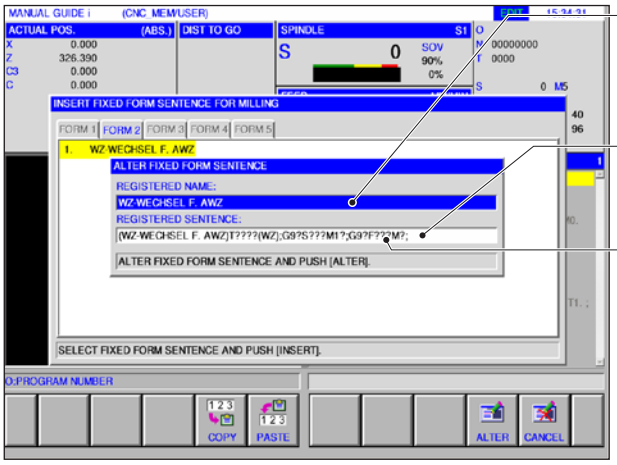




4 Press the softkey to create a new fixed form or to process an existing fixed form.



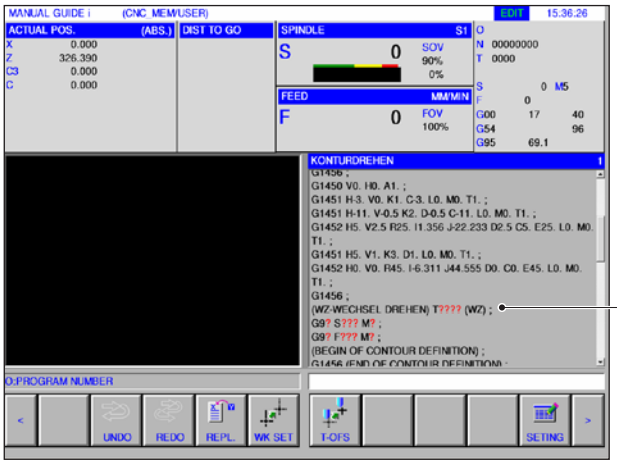
5 Press softkey to delete a fixed form.



6 Enter or change registration name.

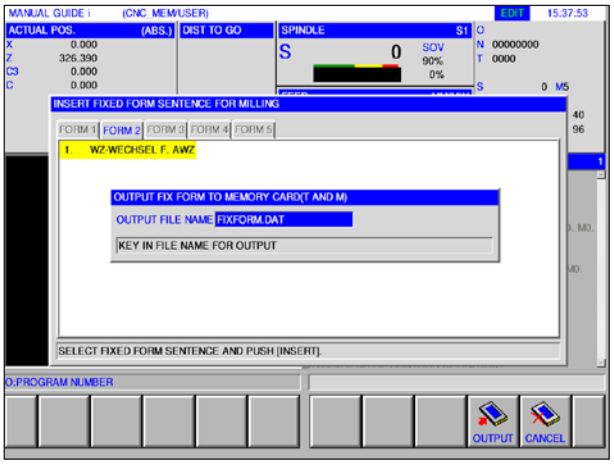
7 Enter or change registration set.

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

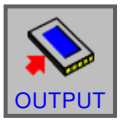


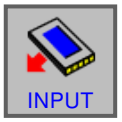
Example: Tool change for turning

(TOOL CHANGE TURNING) T???? (TOOL);
G9? S??? M?;
G9? F??? M?;



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





9 Press softkey to open dialogue for saving or uploading.

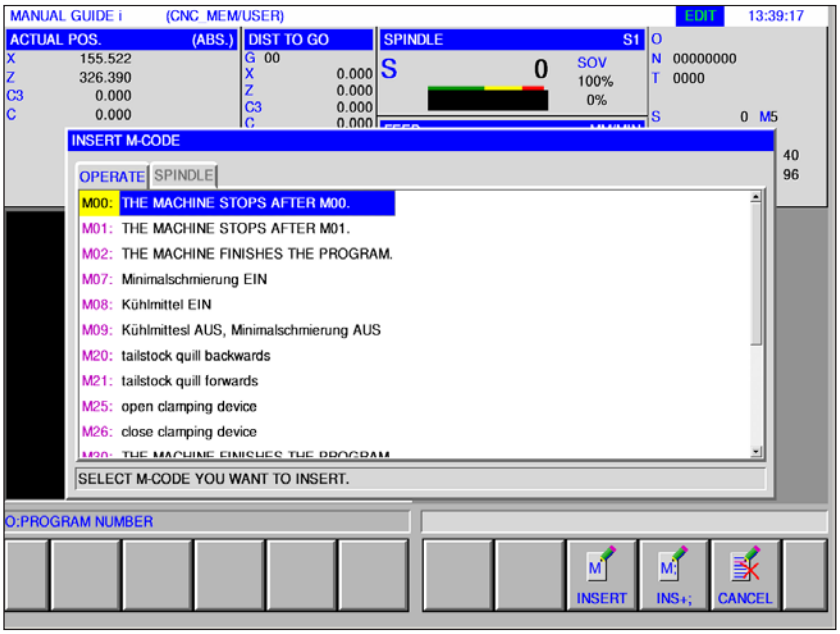


10 The softkey "STAND." performs a reset of the fixed forms to the condition after the software installation (corresponds to the condition pre-set by the factory).
Fixed form sets that have been entered or altered will be deleted or reset.



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 TURN 60 turning machine has no position-controlled main spindle, so neither is it possible to program spindle positions.

M- commands overview

M00	Programmed stop
M01	Optional stop
M02	End main program
M03	Main spindle ON, clockwise
M04	Main spindle ON, counterclockwise
M05	Main spindle OFF

Commands for the spindles of power tools in the different M-code systems: (Switched in EmConfig)	FANUC standard *)	EMCO Training	EMCO Industry
AWZ spindle ON clockwise	M103	M13	M303
AWZ spindle ON counterclockwise	M104	M14	M304
AWZ spindle OFF	M105	M15	M305

M07	Minimal lubrication ON
M08	Coolant ON
M09	Coolant OFF, minimal lubrication OFF

Commands for C-axis mode in the different M-code systems	FANUC standard *)	EMCO Training	EMCO Industry
C-axis mode activation	M14	M52	M52
C-axis mode deactivation	M15	M53	M53

M20	Quill BACK	M71	Purge ON
M21	Quill ADVANCE	M72	Purge OFF
M25	OPEN clamping device	M90	Manual chuck
M26	CLOSE clamping device	M91	Draw clamp
M29	Rigid tapping without length compensation	M92	Pressure clamp
M30	End main program	M98	Call sub-routine
M32	End main program with restart	M99	Jump back to calling program

*) FANUC standard is the default setting.

Command abbreviations overview

Part 1 valid for turning and milling

Command	Meaning
AND	Logical AND link
DIV	Integer division
DO	Loop construct
END	Loop construct
EQ	Equal
FUP	Round up
GE	Greater than or equal to
GT	Greater than
GOTO	Loop construct
IF	Loop construct
LT	Less than
LE	Less than or equal to
NE	Not equal to
OR	Loop construct
POW	Power
THEN	Loop construct
ROUND	Round down
WHILE	Loop construct
XOR	Exclusive OR

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

Overview of machine G-commands

The Fanuc controller uses the assignment groups A, B and C for the G-commands; i.e. the same command has different numbers in the various groups.
The Fanuc controller is programmed with the C assignment group commands, but it is group A that is described in the Fanuc documents. Go to the following table to find the description of the G-commands in the Fanuc documents.

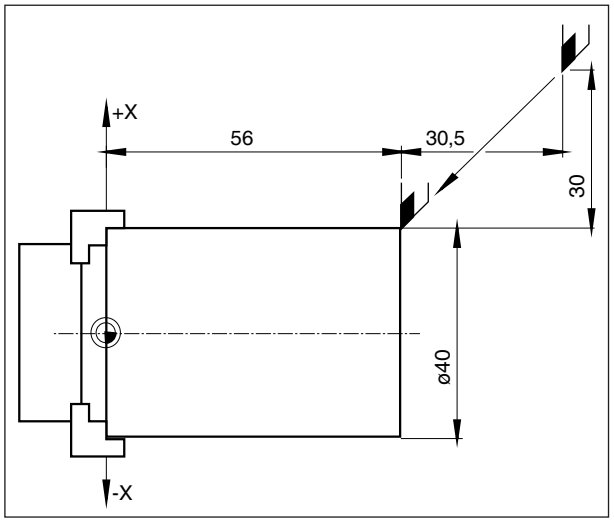
Code Grp. A	Code Grp. B	Code Grp. C	Meaning
G00	G00	G00	Rapid traverse
G01	G01	G01	Straight interpolation
G02	G02	G02	Circular interpolation/clockwise
G03	G03	G03	Circular interpolation/counterclockwise
G04	G04	G04	Dwell time
G07.1	G07.1	G07.1	Cylindrical interpolation
G10	G10	G10	Data setting
G12.1	G12.1	G12.1	"Polar coordinates interpolation" mode
G13.1	G13.1	G13.1	Real coordinates activation
G17	G17	G17	XY plane selection
G18	G18	G18	ZX plane selection
G19	G19	G19	YZ plane selection
G20	G20	G70	Switch to imperial (inches) input
G21	G21	G71	Switch to metric input
G32	G33	G33	Thread cutting
G40	G40	G40	Cutting radius offset deselection
G41	G41	G41	Cutting radius offset left
G42	G42	G42	Cutting radius offset right
G52	G52	G52	Additive zero point offset
G53	G53	G53	Movements related to machine coordinates system
G54	G54	G54	Workpiece coordinates system 1 selection
G55	G55	G55	Workpiece coordinates system 2 selection
G56	G56	G56	Workpiece coordinates system 3 selection
G57	G57	G57	Workpiece coordinates system 4 selection
G58	G58	G58	Workpiece coordinates system 5 selection
G59	G59	G59	Workpiece coordinates system 6 selection
G65	G65	G65	Macro call
G66	G66	G66	Modal macro call
G67	G67	G67	End modal macro call
G70	G70	G72	Finishing cycle

Code Grp. A	Code Grp. B	Code Grp. C	Meaning
G71	G71	G73	Material degradation while turning
G72	G72	G74	Material degradation while plane turning
G73	G73	G75	Contour repetition
G74	G74	G76	Axial piercing cycle
G75	G75	G77	Radial piercing cycle
G76	G76	G78	Thread cycle in more than one process
G80	G80	G80	Delete modal call
G83	G83	G83	Frontal drilling cycle
G84	G84	G84	Frontal tapping
G87	G87	G87	Radial drilling cycle
G88	G88	G88	Radial tapping cycle
—	G90	G90	Absolute dimension programming
—	G91	G91	Incremental dimension programming
G50	G92	G92	Speed limitation
G98	G94	G94	Feed in mm/min
G99	G95	G95	Feed in mm/revolution
G96	G96	G96	Constant cutting speed
G97	G97	G97	Spindle rotation as constant rpm
—	G98	G98	Fixed cycle: Return to start plane
—	G99	G99	Fixed cycle: Return to plane with point R

Brief description of G-commands

There is a brief description of the G-commands for the WinNC controller for Fanuc 31i, assignment group C.
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.

Instructions:
These instructions describe the command breakdown for assignment group C, as it is used on the Fanuc controller on the EMCO machine.



Absolute and incremental values for G00

G00 Rapid traverse

Format
N.... G00 X... Z...

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

Notes

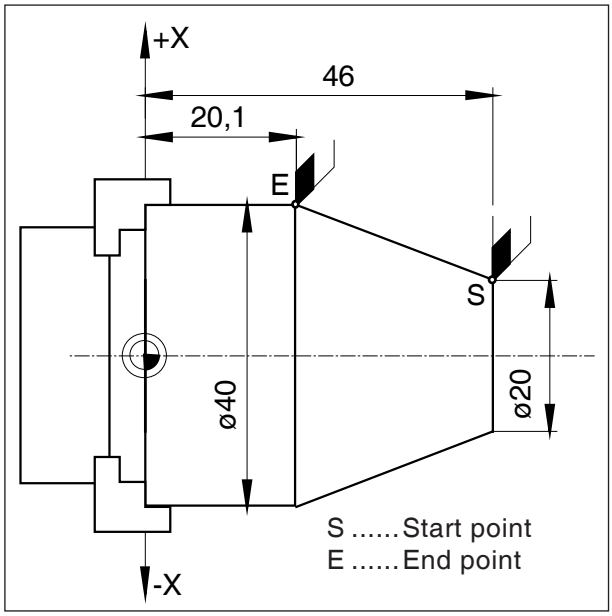
- A programmed carriage feed F is suppressed during G00.
- The rapid traverse speed is set to fixed.
- The feed offset switch is limited to 100%.

Example

absolute G90
N50 G00 X40 Z56

incremental G91
N50 G00 X-30 Z-30.5

G01 Straight interpolation



Absolute and incremental values for G01

Format
N... G01 X... Z.... F....

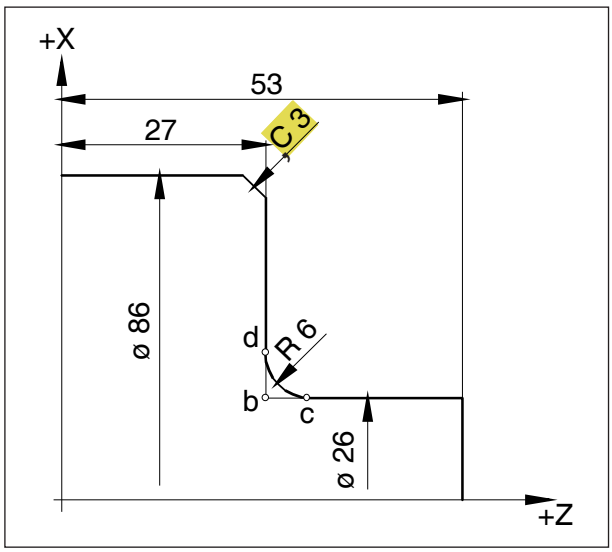
Straight line movement with programmed feed speed (plane, longitudinal, conical turning)

Example

absolute G90
N.. G95
.....
N20 G01 X40 Z20.1 F0.1

incremental G91
N.. G95 F0.1
.....
N20 G01 X20 Z-25.9

Entering chamfers and radii



Entering chamfers and radii

Example
....

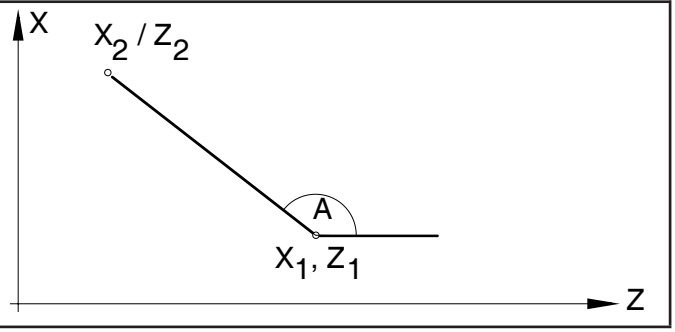
N 95	G 01	X 26	Z 53	
N 100	G 01	X 26	Z 27	,R 6
N 105	G 01	X 86	Z 27	,C 3
N 110	G 01	X 86	Z 0	

....

Notes

- Chamfering and radii can be inserted only between two G01 movements.
- The movement which is programmed in the second block must begin with point b in the diagram.
  With incremental value 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.
- 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 is generated.

Direct drawing dimension input

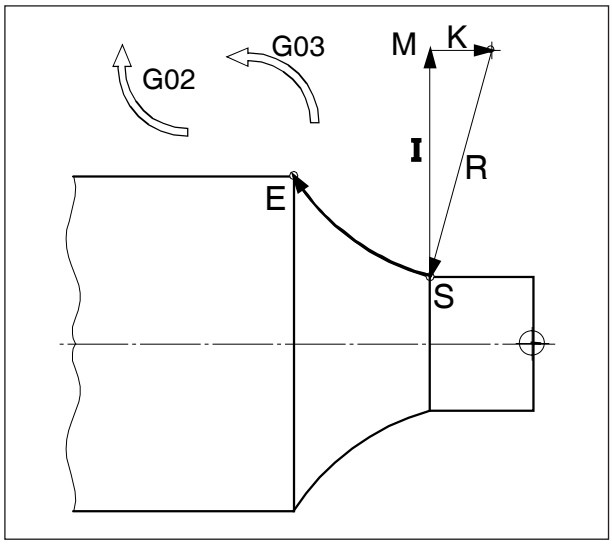
	Commands	Tool movements
1	***X_2... (Z_2...) ,A...***	

Information:

- 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.
- The following G-commands should not be used for blocks with chamfer or radius:

  G7.1, G10, G11,
  G52, G53,
  G72, G73, G74, G75, G76

  G02, G03, G20, G21
- These must not be used between blocks with chamfer or radius which define the sequence numbers.



Direction of rotation and parameters of an arc

G02 Circular interpolation, clockwise

G03 Circular interpolation, counterclockwise

Format

N... G02 X... Z... I... K... F...
or
N... G02 X... Z... R... F...

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

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

R ..... Radius of the arc

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

Notes

- If either I or K is equal to 0, the relevant parameters must not be given.
- If R < 0: Arc ≥ 180°
- If R > 0: Arc < 180°.

G04 Dwell time

Format

N... G04 X... [sec]
or
N... G04 P... [msec]

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

G7.1 Cylindrical interpolation

Note:
The unit for cylindric interpolation with G07.1. is preselected in mm. The setting can be changed in EMConfig. Select between mm and degree.

This function implements a cylindrical surface in the programming.
This is how, for example, programs for cylindrical cam machining can be produced on lathes.

The traverse of the turning axis C programmed by the angle data is changed internally by the controller into the distance of a virtual linear axis along the outer surface of the cylinder.
This is how it is possible for linear and circular interpolations to be executed on this surface with a different axis.

Format

G1 G91 Z0 C0;	Determine the centre point of the reference plane (PRM1022#C=6).
G7.1 C[r];	activates cylindrical interpolation mode with cylindrical radius information (in [mm] for computing the movement feed).
:	Geometry program.
G7.1 C0;	ends cylindrical interpolation mode.
G18;	back to main plane

Computing Y [mm]- in C[°] coordinates for path programming

$$C_p = \frac{Y_p[mm] \cdot 360[°]}{2\pi \cdot R_{cyl}[mm]}$$

C_p [°].....path to be traversed in C-axis
Y_p [°].....drawing dimension
R_{cyl} [mm].....Radius of the cylinder jacket

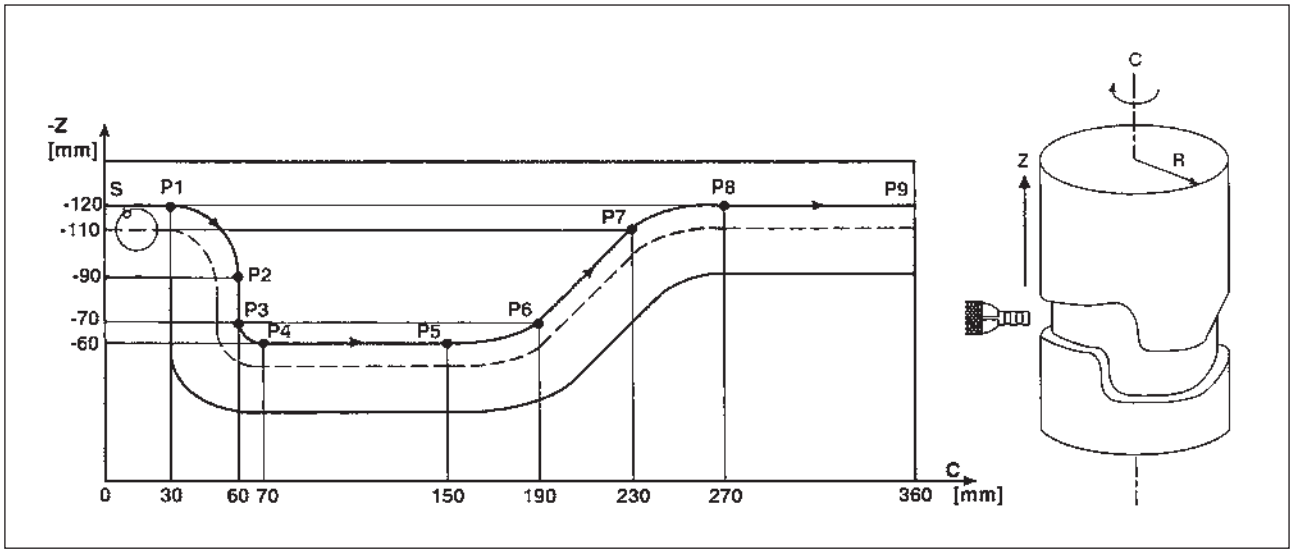
G19 determines the plane in which the rotating axis C is stipulated as a linear axis parallel to the Y-axis. The block structure for the geometry program is then as follows:
G1 Z.. [in mm] C.. [in °]; straight interpolation.
G2 (G3) Z.. [in mm] C.. [in °] R.. [in mm]; circular interpolation.

Information:

- The reference point of the cylinder must be entered incrementally, as otherwise this would be approached instead of the tool!
- Cutting position 0 must be assigned to the tool in the offset data. The mill radius, however, must be entered.
- In G7.1 mode, the coordinate system must not be changed.
- G7.1 C.. or G13.1 C0 must be programmed in "Cutting radius offset OFF" (G40) and cannot be started or ended within "Cutting radius offset ON" (G41 or G42).
- G7.1 C.. and G7.1 C0 are to be programmed in separate blocks.
- In a block between G7.1 C.. and G7.1 C0, an interrupted program cannot be restarted.
- The arc radius for circular interpolation (G2 or G3) must be programmed via a R-command and must not be programmed in degrees or via K and J coordinates.
- In the geometry program between G7.1 C.. and G7.1 C0 you must nor program rapid traverse (G0)or any position processes or boring cycles (G83 to G89).
- The feed entered in cylindrical interpolation mode is to be understood as a traverse speed on the developed cylindrical surface.

Example - Cylindrical interpolation at the main spindle

X-axis with diameter and C-axis with angle programming.



N2 T0202 (mill diameter 12); AWZ- radial/ cutting radius 6.0, cutting position 0.

G97 S1000 M13;	constant tool speed (...or M14)...AWZ selection and direction of rotation of the tool.
M52;	Main spindle selected as C-axis.
G52 C.. ;	any offset of the C-axis angle.
G28 C0;	Reference. C-axis (only after offset and needed once after the first call by M52).
G40 G0 X120 C0 Z-100;	Position the tool in rapid traverse (possibly deselect cutting radius offset).
G1 G91 Z0 C0;	Determine the centre point of the reference plane
G7.1 C57.299;	Start cylindrical interpolation with cylindrical radius information in [mm].
G90 G42 G1 G94 Z-120 F350;	Select cutting radius offset outside the workpiece on point S.
X96 F100;	deliver in X.
C30 F250;	at point 1.
G2 Z-90 C60 R30 F250;	at point 2.
G1 Z-70;	at point 3.
G3 Z-60 C70 R10;	at point 4.
G1 C150;	at point 5.
G3 Z-70 C190 R75;	at point 6.
G1 Z-110 C23;	at point 7.
G2 Z-120 C270 R75;	at point 8.
G1 C360;	at point 9.
X120 F350;	raise in X.
G40 Z-100;	deselect cutting radius offset outside the workpiece.
G7.1 C0;	End of cylindrical interpolation by deselecting the cylinder radius'.
G95;	Deselect G94 (feed in mm/min).
G52 C0;	possibly reset the angular offset in the C-axis.
M53;	Main spindle selected as C-axis.
G0 X150 Z150 M15;	Freewheel in rapid traverse and AWZ speed off.
G18;	back to main plane

G10 Data setting

The G10 command can overwrite control data, program parameters and write tool data, etc.
In user-related practice, G10 is recommended mainly for programming the workpiece zero point.

Zero point offset with workpiece coordinate systems G54 to G59

In general, with this controller, it is recommended on reaching a workpiece zero point offset that you use one of the six preset workpiece coordinate systems G54 to G59, as here all overwritten values can be checked on screen at any time and are comprehensible. Commands G54 to G59 are described in more detail later in this chapter.)

Format

N.. G10 L2 P1 X.. Z
(overwrites G54 workpiece coordinates system)

N.. G54
(calls up G54 workpiece coordinates system into the program)

G10 L2 .....Overwrite the entered workpiece coordinates systems.
P0 .....Selection of the external workpiece coordinates system (EXT).
P1 (...to P6).....Selection of the workpiece coordinates system G54 (...to G59).

Information:
- The selected workpiece coordinates system in the program must be called up in the next block.
- Calling up the external workpiece coordinates system offsets the basis for all coordinates systems called up subsequently around the dimensions entered there.
- The selected workpiece coordinates system can be replaced via the a/m format inside a program as often as you like.

Simulation with Manual Guide-i

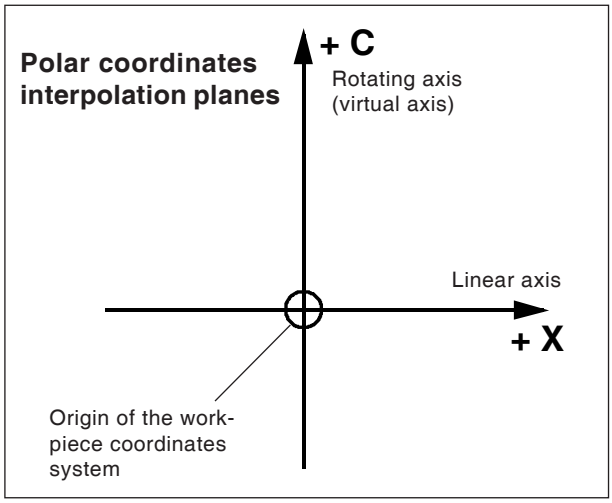
So that the simulation starts correctly with Z0, the workpiece coordinate system (e.g. G54) must be programmed before defining the unmachined part.

Program

O1234
G54 (set workpiece coordinate system)
G1900D100L100.K0.5W0. (Unmachined part definition)
G92S3500 (HSP.LIMIT)
(FREEWHEEL TOOL TURRET)
G18G40
G90T0
G0G53X245Z400M9

....
(Machining)
....
M30
%

G12.1/G13.1
Polar coordinates interpolation



Polar coordinates interpolation is suitable for machining the plane surface of a rotating part or grinding a camshaft on lathes.

It converts a command programmed in a Cartesian coordinates system into the movement of a linear axis X (tool movement) and a rotating axis C (workpiece rotation) for path control.

The rotating axis C also acts as the axis address for the second (virtual) axis.
This is created in coordinate C0 directly after programming G12.1.

Format

N.. G12.1;	starts the mode and allows polar coordinates interpolation
:	Geometry program (based on Cartesian coordinates)
N.. G13.1;	ends polar coordinates interpolation mode.

G12.1 selects a plane (G17) in which the polar coordinates interpolation is executed.
The plane G18 used before programming by G12.1 will be deleted.
It is reproduced via the command G13.1 (Polar coordinates interpolation end).

After switching on the machine or on a system RESET, the status "Polar coordinates interpolation" is deleted (G13.1) and the plane defined via G18 used.

Information:

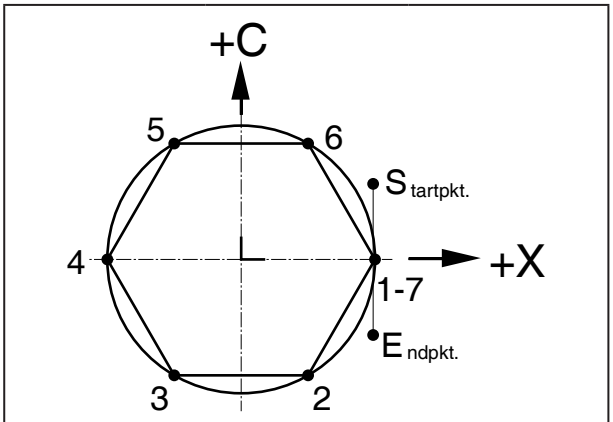
- Cutting position 0 must be assigned to the tool in the offset data.
  The mill radius, however, must be entered.
- In G12.1 mode, the coordinate system must not be changed.
- G12.1 or G13.1 must be programmed in "Cutting radius offset OFF" (G40) mode and cannot be started or ended within "Cutting radius offset ON" (G41 or G42).
- G12.1 and G13.1 are to be programmed in separate blocks.
  In a block between G12.1 and G13.1, an interrupted program cannot be restarted.
- The arc radius for circular interpolation (G2 or G3) can be programmed via a R-command or via I and J commands.

G-codes, which must be programmed in the "Polar coordinates interpolation" mode:

G-code	Use
G01	Linear interpolation
G02, G03	Circular interpolation
G04	Pause
G40, G41, G42	Tool radius offset (Polar coordinates interpolation is used after tool offset on the toolpath)
G65, G66, G67	User macro command
G98, G99	Feed per minute, feed per revolution

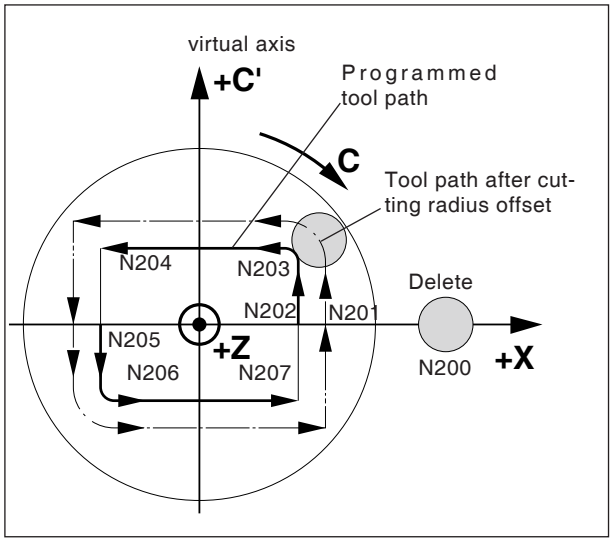
Example 1 - Polar coordinates interpolation

X- and C-axes with radius programming.



Point	X	C
S	17.32	10
1	17.32	0
2	8.66	-15
3	-8.66	-15
4	-17.32	0
5	-8.66	15
6	8.66	15
7	17.32	0
E	17.32	-10

N1 T0101 (mill diameter 10);
AWZ- axial/mill radius 5.0, cutting position 0.
G97 S1000 M13;
constant tool speed.
(...or M14)...AWZ selection and direction of rotation of the tool.
M52; Main spindle selected as C-axis.
G52 C.. ; any offset of the C-axis angle.
G0 C0; Reference. C-axis(needed only once after first call by M52 or after offset).
G40 G0 X30 C0 Z10;
Position the tool in rapid traverse (possibly deselect cutting radius offset).
G12.1; Start polar coordinates interpolation.
G41 G1 X17.32 C10 F0.1;
At point S in feed and cutting radius offset selection.
Z-6
C0; At point 1.
X8.66 C-15; At point 2.
X-8.66; At point 3.
X-17.32 C0; At point 4.
X-8.66 C15; At point 5.
X8.66; At point 6.
X17.32 C0; At point 7.
C-10; At point E.
Z5
G40 X30 Z10; move away from part in feed and deselect cutting radius offset.
G13.1; End polar coordinates interpolation.
G52 C0; possibly reset the angular offset in the C-axis.
M53; Main spindle selected as C-axis.
G0 X150 Z150 M15;
Freewheel in rapid traverse and AWZ rpm off.



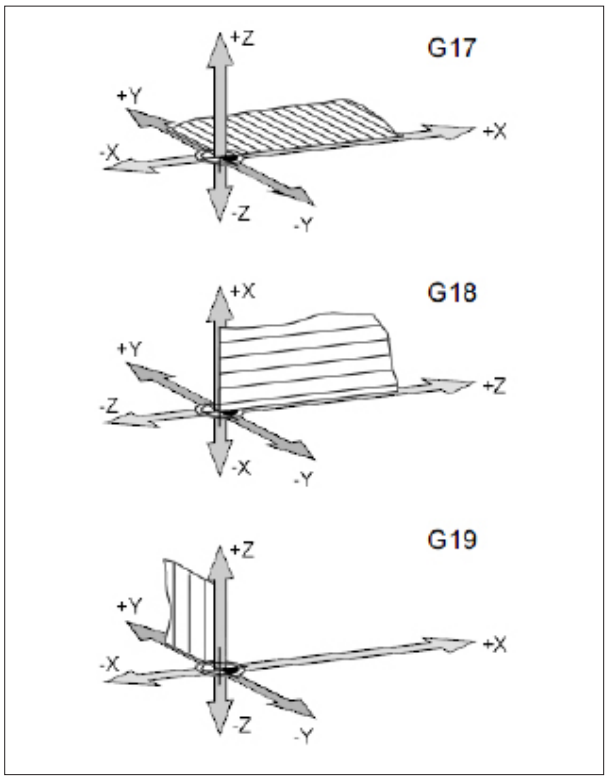
Polar coordinates interpolation with cutting radius offset

Example 2 - Polar coordinates interpolation with cutting radius offset

X- and C-axes with radius programming.

N1 T0101	Shaft mill ø10, powered, cutting radius 5,0, cutting radius position 0
G97 S1000 M13	constant spindle speed (or M14) direction of rotation
M52	C-axis selection
G0 X120.0 C0 Z5;	Positioning at the start position
G12.1;	Start polar coordinates interpolation
G42 G01 X20.0 F... ;	Geometry program. (based on Cartesian coordinates in the X-C' plane)
Z-10	
C10.0	
G03 X10.0 C20.0 R10.0 ;	
G01 X-20.0 ;	
C-10.0 ;	
G03 X-10.0 C-20.0 I10.0 J0 ;	
G01 X20.0 ;	
C0 ;	
G40 X60.0;	
G13.1;	End polar coordinates interpolation
M15	
Z... ;	
X... C... ;	
M30;	

G17-G19 Plane selection



Planes in the workspace

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.

Only for mill control:
The tool length offset is executed in the vertical axis on the active plane (G43 H..).

G17 XY plane
G18 ZX plane
G19 YZ plane

G28 Return to reference point

Format
N... G28 X... Z...

X, Z, ...Absolute intermediate coordinates

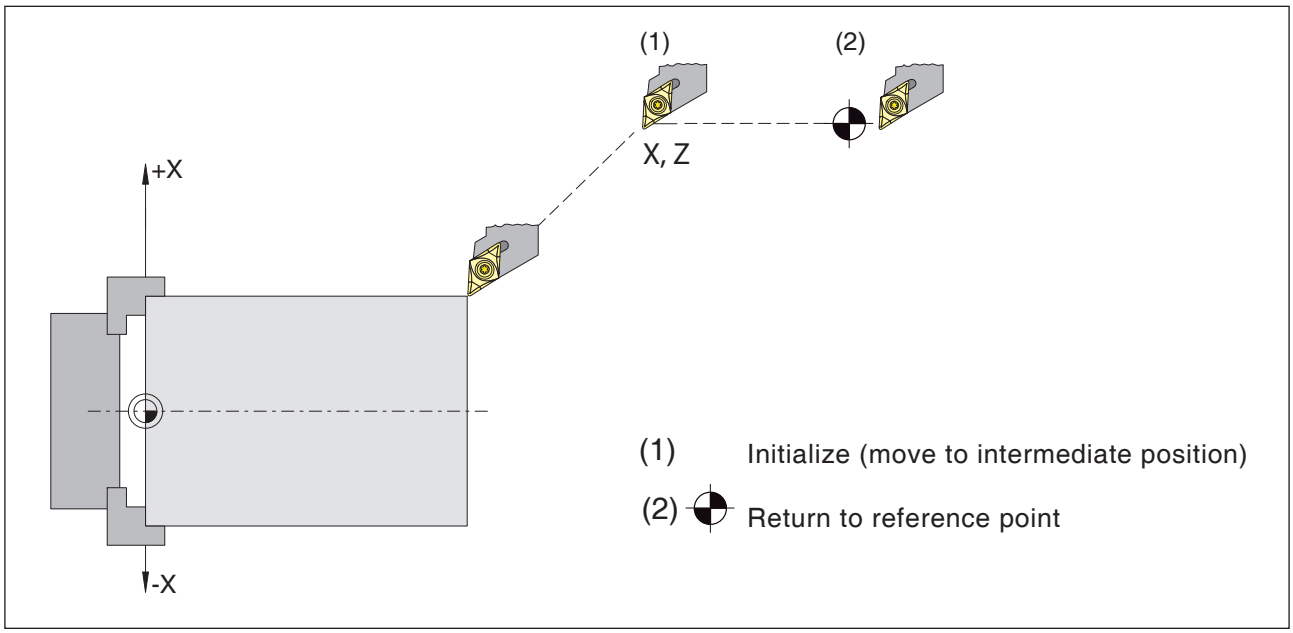
The G28 command is used to approach the reference point via an intermediate position (X, Z). First the movement to X, and Z, then movement to the reference point.

Both movements with G0.

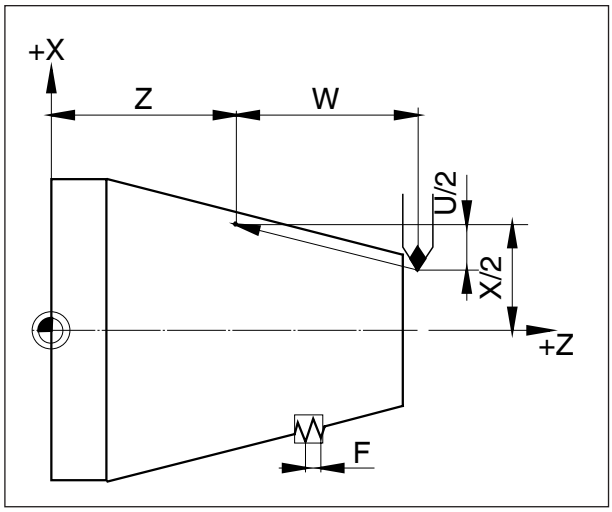
The movement to the intermediate position can be programmed incrementally:

G-Code group B/C
G91 G28 X10 Z10
G90

G-Code group A
G28 U10 W10



Return to reference point



Dimensions for thread cutting

G33 Thread cutting

Format
N... G33 X... Z... F... Q (option)

F .....Thread pitch [mm]
Q.....Starting point offset [mm]

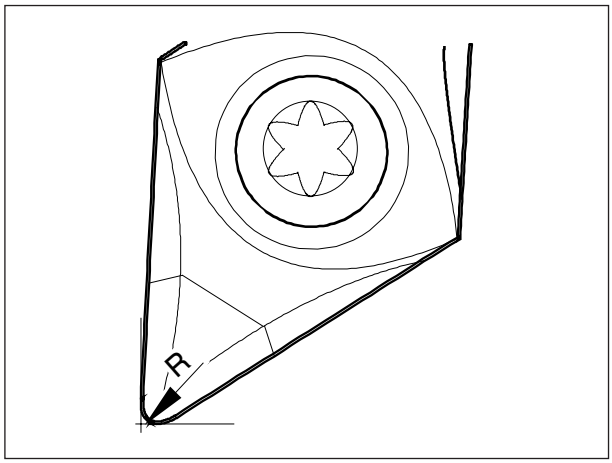
Straight, tapered and transverse threads can be cut.

As there is no automatic retraction at the start point, the multiple thread cycle G78 is mainly used.

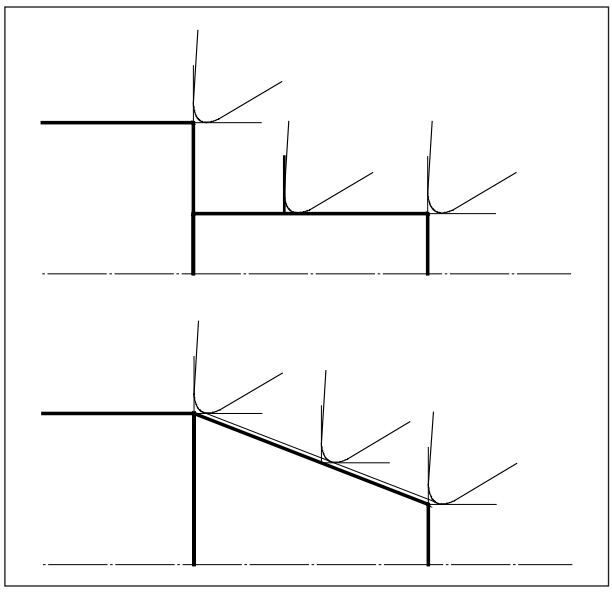
Machining processes such as knurling and diamond knurling are also feasible.

Notes

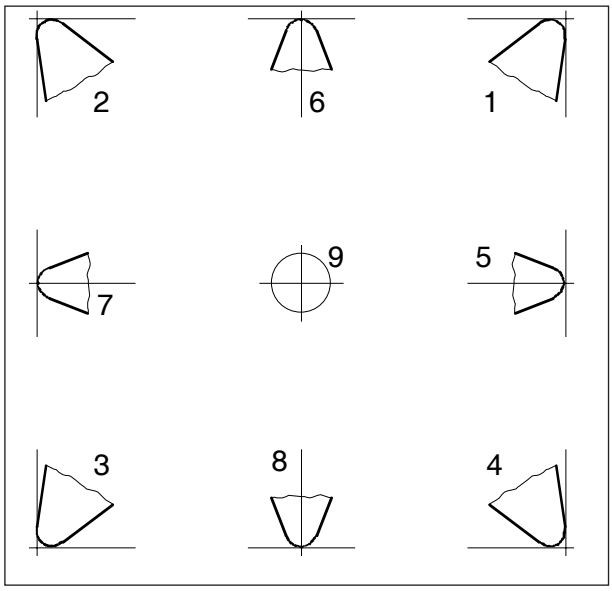
- With conical thread cutting, the pitch must be determined with the larger value in the X- or Z-axis.
- Continuous thread cutting is possible (multiple thread)
- The starting point offset Q applies only to multiple threads.



Tip radius and theoretical tooltip.



Axis parallel and oblique cutting movements



Cutting position

Cutting radius offset

During the tool measurement, the cutting plate is measured only at two points (tangential to the X- and Z-axes).

The tool correction therefore describes only a theoretical tooltip.

This point is moved to the workpiece on the programmed paths.

Movements in the axial directions (longitudinal or traverse turning) are made with the tangential points on the cutting plate.

There are therefore no dimensional errors at the workpiece.

For concurrent movements in both axial directions (cone, radii) the position of the theoretical cutting point is no longer the same as the actual cutting point on the tool plate.

This generates dimensioning errors at the workpiece.

Μαξιμυμ χοντουρ ερρορ ωιτηουτ χυττινγ ραδιυσ οφφσετ ωιτη 45° μοωεμεντσ:
χυττινγ ραδιυσ 0.4 μμ ≡ 0.16 μμ πατη σεπαρατιον ≡ 0.24 μμ σεπαρατιον ιν Ξ ανδ Z

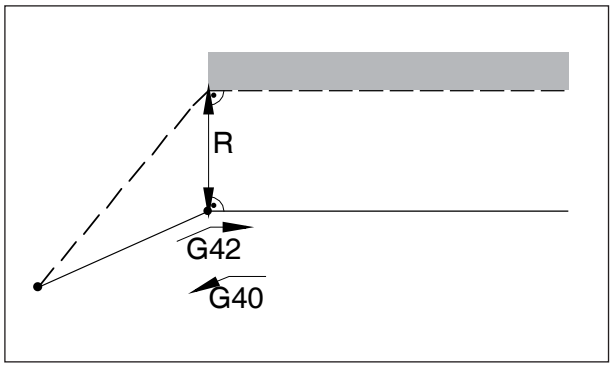
When using cutting radius correction, these dimensioning errors are computed and compensated by the control system automatically.

The cutting radius R and cutting position T are essential information during tool dimensioning for cutting radius offset.

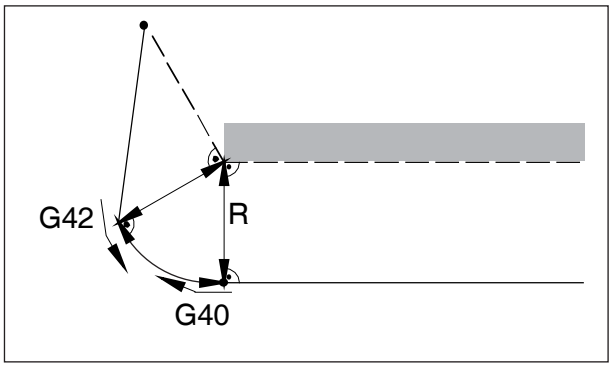
The cutting position is given by a variable (see sketch).

Consider the tool as if it is clamped to the machine to determine the cutting position.

Tool paths when selecting/deselecting cutting radius offset



Approach or move away from a corner point from the front

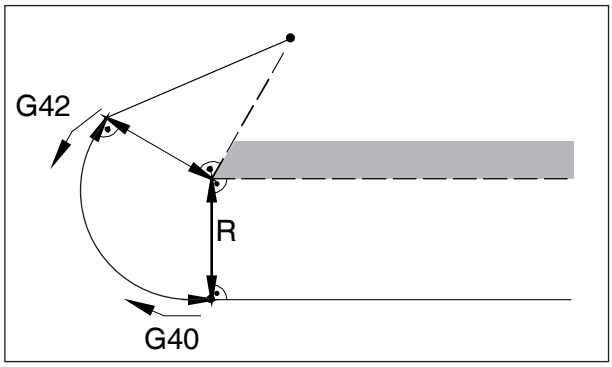


Approach or move way from the side to the rear

— — — programmed tool path
_____ actual tool path

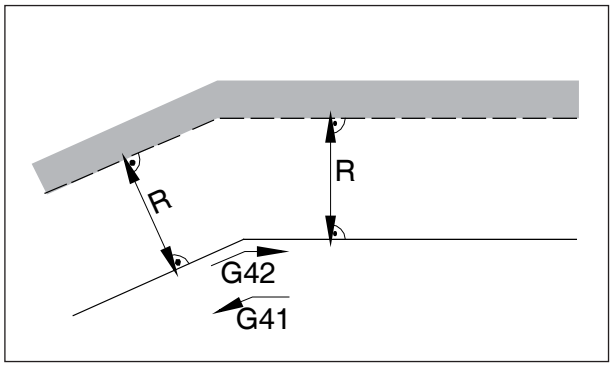
For arcs, you always approach on the tangents in the circle start/end point.

The approach path to the contour and the path away from the contour must be greater than the cutting radius R.

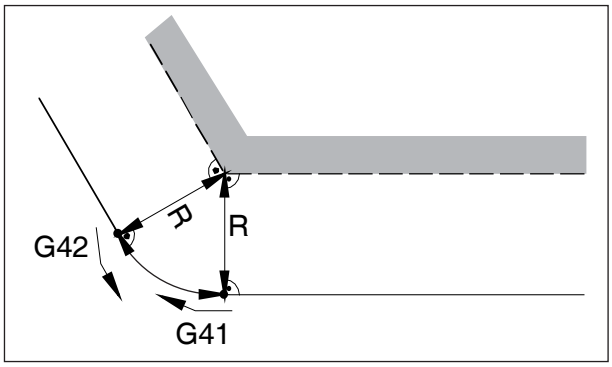


Approach or move away from a corner point from behind

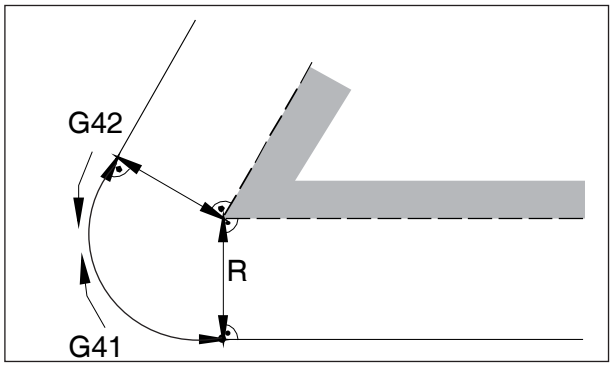
Tool paths in the program sequence with cutting radius offset



Tool path at an inside corner



Tool path at an outside corner > 90°



Tool path at an outside corner < 90°

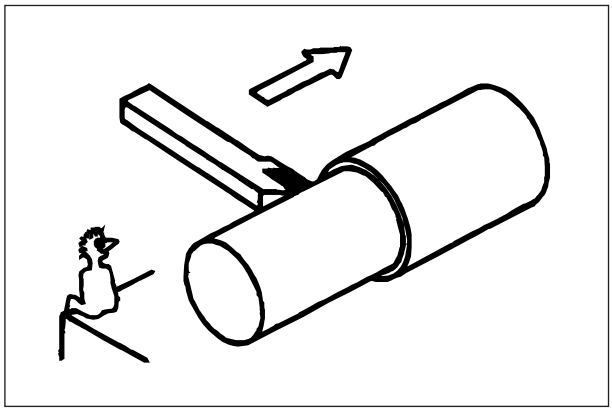
G40 Deselection mill radius offset

G40 deselects cutting radius offset.
Deselection is only permitted 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.

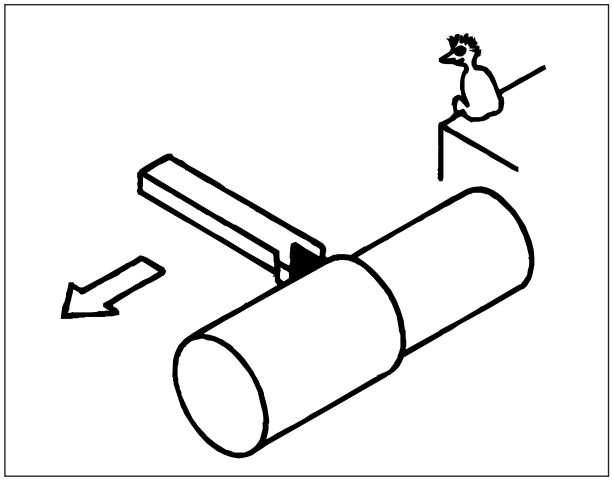
G41 Mill radius offset left

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

Notes
- Directly switched between G41 and G42 is not allowed - prior deselection with G40.
- The statement of the cutting radius R and the cutting position T is essential.
- Selection in connection with G00 or G01 required.
- Change of tool offset is not possible when tool radius offset selected.



Definition G41 Mill radius offset left



Definition G42 Mill radius offset right

G42 Mill radius offset right

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

For instructions see G41!

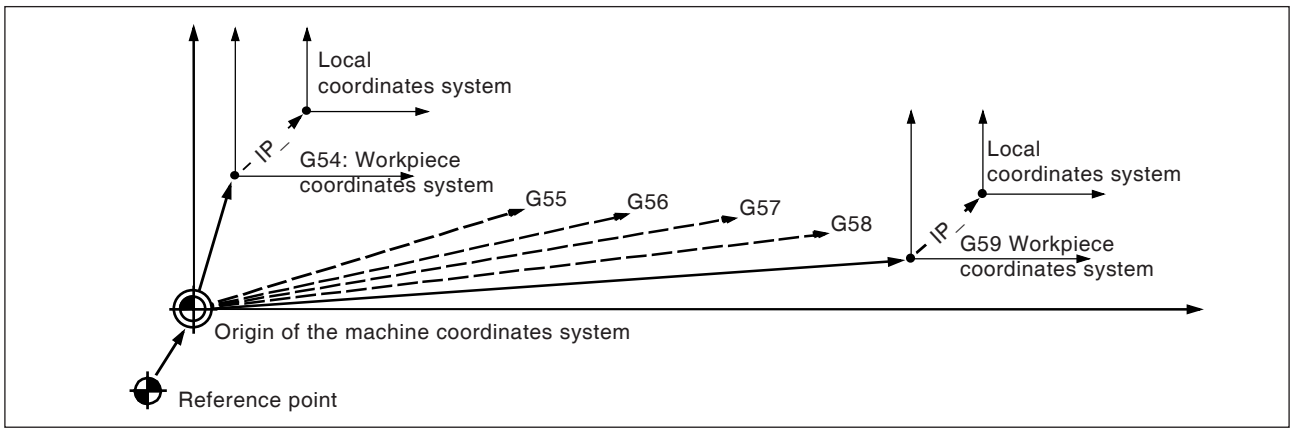
G52 Local coordinates system

When programming in a workpiece coordinates system (G54 to G59), a "daughter coordinates system" can be established within the selected workpiece coordinates system, in order to simplify the programming of certain functions.
Such a coordinates system is described as a local coordinates system.

Format
N.. G52 X.. Z.. C.. ;
(Establishing a local coordinates system)

N.. G52 X0 Z0 C0 ;
(Deleting a local coordinates system)

The workpiece zero point set in the selected workpiece coordinates system is offset by the the positive or negative dimension given in the local coordinates system.
After deleting the local coordinates system, the zero point on the workpiece zero point set originally in the selected workpiece coordinates system is reset.



Notes
- When establishing local coordinates systems, workpiece and machine coordinates remain unchanged.
- Traverse commands directly after selecting or deselecting the local coordinates system must be absolute dimension commands.

Caution:
Pressing the RESET button deletes the existing local coordinates system and overreads during the subsequent block process.
Even if coordinates for any axis of the local coordinates system are restated, the existing local coordinates system is deleted.

G53 Selecting the machine coordinates system

A coordinates system whose origin is on the machine zero point, is designated as the machine coordinates system.
G53 is a simply more effective (modally more effective) G-code for selecting the machine coordinates system.

Format
N.. T0 (tool offset deselection)
N.. G53 X.. Z..



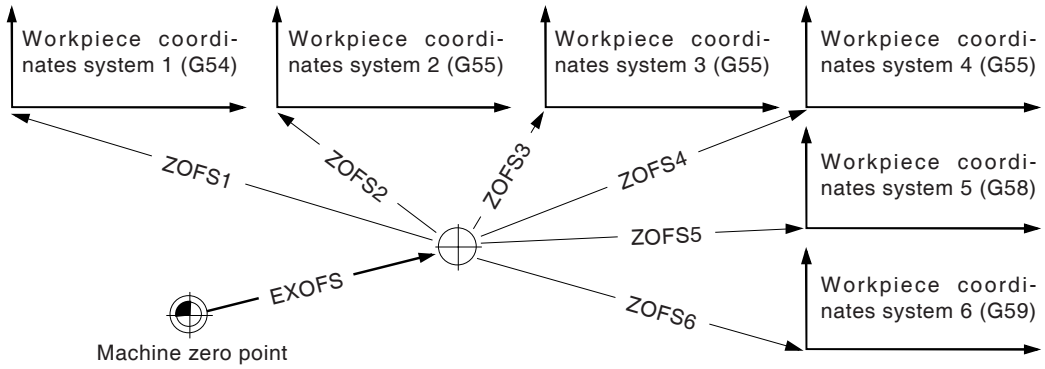
Information:
- "T0" must not be in the same block with G53!
- G53 must be stated in absolute dimensions. For incremental dimensional values, the G53 command is ignored.

G54 to G59 Selecting a workpiece coordinates system

A coordinates system which is used for machining workpieces is called a workpiece coordinates system. It is called up in a machining program and must be set up in the CNC before machining starts. A workpiece coordinates system set up earlier can be changed by offsetting the origin.
There are available six (specified via G54 to G59) preset workpiece coordinates systems and an external workpiece coordinates system:

EXT workpiece coordinates system 0
G54 workpiece coordinates system 1
G55 workpiece coordinates system 2
G56 workpiece coordinates system 3
G57 workpiece coordinates system 4
G58 workpiece coordinates system 5
G59 workpiece coordinates system 6

The EXT workpiece coordinates system offsets all workpiece coordinates systems specified via G54 to G59.



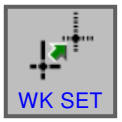
EXOFS: ..... External workpiece zero point offset value
ZOFS1 Ù ZOFS6 ..... Workpiece zero point offset value

There are two options for setting up the workpiece coordinates system:

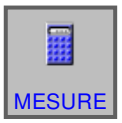
Entry via manual entry keyboard



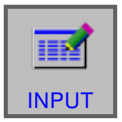
- MDI mode



- Press softkey.



- Measure tool

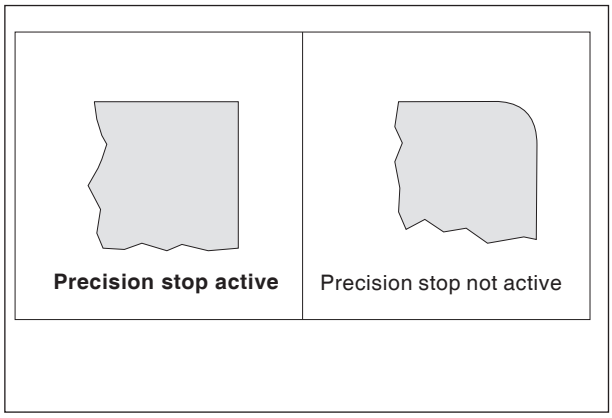


- Conclude measurement with "INPUT".

See details in chapter "G Tool management"

Programming via G10
Separate offsetting of the individual workpiece coordinates systems, including the external workpiece coordinates system (see under G10, data setting).

Notes
- The workpiece coordinates systems 1 to 6 are created after moving to the reference point.
- When switching on, the G54 coordinates system is selected.

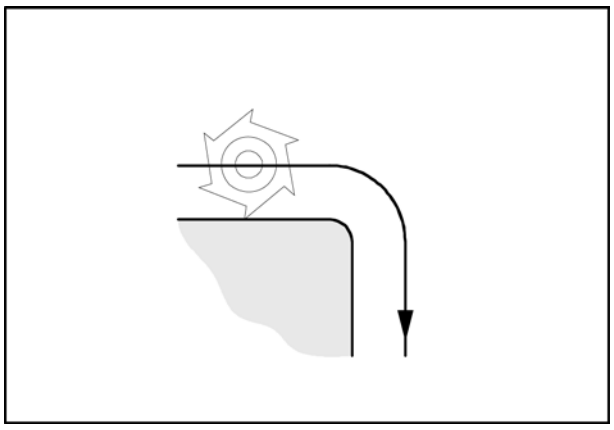


G61 Precision stop (modally effective)

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.

The tool is decelerated at the end point of a block, then an in-position check is made.
Then the next block is executed.



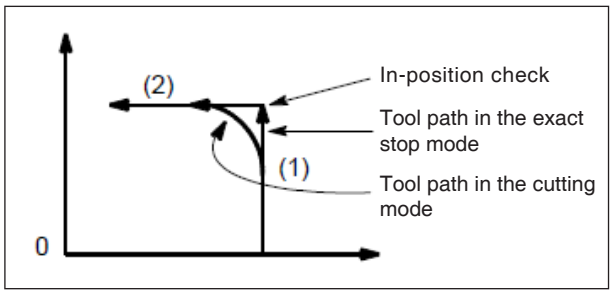
G64 Path control mode

G64 Path control operation

The contour will be created with as constant a path speed as possible.
Shorter processing times and rounded corners are created.

In tangential contour transitions, the tool moves with as constant a path speed as possible; in corners, the speed is reduced correspondingly.

The larger the feed F is, the greater the smoothing down of the corners (contour error).



Tool path

G65 Macro call

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

Format
G65 Pxxxx Lrrrr Arguments

or
G65 Pxxxx Lrrrr Arguments

Xxxx is the macro number (e.g. O0123)
- rrrr the repetition value
- "program.CNC" is the name of the macro file
- Arguments is a list of variable designators and values.

Arguments for macro calls are given by using the letters A-Z, excluding G, L, N, O and P.
Macros are written like normal programs. However, macro programs can access their arguments with numbers: #1 for A, #2 for B etc. (exceptions: # 4-6 for I-K, # 7-11 for D-H).

A macro can use the negative of an argument with a minus sign in front of the '#'. Other computing operations are not supported.
Macros can call other macros (up to 4 levels in depth), macro M-functions and sub-routines. Macro M-functions and sub-routines can call macros.

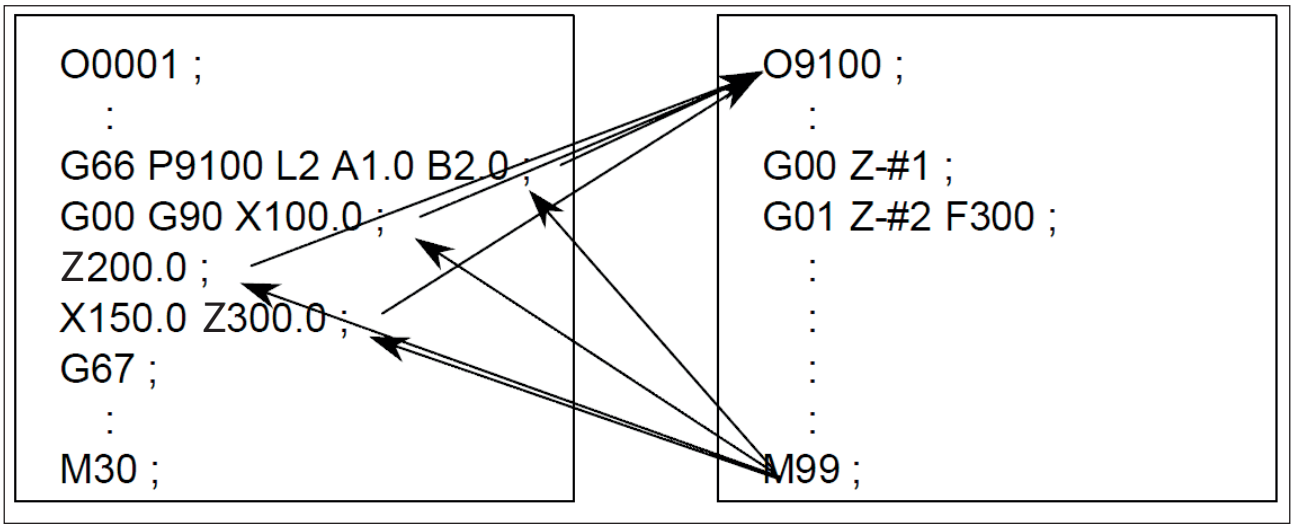
Example for a main program

G65 <TEST.CNC> A5 B3 X4
Macro TEST.CNC:
G1 X#X Y#A Z-#B
This call will produce
G1 X4 Y5 Z-3

G66 Macro call (modal)

Format
N.. G66 P.. L..Argument
..

P ..... Program number
L..... Number of repetitions (default is 1)
Argument... A list of variable designators and values which are transferred to the macro.

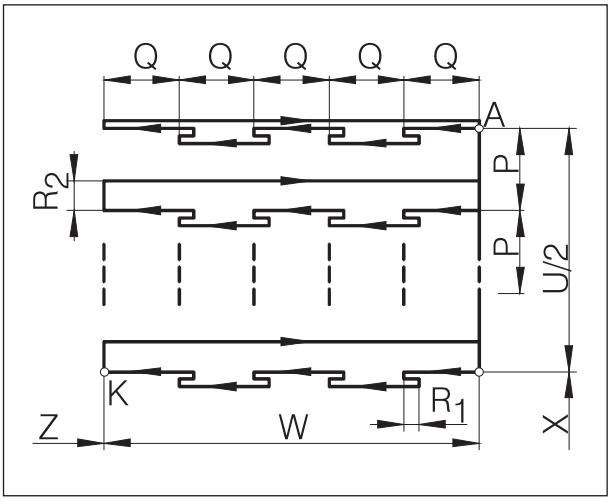


Example of macro call programming

G67 Macro call (modal) End

Format
N.. G67

This function ends the macro call.



Deep Hole Drilling / Face Grooving

G76 Deep Hole Drilling / Face Grooving Cycle

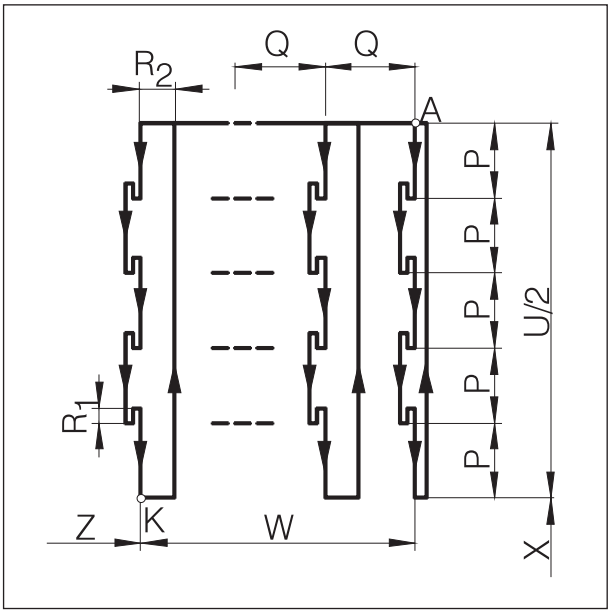
Format

N... G76 R...

N... G76 X(U)... Z(W)... P... Q... R_2... F...

first set	R_1 [mm]	Retraction hight for chip crushing (incremental without sign)
second set	X(U), Z(W)	Absolute (incremental) coordinates of the contour corner point K
	or	
	Z(W)	Absolute (incr.) Drill depth
	P [µm]	Incr. Infeed in X-direction (without sign); P < Tool width!
	Q [µm]	Cutting depth in Z-direction (without sign)
	R_2	Undercut amount at Z end point
	F	Feed

- If X(U) and P are not specified, the G76 command can be utilized as drilling cycle (Tool needs to be positioned on X0 beforehand!)
- When using the grooving cycle, make sure that the infeed P is smaller than the tool width B
- During the first cut, no undercut at the Z end point is performed
- Always enter a positive undercut amount



Longitudinal grooving

G77 Grooving cycle (X-axis)

Format

N... G77 R_1...

N... G77 X(U)... Z(W)... P... Q... R_2... F...

first set R_1 [mm] Retraction hight for chip crushing,

second set X(U), Z(W) Absolute (incremental) coordinates of K

P [µm] Cutting depth in X-direction (without sign); Plus infeed < Tool width

Q [µm] Incremental infeed in Z-direction (without sign)

R_2..... Undercut amount at X end point

F ..... Feed

- When using the grooving cycle, make sure that the infeed Q is smaller than the tool width B
- Tool width is not taken into consideration in the cycle
- During the first cut, no undercut is performed
- Always enter a positive undercut amount

G78 Multiple thread cutting cycle (ID or AD cycle)

Format

N... G78 P_1... Q_1... R_1...
N... G78 X... Z... R_2... P_2... Q_2... F...

first block:

P_1 ..... is a six-figure parameter, split into 2 groups:

PXXxxxx

The first two figures of this parameter define the number of finishing cuts

PxxXXxx

The centre two figures define the ***discharge amount*** P_F (see sketch)

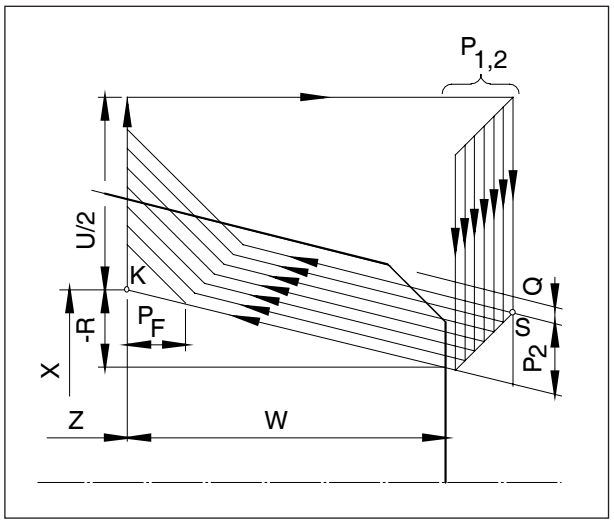
$$P_{xxXXxx} = \frac{P_F\ [mm] \times 10}{F}$$

PxxxxXX

Defines the edge angle of the thread in [°] .
(allowed: 80, 60, 55, 30, 29, 0)

Q_1 ..... Minimum cut depth [µm] incremental

R_1 ..... Finishing dimension [mm] incremental



Multiple thread cutting cycle

second block X, Z — Absolute coordinates of the point K

R_2 [mm] Incremental taper dimension with sign (R=0 cylindrical thread)

P_2 [µm] incremental Thread depth (always plus), represented in the drawing as P_2

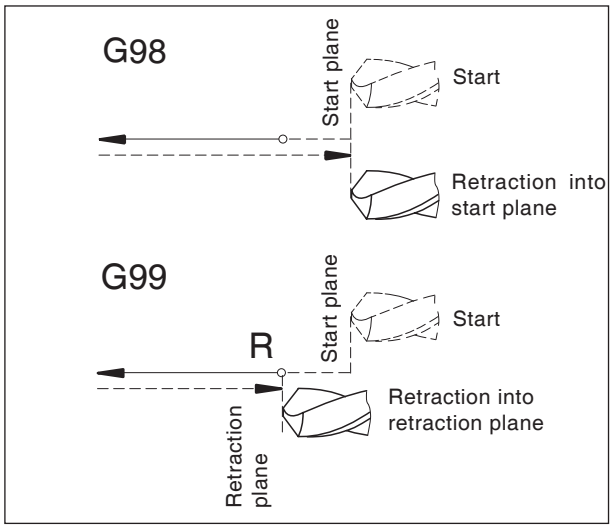
Q_2 [µm] incremental cutting depth in the first cut (radius value) without sign

F [mm] Thread pitch

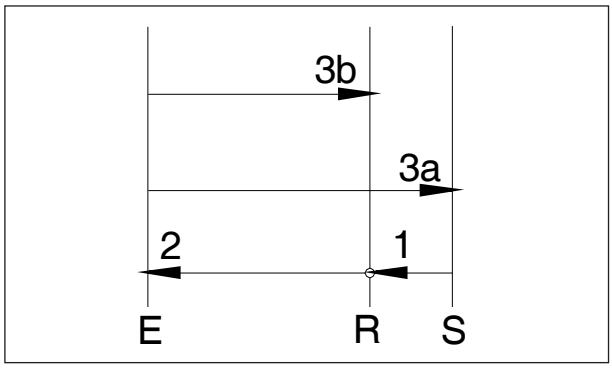
Boring cycles with power tools

Overview of the boring cycles

G-code	Boring axis	Hole machining (– direction)	Procedure at the bottom of the boring	Retract (+ direction)	Uses
G80	———	———	———	———	End of cycle
G83	Z-axis	Cutting feed intermittent	Dwell time	Rapid traverse	Plane boring cycle
G84	Z-axis	Cutting feed	Pause ➙ Spindle counterclockwise	Cutting feed	Plane thread tapping cycle
G85	Z-axis	Cutting feed	———	2× cutting feed	Plane reaming cycle
G87	X axis	Cutting feed intermittent	Dwell time	Rapid traverse	Side boring cycle
G88	X axis	Cutting feed	Pause ➙ Spindle counterclockwise	2× cutting feed	Side thread tapping cycle



Retraction patterns G98, G99



Retraction sequence G98, G99

G98/G99 systems

G98 .....After reaching the boring depth, the tool moves to the start plane
G99 .....After reaching the boring depth, the tool moves to the retraction plane - defined by the R parameter

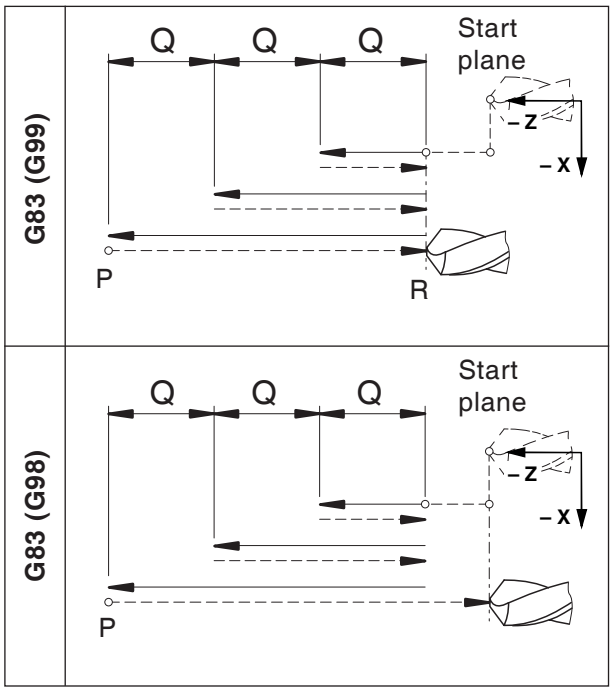
If G98 or G99 not active, the tool moves back to the start plane. If G99 (retraction to retraction plane) is programmed, the address R must be defined. With G98, R can be discarded!

R defines the position of the retraction relative to the last Z-position (start position for boring cycle). With a negative value for R, the retraction plane is under the start position, with a positive value above it.

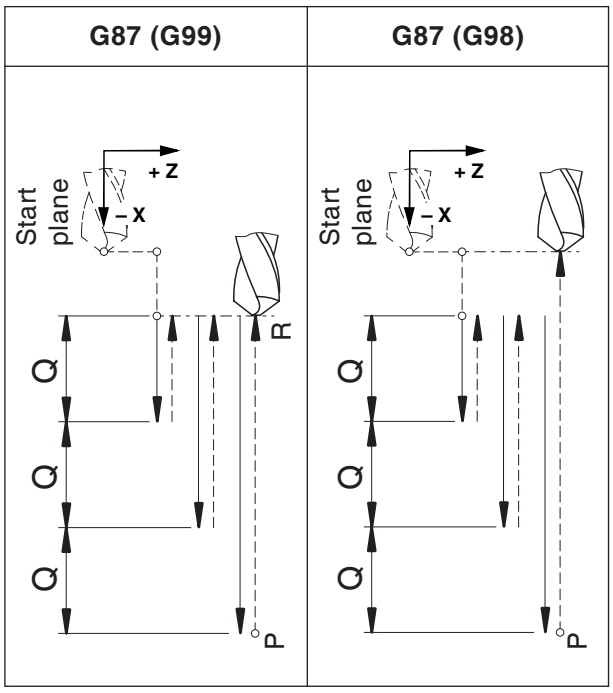
Movement sequence

1: From the start position (S), the tool moves in rapid traverse on to the plane (R) defined by R.
2: Cycle-specific boring machining until the end depth (E) is reached.
3: a: Retraction takes place with G98 up to the start plane (start position S).
b: with G99 up to the retraction plane (R).
4: The NC parameter 5102#6" defines how the parameter R must be programmed in dependency on G90/G91:
Incremental..... Value 0
Absolute..... Value 1

Instructions:
If the machine is set on 10^{-4} (0.0000) for the smallest increment, the incremental parameters in [µm] must be set with one extra decimal place.



Plane boring cycle G83



Side boring cycle G87

G80 Delete boring cycle (G83 to G89)

Format
N... G80
Boring cycles must be deselected, as they are modally effective through G80 or another Group 1 G-code (G00, G01, etc.).

G83 Plane boring cycle
G87 Side boring cycle

G83 format
N... G98(G99) G83 X... C... Z... R... Q... P... F... (M...) (K...)

G87 format
N... G98(G99) G87 Z... C... X... R... Q... P... F... (M...) (K...)

G98(G99) .. Retraction to start plane (retraction plane)
X ..... G83: Absolute hole position in the X-axis
G87: Absolute boring depth in the X-axis
Z ..... G83: Absolute boring depth in the Z-axis
G87: Absolute hole position in the Z-axis
C [°]..... Boring position
R [mm] ..... Incremental value of the retraction plane related to the start point in the Z/X-axis (with sign)
Q [µm] ..... incremental boring depth per delivery (without sign)
P [msec] .... Dwell time at bottom of hole:
P1000 = 1 sec
F ..... Feed
(M) ..... M-command for C-axis clamping (with defined parameter "5110").of the cycle repetitions
(K) ..... Number of cycle repetitions (only for incremental programming)

Notes
- The positions for borings C, X and Z in the cycle must not be programmed if the tool was traversed to the boring position in the preceding block. In this case, only the boring depth should be programmed (Z for G83, X for G87).
- If Q is not entered, no cutting division takes place and the Z end position is approached with a movement.
- Q must therefore be programmed together with each subsequent boring positioning .

Example - G83 boring cycle/deep hole boring cycle in Z-direction with power tool (plane boring cycle) on the main spindle

N3 T0303;	(Borer, AWZ-axial)
M52;	Main spindle selected as C-axis.
G97 S2000 M13	(...or M14)...AWZ selected and tool rotates clockwise (see note).
G0 X50 Z10 C30;	Positioning the tool.

G83 Z-42 R-8 Q6000 F0.5;
(Boring 1 (Q6000 = 6 mm cut depth per feed in deep hole boring)).

C150 Q6000;	Boring 2.
C270 Q6000;	Boring 3.
G80 M15;	Deselection boring cycle and switch off tool speed.
M53;	Main spindle deselected as C-axis.



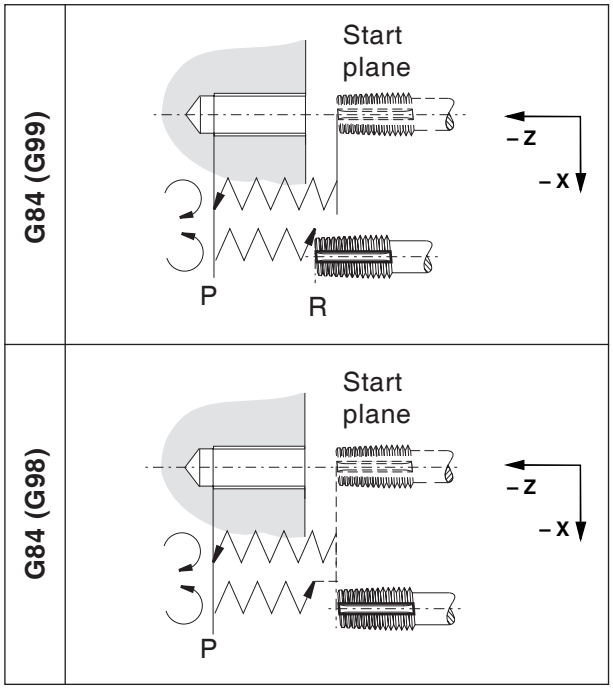
Instructions:
The default direction of rotation for power tools is for right tools.
The method of construction of the tool mounting used may require the direction of rotation to be reversed.

Example - G87 boring cycle/deep hole boring cycle in X-direction with power tool (side boring cycle) on the main spindle

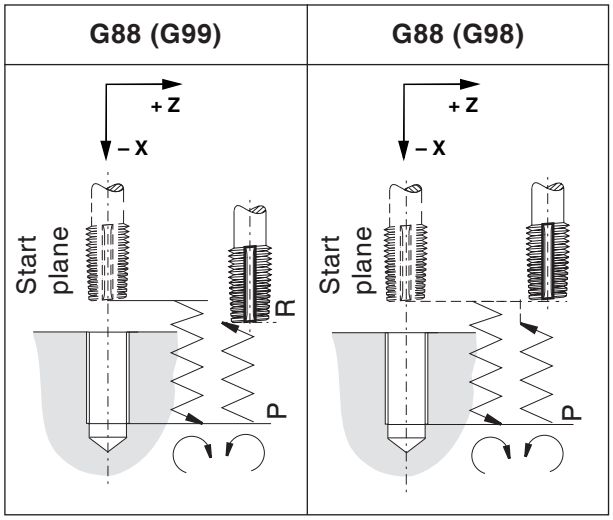
N7 T0707	(Borer, AWZ-radial);
M52;	Main spindle selected as C-axis.
G97 S2000 M13;	constant tool speed (...or M14)...AWZ selected and tool rotates clockwise (see note).
G0 X50 Z-20 C30;	Positioning the tool.

G87 X18 R-6 Q6000 F0.5;
(Boring 1 (Q6000 = 6 mm cut depth per feed in deep hole boring)).

C150 Q6000;	Boring 2.
C270 Q6000;	Boring 3.
G80 M15;	Deselection boring cycle and switch off tool speed.
M53;	Main spindle deselected as C-axis.



G84 plane thread tapping cycle



G88 side thread tapping cycle

G84 Plane thread tapping cycle with/without compensating chuck

G88 Side thread tapping cycle with/without compensating chuck

Thread tapping without compensating chuck is designated as "rigid thread tapping".
The cycles G84 and G88 can be executed with or without compensating chuck.

G84 format
N... G98(G99) G84 X... C... Z... R... P... F... (M...) (K...)

G88 format
N... G98(G99) G88 Z... C... X... R... P... F... (M...) (K...)

G98(G99) .. Retraction to start plane (retraction plane)
X ..... G84: Absolute hole position in the X-axis
G88: Absolute boring depth in the X-axis
Z ..... G84: Absolute boring depth in the Z-axis
G88: Absolute hole position in the Z-axis
C [°]..... Boring position
R [mm] ..... Incremental value of the retraction plane related to the start point in the Z/X-axis (with sign)
P [msec] .... Dwell time at bottom of hole:
P1000 = 1 sec
F ..... Feed
(M) ..... M-command for C-axis clamping (with defined parameter "5110").of the cycle repetitions
(K) ..... Number of cycle repetitions (only for incremental programming)

- The spindle turns clockwise for right tool mounting. The direction of rotation changes at the bottom of the boring, in order to feed the tool out of the thread tapping.

Instructions:

- The default direction of rotation for power tools is for right tools.
  The method of construction of the tool mounting used may require the direction of rotation to be reversed.
- No dwell time should be programmed when thread tapping with length compensation.
- Thread tapping without length compensation is also described as "Rigid Tapping".

- Thread tapping with compensating chuck:
  The power tool is driven or stopped synchronously in the boring axis (in the Z-axis for G84, X-axis for G88). In thread tapping, feed is set to 100% (override switch is disabled).
- Rigid thread tapping (without compensating chuck):
  M29 Sxxx must be programmed in a block before the cycle.
  The power tool motor is controlled like a servo motor, therefore faster thread tapping is possible. The feed is limited to 100% (the value can be less, but not higher than 100%).
- Feed in [mm/min]:
  The thread pitch is generated from the feed, divided by the spindle speed
- Feed in [mm/rotation]:
  The feed corresponds to the thread pitch.

Example - G84 tapping cycle for thread cutting in Z-axis (axial tapping cycle) with the main spindle

N10 T1010	(thread tap M6x1, AWZ-axial);
G97;	Constant spindle speed selected
G0 X0 Z10 ;	Positioning the tool.
M29 S1000;	Thread tapping without length compensation and spindle speed selected.
G84 Z-20 R-6 P0 F1;	Thread
G80 M5;	Tapping cycle deselected and spindle stopped essential, in order to reset a potential reversal of the direction of rotation (for a left hand thread) caused by programming [M330]

Example - G84 tapping cycle for thread cutting in Z-direction with power tool (axial tapping cycle) on the main spindle

N4 T0404	(thread tap M6x1, AWZ-axial);
M52;	Main spindle selected as C-axis.
G97;	Constant spindle speed selected
G0 X50 Z10 C30;	Positioning the tool.
M29 S1000;	Thread tapping without length compensation and tool speed selected.
G84 Z-20 R-6 P500 F1;	Thread 1.
C150;	Thread 2.
C270;	Thread 3.
G80 M15;	Boring cycle deselected and tool speed switched off essential, in order to reset a potential reversal of the direction of rotation, caused by programming [M330]
M53;	Main spindle deselected as C-axis.

Example - G84 tapping cycle for thread cutting in Z-direction with power tool (axial tapping cycle) on the main spindle

N8 T0808	(Tap M6x1, AWZ-radial);
M52;	Main spindle selected as C-axis.
G97;	Constant spindle speed selected
G0 X80 Z-30 C30;	Positioning the tool.
M29 S1000;	Thread tapping without length compensation and tool speed selected.
G88 X50 R-6 P500 F1;	Thread 1.
C150;	Thread 2.
C270;	Thread 3.
G80 M15;	Boring cycle deselected and tool speed switched off essential, in order to reset a potential reversal of the direction of rotation, caused by programming [M330]
M53;	Main spindle deselected as C-axis.

G90 Absolute programming

Format

N... G90

Addresses are to be programmed as follows:
X .....Diameter

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... Z...).

G91 Incremental programming

Format

N... G91

Addresses are to be programmed as follows:
X .....Incrementally in diameter
Z .....Incremental (actual) traverse path

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... Z...).

G92 Setting maximum spindle speed

Format

N... G92 S.. Speed limitation

The G92 command determines the maximum spindle speed (rpm) for constant cutting speed (G96).

G94 Feed per minute

G94 means all values defined under F (feed) as mm/min.

Format
N... G94 F..

G95 Feed per revolution

G94 means all values defined under F (feed) as mm/rev.

Format
N... G95 F..

G96 Constant cutting speed

Unit: m/min
The controller continuously computes the spindle speed corresponding to the relevant diameter.

Format
N... G96 S..

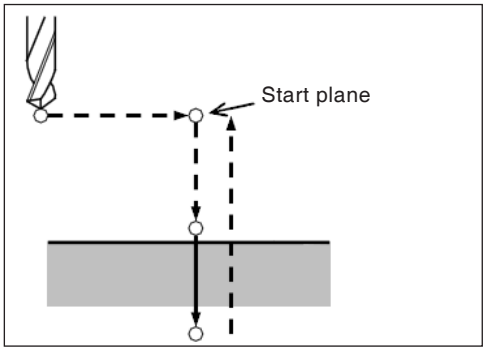
G97 Constant spindle speed deselected

Unit: m/min
Format
N... G97 S..

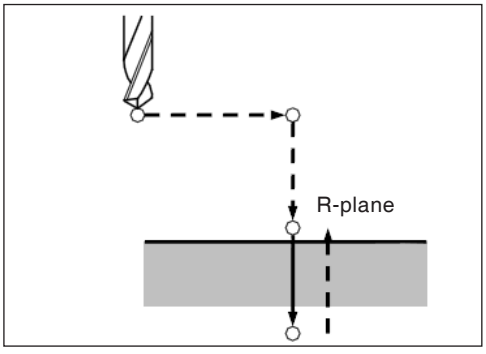
G98/G99 Return plane

G98 returns the tool from the bottom of the boring to the start plane.
G99 returns the tool from the bottom of the boring into the plane with point R.

In general, G99 is used for the first boring process and G98 for the last boring process.



Return to start plane



Return to R-plane

Bar feeder - / bar loader interface (option)

Bar feeder general

- The signals from the machine to the loader are freely programmable in the parts program via M functions.
- The signals from the loader magazine to the machine can be freely querried in the parts program and can be used for certain jumps (branches) or wait loops.
- The parts program is always started automatically by the machine, even at the bar end.

M functions

M57 Spindle turning or levelling process ON
M58 Main spindle levelling OFF
This function is mainly used in bar loader program if a new bar is loaded into the main spindle.
M67 Bar feed ON
The machine immediately continues working without waiting for a feedback of the loader (active until M68).
M68 Bar feed OFF
M69 Initializing the bar change
The machine immediately continues working without waiting for a feedback of the loader (active until M30 or "start after bar end").



Danger:

The bar loader has to be connected in a way that bar feeding is only possible when the machine door is closed!

Control signals (MACRO Variable)

bar end ..... #1000
loader has feeded ..... #1001
start after bar change ..... #1002

By means of these signals (MACRO Variable) you can branch accordingly in the parts program and either call the corresponding program parts or hide them, or program a wait loop.

In cycle O9100 variables are defined. Refer to "Used Variables" for more information.

Programming control signals

Bar end

Query whether there is a "bar end" or not:

```
IF[#1000EQ0]GOTO100
.....
.....
N100 (target when there is no bar end)
```

These program blocks are worked off at "bar end".
When there is no bar end, the program always jumps to line N100.

Start after bar change

Wait loop until the bar change is completed.

```
N200
G4 X0.5
IF[#1002EQ0]GOTO200
```

Loader has feeded

Wait loop until the signal "loader has feeded" will be given.

```
N300
G4 X0.5
IF[#1001EQ0]GOTO200
```

Example

```
M67
N300
G4 X0.5
IF[#1001EQ0]GOTO300
M68
```

Used variables

The following variables are used in the cycle O9100:

#100 Bar catching program
Example for input 5432 (= program-no.)
#101 Remaining part - discharge program
Example for input 3444 (= program-no.)
#102 Pick-up position in Z
Example for input -62
#103 Stop position in X
Example for input 0
#104 Final position in Z
Example for input 0.5
#105 Free standing position in Z
Example for input 100
#106 Final position in Z, new bar
Example for input 1
#107 Feed rate in [mm/min]
Example for input 3000
#108 Dwell time in [s], collection device in front position
Example for input 2
#109 Dwell time in [s], ejection of remaining part, collection device at the back
Example for input 4
#110 Spindle oscillation OFF/ON
Example for input 0 = OFF / 1 = ON
#515 part disposal program
1=chip tray
2=program for remaining parts
0=collection tray
#516 bar end marker

Specimen program for bar loaders LM800

```
%

O0001 (LOADER PROGRAM)
G10L2P1Z200 (FOR G54)
G92S2000 (HSP.LIMIT)
(freewheel position tool changer)
G18G40G80
G90G95T0
G0G53X440Z430M9
M5

#100=0 (bar begin program)
#101=0 (remaining part PRG)
#102=-10 (pickup position Z)
#103=0 (gauge position X)
#104=1 (ENDPOSITION Z)
#105=100 (release position Z)
#106=1 (ENDPOSITION Z NEW BAR)
#107=3000 (FEED MM/MIN)
#108=3 (dwelltime collection tray)
#109=3 (dwelltime.remaining part.)
#110=0 (spindle leveling)
#515=1 (part disposal)

(STOPPER)
T0606
G54

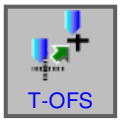
M98P9100

(freewheel position tool changer)
G18G40G80
G90G95T0
G0G53X440Z430M9

M30
%
```

F: Tool management

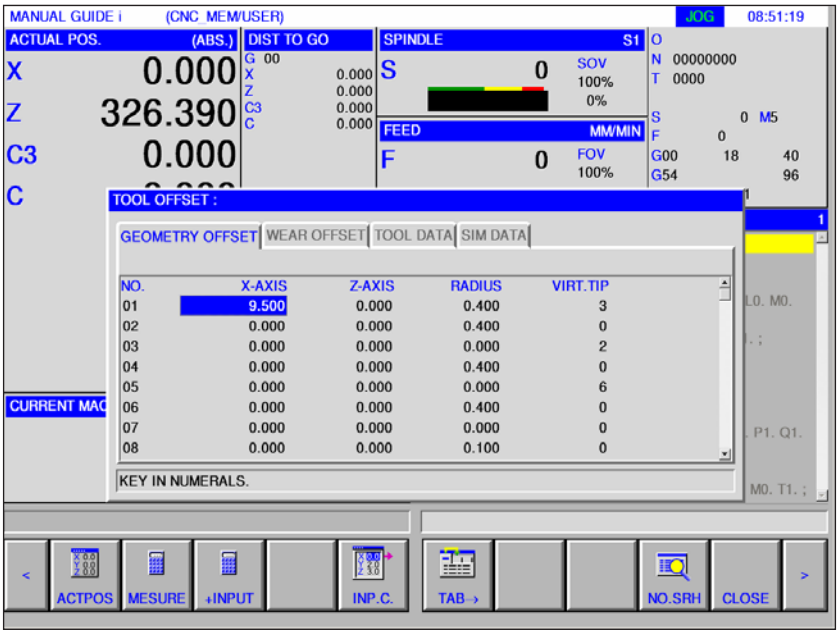
Tool settings



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

4 register cards are available for selection:

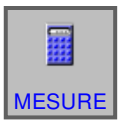
- Geometry data offset
- Wear offset
- Tool data
- Simulation data



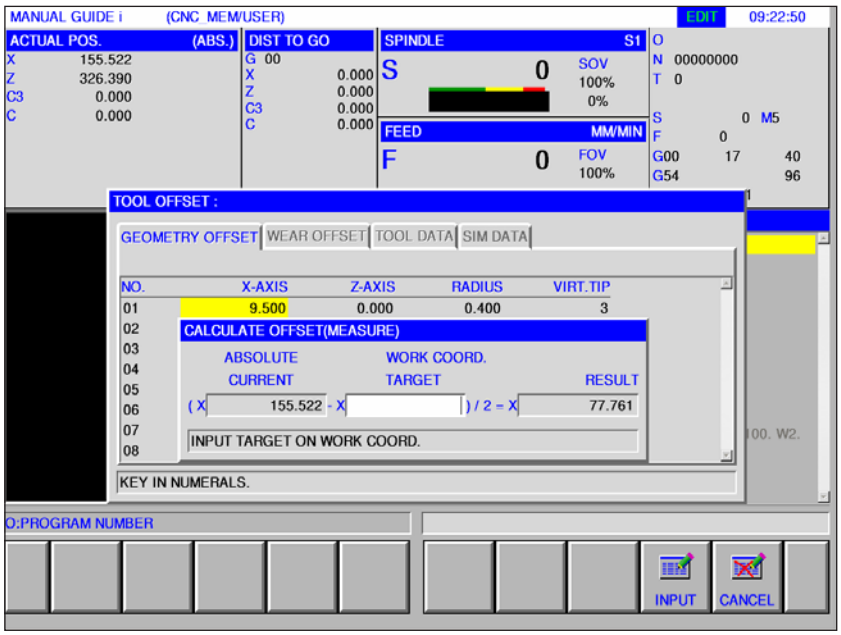
Parameter	Description
No.	Tool number. A maximum of 99 tool offsets is available
X-AXIS	X-axis geometry offset
Z-AXIS	Z-axis geometry offset
RADIUS	Tool radius offset
VIRT.TIP	Cutting position

Measuring

Geometry offset for the X- and Z-axes



- Press softkey.



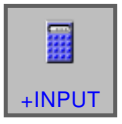
"MEASURE" performs the following offset calculation.

Current value of the absolute coordinates - target value of the workpiece coordinates).

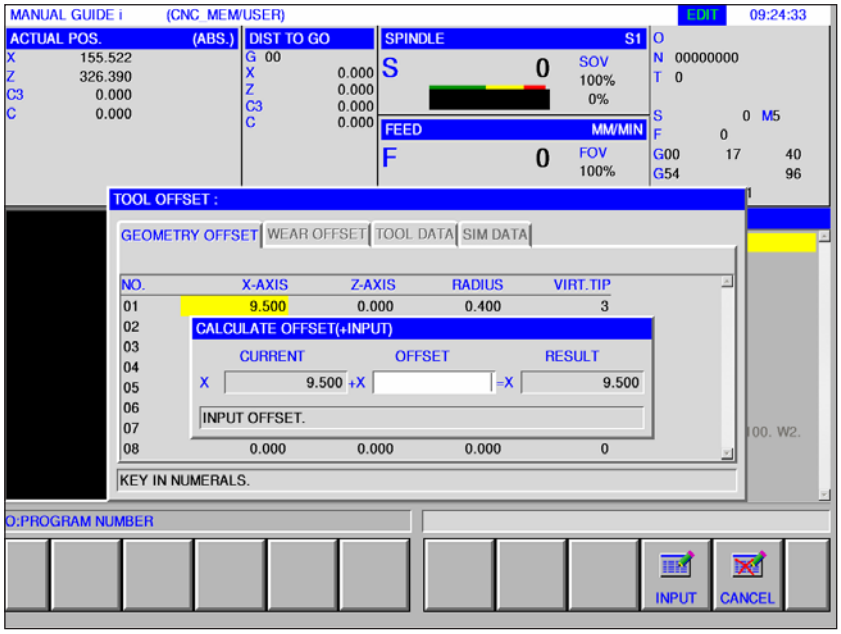
Repeat the measurement for the Z-axis in the same way.

Calculating

Geometry offset for the X- and Z-axes



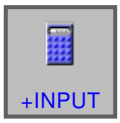
- Press softkey.
- "+INPUT" performs the following offset calculation.



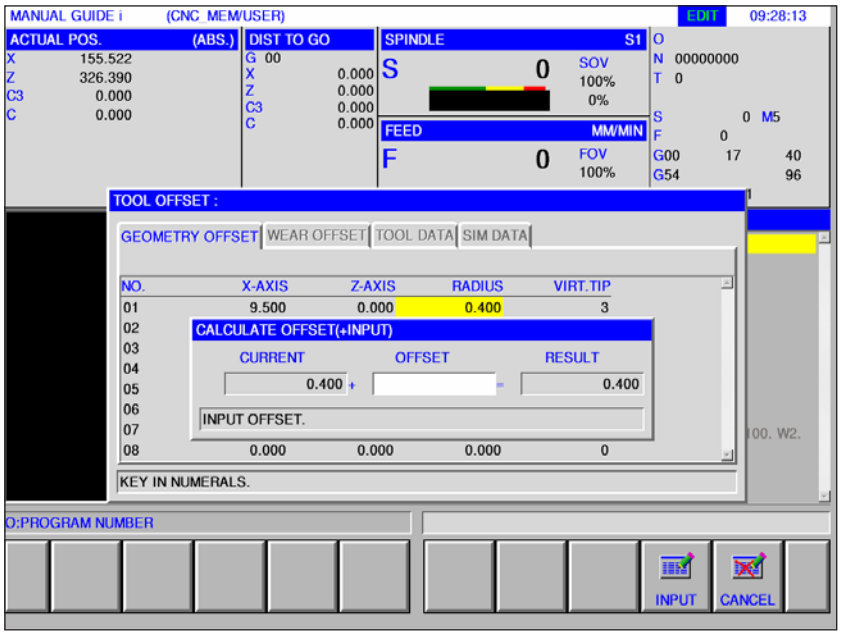
Current value + offset.

Repeat the measurement and calculation for the Z-axis in the same way.

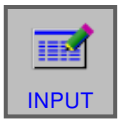
Radius offset



- "+INPUT" performs the following offset calculation.

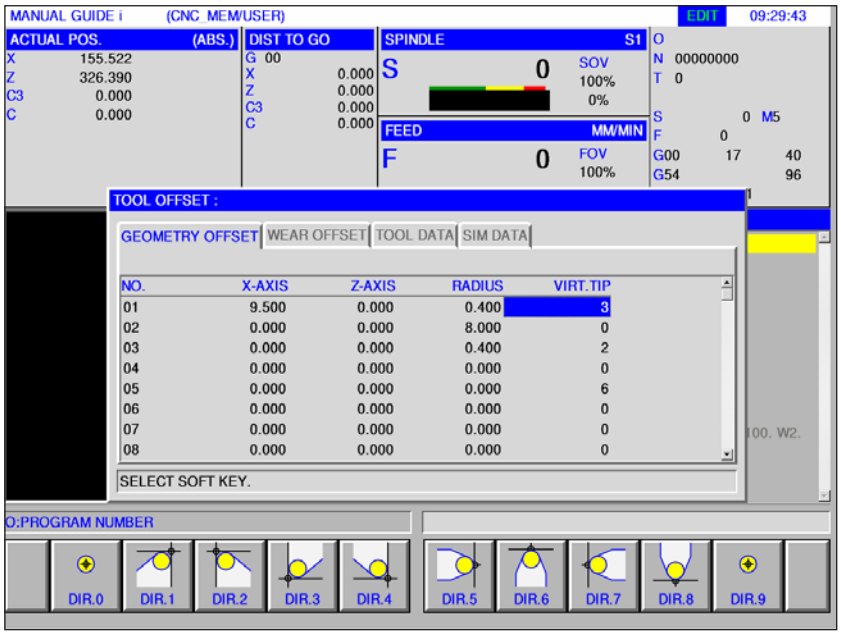


Enter the current value of the radius + offset.



- Complete the calculation with "INPUT".

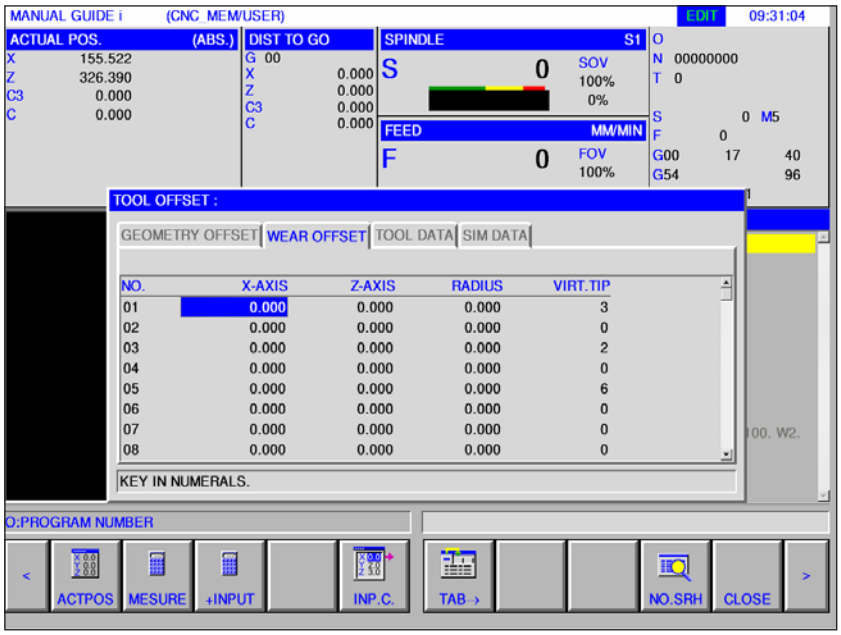
Cutting position



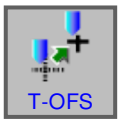
- Select the desired cutting position in the softkey toolbar.

Tool wear offset

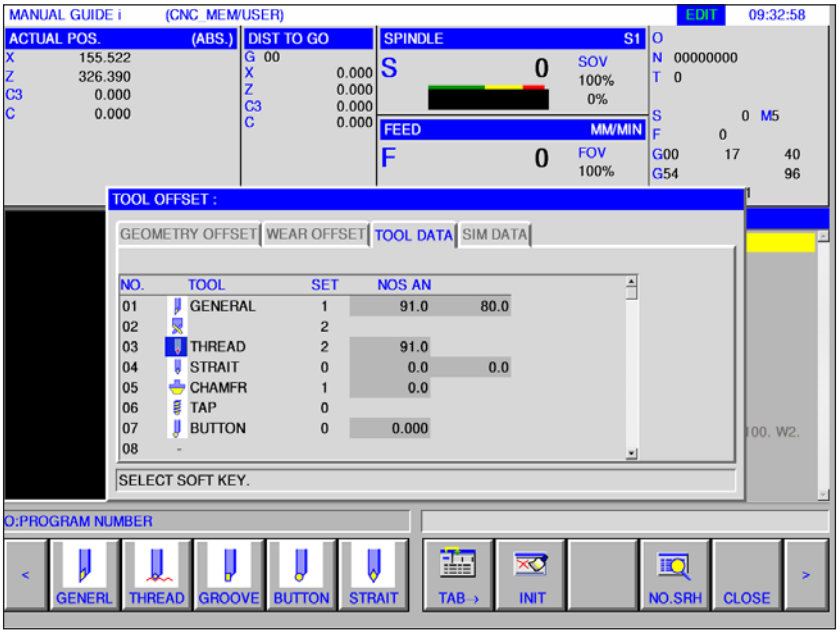
The entry for measurements and calculations of tool wear offset, radius offset and setting cutting positions are made in the same way as those for geometry offset.



Tool data



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



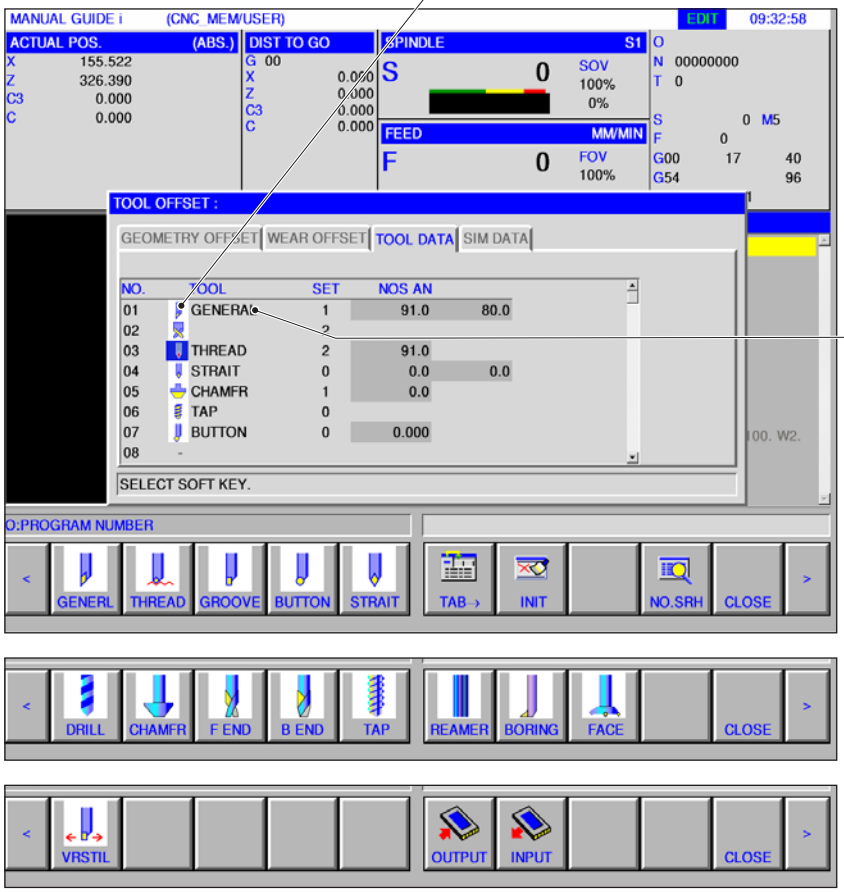
Parameter	Description
No.	Tool number
	Use the softkey to select the tool
TOOL	Tool name
SET	Setting 1 to 16. The tool tip is defined by the tool mounting position.
CUTAN	Cutting angle
NOS AN	Corner angle
TIP W	Width
TIP L	Length
CUT D	Diameter



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

Select tool

- Click on the tool symbol or move the cursor on to it with the arrow keys and select the wanted tool.
  The following tools are available for lathes.



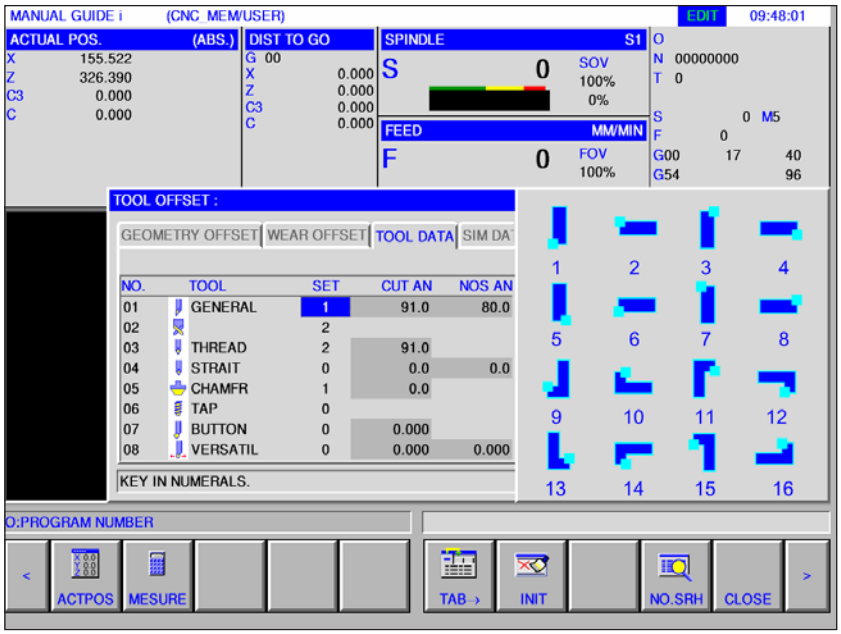
Enter tool name

Softkey bar with the available tools.
Scroll the pages forwards and backwards with the expansion button.

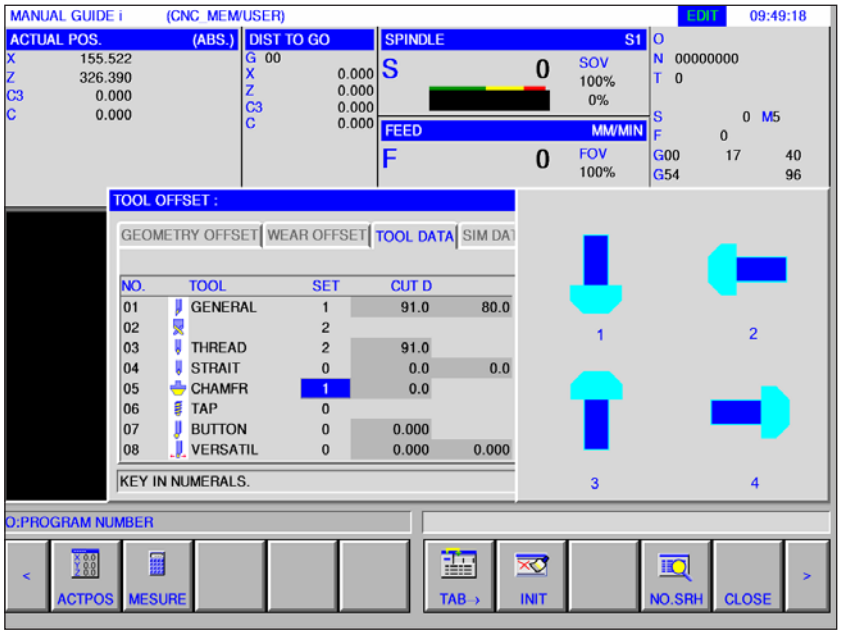
Instructions:
The available parameters can differ, according to the tool selected.

Tool setting number

- Mark a tool setting number with the cursor for an instruction window to pop up in the right hand portion of the screen.
  The following is true for a general rotary tool:
  Enter the tool setting number (1 to 16) to make the setting.



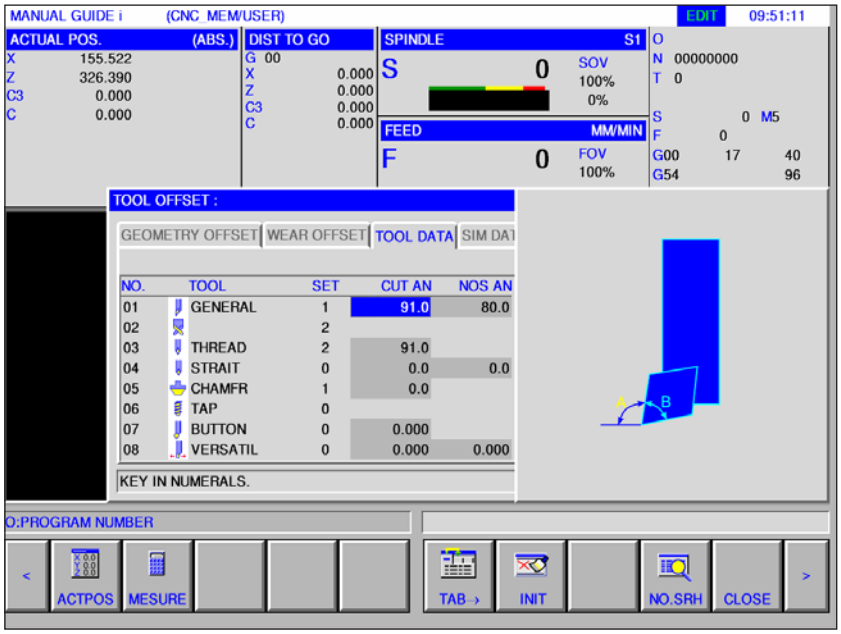
Settings for a general rotary tool.



Settings for a milling tool

Setting angle, corner angle, diameter, width and length

- 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 turning

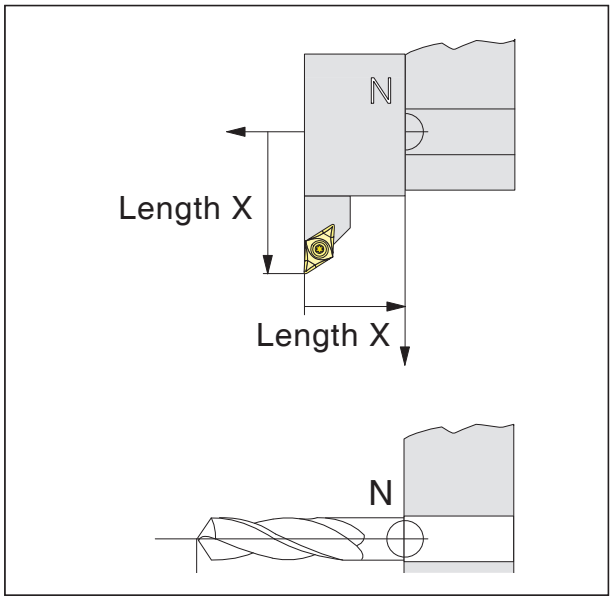
Tool type	General	Tap	Punch
Data 1	Cutting angle	Tip angle	Cutting width
Data 2	Tip angle		Cutting length *)

Tool type	Rotary tool form	Straight rotary tool	Multi-function tool
Data 1	Cutting length *)	Cutting angle	Cutting width
Data 2		Tip angle	Cutting length *)

Tool form data for milling

Tool type	Borer	Countersink	
Data 1	Tip angle *)	Mill diameter	

*) the cycle can be machined without a setting.

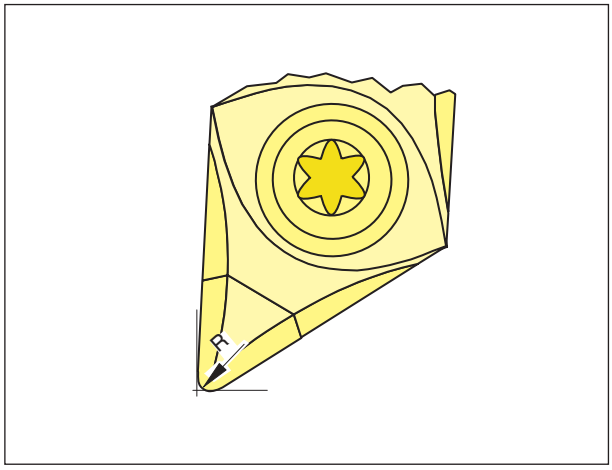


Direction of the length offset for tool types

Tool length offset

The tool length offset moves the tool zero point from the tool housing reference point N to the tool tip.
This is how all position details are related to the tool tip.

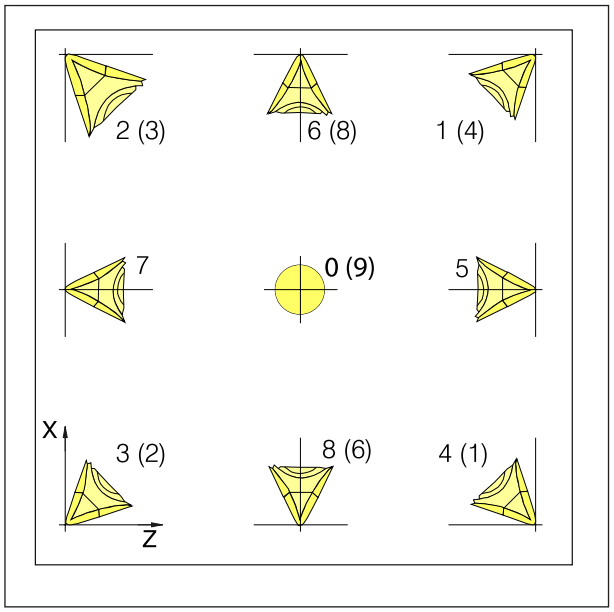
The tool housing reference point N in most machines is on the front face of the tool changer disk or the tool holder.



Cutting radius R

Cutting radius

The cutting radius data is needed only if a cutting radius compensation (G41, G42) is selected for the relevant tool.



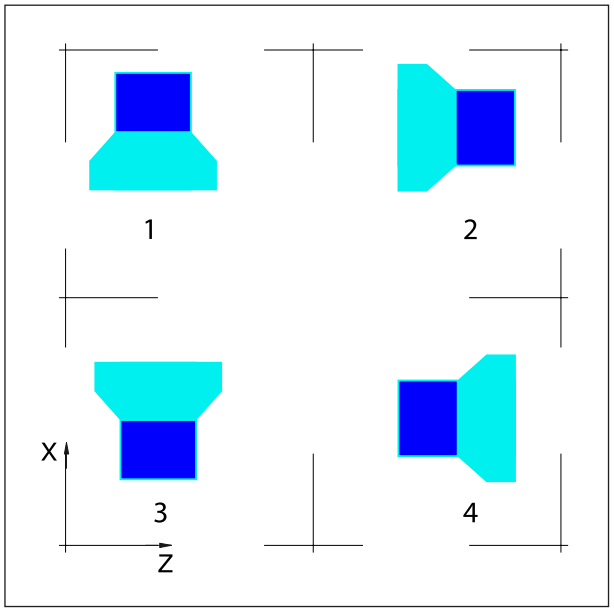
Cutting position of rotary tools

Cutting position of rotary tools

Consider the tool as if it is clamped to the machine to determine the type.
For machines, in which the tool is under (in front of) the rotary centre (e.g. Concept Turn 55), the values in brackets must be used, owing to the reversal of the +X direction.

Rotary tools

General tool
Tap
Punch
Circular tool
Straight tool
Boring tool
Tap
Reamer
Drill rod B
Multi-purpose rotary tool



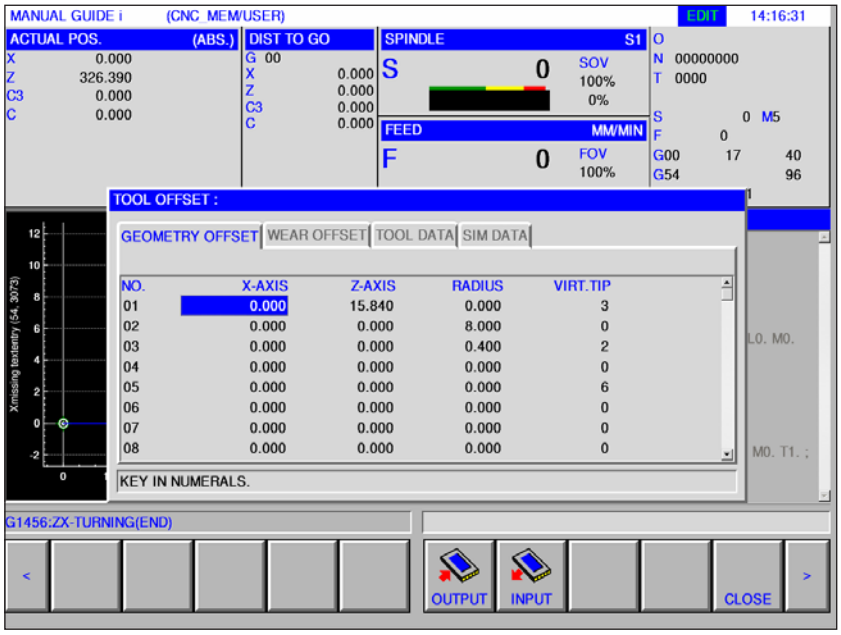
Cutting position of drilling and milling tools

Cutting position of milling and drilling tools

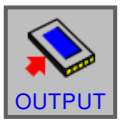
Milling tools

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

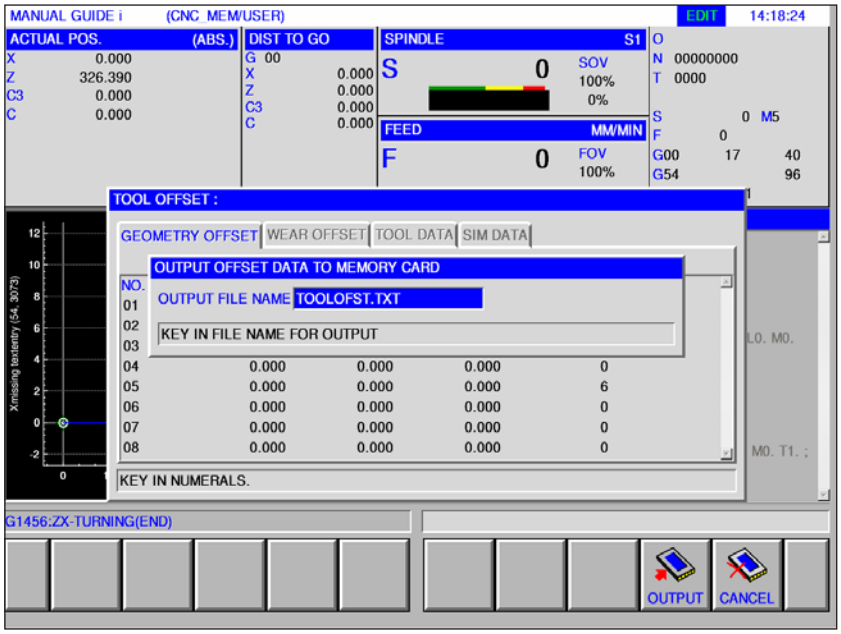
Geometry correction, wear correction and tool data input and output



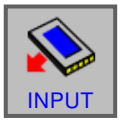
The path to save and read the data is set in EMConfig under the "Directory exchange" heading.



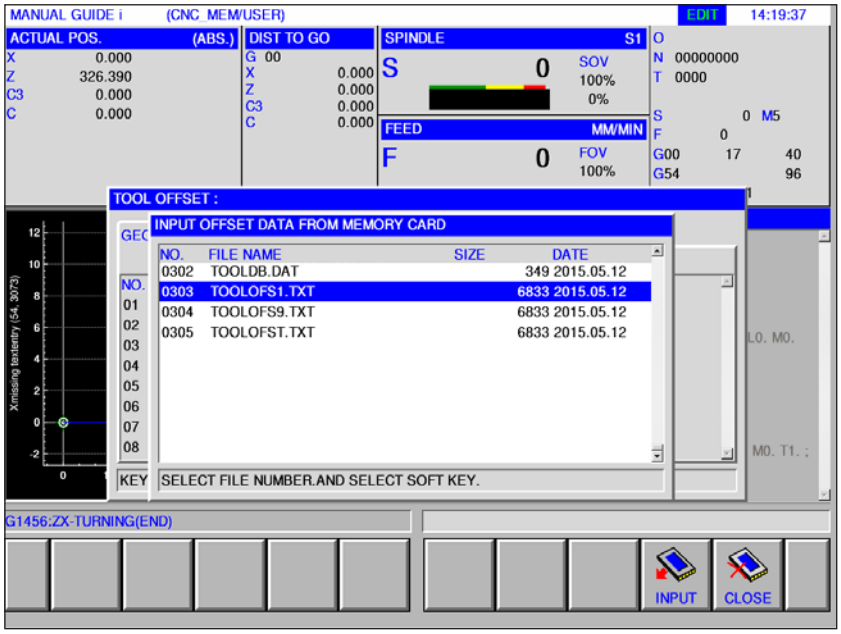
- Press the data output softkey



The default save for geometry correction and wear 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

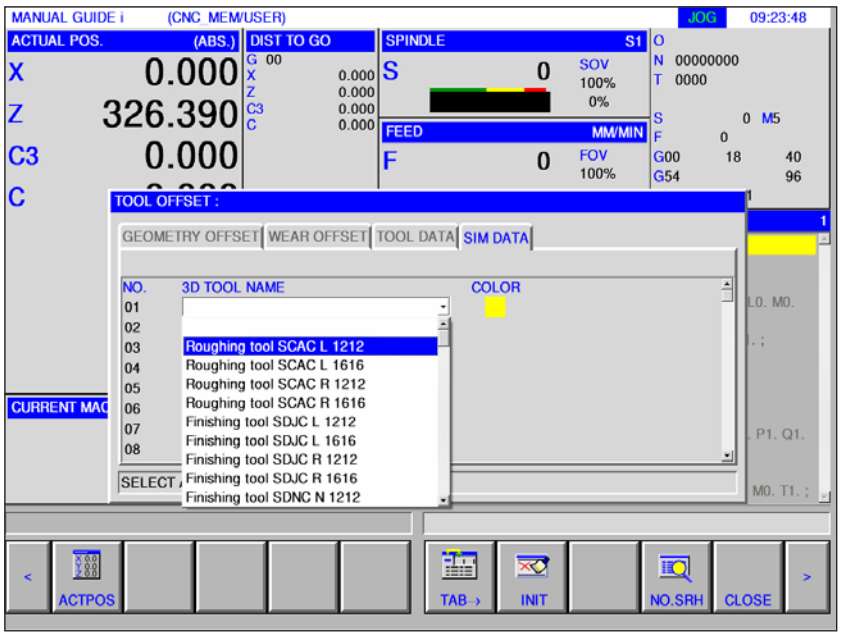


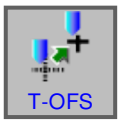
Select the desired file and open with "INPUT".

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.

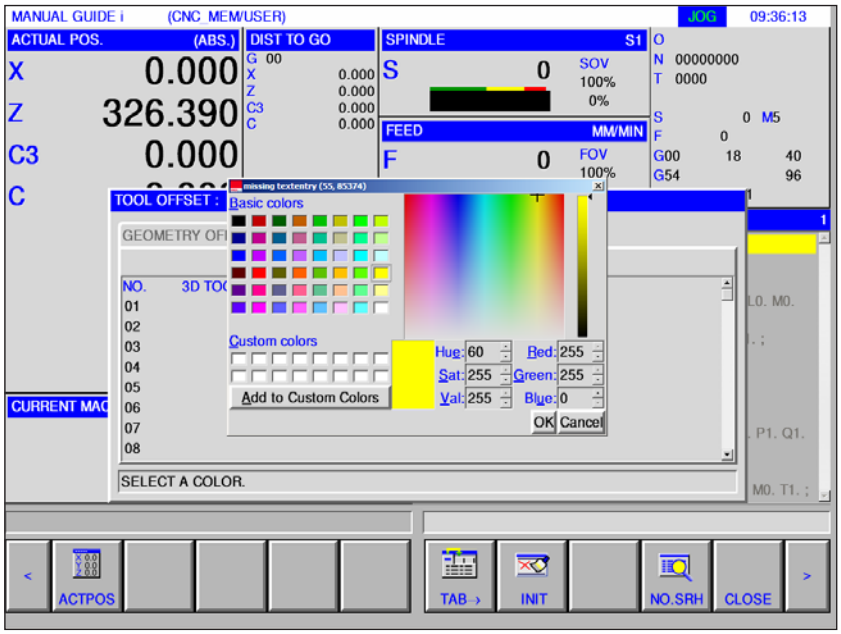
2 Select simulation data.

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.



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 an click to select. Alternatively the values for R,G,B can be entered manually.
- By clicking "Add to Custom Colors" a new color is added.

3 To deselect a color, select black.

4 Confirm with OK or quit with Cancel.

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

Measuring tool manually

Instructions:
Measuring tools is only possible with an active tool.

Precondition

The tool to be measured has already been created in the tool list and is physically located at the magazine place. The created tool should now be measured.



1 Call "JOG" mode.

For measuring tools, the workpiece for machining will be used.



2 Tilt the tool to be measured.



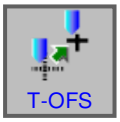
3 Switch the spindle into MDA mode.
G97 S1000 M4

4 Plane turning of a workpiece.

5 Longitudinal turning of a workpiece.



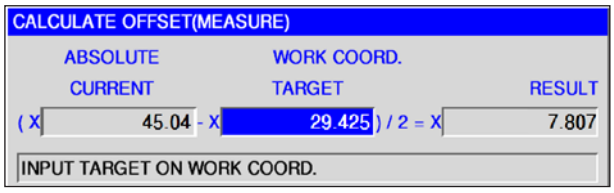
6 Measure the workpiece diameter.



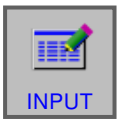
7 Press softkey.



8 Place the cursor on the right tool number in X and press the "MEASURE" softkey.



9 Enter the workpiece diameter measured earlier.

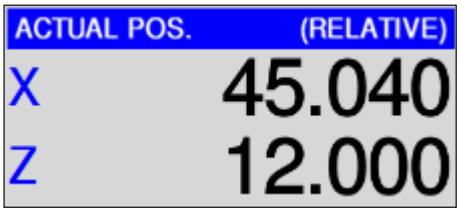


10 Complete the entry with the "INPUT" softkey.
The tool is now measured in X.



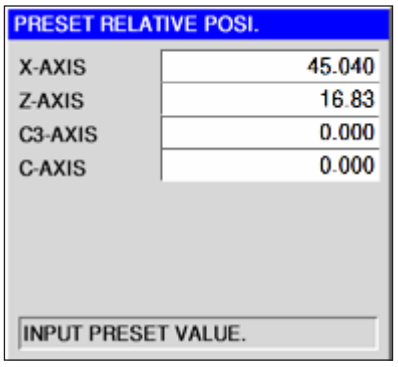
Set up the tool changer in a way such that the length from the front to the changer can be measured.





11 Press the "ACTPOS" softkey to display the relative coordinates.





12 Press the "PRESET" softkey and enter the measured value in Z.



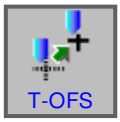
Accept changes with the "ALTER" softkey.



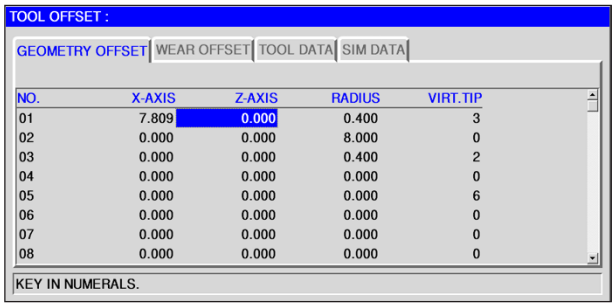
Call "JOG" mode.

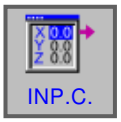


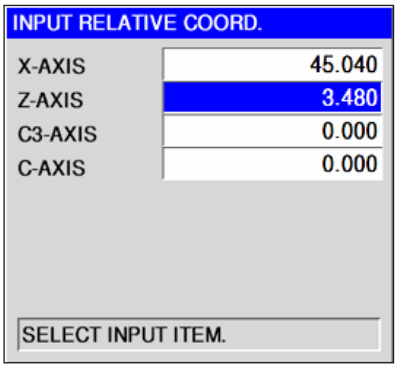
13 Scratch the tool from the front in Jog mode.



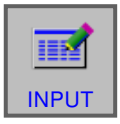
14 Press the "T-OFS" softkey and place the cursor on the Z-value of the tool to be measured.





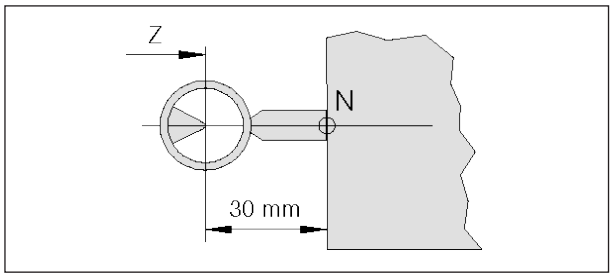


15 Press the "INP.C" softkey and place the cursor on the Z-axis.



Complete the entry with the "INPUT" softkey.

16 The tool is now measured in X and Z.



Reference tool measurement Concept Turn 55

Measure the tool with the optical default setting device at the machine

1 This method is more accurate, because contacts are avoided and the tool is shown magnified in the optics

The length must be observed for the reference tool. The exact length must be known.

The reference tool is used only to determine the position of the optic cross-hairs in the MKS

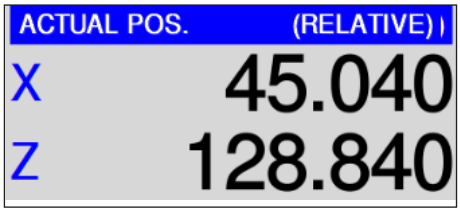
2 Procedure:

- Call "JOG" mode.

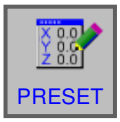


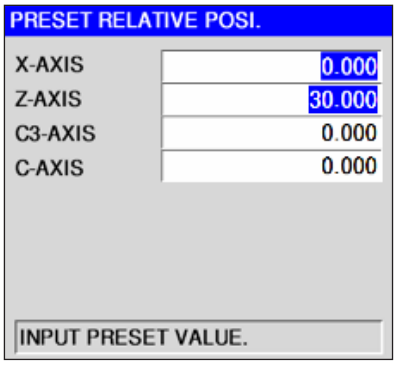
- Mount the reference tool and tilt it into the working position.
- Move the reference tool behind the cross hairs.





- Press the "ACTPOS" softkey to display the relative coordinates.





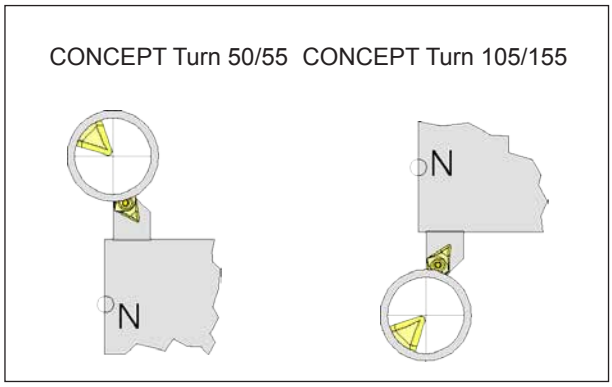
- Press the "PRESET" softkey and enter the value 0 in X and the length of the reference tool in Z.
  The reference tool is not the same length on all machines.



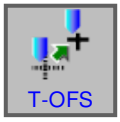
Accept changes with the "ALTER" softkey.

3 Measure tool

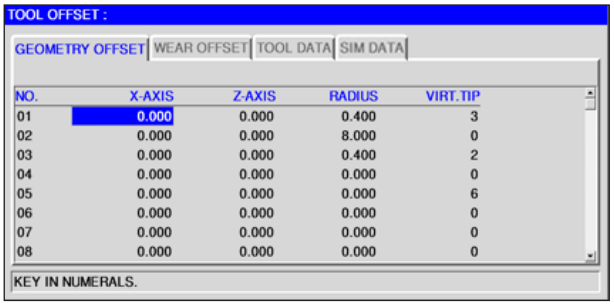
- Tilt the tool to be measured into the working position.
- Move the tool behind the cross hairs

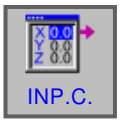


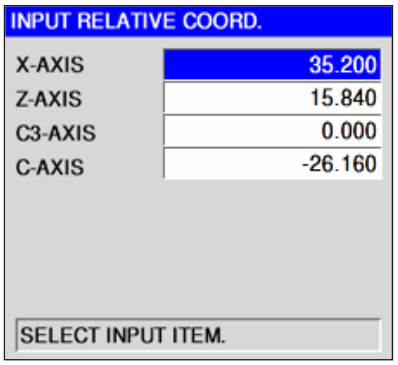
Move the tool behind the cross hairs



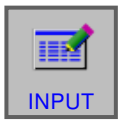
- Press the "T-OFS" softkey and place the cursor on the right offset number.



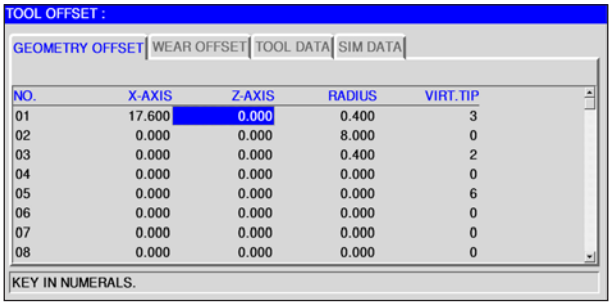




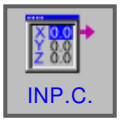
- Press the "INP.C." softkey and set the cursor on X-axis.

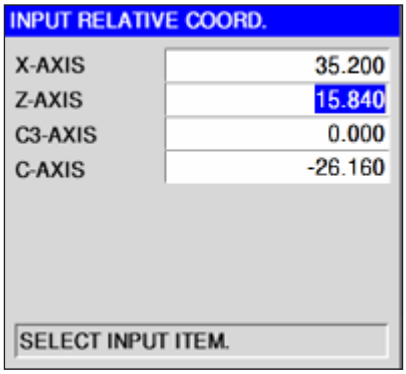


- Complete the entry with "INPUT".

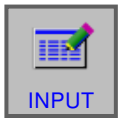


- Place the cursor on the Z-axis.

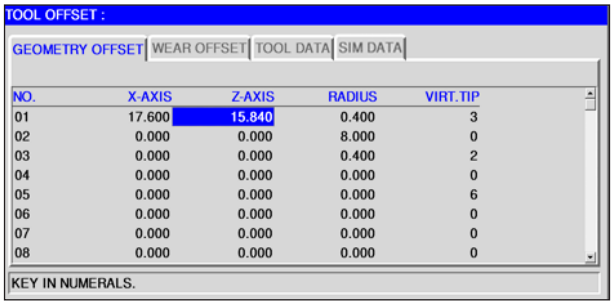




- Press the "INP.C." softkey and set the cursor on the Z-axis.



- Complete the entry with "INPUT".



- The tool is now measured in X and Z.

G: Program sequence

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

Preconditions

Set zero points

The zero points used must be measured and entered.

Tools

The tools used must be measured and entered.
The tools must be in the corresponding positions (t) in the tool changer.

Reference point

The reference point must be approached in all axes.

Machine

The machine must be ready for operation.
The workpiece must be securely clamped.
Loose parts (spanners, etc.) must be removed from the work area to avoid collisions.
The machine doors must be closed at the start of the program.

Alarms

There must be no alarms pending.



NC start

For the NC program run to be started, a Fanuc program must be open. The file name of the currently open program is shown in the centre of the simulation window.

NC reset



Use this button to stop the NC program run and reset to the start conditions.



NC stop

This button stops the NC program run. The simulation can be continued with this "Cycle start" button.

Program start, program stop

- Open a program for execution.



- Select "AUTO" mode.



- Open program.
- This softkey opens the selected program.



- Press the "Cycle start" button.





- Stop program with "Cycle stop", continue with "Cycle start".



- Cancel program with "NC reset".

Repositioning

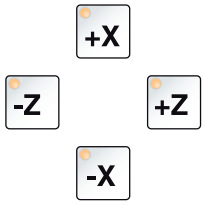


If a program interruption occurs in automatic mode, e.g. after a tool break, then the tool can be moved away from the contour in manual mode.
To avoid later collisions the axes must be moved to a safe position.
The coordinates of the interruption position will be saved.
The travel difference between the axes that has been brought about in manual mode is shown in the current values window. This travel 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".

Set flow



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

6043-6046: TOOL DISK POSITION FAULT
Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.
The Z axis could have been slipped out of the toothing while the machine was switched off.
A running CNC program will be stopped.
Contact EMCO service.

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

6048: DIVIDING TIME EXCEEDED
Dividing head stucks, 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 stucks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN
When opening the clamping pressure switch does not fall within 400ms. Pressure switch defective or mechanical problem. E22.3

6070: PRESSURE SWITCH FOR TANI MISSING
When closing the clamping pressure switch does not respond. No compressed air or mechanical problem. E22.3

6071: DIVIDING DEVICE NOT READY
Servo Ready Signal from frequency converter missing. Excess temperature drive TANI or frequency converter not ready for operation.

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

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

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective
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 EMConfig.

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 xxxxx.pls.

7057: TOOLHOLDER OCCUPIED
The clamped tool cannot be positioned in the tool turret since the position is occupied.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7058: RETRACTING THE AXES
The position of the tool turret arm cannot be clearly defined during the tool change.
Open the machine door, push the tool turret magazine backwards to the stop. Move the milling head in the JOG mode upwards to the Z reference switch and then traverse the reference point.

7087: MOTOR PROTECTION HYDRAULIC CLAMPING RELEASED!
Hydraulic motor is defective, stiff, circuit breaker is set incorrectly.
Replace motor or check circuit breaker and replace if necessary.

7090: ELECTRICAL CABINET OVERRIDE SWITCH ACTIVE
The cabinet door can only be opened when the key switch is switched on without raising an alarm.
Switch off key switch.

7107: OTOR PROTECTION RELEASED
A motor protection switch has triggered. A possibly running NC program will be completed. A new NC start will be prevented.
Check the device connected to the motor protection switch and switch on again. If recurring, contact EMCO service department.

7270: OFFSET COMPENSATION ACTIVE !
Only with PC-MILL 105
Offset compensation activated by the following operation sequence.
- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4

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

7271: COMPENSATION FINISHED, DATA SAVED !
see 7270

PC TURN 50 / 55 / 105 / 120 / 125 / 155
Concept TURN 55 / 60 / 105 / 155 / 250 / 260 / 460
Concept MILL 250
EMCOMAT E160
EMCOMAT E200
EMCOMILL C40
EMCOMAT FB-450 / FB-600

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed.
The reference position will be lost, the auxiliary drives will be switched off.
Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
The auxiliary drives will be switched off.
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
The auxiliary drives will be switched off.
Contact EMCO Service.

6003: PLC - NO DATA UNIT
The auxiliary drives will be switched off.
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
The auxiliary drives will be switched off.
Contact EMCO Service.

6005: K2 OR K3 NOT DE-ENERGIZED
Turn machine on/off. Defective security board.

6006 EMERGENCY-OFF RELAY K1 NOT DE-ENERGIZED
Turn machine on/off. Defective security board.

6007 SAFETY CIRCUIT FAULT

6008: MISSING CAN SUBSCRIBER
The PLC-CAN board is not identified by the control.
Check the interface cable and the power supply of the CAN board.

6009: SAFETY CIRCUIT FAULT

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6011: DRIVE Z-AXIS NOT READY
see 6010.

6012: DRIVE Z-AXIS NOT READY
see 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This alarm will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxiliary drives will be switched off.

6015: NO DRIVEN TOOL SPINDLE SPEED
see 6014.

6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING

6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING
In the tool turret that can be coupled, the position of the coupling and uncoupling magnet is monitored by means of two proximity switches. It has to be made sure that the coupling is in the rear stop position so that the tool turret can get to the next tool position. Equally, during operation with driven tools the coupling has to be safe in the front stop position.
Check and adjust the cables, the magnet and the stop position proximity switches.

6018: AS SIGNALS, K4 OR K5 NOT DE-ENERGIZED
Turn machine on/off. Defective security board.

6019: POWER SUPPLY MODULE NOT READY
Turn machine on/off. Power supply module, defective axis controller 6020 AWZ drive failure turn machine on/off, defective axis controller.

6020: DRIVEN TOOL DRIVE MALFUNCTION
The driven tool power supply defective or the driven tool drive is too hot, fuse defective, mains overvoltage or undervoltage. A running CNC program will be aborted, the auxiliary drives will be stopped. Check the fuses or contact EMCO service.

6021: COLLET TIME OUT
During closing of the clamping device the pressure switch has not reacted within one second.

6022: CLAMPING DEVICE BOARD DEFECTIVE
The signal "clamping device clamped" is constantly released, even though no command has been given. Replace the board.

6023: COLLET PRESSURE MONITORING
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

6024: MACHINE DOOR OPEN
The door was opened while a machine movement. The program will be aborted.

6025: GEARBOX COVER NOT CLOSED
The gearbox cover was opened while a machine movement. A running CNC program will be aborted.
Close the cover to continue.

6026: MOTOR PROTECTION COOLANT PUMP RELEASED

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

6028: DOOR TIMEOUT
The automatic door stucks, 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 stucks (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 stucks (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 stucks 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 stucks 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: Unexpeced 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 oY axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6922 INVERSION OF Z-DIRECTION TOO LONG "1"

Cause: The change in direction of Z axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6923 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC

Cause: ACC-PLC and USBSPS receive different door status reports.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6924 ERROR ENABLE PULSES MAIN DRIVE

Cause: The pulse release on the main spindle control element was interrupted by the USBSPS, as the PLC did not shut it down in a timely fashion.

Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

6925 GRID PROTECTION ERROR!

Cause: Grid protection does not drop out in current operating state, or does not engage.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6926 MOTOR PROTECTION ERROR!

Cause: Motor protection drops out in current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6927 EMERGENCY OFF ACTIVE ERROR!

Cause: Emergency off button was pressed.

Remedy: Restart the machine.

6928 TOOL CHANGER SHUTDOWN MONITORING ERROR

Cause: Unauthorised tool changer movement in the current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6929 MACHINE DOOR CLOSING/LOCKING ERROR

Cause: State of the door lock not plausible or door closure unserviceable.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6930 BEROS MAIN SPINDLE PLAUSIBILITY ERROR

Cause: Beros main spindle signal different.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6931 MAIN DRIVE QUICK STOP FUNCTION PLAUSIBILITY ERROR

Cause: Main drive actuator does not confirm the quick stop function in the current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6999 USB-EXTENSION FOR ROBOTIK NOT AVAILABLE

Cause: The USB extension for robotics cannot be addressed by ACC.

Remedy: Contact EMCO.

7000: INVALID TOOL NUMBER PROGRAMMED

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

7007: FEED HOLD

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

7016: SWITCH ON AUXILIARY DRIVES

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

7017: REFERENCE MACHINE

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

7018: TURN KEY SWITCH

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

7019: PNEUMATIC LUBRICATION MONITORING!

Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE

Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: TOOL TURRET NOT LOCKED

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

7022: COLLECTION DEVICE MONITORING

Time exceed of the swivelling movement.
Check the pneumatics, respectively whether the mechanical system is jammed (possibly a workpiece is jammed).

7023: ADJUST PRESSURE SWITCH !

During opening and closing of the clamping device the pressure switch has to turn off and on once.
Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !

When the clamping device is open and the position stop control is active, the respective proximity switch has to feed back that the clamping device is "Open".
Check and adjust the clamping device proximity switch, check the cables.

7025 WAITING TIME MAIN DRIVE !

The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again.
This message will appear when the door is quickly 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 a unaccepted socket wrench or a sheath which was left open. Remove the socket wrench after clamping the tool and close the sheath.

7072 HORIZONTAL TOOL CHANGE
The turning knob for manual tool clamping on the horizontal spindle is monitored by a switch. The switch reports a tightened turning knob. The spindle gets locked. Release the turning knob after clamping the tool.

7073 RETRACT Y AXIS LIMIT SWITCH !
see 7061

7074 CHANGE TOOL
Clamp programmed tool.

7076: SWIVEL UNIT VOR MILLING HEAD UNLOCKED
The milling head is not fully swung. Fix the milling head mechanically (the end switch must be pushed).

7077: ADJUST TOOL TURRET
No valid machine data for tool change are available. Contact EMCO.

7078: POCKET NOT IN HOME POSITION
Cancel during tool change. Swing back tool recessed in setup operation.

7079: TOOL ARM NOT IN HOME POSITION
see 7079

7080: INCORRECT TOOL CLAMPED !
The tool cone lies beyond tolerance. The clamped tool is twisted by 180°. Bero tool clamping is displaced. Check the tool and clamp it again. If this problem occurs with more tools, contact EMCO.

7082: MOTOR PROTECTION CHIP 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.

7099QUILL DRIVES FORWARD

Effect: Feed stop, Read in lock
Meaning: M21 was programmed ➙ pressure switch "tailstock in front position" is not yet 1.
Remedy: Disappears with pressure switch front position.

7100 QUILL DRIVES BACKWARD

Effect: Feed stop, Read in lock
Meaning: M20 was programmed ➙ limit switch "tailstock back" is not yet 1
Remedy: Disappears with limit switch "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 Easy-2control is missing.
Remedy: Reinstall software and/or contact EMCO Customer Service.

1712 USB-Dongle for Easy2control could not be found!
Cause: USB-Dongle for Easy2control is not connected. Easy2control is displayed but can not be operated.
Remedy: Connect USB-Dongle for Easy2control.

1801 Keytable not found!
Cause: The file with the keytable couldn't be found.
Remedy: Reinstall software and/or contact EMCO Customer Service.

1802 Connection to keyboard lost
Cause: Connection to the serial keyboard was interrupted.
Remedy: Turn keyboard off/on and check the cable connection.

Axis Controller Alarms 8000 - 9000, 22000 - 23000, 200000 - 300000

8000 Fatal Error AC

8100 Fatal init error AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8101 Fatal init error AC
see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

8106 No PC-COM card found
Cause: PC-COM board can not be accessed (ev. not mounted).
Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

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

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

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

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

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

8120 PC interrupt no. unmaskable
see 8119

8121 Invalid command to PC-COM
Cause: Internal error or defective cable
Remedy: Check cables (screw it); Restart software or reinstall when necessary, report to EMCO, if repeatable.

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

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

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

8125 Invalid memory for record buffer
Cause: Insufficient RAM, record time exceeding.
Remedy: Restart software, ev. remove drivers etc. to gain more RAM, reduce record time.

8126 AC Interpolation overrun
Cause: Ev. insufficient computer performance.
Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

8127 Insufficient memory
Cause: Insufficient RAM
Remedy: Close other programs, restart software, ev. remove drivers etc. to gain more RAM.

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

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8130 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The combination of linear and circular axes does not match.
Remedy: Program correction.

8140 Maschine (ACIF) not responding
Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

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

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

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

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

8146 Multiple request for axis
see 8142.

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

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

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

8150 Fatal ACIF error
see 8142.

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

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

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

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

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

8157 Data record done
see 8142.

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

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

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

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:
- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm) or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost
see 8161

8163 Z-Axis synchronization lost
see 8161

8164 Software limit switch max axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8168 Software limit overtravel axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.
Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running
Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

8174 INC not allowed
Cause: At the moment the axis is in motion.
Remedy: Wait until the axis stops and then traverse the axis.

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

8183 Gear too high
Cause: The selected gear step is not allowed at the machine.

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message
see 8175.

8191 Invalid jog feed unit
Cause: The machine does not support the rotation feed in the JOG operating mode.
Remedy: Order a software update from EMCO.

8192 Invalid axis in command
see 8175.

8193 Fatal PLC error
see 8175.

8194 Thread without length
Cause: The programmed target coordinates are identical to the starting coordinates.
Remedy: Correct the target coordinates.

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

8196 Too 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 cant't be interpolated

8215 Invalid state
see 8199.

8216 No rotation axis for rotation axis switch
see 8199.

8217 Axis type not valid!
Cause: Switching during the rotary axis operating mode when the spindle is running.
Remedy: Stop the spindle and switch over to the rotary axis operating mode.

8218 Referencing round axis without selected round axis!
see 8199.

8219 Thread not allowed without spindle encoder!
Cause: Thread cutting, respectively tapping is only possible with spindles with encoders.

8220 Buffer length exceeded in PC send message!
see 8199.

8221 Spindle release although axis is no spindle!
see 8199.

8222 New master spindle is not valid
Cause: The indicated master spindle is not valid when switching over to the master spindle.
Remedy: Correct the spindle number.

8224 Invalid stop mode
see 8199.

8225 Invalid parameter for BC_MOVE_TO_IO!
Cause: The machine is not 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[mm/min] * 0.0016 = distance [mm]
for CT250:
F[mm/min] * 0.00016 = distance [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 diferent rotations saved.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

8270 Reference switch defective
Cause: The reference switch did not switch within the specified range.
Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

8271 Tool load in locked place not possible
Cause: There was an attempt to swing a tool into a locked place in the magazine.
Remedy: Choose a free, unlocked place in the magazine and then swing the tool into the magazine.

8272 Old PLC version, update necessary
Cause: The PLC version is too old to fully support randomised tool management.
Remedy: Update the PLC.

8273 Spindle overload
Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).
Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

8274 Define tool before loading
Cause: The tool must be defined in the tool list before it is possible to transfer the tool into the spindle.
Remedy: Create the tool in the tool list, then load.

8275 Could not read position from absolute encoder
Cause: The position of an absolute encoder could not be read.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times

8276 Physical axis left the valid travel range
Cause: An axis with absolute encoder is outside the valid range for travel.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times.

8277 Sinamics drive package error active
Cause: Error in Sinamics drive.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times.

8278 Control not compatible with ACpn
Cause: The WinNC control is incompatible with the ACpn machine.
Remedy: Install a WinNC control which is compatible with the Acpn.

8279 Connection to drive got lost
Cause: the connection between Acpn and CU320 was lost.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times

8280 Reference point in setting data not equivalent with machine data, please check!
Cause: The reference point saved in the AC setting data does not match the reference point in the machine data (ACC_MSD).
Remedy: Measure the reference point of all axes anew and enter into EMConfig.

8704 Feed override absent, REPOS is not executed
Cause: The REPOS command is not executed because the feed override is set to 0%.
Remedy: Change the feed override and restart REPOS.

8705 Tool sorting active
Cause: The tools will be re-sorted with random tool management to facilitate non-random operation (tool 1 at place 1, tool 2 at place 2, etc.).
Remedy: Wait until sorting has finished. The controller will delete the report independently.

8706 Check new controller - tool table
Cause: The controller was changed with random tool management active.
Remedy: Check the tool or place table to clear the alarm.

8707 Ending with auxiliary drives switched on not possible
Cause: An attempt was made to end the controller, although the auxiliary drives are still switched on.
Remedy: Switch off the auxiliary drives and then end the controller.

8710 Initiating communication with drives
Cause: Acpn connects to the Sinamics drives
Remedy: Wait until the connection was established.

8712 JOG in X and C deactivated during Transmit
Cause: JOG in X and C 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 restarting 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 diamter 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 occured 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 cyle has not been found.

4025 internal calculation error
Cause: An unexpected error has occured 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: Correc the finishing offsets.

4028 pitch 0 not permitted
Cause: The thread was programmed with pitch zero.

4029 undefinded 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 cylce
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 TRANSMIT!

Cause: Travel comes too close to the X0 Y0 coordinates.

Remedy: Change travel.

I: Fanuc 31i controller alarms

Controller alarms 0001 - 88000

The controller triggers these alarms. These are the same alarms as would have occurred on the Fanuc 31i controller.

0006 ILLEGAL USE OF MINUS SIGN
Declaration: Illegal minus sign (-) in a NC command word or a system variable.

0010 IMPROPER G-CODE
Declaration: Improper G-Code commanded.
The parameter for piercing with continuous circular motion is not effective.
The activation signal for piercing with continuous circular motion is "0".

0030 ILLEGAL OFFSET NUMBER
Declaration: Illegal offset number specified.
Alarm also comes on if, in tool offset memory B, the number of tool form offsets exceeds the maximum number of tool offset sets.

0051 MISSING MOVE AFTER CNR/CHF
Declaration: Improper move or travel in the set after chamfering or rounding. Correct program.

0055 MISSING MOVE VALUE IN CHF/CNR
Explanation: In the set for chamfering/rounding, the path is less than the amount of the chamfer or rounding. Correct program.

0077 TOO MANY SUB,MACRO NESTING
Declaration: In total, more subroutines and macro calls have been commanded than allowed.
Further subroutine calls during subroutine call from external memory

0114 ILLEGAL EXPRESSION FORMAT
Declaration: Format error in a printout of a client macro instruction.
The punched tape format parameter is incorrect.

0115 VARIABLE NO. OUT OF RANGE
Declaration: A local, global or system variable in a client macro contains an out of range number.
A non-existent client macro variable number has been given in the "EGB axis fadeout" function (G31.8). Or the number of client macro variables for saving jump positions is insufficient.
High speed cycle machinings are incorrect. The alarm is triggered in the following cases:
1) The program head corresponding to the called machining cycle number is missing.
2) The value of the cycle connection information is outside the permitted range (0 to 999).
3) The number of data elements in the program head is outside the permitted range (1 to 65535).
4) The memory start data variable number of the executable data is outside the permitted range (#20000 to #85535/#200000 to #986431/#2000000 to #3999999).
5) The memory end data variable number of the executable data is outside the permitted range (#85535/#986431/#3999999).
6) The memory start data variable number of the executable data is the same variable number as that used by the program head.

0116 WRITE PROTECTED VARIABLE
Declaration: In a client macro, a variable should be used on the left page of a printout that may only appear on the right page.

0128 ILLEGAL MACRO SEQUENCE NUMBER
Explanation: The figure program includes The given set number was not found in the set number search.
The set number given in GOTO-- and M99P-- as the jump destination was not found.

0175 ILLEGAL G07.1 AXIS
Declaration: Axis with which cylindrical interpolation is impossible. Several axes in a G07.1 set. Cylindrical interpolation is ended for an axis that is not in this mode.
Set the axis for cylindrical interpretation in parameter 1022 not on 0, but on 5, 6 or 7 (parallel axis), in order to describe the arc with the rotation axis (ROT parameter 1006#1 on 1, the parameter 1260 is configured).

0310 FILE NOT FOUND
Declaration: The file was not found when calling up the subroutine or macro.

A 2014-12

0312 ILLEGAL COMMAND IN DIRECT DRAWING DIMENSIONS PROGRAMMING
Declaration: Illegal command format for direct drawing dimensions programming. An illegal G-Code was used for direct drawing dimensions programming.
There are two or more sets without a positioning command between two commands for direct drawing dimensions programming. Decimal point in direct drawing dimensions programming, although decimal points must not be used (bit 4 of parameter no. 3405 = 1).

1330 ILLEGAL OFFSET NUMBER
Declaration: Spindle number higher than the number of controlled spindles while loading parameters or lead error offset data from punched tape or with G10.

1960 ACCESS ERROR (MEMORY CARD)
Declaration: Illegal memory card access.
This alarm is also triggered while reading if the end of the file is reached and no EOR code '%' is found.

3506 WRONG CUTTING AREA
Declaration: The cutting area is illegal.
Remedy: Change the cutting program so that a correct cutting area is given, so that part contour and raw part contour match with the cutting profile.

3507 WRONG CUTTING CONDITION
Declaration: The cutting conditions are illegal.
Remedy: Change the cutting program so that the cutting conditions, such as feed speed, are normal.

3514 WRONG FIGURE DATA
Declaration: The figure data is illegal.
Remedy: Change the cutting program so that the figure data is correct.

3516 NO NECESSARY ADDRESS
Declaration: The necessary addresses for a cycle cutting command or other 4-digit G-commands were not entered.
Remedy: Change the cutting program, e.g. by adding the necessary addresses.

3530 WRONG MACHINING TYPE
Explanation: The machining type specification is illegal.
Remedy: Change the machining program so that the machining type is appropriate.

3531 WRONG RETURN MODE
Explanation: The return mode is illegal.
Remedy: Change the machining program so that the return mode is appropriate.

3535 WRONG THREADING LEAD
Declaration: The threading lead setting is illegal.
Remedy: If, say, a negative value was entered as the threading lead,
change the machining program so that the threading lead is correct.

3538 WRONG CUTTING DIRECTION
Explanation: The cutting direction setting is illegal.
Remedy: A value was entered that must not be given for turning or other machining directions. Change the machining program so that the machining direction is permitted.

3539 WRONG CUT DEPTH DIRECTION
Declaration: The cut depth direction setting is illegal.
Remedy: A value was entered that must not be given for turning or other cut depth directions. Change the machining program so that the cut depth direction is permitted.

3541 WRONG CHAMFERING AMOUNT
Explanation: The chamfering amount setting is illegal.
Remedy: A value was entered that is not permitted for chamfering or other chamfer values, e.g. a negative value. Change the machining program so that the chamfering amount is permitted.

3542 WRONG EJECTION STROKE
Declaration: The ejection stroke setting is illegal.
Remedy: A value was entered that is not permitted for chamfering or other plane area ejection stroke amounts, e.g. a negative value.
Change the machining program so that the ejection stroke amount is permitted.

3543 WRONG THICKNESS
Explanation: The given machining thickness is illegal.
Remedy: A value was entered that is not permitted for pocket milling or other machining thicknesses, e.g. a negative value. Change the machining program so that the machining thickness is permitted.

3547 WRONG CUTTING DEPTH ANGLE
Declaration: The cutting depth angle is illegal.
Remedy: A value was entered that is not permitted for pocket milling or other cutting depth angles. Change the machining program so that the cutting depth angle is permitted.

3548 WRONG CLEARANCE
Declaration: - The clearance is illegal.
Remedy: A clearance value was entered that is illegal, e.g. a negative value. Change the machining program so that the clearance is permitted.

3551 WRONG FINISHING NUMBER
Declaration: - The number of finished machining processes is illegal.
Remedy: An illegal value for the number of thread cutting processes or for other finished machining process was entered, e.g. 0. Change the machining program so that the number of finished machining processes is permitted.

3552 WRONG APPROACH SETTING
Declaration: The approach setting is illegal.
Remedy: A value was entered that is not permitted for contour cutting or other approaches. Change the machining program so that the approach is permitted.

3553 WRONG ESCAPE SETTING
Declaration: The escape setting is illegal.
Remedy: A value was entered that is not permitted for contour cutting or other escapes. Change the machining program so that the escape is permitted.

3575 WRONG FIGURE TYPE
Explanation: The figure type is illegal.
Remedy: A figure type that is not permitted was selected for a figure set.
Change the machining program so that the figure type is permitted.

3579 WRONG CORNER SETTING
Declaration: The corner rounding setting is illegal.
Remedy: A value for rounding the corner of a figure type that is not permitted was entered. Change the machining program so that the entered value is permitted.

3582 WRONG FIGURE RADIUS
Declaration: The figure radius setting is illegal.
Remedy: A value for rounding the corner of a figure type that is not permitted was entered. Change the machining program so that the entered value is permitted.

3584 WRONG PITCH SETTING
Declaration: The pitch setting is illegal.
Remedy: A value for a pitch for a figure set that is not permitted was entered. Change the machining program so that the set value is permitted.

3585 WRONG HOLE/GROOVE NUMBER
Declaration: The number of holes/grooves is illegal.
Remedy: A value for the number of holes or grooves in a figure set that is not permitted was entered. Change the machining program so that the set value is permitted.

3586 WRONG COORDINATE POSITION
Declaration: The coordinate positions are illegal.
Remedy: A value for the coordinates of a figure set that is not permitted was entered. Change the machining program so that the set value is permitted.

3592 WRONG OMITTED POINT
Declaration: A random figure is not closed.
Remedy: A random figure that was entered for plane cutting, pocket milling or turning was not closed. Change the machining program so that there is a closed figure that has identical start and end points.

5010 END OF RECORD
Declaration: The EOR (end of record) code was in the middle of a set. An alarm is also generated if the percent sign is read at the end of the program.

5044 G68 FORMAT ERROR
Declaration: Error in the three-dimensional coordinate conversion command:
(1) No I, J or K in the three-dimensional coordinate conversion command string (without the "Coordinate rotation" option).
(2) I, J or K are all 0 in the three-dimensional coordinate conversion command string.
(3) No rotating angle R in the three-dimensional coordinate conversion command string.

W: Accessory Functions

Activating accessory functions

According to the machine (turn/mill) the following accessories can be taken into operation:
- automatic tailstock
- automatic vice/clamping device
- Air purge system
- Dividing attachment
- Robot interface
- Automatic doors
- Win3D view simulation software
- DNC interface

The accessories are activated with EMConfig.

Robotic Interface

The robotic interface is used to connect the concept machines to an FMS/CIM system.

The most important functions of a concept machine can be automated via the inputs and outputs of an optional hardware module.
The following functions can be controlled via the robotic interface:
- Program START / STOP
- Door open / closed
- Quill clamp / back
- Clamping device open / closed
- Feed STOP



Note:
The functionality of the Win3D-View is depending on the control software that is used.

Automatic doors

Preconditions for activation:
- The auxiliary drives must be switched on.
- The main spindle must be still (M05 or M00) - this also means that the run-out phase of the main spindle must be ended (program dwell time if required).
- The feed axes must be still.
- The tool changer must be still.

Behavior when automatic doors active:

Opening door
The door can be opened manually, via the robot interface or DNC interface.
In addition, the door opens when the following commands are executed in the CNC program:
- M00
- M01
- M02
- M30

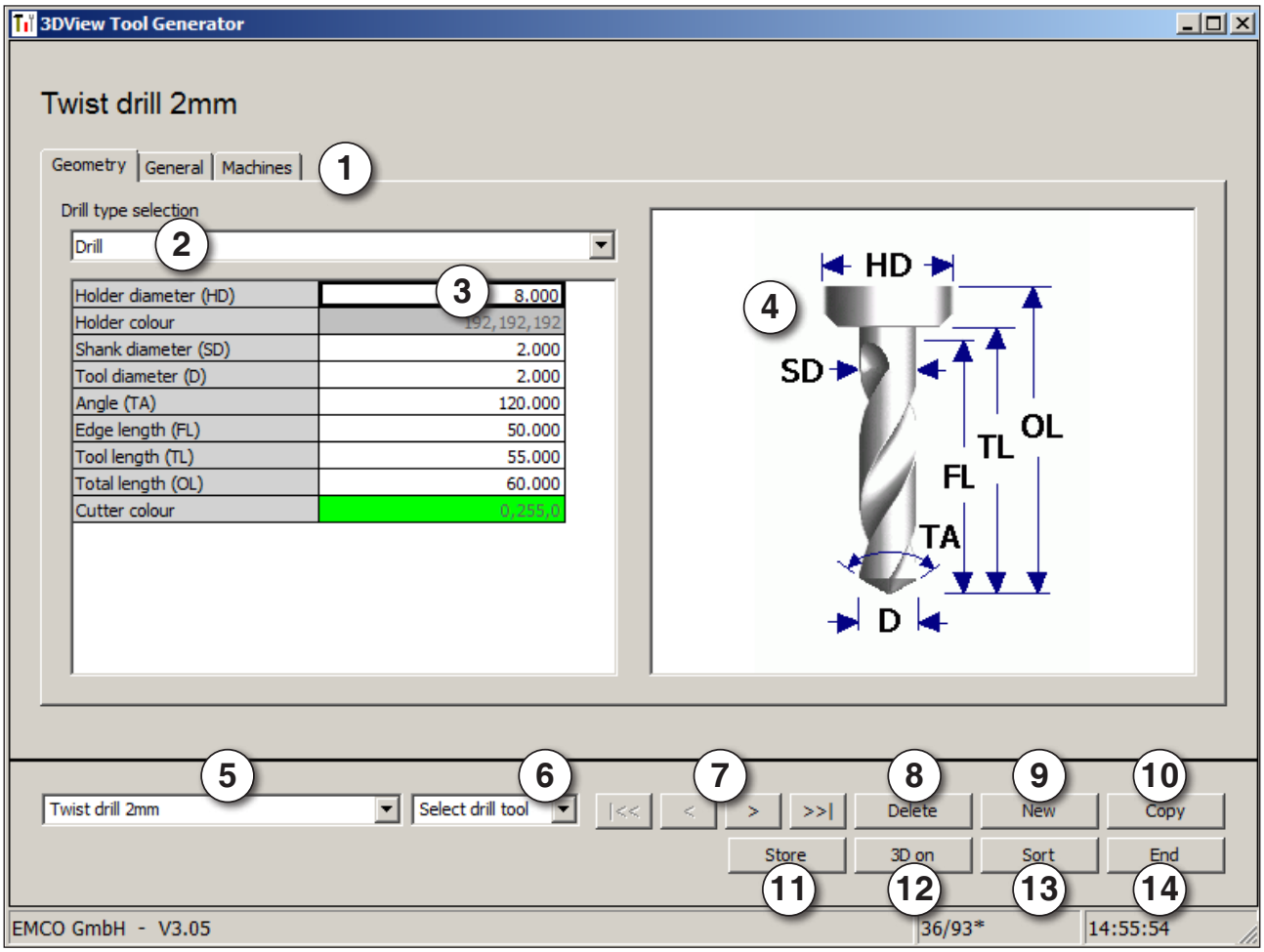
Closing door:
The door can be closed by manually pressing the button via the robot interface. It is not possible to close the door via the DNC interface.

Win3D View

Win3D View is a 3D simulation for turning and milling, which is offered as an additional option for the WinNC product. Graphical simulations of CNC controls are primarily designed for industrial practice. The Win3D View screen representation goes beyond the industrial standard. Tools, raw parts, clamping devices and the processing steps are represented extremely realistically. The programmed movement paths of the tool are checked by the system for a collision with clamping device and raw part. A warning message is issued when there is danger. This makes possible to have understanding and control of the manufacturing process already on the screen.
Win3D View is used to visualize and prevent costly collisions.
Win3D View offers the following advantages:
- Extremely realistic representation of workpiece
- Tool and clamping device collision control
- Cut representation
- Zoom functions and turning of views
- Representation as solid or wireframe model

Tool modelling with the 3D-Tool Generator

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

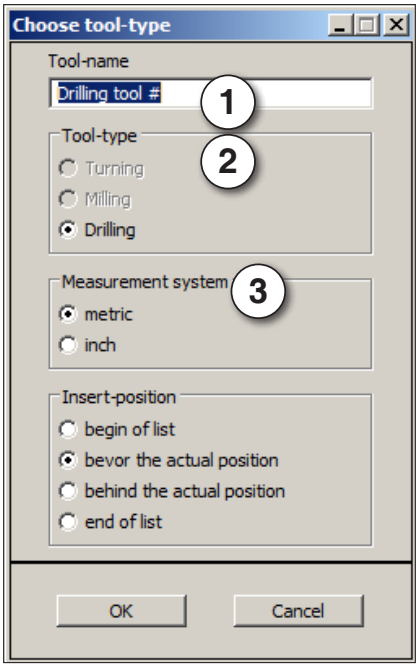


1 Register cards for "Geometry", "General" and "Machines" for drilling and milling tools and "Tip", "Holder", "General" and "Machines" for turning tools.
2 Selection of tool types
3 This window enables the input of tool dimensions.
4 Graphical support for the tool dimensioning
5 Choice of tools for the 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

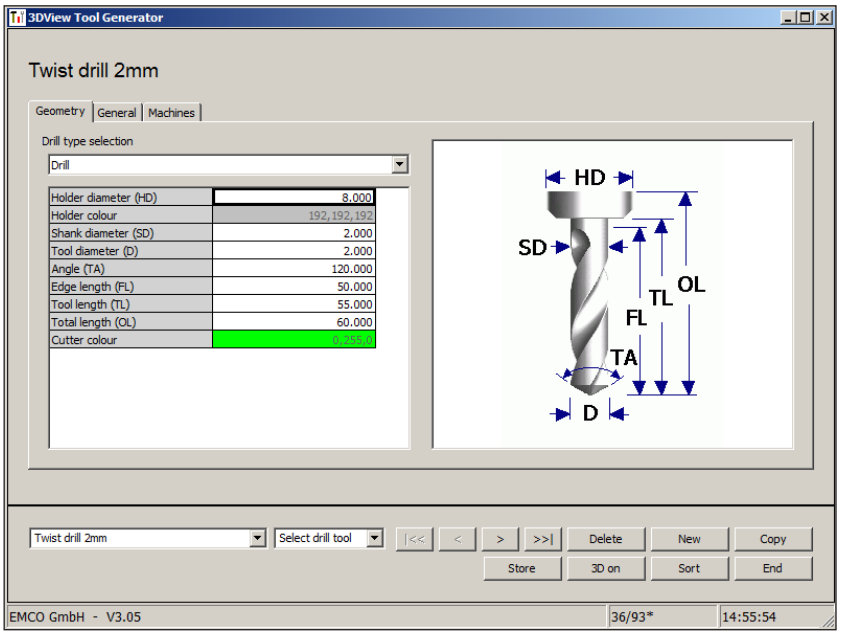
- Set the selection for tool types to "Selection all".
- Press the button to generate new tools.
- Select the tool name (1), the tool type (2) and the measurement system (3).

New



OK

- Confirm the entries with "OK".



- Define all tool dimensions.
- Define all tool colours (see "select tool colour").

Store

- Confirm the entries with "Store".

Copying a tool

Copy

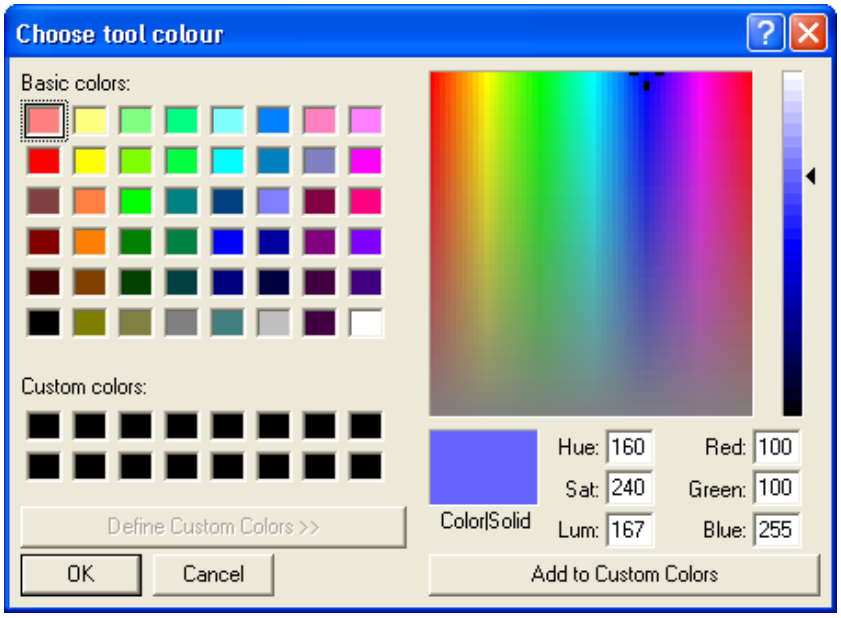
- Call the tool you wish to copy.
- Press the button to copy tools.
- Enter the new tool name.
- Confirm the inputs with "Save".

Changing an existing tool

Store

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

Selecting a tool colour



- Make a double click with the mouse pointer in the coloured box of the tool colour. The window "Select tool colour" will appear.
- Select the required colour.

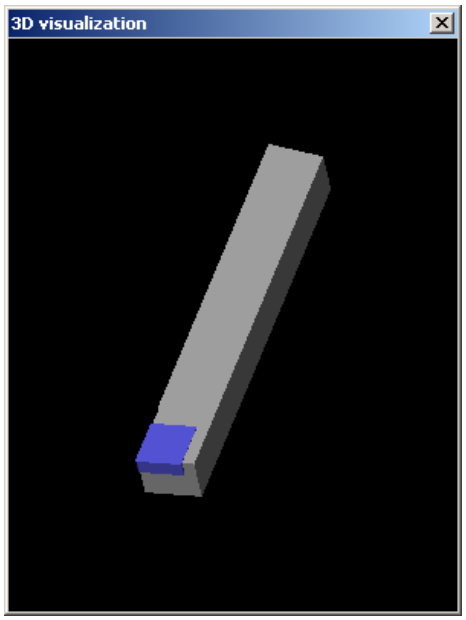
OK

- Confirm the entries with "OK".

Visualizing a tool

3D on

- 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.

Strg

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

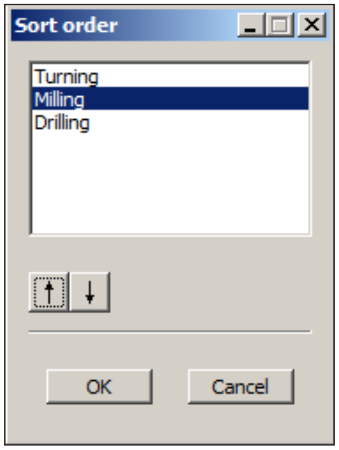
Shifting
Press the right mouse button + mouse movement in the required direction to shift the simulation image.

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.

Sort

- Press the button for sorting.



- Define new sorting sequence.

OK

- 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.



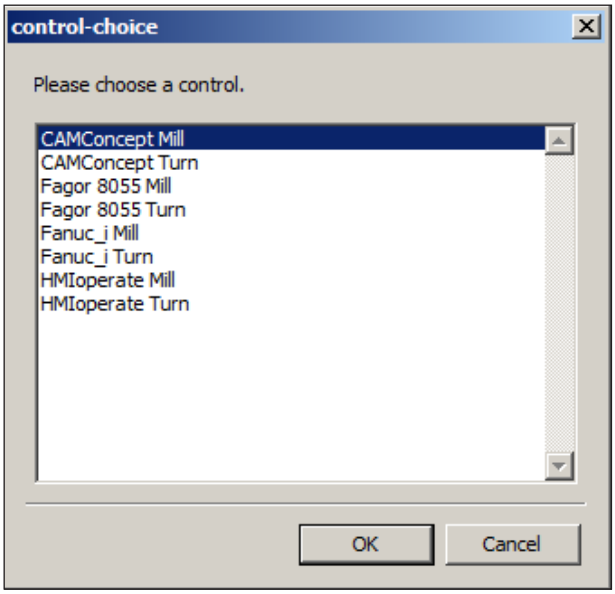
Safety-related parameters are protected by a password. They can only be activated by set-up technicans or by technical support representatives.



Icon for EMConfig

How to start EMConfig

Open EMConfig.



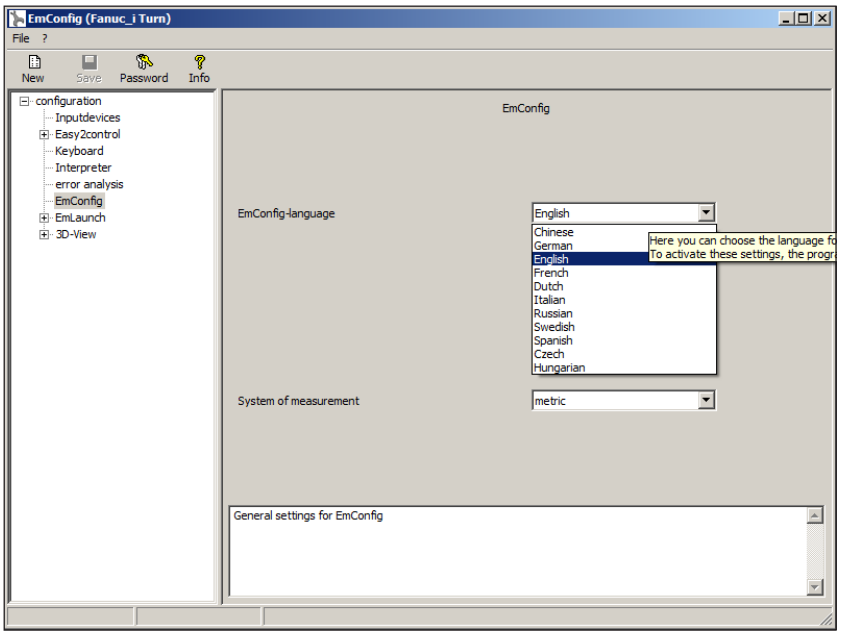
Selection box for control type

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.



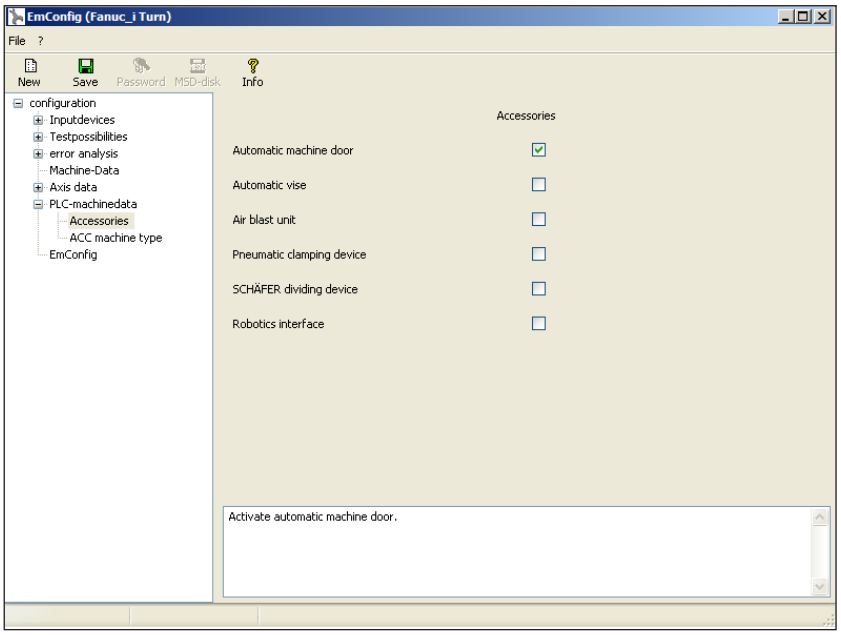
Change the language of EMConfig

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

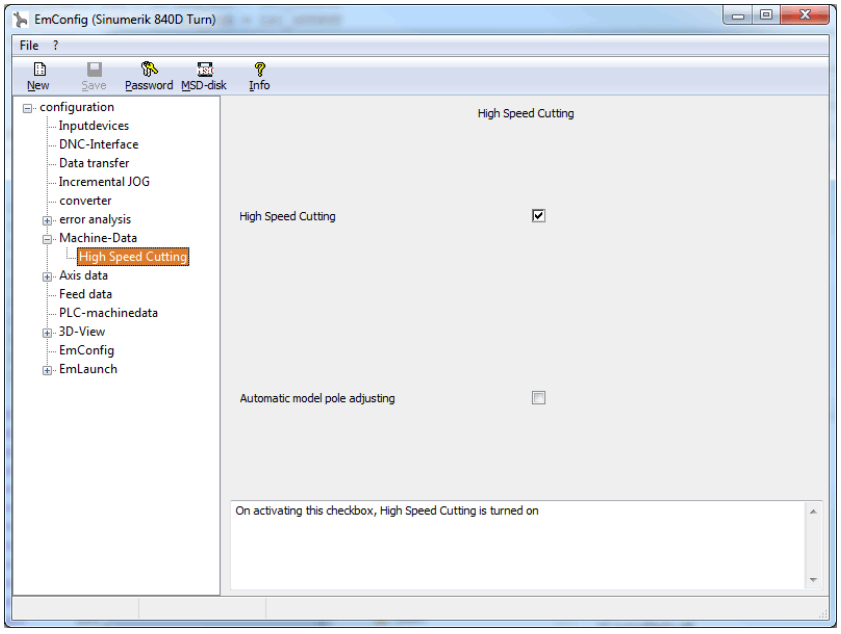
When you install accessories on your machine, you need to activate them here.



Activate accessories

High Speed Cutting

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



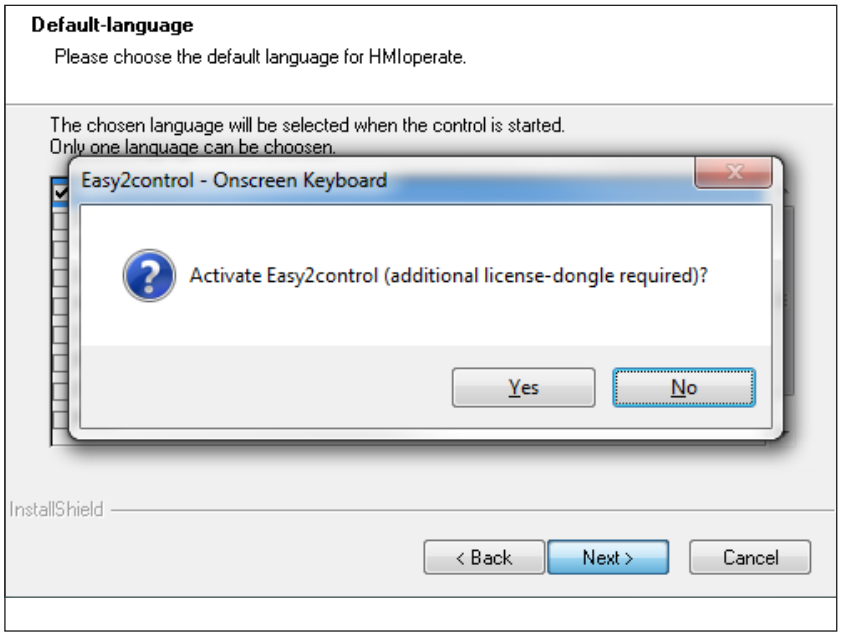
Activate High Speed Cutting

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.

Note:
If Easy2control is used without the hardware-dongle, the controls are diactivated and an appropriate alarm is output by the controller. However, the virtual keyboard is displayed completely.

Easy2control on screen operation

Installation and activation using the example of WinNC for Sinumerik Operate.

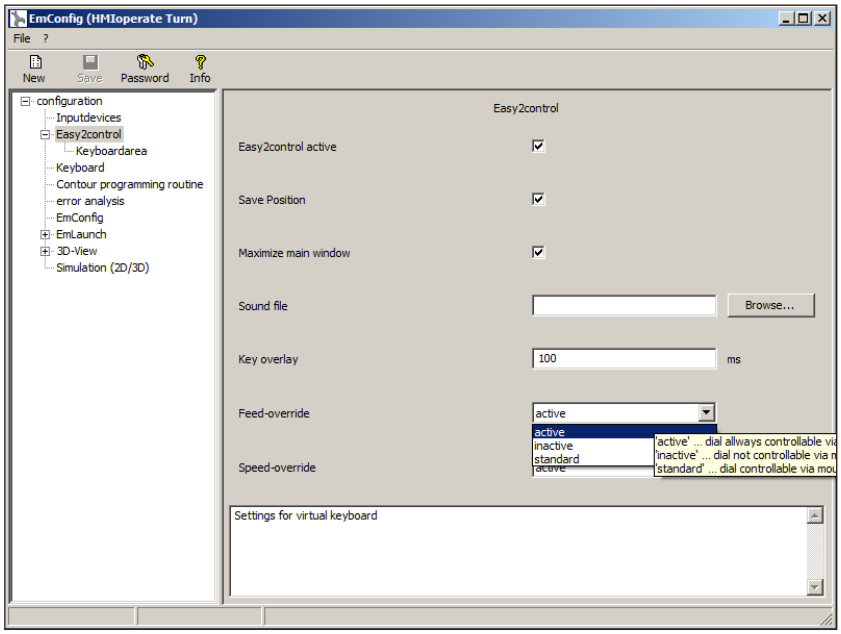


Activating Easy2control

When installing the software WinNC for Sinumerik Operate you will be prompted to activate Easy2control. In order to use the Software without restriction, the supplied dongle must be connected to a free USB port.

Settings

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



Easy2control settings

Dial feed-override and dial speed-override:

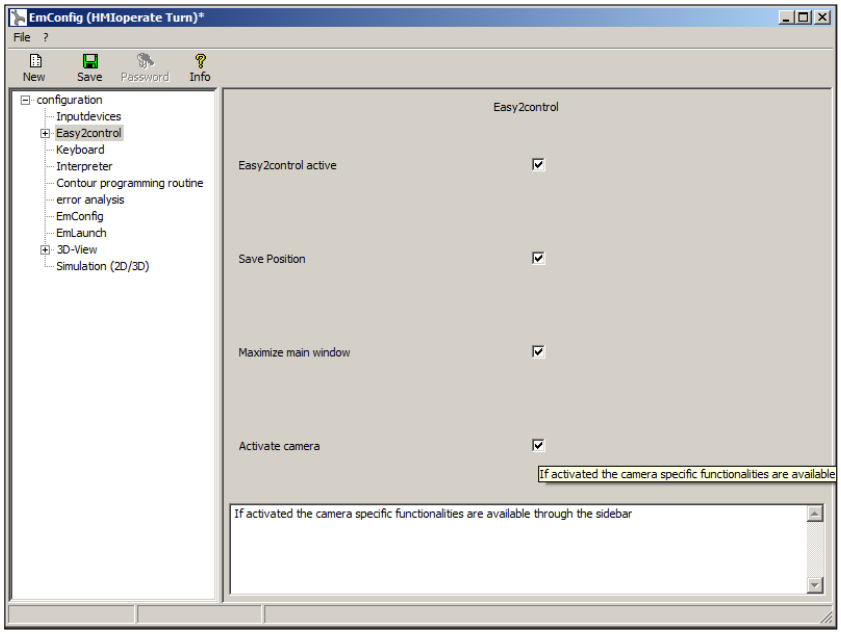
- **Aktive**: dial always controllable via mouse/touchscreen (even with available hardware- version).
- **Inaktive**: dial not controllable via mouse/touchscreen.
- **Standard**: dial controllable via mouse/touchscreen when no hardware-version is available.

Machine room camera

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.

The accessory machine room camera is available for all controls which support Easy2control.



Activate camera

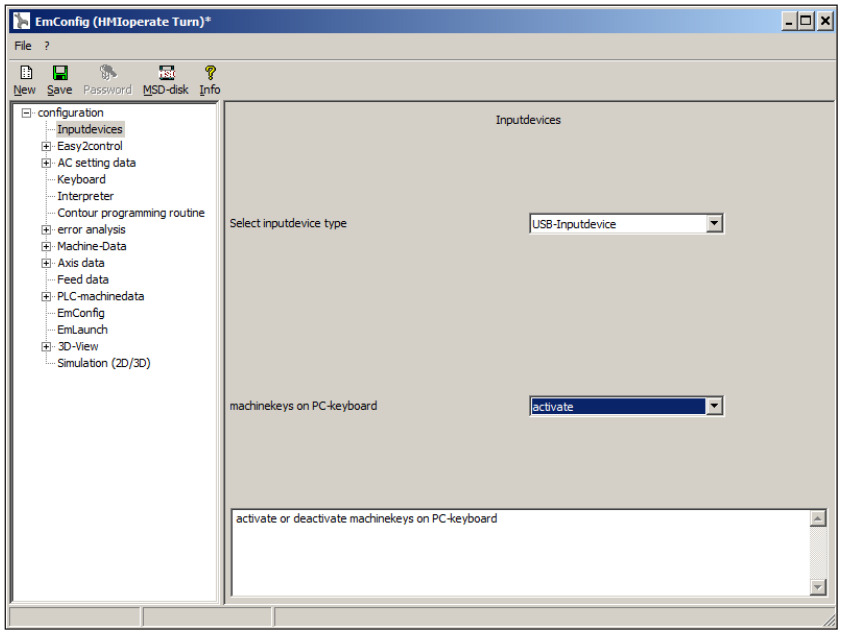
Please refer to chapter Y “external input devices” for more information about the installation guide for the the camera.

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



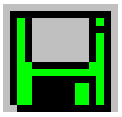
Setting for the machinekeys

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



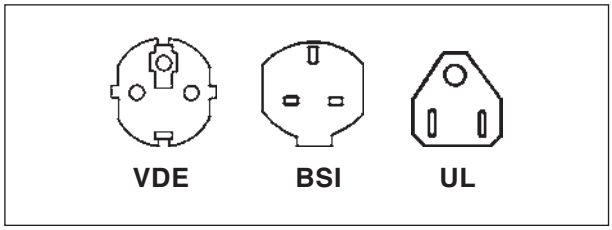
After having changed the machine data, the machine data floppy disk or the machine data USB flash drive must be in the appropriate drive.
Otherwise your changes cannot be saved and get lost.

Y: External Input Devices

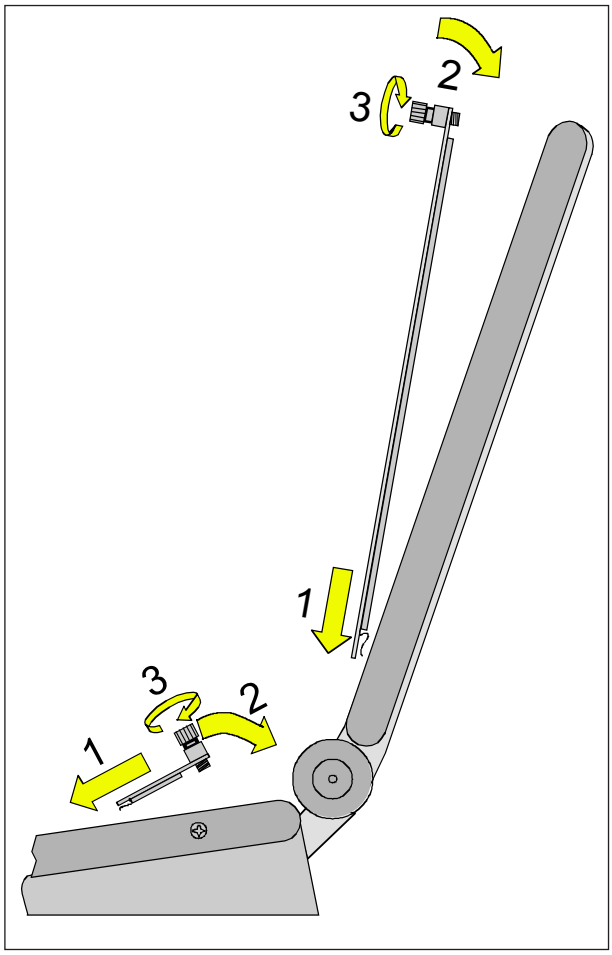
EMCO Control Keyboard USB

Scope of supply

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



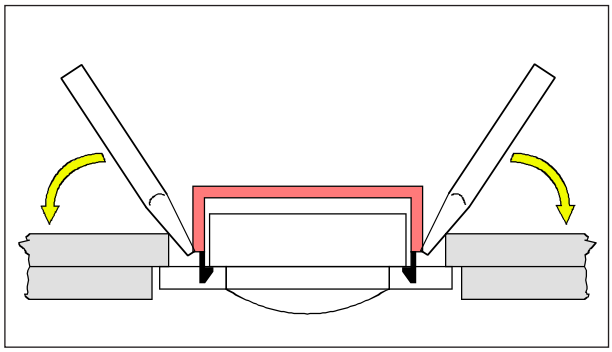
Ref. No.	Description
X9B 000	Basic unit with USB cable
X9Z 600	TFT Display with screen cable and power supply unit
A4Z 010	Mains cable VDE
A4Z 030	Mains cable BSI
A4Z 050	Mains cable UL
X9Z 050N	Key module FAGOR 8055 TC 2 key sheets with keys
X9Z 055N	Key module FAGOR 8055 MC 2 key sheets with keys
X9Z 426N	Key module HEIDENHAIN 426/430 2 key sheets with keys 1 package exchange keys
X9Z 060	Key module WinNC for SINUMERIK OPERATE 2 key sheets with keys
X9Z 030	Key module WinNC for FANUC 31i 2 key sheets with keys 1 package exchange keys
X9Z 640	Key module Emco WinNC for HEIDENHAIN TNC 640



Assembling

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

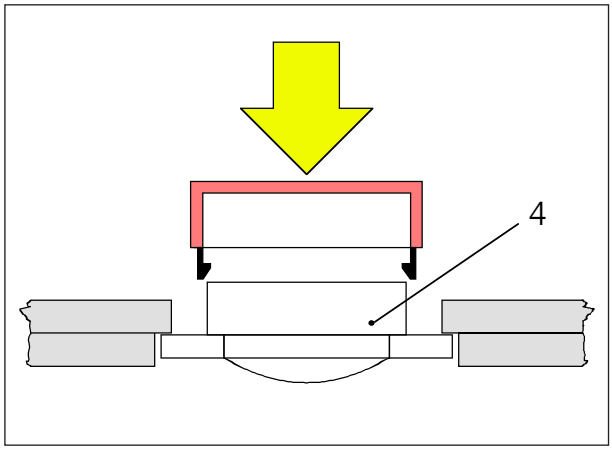
Note:
The key sheets must not be bended, otherwise the switching function can not be warranted.



Exchange of single key caps

Off works the keyboards are equipped with the keys for turning.
The scope of supply includes a package of exchange key caps to equip the keyboard for milling.
If you want to use the control keyboard for milling, you have to exchange a part of the key caps. Exchange them as shown on the following pages.

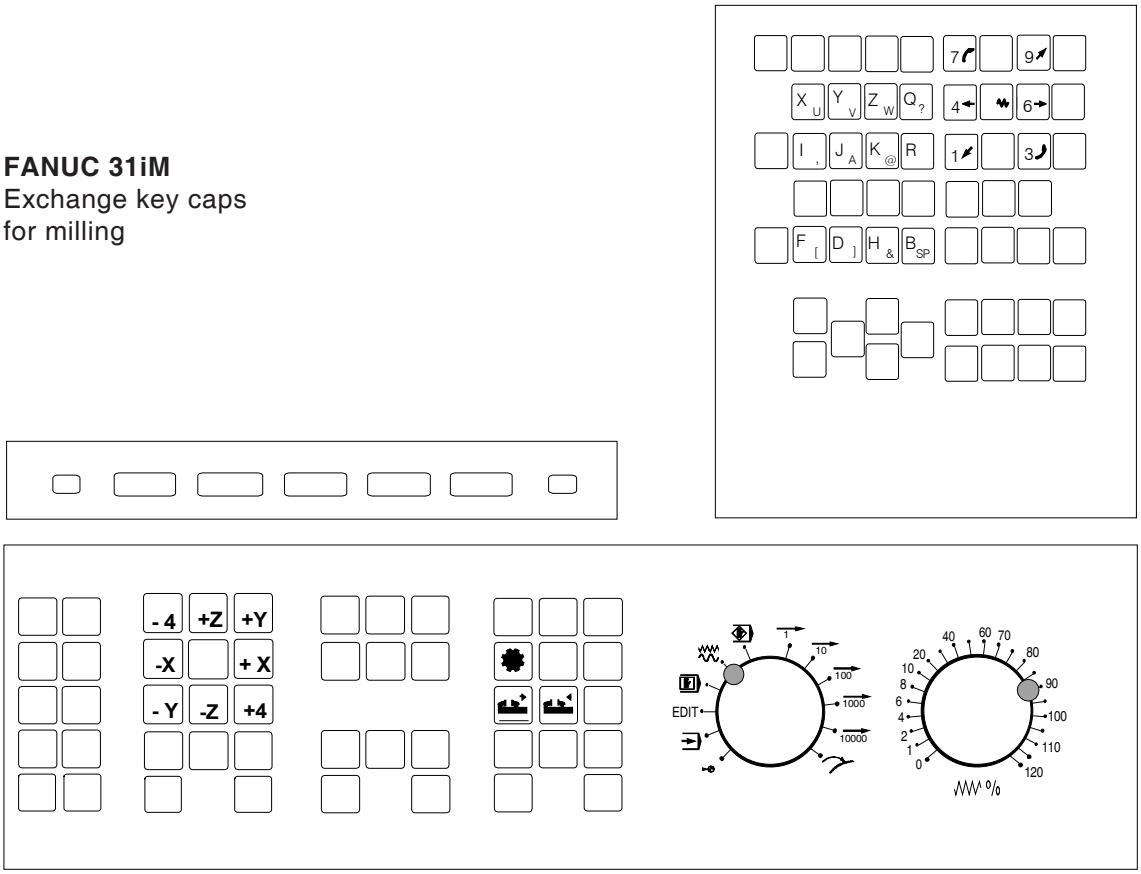
Note:
For the control type Emco WinNC for HEIDENHAIN TNC 640 only a milling version is available.



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

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

FANUC 31iM
Exchange key caps
for milling



Connection to the PC

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

Settings at the PC software

Setting during new installation of the PC software

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

Setting in case of PC software already installed

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

Easy2control On Screen operation

Easy2control adds a range of attractive applications to the successful interchangeable control system used in EMCO training machines. Suitable for use in machines and simulation workplaces alike, it displays additional control elements directly on the screen and, when used together with a touchscreen monitor, provides the ideal input interface.

Scope of supply

The software for Easy2control is part of the control software.
The dongle for a workstation license is delivered:

Ref. No. X9C 111

Technical data for the screen:

16:9 Full-HD Monitor (1920x1080) at the minimum

Currently available controls (T and M):

- Sinumerik Operate
- Fanuc 31i
- Emco WinNC for Heidenhain TNC 640 (M only)
- Emco WinNC for Heidenhain TNC 426 (M only)
- Fagor 8055

Note:
If a Full HD monitor is used without touchscreen function, the control is operated just with mouse and keyboard.

Operating areas

Sinumerik Operate



Machine control panel

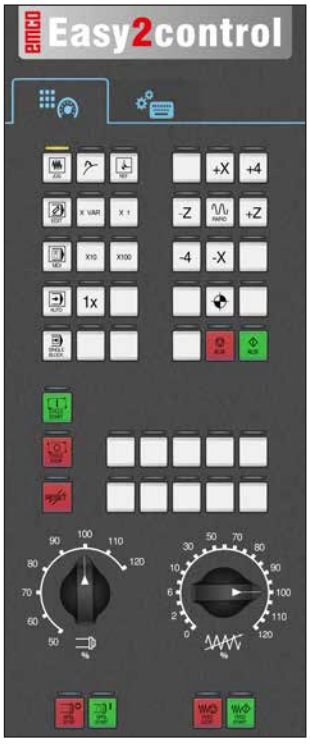


Controller-specific operation



Control operation complete

Fanuc 31i



Machine control panel



Control operation complete

Emco WinNC for Heidenhain TNC 640



Machine control panel

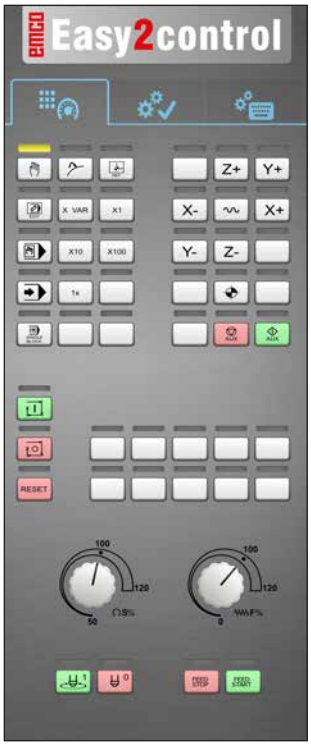


Controller-specific operation

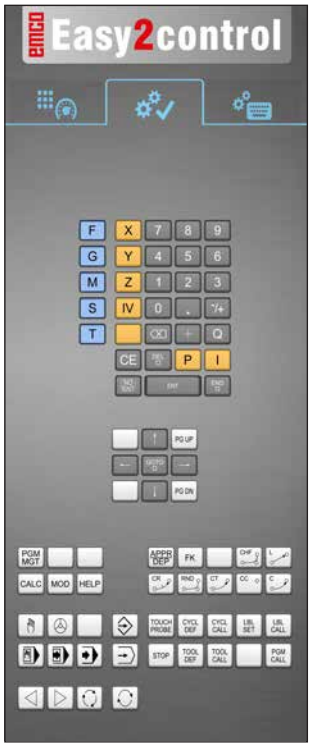


Control operation complete

Heidenhain TNC 426



Machine control panel

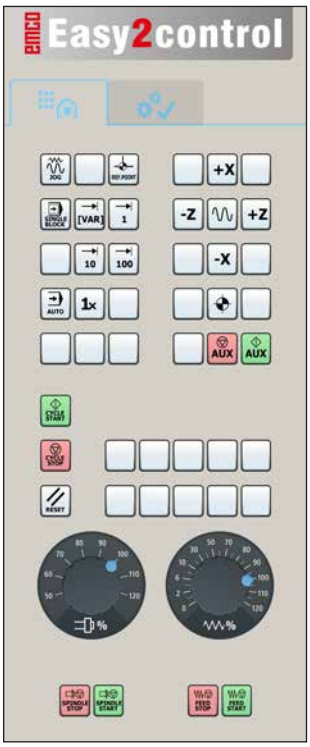


Controller-specific operation

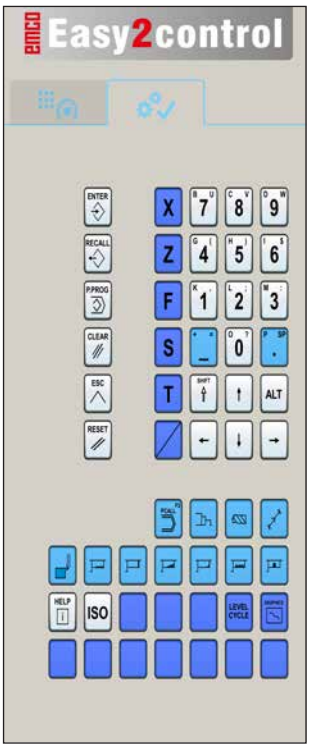


Control operation complete

Fagor 8055



Machine control panel



Control operation complete

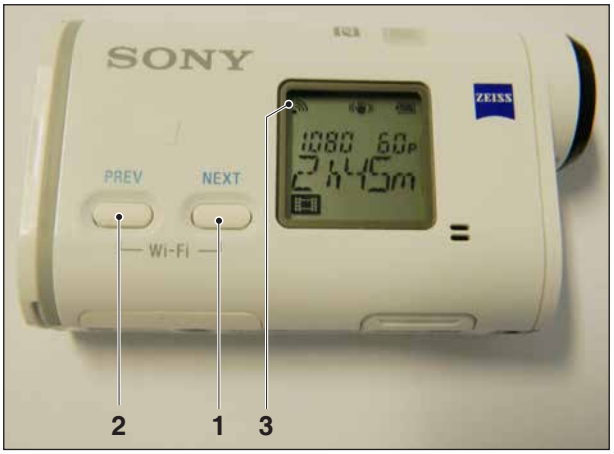
Refer to the chapter “Key Description” of the respective control description for operation and key function.

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

Machine room camera

This accessory can be ordered using the following order number:

Ref. No.: S4Z750



Activating WLAN



Connect WLAN

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).



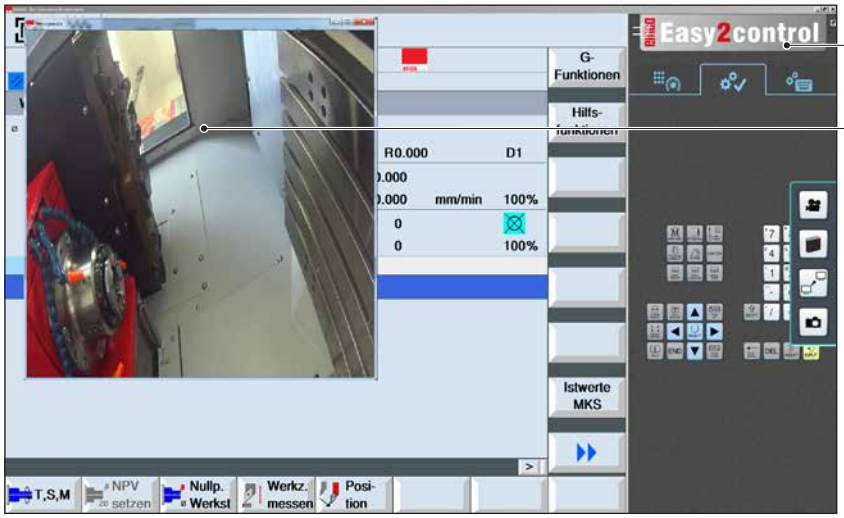
- Open the operating manual of the control.



- Option for the second screen:
  - duplicate screen
  - screen extention to two monitors



- Creates a screenshot of the control in *.png format.



Operating the machine room camera

Note:
The option for the second screen is only available for machines CT/CM 260 and 460.

Attention:
The camera may not be operated without the supplied waterproof case.

If the camera is operated without the waterproof case it may be damaged by coolant and chips.

Z: Software Installation Windows

System prerequisites

Machine with integrated control PC
- All Concept machines
- Machines that were converted to ACC
- MOC with Windows 7 or higher (32 / 64 Bit)

Machines with included control PC and programming stations
- Windows 7 or higher (32 / 64 Bit)
- free hard drive space 400 MB
- Programming station: 1*USB, machine version: 2*USB
- TCP/IP-capable network card for machine version)

Recommended system prerequisites
- PC Dual Core
- Working memory 4 GB RAM
- free hard drive space 2 GB

Software installation

- Start Windows
- Start the installation application on the USB stick or your download file.
- Follow the instructions from the installation guide.

For more informations regarding software installation and / or software update please refer to the documentation “short description for WinNC update installation”.



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

Variants of WinNC

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

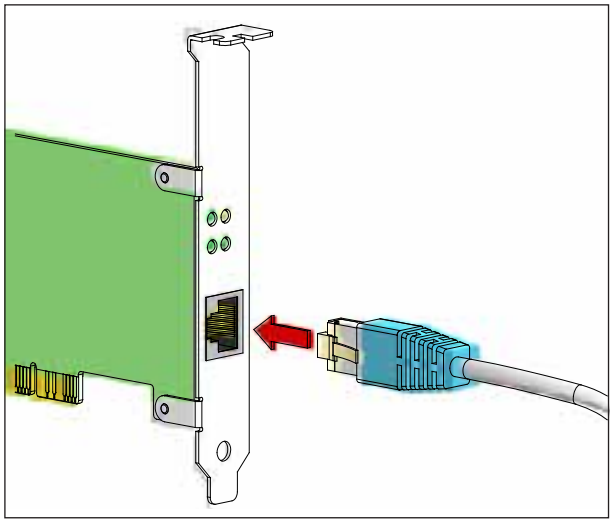
- WinNC for SINUMERIK Operate T and M
- WinNC for FANUC 31i T and M
- HEIDENHAIN TNC 426
- Emco WinNC for HEIDENHAIN TNC 640
- FAGOR 8055 TC and MC
- CAMConcept T and M

In case there are several control types installed, a menu appears when starting EM Launch from which you can select the desired type.
The following versions can be installed from the WinNC variants:

- Demo licence:
  The demo licence is valid for 30 days after the first use. 5 days before the demo licence expires, you can enter another valid licence key (see licence manager)
- Programming station:
  Programming and operation of the appropriate CNC control type is simulated by WinNC on your PC.
  - Single user licence:
    Authorizes to external programming of CNC-controlled machine tools on one PC workstation.
  - Multi-user licence:
    Authorizes to external programming of CNC-controlled machine tools. The multi-user licence can be installed on an unlimited number of PC workstations or in a network within the institute registered by the licensor.
  - Educational licence version:
    Is a time-limited multi-licence especially for schools and educational institutes.
- Machine licence:
  This licence allows to directly operate a PC-controlled machine (PC TURN, Concept TURN, PC MILL, Concept MILL) of WinNC as if it was operated by an ordinary CNC control.

Danger:
Removal and installation of the network card must only be carried by skilled personnel. The computer must be disconnected from the power supply (pull the power plug).

Note:
During a machine installation one network-card is reserved exclusively for the control of the machine.



Connection of the machine to the PC

Network card (ACC)

for:
Concept Turn 55
Concept Mill 55
Concept Turn 105
Concept Mill 105
Concept Turn 60

Only for machines with ACC kit:
PC Turn 50
PC Mill 50
PC Turn 100
PC Mill 120

Network card type: TCP/IP compatible network card

Setting the network card for the local connection to the machine:

IP address: 192.168.10.10
Subnetmask 255.255.255.0

In case of problems observe the instructions of your operating system (Windows help).

Instructions:
If the network connection to the machine could not be established at the start, the above adjustments are to be made.

Starting WinNC

If you choose AUTO START YES during the installation of your machine version, WinNC starts automatically after switching on the PC.

Otherwise proceed as follows:

1 Switch the machine on.

2 Wait for 20 seconds to ensure that the machine operating system is running before the network connection to the PC is established. Otherwise it is possible that no connection can be established.

3 Switch the PC on and start Windows.

4 Click on the start symbol at the bottom.

5 Select program, EMCO and click on WinNC.

6 The start image will be shown on the screen. The licence holder is registered in the start screen.

7 If you have only installed one CNC control type, it starts immediately.

8 If you have installed several CNC control types, the selection menu appears.

9 Select the desired CNC control type (use cursor buttons or mouse) and press ENTER to start it.

10 If you use the control keyboard, you can select the desired CNC control type with the cursor buttons or mouse and start with the "NC-Start" button.



Selection menu EMLaunch

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

Terminating WinNC



1 Switch off auxiliary drive with AUX OFF.
Only for machine places, not for programming stations.



2 By simultaneously pressing these buttons WinNC for Sinumerik Operate will be terminated specifically.
This corresponds to Alt+F4 on the PC keyboard.

Checks by EmLaunch

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



DHCP disabled

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.



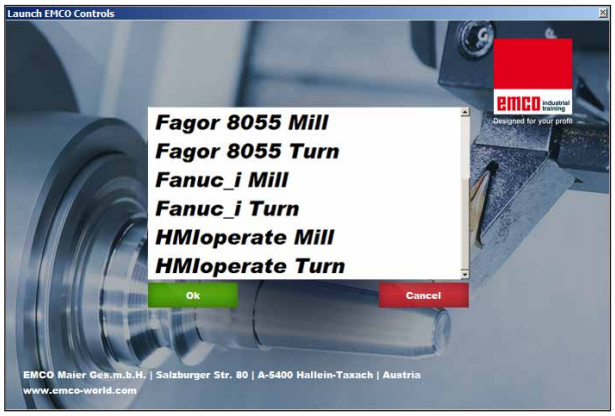
IP-address configuration

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



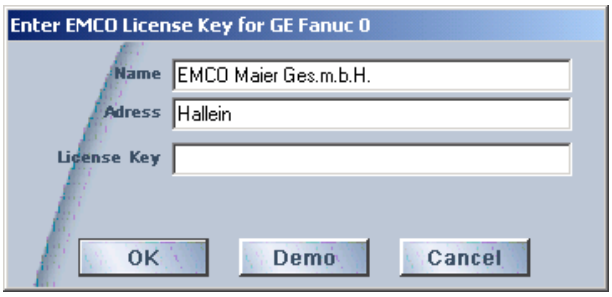
Setup the connection to the machine

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.



Conncection to the machine is OK

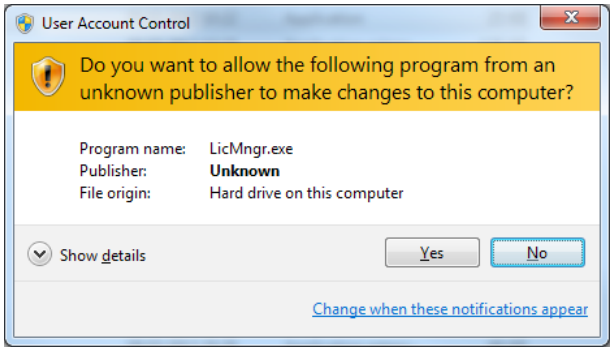
The connection to the machine is completed and the corresponding control can be started.



Input window licence key enquiry

Licence input

After the installation of an EMCO software product, an input window appears during initial operation and asks for name, address and licence key. This input window appears for every software product that is installed. In case a demo licence is desired (see page Z1), please select "DEMO". Then the input window reappears only 5 days before the expiry date of the demo licence. A subsequent input of a licence key is also possible via the licence manager (see licence manager below).



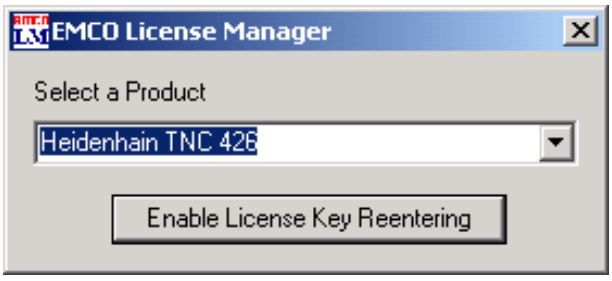
Run EMCO licence manager as an administrator

Licence manager

The query in the UAC dialog box must be confirmed with Yes in order to start the Licence Manager.

For the release of additional function groups of existing EMCO software products it is necessary to enter a new licence key (exception: demo licence).

The ***EMCO Licence Manager*** (see picture on the bottom on the left) enables the input of further new licence keys. For this purpose select the new product in the selection window and confirm the input.



EMCO Licence Manager

The next time you start your control software an input window appears and asks you to enter name, address and licence key (see picture on the top left).

Please note that the licence key is asked for each software product individually. The picture on the left shows e.g. the input prompt for the licence key for the software product "Heidenhain TNC 426".

Input licence key:
Start the WinNC with the option “Run as Administrator” right after installing the programm or launching the licence manager.