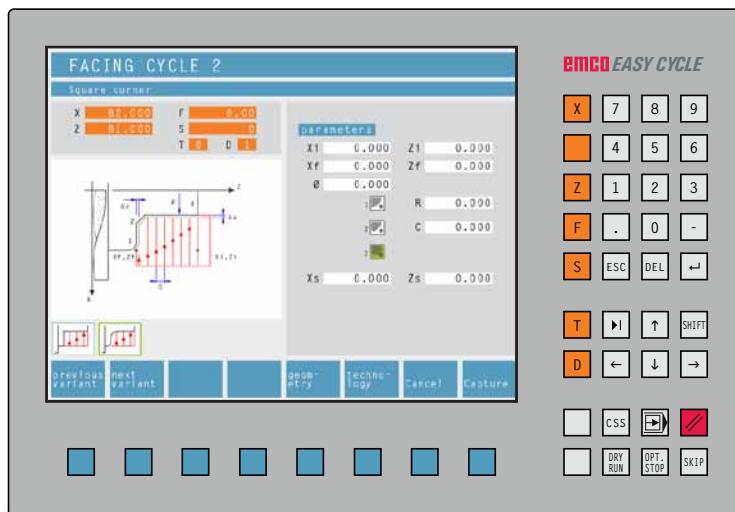


EMCO EASY CYCLE T

Software description software version from 1.14



Software description EMCO EASY CYCLE Turning

Ref.-No. EN 1839
Edition C 2013-06

This instruction manual is also available on the
EMCO homepage in electronic form (.pdf).

Original instruction manual

EMCO Maier Ges.m.b.H.
P.O. Box 131
A-5400 Hallein-Taxach/Austria
Phone ++43-(0)62 45-891-0
Fax ++43-(0)62 45-869 65
Internet: www.emcoworld.com
E-Mail: service@emco.at



Designed for your profit



Please note:

This instruction manual does not include the whole functionality of the control software EMCO EASY CYCLE. In fact, emphasis was put on the clear and simple illustration of the important functions so as to achieve a most comprehensive learning success.
Depending on the machine you operate with EASY CYCLE, not all functions are available.

Preface

The software EMCO EASY CYCLE is part of the EMCO training concept.

With the EMCO EASY CYCLE, cycle controlled and CNC turning and milling machines are easy to operate. Background knowledge of ISO programming is not necessary.

An interactive contour programming serves to define workpiece contours with linear and circular contour elements.

The programming of a cycle is interactive and with graphic support. A large number of machining cycles that can be combined to a program, are available for the customer. Single cycles or the created cycle programs can be graphically simulated on the screen. The possibility for simulation of DIN programs is limited.

This instruction manual does not include the whole functionality of the control software EMCO EASY CYCLE. However, emphasis was put on the simple and clear illustration of the important functions, so as to achieve a most comprehensive learning success.

If you have any further queries or suggestions for improvement regarding this instruction manual, please do not hesitate to contact us.

**EMCO MAIER Gesellschaft m. b. H.
Dep. Technical Documentation
A-5400 HALLEIN, Austria**

emco group
Designed for your profit

EC conformity



The CE mark certifies, together with the EC declaration of conformity, that the machine and the guidelines are in conformity with the regulations of the directives applicable to the products.

All rights reserved, duplication only upon authorization of Messrs. EMCO MAIER
© EMCO MAIER Gesellschaft m. b. H. , Hallein/Taxach - Austria

Content

A: Fundamentals A1

Reference Points of the EMCO Lathes.....	A1
M = Machine zero point	A1
R = Reference point.....	A1
N = Tool mount reference point	A1
W = Workpiece zero point.....	A1
Absolute workpiece positions	A2
Zero offset.....	A3
Setting the reference point.....	A3

B: Key Description B1

EASY CYCLE Control Keyboard	B1
Screen layout.....	B2
PC keyboard.....	B3
Overview button assignment control keyboard.....	B4
Overview Button Assignment Machine Operating Elements	B6
Key Description	B8
Constant Cutting Speed (CCS) / Constant Spindle Speed..	B8
Single block	B8
Reset key (Reset)	B8
Dryrun (test-run feed)	B8
Selectable Stop.....	B9
Skip (block mask)	B9
NC Stop	B9
NC Start.....	B9
Direction buttons.....	B9
Rapid Traverse	B10
Feed Stop	B10
Feed Start.....	B10
Spindle Stop	B10
Spindle Start	B10
Spindle speed correction	B10
Tailstock forwards, tailstock backwards.....	B10
Coolant	B10
Clamping device	B10
Tool turret	B11
Auxiliary OFF	B11
Auxiliary ON.....	B11
EMERGENCY OFF	B11
Key Switch Special Operations Mode.....	B11
Types of operation.....	B12
Override switch (feed rate override)	B13
USB connection (USB 2.0)	B13

C: Operation C1

Feed F [mm/U].....	C1
Speed fundamentals.....	C2
Spindle speed S [rpm]	C3
Constant cutting speed [m/min]	C4
Moving coordinate axes.....	C5
Setting / resetting the reference point.....	C5
Tool change	C5
Define tool change point.....	C6
Program management.....	C7
Creating a program.....	C8
Deleting a program	C8
Copying a program	C9
Renaming a program	C9
Selecting / deselecting a program at the machine.....	C9
Editing the program	C10
Shifting the cycle.....	C10

Editing the cycle.....	C10
Deleting the cycle	C11
Adding a new cycle	C11
Copying the cycle	C11
NC import / export.....	C12
Export	C12
Import.....	C13
Graphical simulation	C14
Simulate cycle.....	C14
Simulate program	C15
General 2D/3D Simulation.....	C15
Processing time	C15
Frame	C16
Starting the simulation	C16
Stopping the simulation	C16
Clear screen	C17
Single block	C17
Zoom.....	C17
3D simulation with EMCO Win 3D-View	C18
Entering the basic settings.....	C18
Raw part definition	C20
Examples for raw parts definition.....	C21
Simulation	C22
Tool modelling with the 3D-Tool Generator	C23
Generating a new tool.....	C24
Copying a tool	C24
Changing an existing tool	C25
Selecting a tool colour	C25
Visualizing a tool	C26
Sorting function	C27
Contour management.....	C29
Screen layout.....	C30
Define / create a contour	C31
Creating the element starting point.....	C31
Editing an element	C32
TeachIn	C32
Deleting an element	C33
Creating the element straight line	C33
Creating the arc element	C34
Defining contours clearly	C35
Copying, deleting, renaming, chaning and importing / exporting a contour	C35
System.....	C36
Diagnosis function PLC	C36
Diagnosis function USB-SPS.....	C36
Parameters	C36
Security	C36
Network	C36
Setup a FTP-connection to EACY CYCLE	C37
TCP/IP settings	C37
FTP connection	C38
Alarms	C39
Messages	C39
Date on/off	C39
Version numbers	C39
Switching EASY CYCLE off	C39
Restarting EASY CYCLE	C39
Reference	C40
Referencing the X/Z axis	C40
Referencing all	C40
Reference position / path to go	C40
Block scan	C41
Skipping cycles in the block scan	C41

D: Programming D1

Cycles.....	D1
Overview Cycles.....	D1
Defining a cycle	D3
Input of geometry data	D4
Input of technology data	D5
Turning Cycle 1.....	D6
Turning Cycle 2.....	D8
Facing Cycle 1.....	D10
Facing Cycle 2.....	D12
Thread Cutting 1	D14
Thread Cutting 2	D18
Thread Cutting 3	D22
Multiple Gear Thread.....	D26
Grooving Cycle 1	D30
Grooving Cycle 2	D32
Grooving Cycle 3	D34
Grooving Cycle 4	D38
Cut-Off Cycle	D42
Contour Turning.....	D44
Drilling Cycle.....	D48
Tapping Cycle.....	D52
Positioning	D54
Taper Cycle 1 / 2 / 3.....	D56
Rounding Cycle	D60
Iso Edit.....	D66
Overview M-Commands	D66
Overview G-Commands	D67
Miscellaneous commands	D67
Coordinate transformation	D68
Absolute Zero-Offset.....	D68
Incremental Zero-Offset.....	D68
Call subprogram	D69
Cancel Zero-Offset	D69

E: Tool measurement E1

Defining tool data.....	E1
Editing tool data.....	E2
Tool family / Tool edge position	E3
Tool calibration with optical presetting device	E7
1.) Gauging the optical tool presetting device:.....	E7
2.) Tool measurement:.....	E8
Tool measurement with touching method.....	E9

F: Program run..... F1

Requirements	F1
Datum setting.....	F1
Tools	F1
Reference point	F1
Machine	F1
Alarms.....	F1
NC-Start.....	F2
NC-Reset	F2
NC-Stop	F2
Program start, Program stop	F2
Repositioning	F2

H: Alarms and Messages H1

Machine Alarms 6000 - 7999.....	H1
PC MILL 50 / 55 / 100 / 105 / 125 / 155.....	H1
Concept MILL 55 / 105 / 155	H1
PC TURN 50 / 55 / 105 / 120 / 125 / 155.....	H6
Concept TURN 55 / 60 / 105 / 155 / 250	H6
Concept MILL 250	H6
EMCOMAT E160	H6
EMCOMAT E200	H6
EMCOMILL C40	H6
EMCOMAT FB-450 / FB-600.....	H6
Inputunit alarms 1700 - 1899.....	H15
Axis Controller Alarms	H17
Axis Controller Messages	H24
Control alarms 2000 - 5999	H25
Fagor 8055 TC/MC	H25
Heidenhain TNC 426	H25
CAMConcept	H25
EASY CYCLE	H25
Sinumerik for OPERATE	H25

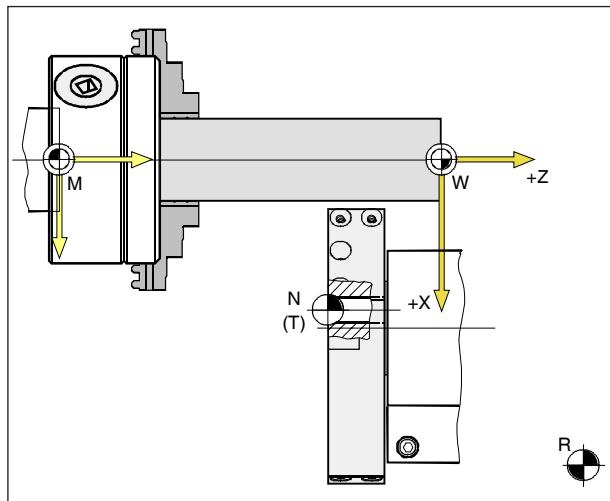
X: EMConfig X1

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

Z: Software Installation Windows Z1

System prerequisites	Z1
Software installation.....	Z1
Variants of WinNC	Z1
Network card (ACC).....	Z2
Starting WinNC	Z3
Terminating WinNC	Z3
Licence input.....	Z4
Licence manager	Z4

A: Fundamentals



Machine points

Reference Points of the EMCO Lathes



M = Machine zero point

An unchangeable reference point established by the machine manufacturer.

Proceeding from this point the entire machine is measured.

At the same time „M“ is the origin of the coordinate system.



R = Reference point

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

Required after every power failure.



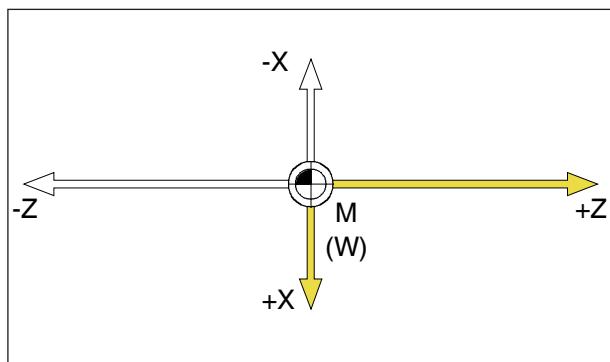
N = Tool mount reference point

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



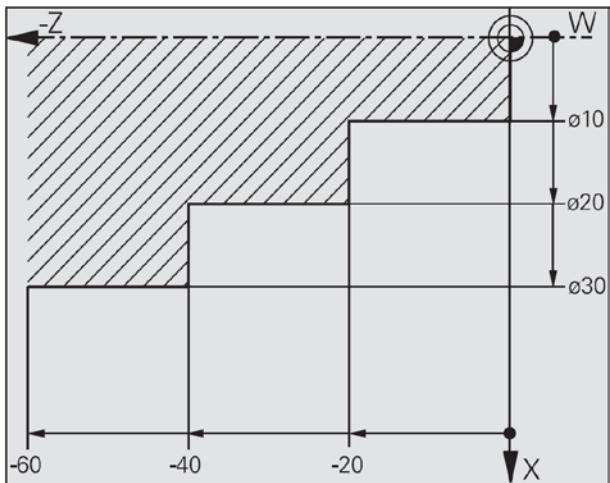
W = Workpiece zero point

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



Coordinate system

Absolute workpiece positions

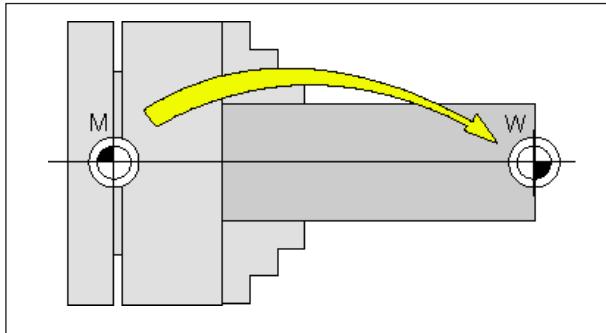


Position coordinates that refer to the workpiece datum are called absolute coordinates.

Every position of a workpiece is clearly determined by absolute coordinates (picture on the top left).

The origin of the coordinate system lies in the machine zero point „M“, respectively in the workpiece zero point „W“ after a programmed zero offset. All target points are described from the origin of the coordinate system by definition of the respective X and Z distances.

X dimensions are indicated as diameter values (like dimensioning on the drawings).



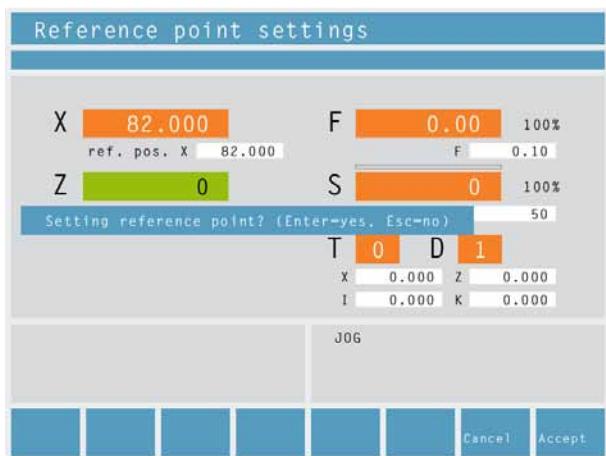
Zero offset from machine zero point M to work-piece zero point W

Zero offset

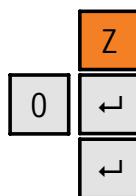
The machine zero point "M" of EMCO lathes is on the turning axis and on the front side of the spindle flange. This position is not suited as starting point for programming. With the so-called zero offset, the coordinate system can be shifted to a suitable point in the working area of the machine.

The control offers two methods to set a zero offset:

- 1.) Set the reference point (see below).
- 2.) Cycle group coordinate transformation. Here, absolute or incremental datum shifts can be used (see chapter D).



Definition of the axis value in Z direction



Setting the reference point

The setting of the reference point is carried out axis by axis and serves to define the workpiece zero point.

- Press the key of the required axis.
- Enter the value with which you want to preset the axis.
- To capture the entry, press "ENTER", to abort the entry, press "ESC". EASY CYCLE opens a dialogue box: (see center of the illustration opposite). To confirm, press "Enter", to abort, press "ESC".

Example:

The workpiece zero point should be defined at the front side of a turning component.

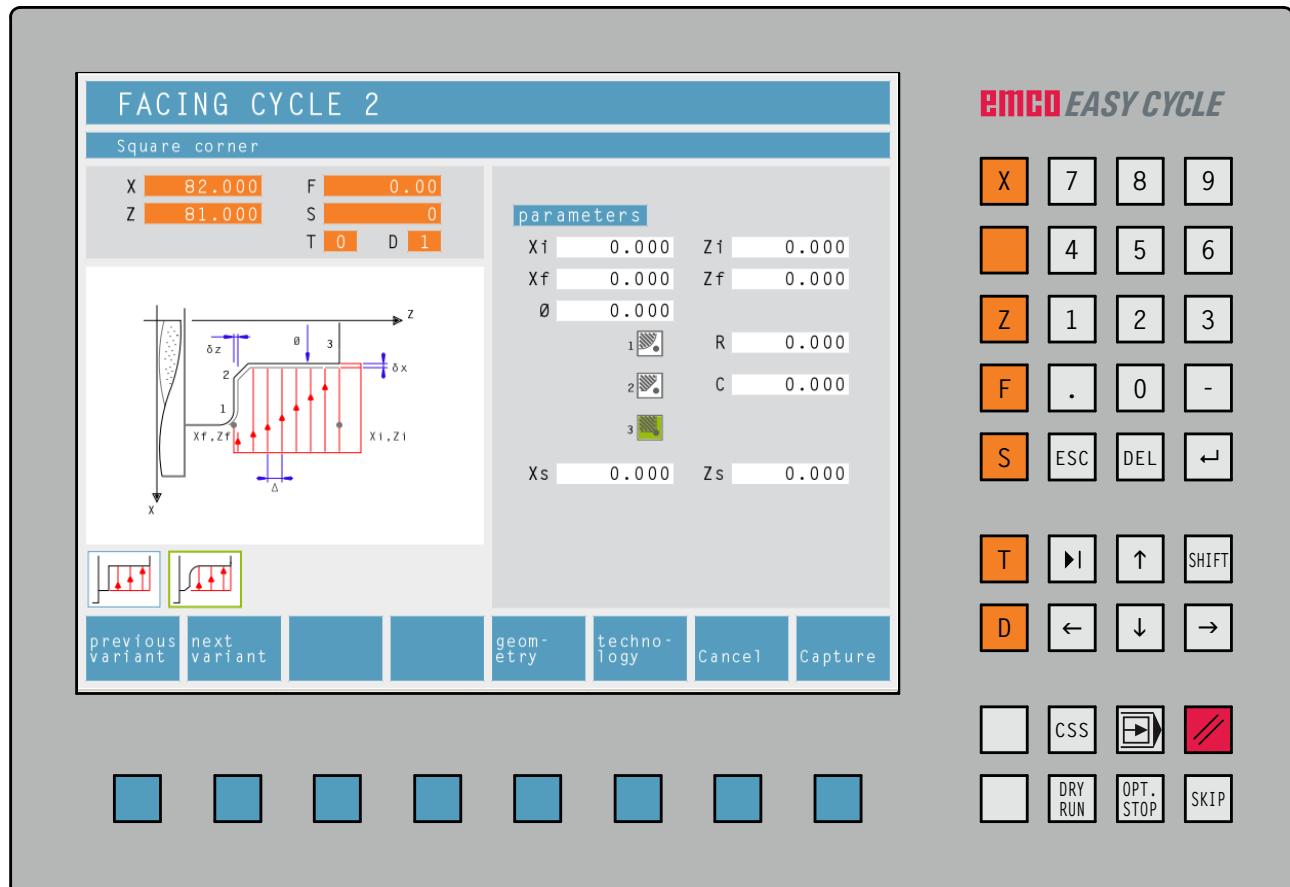
Execution: Touch with a tool on the front side of a faced workpiece.

- Press "Z".
- Enter the value "0" and confirm with "ENTER".
- Press "ENTER" again.

Now, the workpiece zero point W is preset in Z-axis direction with the value "0".

B: Key Description

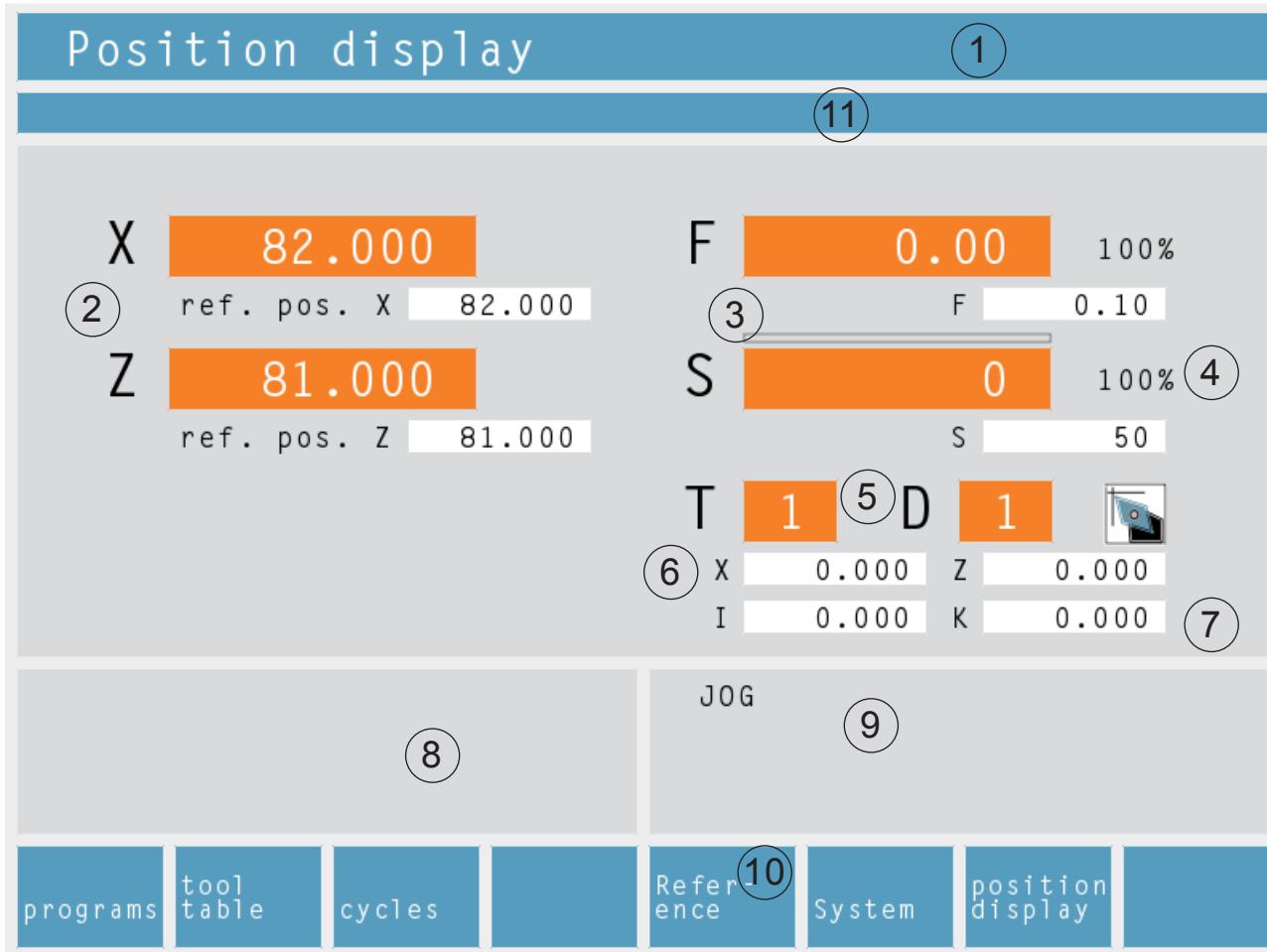
EASY CYCLE Control Keyboard

**Note:**

Depending on the machine you are operating with EASY CYCLE, not all functions and machine buttons may be available.



Screen layout



- 1 Program name and number
- 2 NC position displays; technology data; Softkey allows you to switch between the reference position display and the remaining distance display.
- 3 Display of the actual feed and speed values
- 4 Display of the programmed feed and speed values
- 5 Tool change number; tool correction
- 6 Tool change position
- 7 Tool wear
- 8 Cycle list

The defined cycles of a program will be shown. The just processed cycle will be shown highlighted.

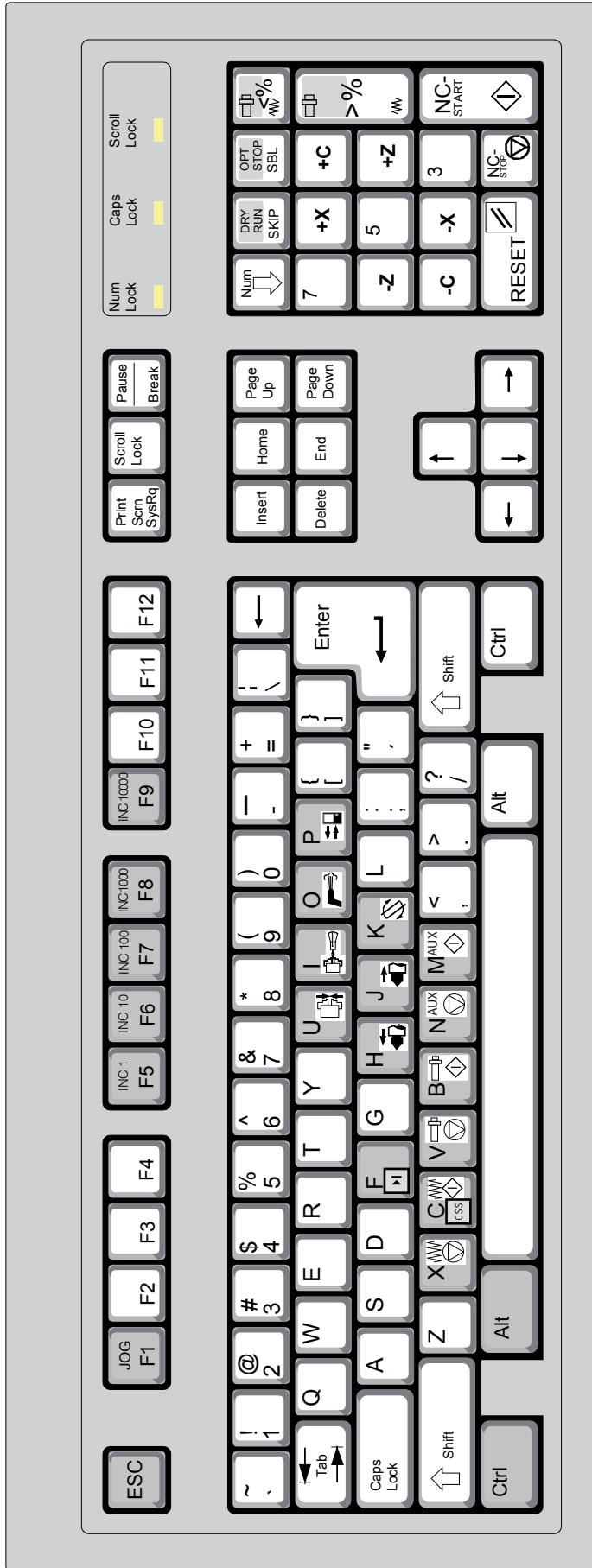
- 9 Status display; operating mode display; processing time;
- 10 Softkeys
- 11 Alarm and notification lines

Note:

Pressing the "ESC" key takes you back to the respective higher-level menu.

KEY DESCRIPTION

PC keyboard



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



Note:

The PC keyboard is only available for EMCO Concept machines.

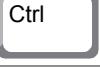
The machine functions in the numerical keypad are only active if NUM Lock is not active.

Overview button assignment control keyboard

PC key	Control system button	Function
X Z	X Z	Select coordinate axes
F	F	Program feed
S	S	Program spindle speed
T	T	Program tool number
D	D	Program correction value
ESC	ESC	Cancel dialog or leave menu.
Delete	DEL	Delete input
Enter	↔	Conclude input and continue dialog.
Ctrl F	▶	Select button
↑ ← ↓ →	↑ ← ↓ →	Move mark
↑	SHIFT	Upper/lower case
↑ X	SHIFT X	Tool change point in X (*)
↑ Z	SHIFT Z	Tool change point in Z (*)
↑ T	SHIFT T	I...tool wear in X (*)
↑ D	SHIFT D	I...tool wear in X (*)

(*)...These functions can only be defined from the Main Menu.
(The Main Menu is the first screen layout which appears after starting EASY CYCLE.)

KEY DESCRIPTION

PC key	Control system button	Function
 		Constant Cutting Speed (CCS) m/min
		Single block
		Reset key (Reset)
 		Dryrun (test-run feed)
 		Selectable Stop
		Skip (block mask)

Overview Button Assignment Machine Operating Elements

PC key	Control elements	Function
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 drives off AUX ON
Enter		NC Start
,		NC Stop

Note:

Selecting the machine buttons via the PC keyboard:

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

KEY DESCRIPTION

PC button	Control elements	Function
	-  + 	Spindle speed correction
	- 	Override (feed rate override)

Key Description

Constant Cutting Speed (CCS) / Constant Spindle Speed



This function offers you the possibility of programming the speed as either constant speed or constant cutting speed (see Chapter C Operation "Speed S").

Single block



This functions allows you to execute a program block by block. The Single Block function can be activated in the automatic mode (a program will be executed automatically) operation type.

When single block processing is active:

- SBL" (=SingleBlock) is shown on the screen.
- the current block of the part program is only processed when you press the NC Start button.
- processing 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.

Reset key (Reset)



- Execution of the current program is broken off.
- Alarm notifications will be cleared.
- The control system is in the starting position and ready for a new program sequence.
- A running program or a positioning movement will be broken off.

Dryrun (test-run feed)



In Dry-run mode, positioning movements will be carried out with the dry-run feed.

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

On starting the NC program, the main spindle will not be activated and the slides will move with dry-run feed speed.

Only perform the test run without a workpiece.

If the test run is engaged, the test "DRY" appears in the simulation window.

Selectable Stop



When this function is active, program processing will stop at blocks in which the M01 special function has been programmed. You start processing again with the NC Start button. If the function is not active, the M01 special function will be ignored.

Skip (block mask)



In Skip mode, program blocks will be jumped over when the program is running.

NC Stop

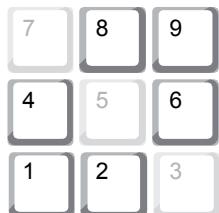


After pressing the NC Stop button the execution of the running program will be broken off. Processing can then be continued by pressing the NC Start button.

NC Start



After pressing the NC Start key the selected program will be started with the current block.



Direction buttons

With these buttons, the NC axes can be moved in JOG operation mode.

Rapid Traverse



If this function is pressed in addition to the movement buttons, the axes concerned move in rapid traverse.

Feed Stop



In "AUTOMATIC" operation mode, this function cancels a slide movement (not with threading).

Feed Start



This function resumes a programmed slide movement which has been interrupted.

If the main spindle motion was also broken off, it must be switched on first.

Spindle Stop



This function interrupts the motion of the main spindle. If it happens during a feed movement, that has to be stopped first.

Spindle Start



This function resumes the programmed spindle motion.

Spindle speed correction

The set spindle speed value S will be shown on the screen as an absolute value and as a percentage (see Chapter C Operation "Spindle Speed Correction").



Tailstock forwards, tailstock backwards

With these functions the tailstock is moved forwards or backwards.

Coolant



This function switches the coolant equipment on or off.

Clamping device



This function activates the clamping device.

Tool turret



In JOG mode, this function swings the tool turret round to the next tool position.

Auxiliary OFF



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

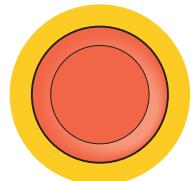
Auxiliary ON



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

The button must be pressed for around 1 second.

Briefly pressing the AUX ON button is a quit function and causes the central lubrication system to perform a lubrication impulse.



EMERGENCY OFF

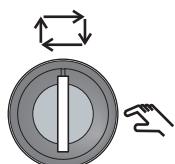
Press the red button only in an emergency.

Effects:

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

Release: Twist button

To go on 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 certain dangerous 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 authorised 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).

Types of operation



REF - Reference mode

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

AUTO

AUTO - Automatic mode

Control the machine by automatically executing programs.

JOG

JOG - Jogging

Standard movement of the machine by continuous movement of the axes via the directional buttons or by incremental movement of the axes via the directional buttons or the handwheel.



Inc 1 - Incremental Feed

Move step by step a predefined distance of 1 increment.

Metrical measurement system: Inc 1 corresponds to 1µm

Imperial (inch-based) measurement system: Inc 1 corresponds to 0.0001 inches



Inc 10 - Incremental Feed

Move step by step a predefined distance of 10 increments.

Metrical measurement system: Inc 10 corresponds to 10µm

Imperial (inch-based) measurement system: Inc 10 corresponds to 0.001 inches



Inc 100 - Incremental Feed

Move step by step a predefined distance of 100 increments.

Metrical measurement system: Inc 100 corresponds to 100µm

Imperial (inch-based) measurement system: Inc 100 corresponds to 0.25 mm



Inc 1000 - Incremental Feed

Move step by step a predefined distance of 1000 increments.

Metrical measurement system: Inc 1000 corresponds to 1000µm

Imperial (inch-based) measurement system: Inc 1000 corresponds to 0.1 inch



Inc 10000 - Incremental Feed

Move step by step a predefined distance of 10,000 increments.

Metrical measurement system: Inc 10000 corresponds to 10000µm

Imperial (inch-based) measurement system: Inc 10000 corresponds to 1 inch

Note:

Switching between the metrical measurement system and the imperial (inch-based) measurement system is carried out with the EmConfig utility software (see Chapter X EmConfig).



Override switch (feed rate override)

The rotary switch enables you to change the programmed feed value F (corresponds to 100%).

The set feed value F in % will be shown on the screen (see Chapter C Operation "Feed Rate Override").



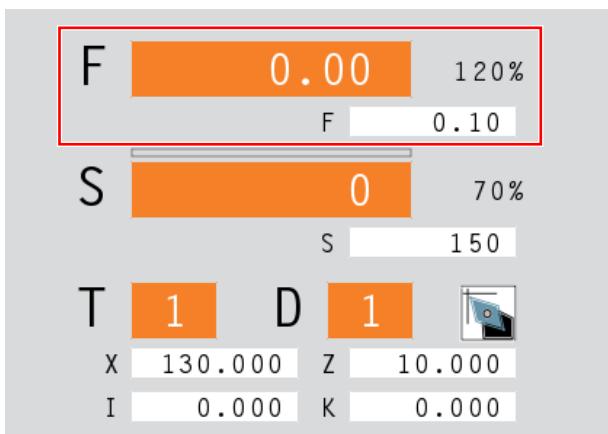
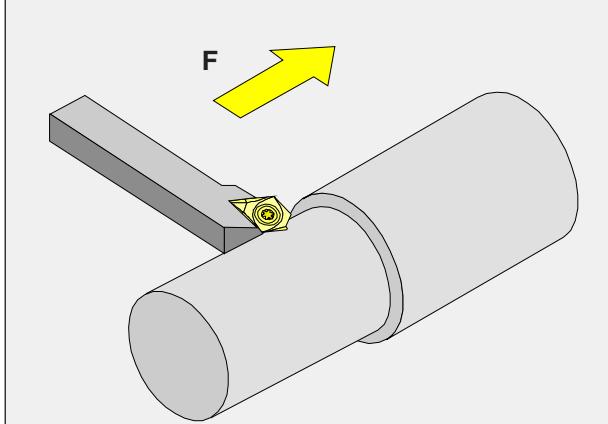
USB connection (USB 2.0)

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

C: Operation

Feed F [mm/U]

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



Input

- Use this key to select the input box for the feed.
- Enter the required feed value.
- Press the enter key. The required value will be automatically entered in the display for the programmed value.

Rapid traverse

For rapid traverse, please enter F99999.

Feed manipulation

The feed value F that you have programmed, corresponds to 100 percent.

The set feed value F can be changed in percent by means of these keys or the feed override switch.

Setting range:

0 % to 120% of the programmed feed rate.
Only the altered percentage value will be displayed, not the resulting effective value.
In rapid traverse, 100% will not be exceeded.

Speed fundamentals

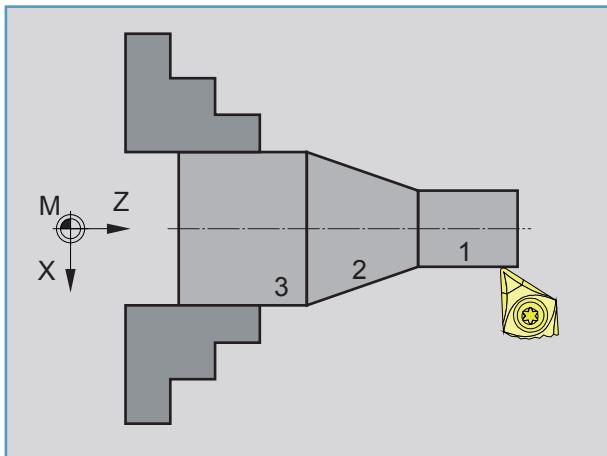
You choose between the following procedures:

- **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:**

Section 1 to 3: same speed.

- **Constant Cutting Speed CCS:**

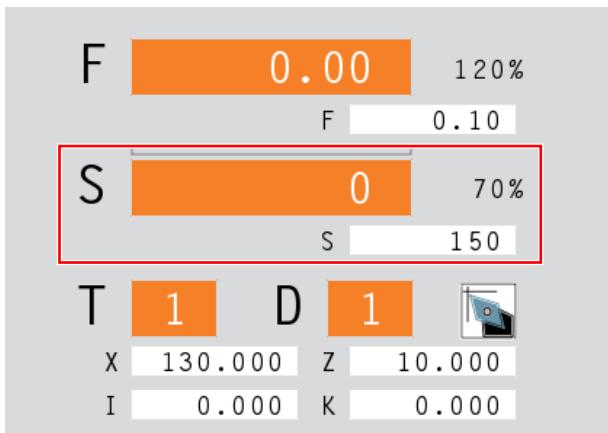
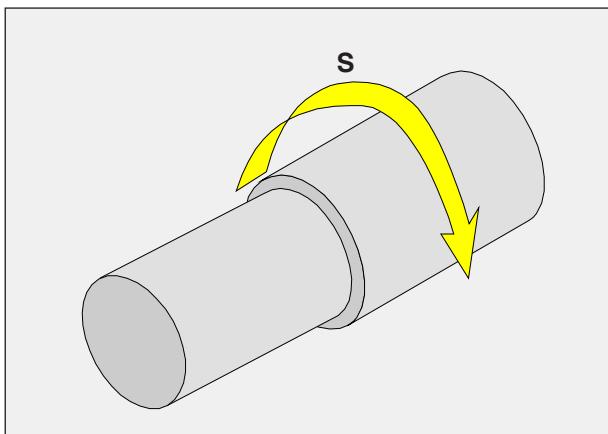
Section 1: high speed.

Section 2: constantly increasing speed.

Section 3: low speed.

Spindle speed S [rpm]

The spindle speed S is entered in rotations per minute (1/min).



Input

- Use this key to select the input box for the spindle speed.
- Enter the required spindle speed.
- Press the enter key. The required value will be automatically entered in the display for the programmed value.

Spindle speed correction

The spindle speed S you have programmed, corresponds to 100%.

The set spindle speed value S can be changed in percent by means of these key sequences or the spindle override switch.

Setting range:

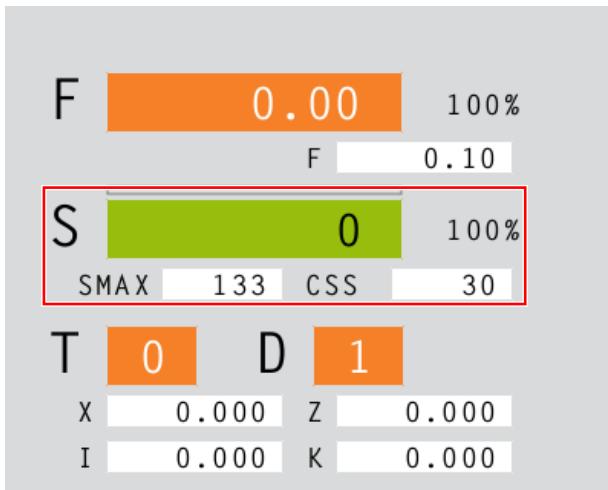
0% to 120% of the programmed spindle speed. Only the altered percentage value will be displayed, not the resulting effective value.

Constant cutting speed [m/min]

Enter the constant cutting speed CSS in (m/min) or in (ft/min).

When the constant cutting speed is activated, the spindle speed is automatically changed in a way, always depending on the workpiece diameter, that the cutting speed S in (m/min or inch/min) will remain constant at the tool edge. Thus, you get smoother turning surfaces and therefore a higher surface quality.

If a workpiece with a large diameter difference is machined, the entry of a spindle speed limitation is recommended. Through this, non-permissibly high spindle speeds for small diameters can be avoided.



- Use this key to select the input box for the spindle speed.
- Enter the required spindle speed.



Activate CSS

- Activation of the constant cutting speed. The programmed spindle speed S is automatically converted into the constant cutting speed CSS.



- Enter the constant cutting speed required.
- Enter the spindle speed limitation Smax.



Deactivate CSS

- By pressing the CSS key again, the constant cutting speed is deactivated. The programmed constant cutting speed CSS is automatically converted into the spindle speed S.

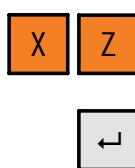
Moving coordinate axes



Example: Move X axis to position 20

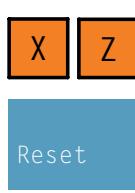
- Select the input box for the axis you wish to traverse.
- Enter the required coordinate value.
- Press the soft key "Move to position", so that the axis is traversed at the set feed rate.

Setting / resetting the reference point



Setting the reference point

- Use the key to select the input box for the required axis.
- Enter the required reference value.
- Press the enter key.
- EASY CYCLE opens a dialogue box. Confirm the prompt with the enter key.



Resetting the reference point

- Use the key to select the input box for the required axis.
- Press the soft key, so that a previously set reference point will be reset to the machine zero point.

Tool change



Example: Select tool 3 with tool edge 2

- Use the key to select the input box for the tool number.
- Enter the number of the required tool (3).
- Press "Enter", so that the tool will be changed.
- Use the key to select the input box for the tool edge number.
- Enter the number of the required tool edge (2).
- Press "Enter", so that EASY CYCLE will consider the selected tool edge 2.



Please note:

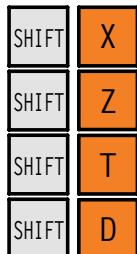
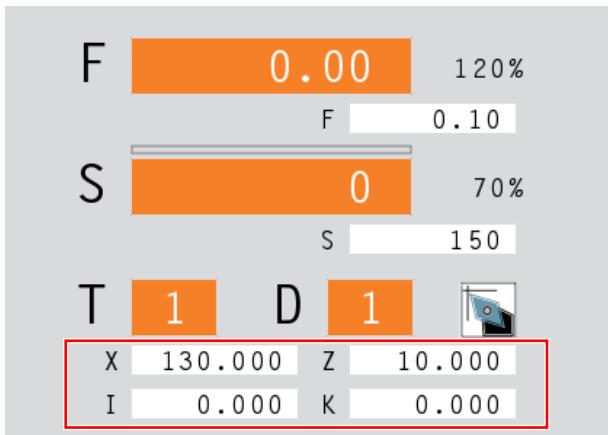
For tool edge number 0, no tool offset will be considered.

Define tool change point

You can define the tool change point with the X and Z coordinates. The coordinates of the work-piece change point are relative to the machine zero point. Prior to the tool change, this position will be approached with the slide reference point.

The tool change point should be selected in such a way that no collisions can occur during the tool change.

The tool change point and the tool wear are defined from the main menu with the following button combinations: (The Main Menu is the first screen layout which appears after starting EASY CYCLE.)

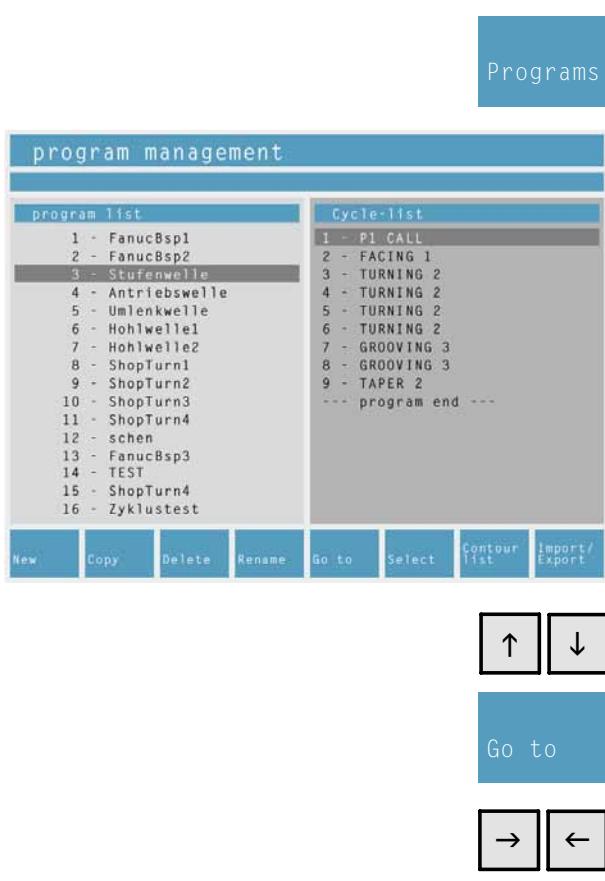


X...tool changing point in X

Z...tool changing point in Z

I...tool wear in X

K...tool wear in Z



Program management

A program consists of a succession of cycles.
Press the soft key in order to get into the program management.

In the program list on the left, the workpiece programs that are stored by EASY CYCLE, are displayed. The cycle list on the right shows the cycles contained.

On the top right you see the program that has been selected at the machine (here: "P3 - Stepped Shaft").

In case there exist more programs than the ones displayed in the window, use the cursor keys in order to move through the list or press the soft key "Go to".

With these cursor keys, you can switch between the lists. The active list is always highlighted in light grey.

The program management offers you the following possibilities:

- creating a program

For marked entries (highlighted in light grey):

- deleting a program
- copying a program
- changing a program
- selecting / deselecting a program at the machine

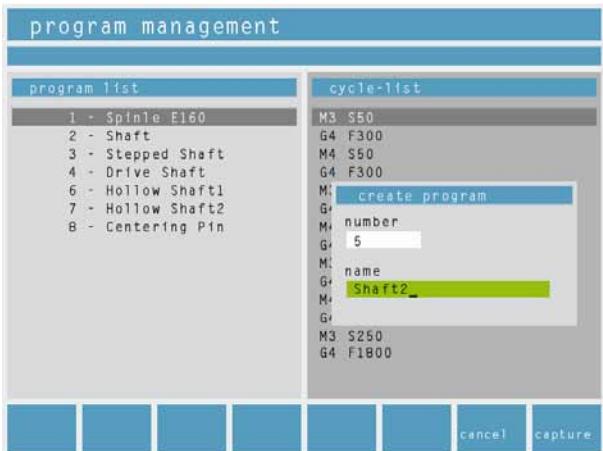
Instructions:

We recommend regular data backup, to prevent the resultant data loss in the event of a defect.



Creating a program

- Press the soft key.



Capture

New

Capture

- Enter the program number and the program name and confirm with the soft key.
- Use the cursor keys to switch to the cycle list and define all values of a cycle (see chapter D, programming).
- Press the soft key in order to capture the cycle into the workpiece program.
- Enter further cycles.

Deleting a program

Only programs that have been deselected can be deleted. This is why the soft key "Delete" is not active when the marked program is selected at the machine.

Deselect

Delete

Capture

- Press the soft key.

- Confirm the selection with the soft key.

Copying a program

- Move the highlight on the required program.
- Press the soft key.
- Allocate a program number and a name.



- Enter the program number and the program name and confirm with the soft key.

Renaming a program

- Move the highlight on the required program.
- Press the soft key. The soft key is only active when the program is not selected.
- Allocate a program number and a name.
- Press the soft key in order to capture the new values.



Selecting / deselecting a program at the machine

- Move the highlight on the required program.
- Press the soft key.
- A marked program is automatically selected by switching to the cycle list.



Please note:

When you create, copy and rename a program, the alarm message line displays "file <number> exists (replace / edit)", if the program number you have allocated does already exist. Program names can be allocated several times.



Editing the program

- Move the highlight on the required program.
- Switch into the cycle list.
- Move the highlight on the required cycle.



Shifting the cycle

- Press the soft key.

- Move the highlight on the position prior to which the cycle shall be inserted.

- Press the soft key.



Editing the cycle

- Press the "ENTER" key.
- Carry out the modifications in the cycle.
- Press the soft key.

Deleting the cycle

- Press the soft key.



Capture

- Confirm the selection with the soft key.

New

Adding a new cycle

- Press the soft key.

Long turning	Facing	Thread cutting	Grooving
Contour turning	Bore	Position	Taper turning
Radius turning	Iso edit	Coord. trans.	Call subprog.

Capture

- Confirm the selection with the soft key.

Copy

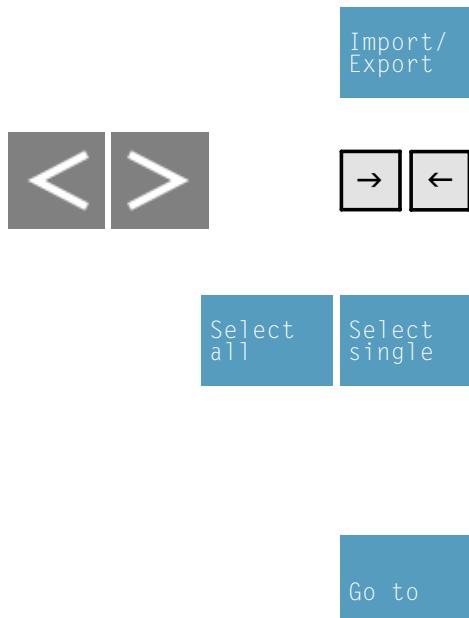
Copying the cycle

- Press the soft key.
- Move the highlight on the position prior to which the cycle shall be inserted.
- Press the soft key.

Insert

NC import / export

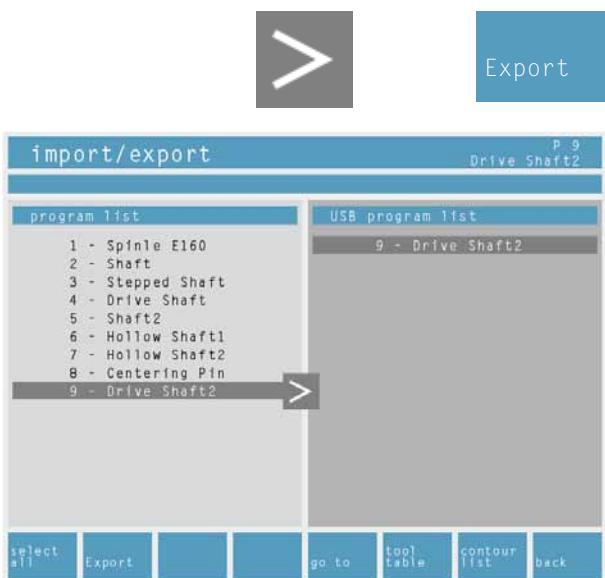
You can import or export programs, contours and the tool table by means of an USB keychain.



- Press the soft key and you will get from the program or contour management to the import / export window.
- Use the cursor keys to select, whether you wish to export or import.
The symbol indicates the file transfer direction.
- In order to export, import or delete files, select all files or single files.

When you press the soft key "Delete" in the program list, all programs will be deleted except for the file you have selected at the machine.

- Press the soft key to select the required file in order to import, export, delete or rename it.



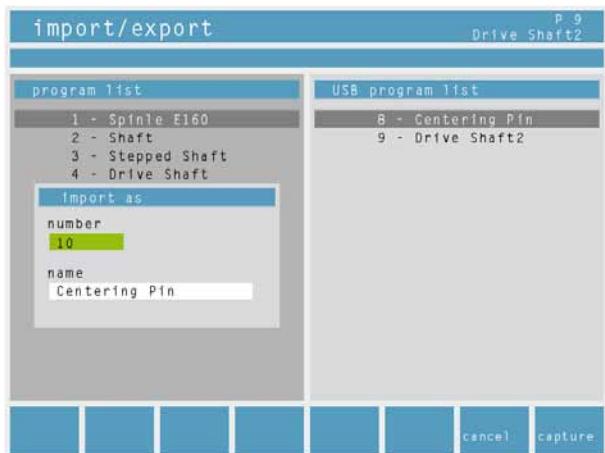
Export

- Press the soft key. The file that has been selected in the program list or in the contour list will be displayed in the USB program list or the USB contour list.



Import

- Press the soft key. The file that has been selected in the USB program list or the USB contour list will be displayed in the program list or in the contour list.



Please note:

When the number of a file in the transfer target has already been allocated, you will be prompted to enter a new number for the target (see left-hand illustration). If you select an already existing number, the alarm message line will display "file <number> exists (replace / change)". You can either overwrite the file that is saved in the target or alter the number (see bottom left-hand illustration). File numbers are definite parameters. File names can be allocated several times.



Graphical simulation

With the graphical simulation, geometrical errors are recognised, e.g. wrong positions, contour infringements, wrong tool etc.

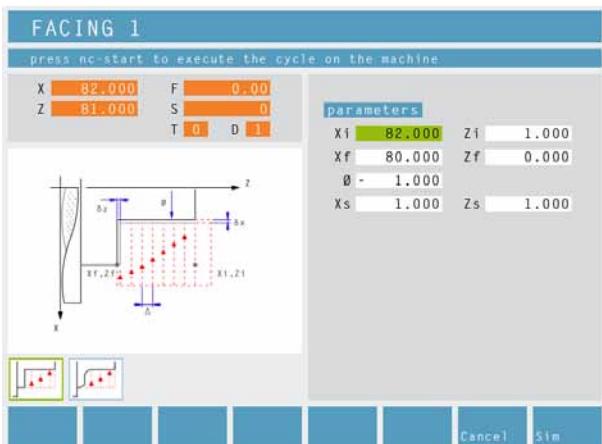
Technological errors will not be recognised e.g. wrong speed or feeds.

There are two possibilities for simulating a cycle in 2D or 3D mode (only with EMCO Win 3D View accessory software):

- in the cycle menu: After definition, the cycle can be simulated immediately.
- in Programme Management: The cycles saved in a program will be simulated.

Note:

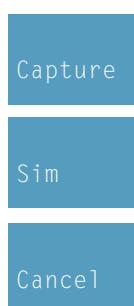
The EMCO Win 3D View software is an accessory to EASY CYCLE and enables the three-dimensional simulation of CNC programs on the screen.



Direct simulation in the Cycles menu

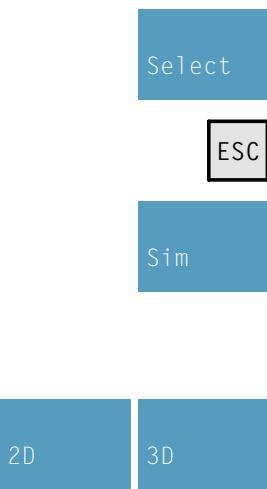
Simulate cycle

- Call up Cycle in the Cycle menu (see Chapter D Programming "Cycles") and enter all required cycle parameters (geometry data and technology data).



Conclude the input with this Softkey.

- Press Softkey.
- This Softkey allows later optimisations to the cycle to be carried out.



Simulate program

- In Program Management (see Chapter C Operation "Program Management") move mark to the program desired.
- Press Softkey.
- Press "ESC" key.
- Press Softkey.

General 2D/3D Simulation

At any time you can check the programmed cycles in a 2D or 3D simulation (only with the software accessory EMCO Win 3D-View).
EASY CYCLE shows all of the tool's position movements in the simulation window.
In addition to the simulation window, the current feeds, spindle speeds, names and position values of the tool are shown.
Error reports also appear here. e.g.: Collision warnings.

Processing time

The processing time is the approximate time EASY CYCLE calculates for the duration of the tool movements executed with feed.
The processing time will be shown in the form (hh:mm:ss).

The time calculated by EASY CYCLE lends itself only to a limited extent to the calculation of the production time, as EASY CYCLE does not take account of machine-dependent times (e.g. for tool change).

For acceleration processes with Constant Cutting Speed activated, approximate values will be used.

The processing time calculated is only valid for a 100% programmed feed value F (Feed Override) and 100% programmed spindle speed S (Speed Override). Non-100% settings cannot be taken into account.



Frame



Start



Stop

Frame

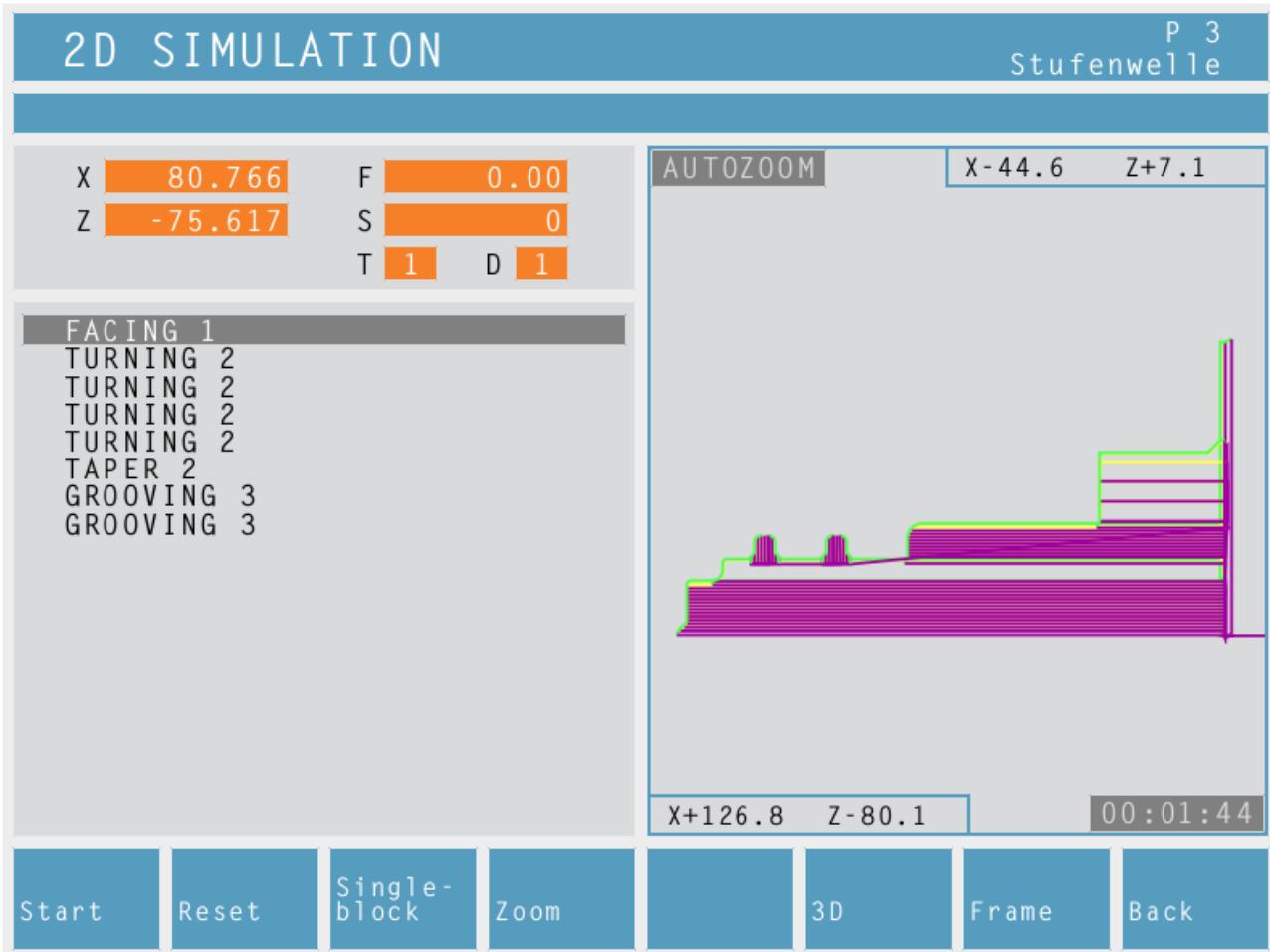
For a better view, you may switch over into the frame mode.

Starting the simulation

The simulation is started by means of this soft key. In order to be able start the simulation, an EASY CYCLE program must have been selected. The program name and the program number of the currently selected EASY CYCLE program are displayed on the top right of the simulation window.

Stopping the simulation

The simulation and the CNC program are stopped by means of this soft key. The simulation can be continued with "Start".





Clear screen

This Softkey breaks off the simulation and the program and resets the initial condition.



S B L

Single block

With this Softkey the simulation stops after every block. The simulation can always be continued with "Start".

Active Single Block mode will be shown in the simulation window.



Zoom

The zoom commands enable the simulation image to be made larger or smaller. It can be moved with the cursor keys.



Larger

After pressing the Softkey the view will be increased by one level in size.

Smaller

After pressing the Softkey the view will be reduced by one level in size.



AUTOZOOM

Automatic

Increases or reduces the display area to the window size.

Active Autozoom will be shown in the simulation window.

3D simulation with EMCO Win 3D-View

- Press Softkey to switch to 3D mode.

3D



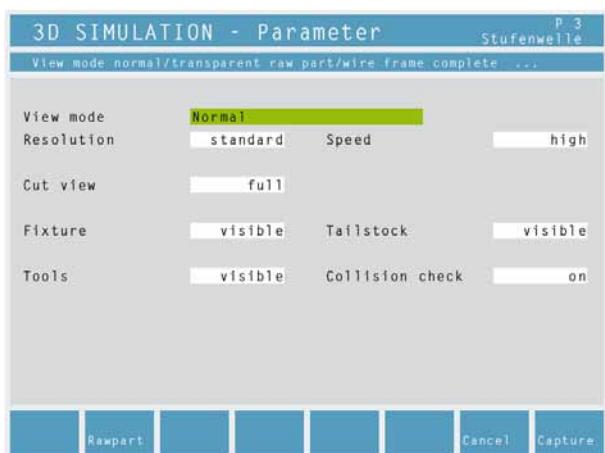
Note:

The EMCO Win 3D View software is an accessory to EASY CYCLE and enables the three-dimensional simulation of CNC programs on the screen.

EMCO Win 3D-View is available only in .programming stations.

Settings

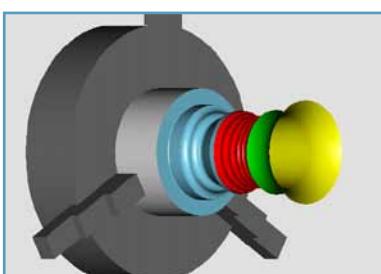
Params



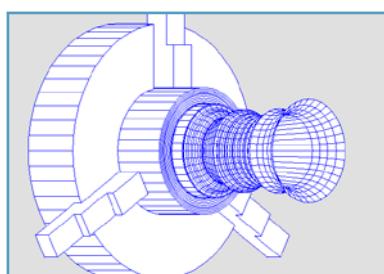
Entering the basic settings

- Press Softkey.
- Press Softkey. You can make the following adjustments:

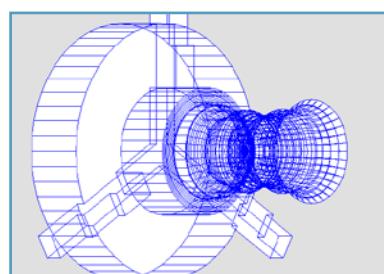
Display type



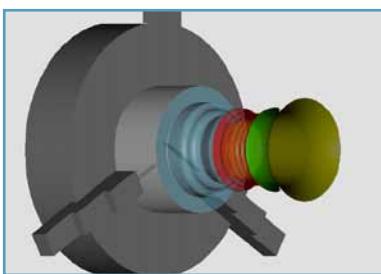
Normal display



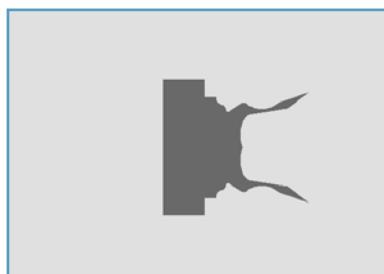
Wireframe



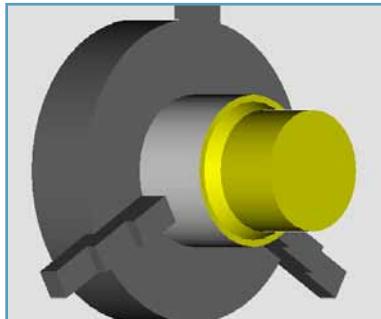
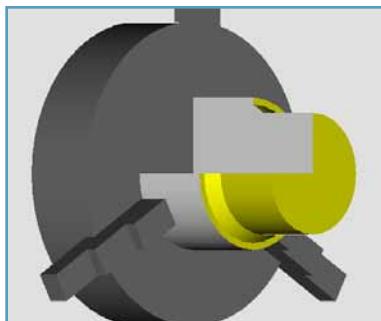
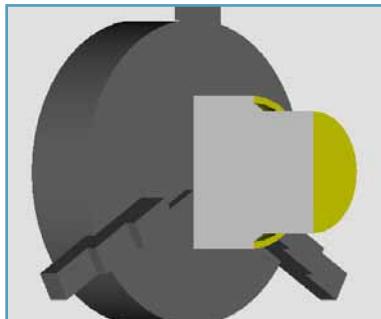
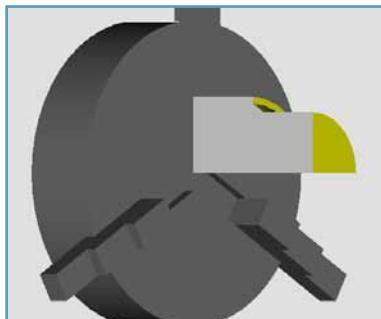
Wireframe complete



Transparent raw part



2D Shaded Profile

*full 3D sectional view**3/4 sectional view**1/2 sectional view**1/4 sectional view***Resolution**

The higher the resolution, the more precise the construction of the 3D image will be.

Speed

With the speed setting the simulation can be slowed down or speeded up.

Sectional view

The sectional display permits the observation of processes which would normally be covered. The representations opposite are available for selection:

Clamping device

visible/invisible

Quill

visible/invisible

Tools

visible/invisible

Collision checking

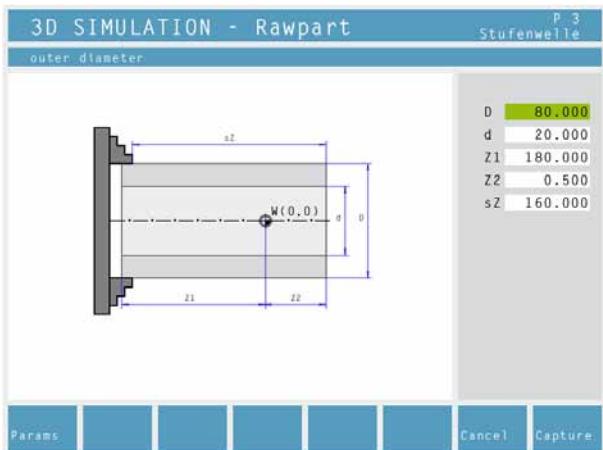
Collision checking monitors the following situations:

- Contacts between tool and clamping device. If clamping device representation is switched off, clamping device collisions will not be monitored.
- Contacts between non-cutting tool parts with the workpiece or the clamping device.

If a collision occurs, the collision type will be shown and the simulation will be broken off.

Settings

Rawpart



Raw part definition

- Press Softkey.
- Press Softkey. You can enter the following values:

D

External diameter.

d

Internal diameter (if available).

Z1

Measurement from workpiece zero point (W) to the rear facing side of the workpiece.

Z2

Measurement from workpiece zero point (W) to the forward facing side of the workpiece.

sZ

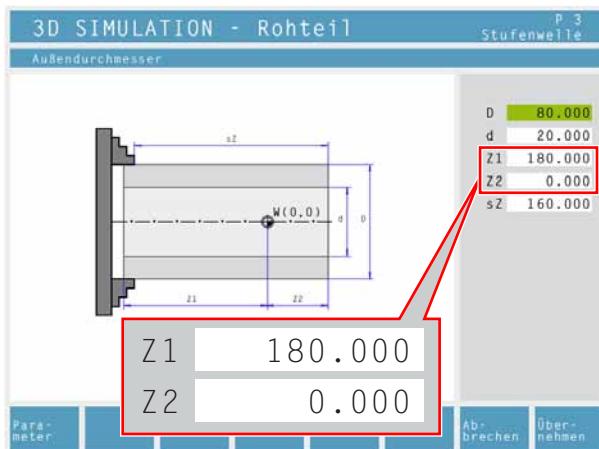
Measurement from the front side of the collet to the forward facing side of the workpiece (jut-out length of the raw part from the clamping device).

Note:

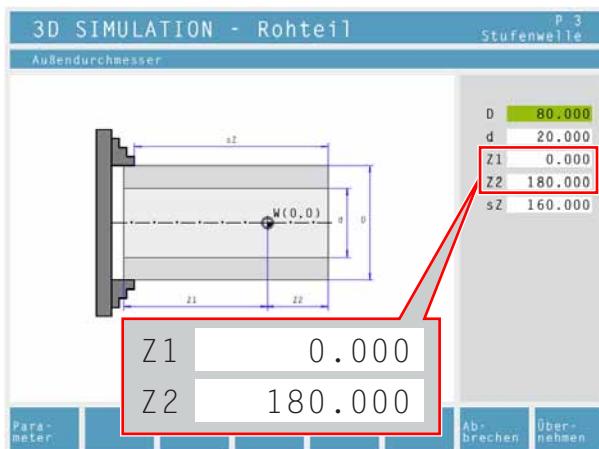
All values are relative to the workpiece zero point (W).

Note:

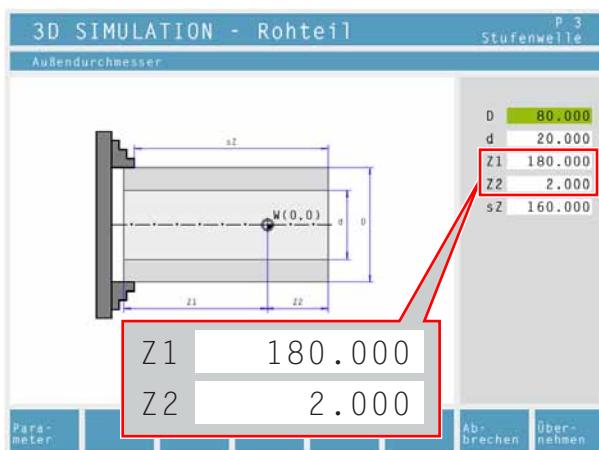
Active zero point shifts (see Chapter D Programming "Coordinate Transformation") will not be taken into account by the program. If there are active zero point shifts, therefore, the values for the raw part definition must be adjusted.

Screen input

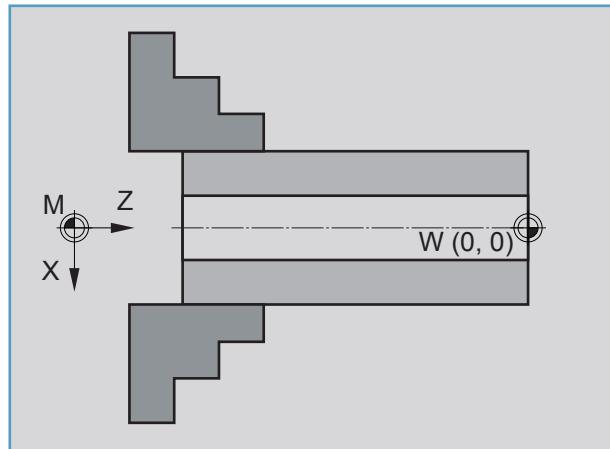
The raw part measurement rightwards from the workpiece zero point W is zero.



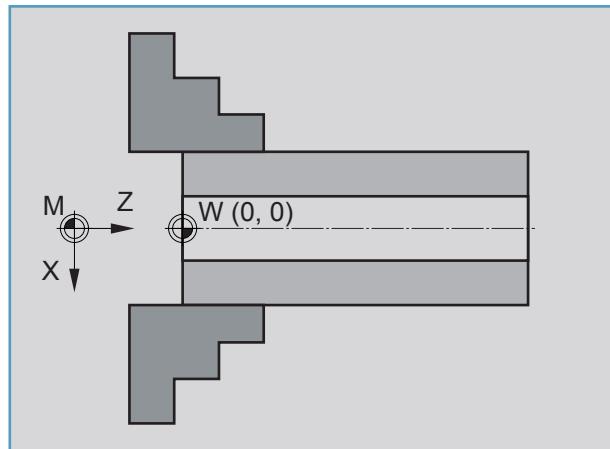
The raw part measurement leftwards from the workpiece zero point W is zero.



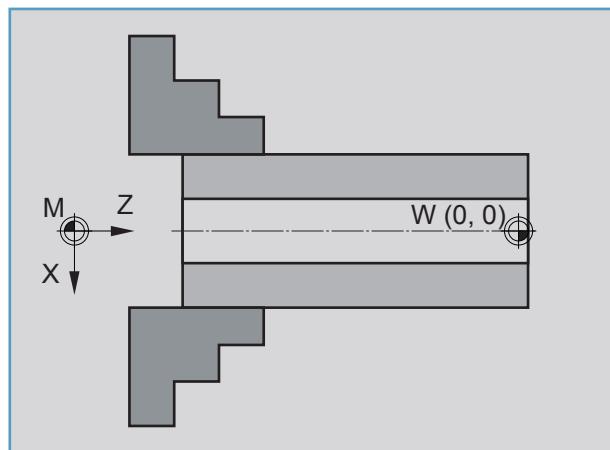
The raw part measurement rightwards from the workpiece zero point W is 2 mm.

Examples for raw parts definition**Actual clamping situation and representation**

The workpiece zero point W lies on the right face of the raw part.



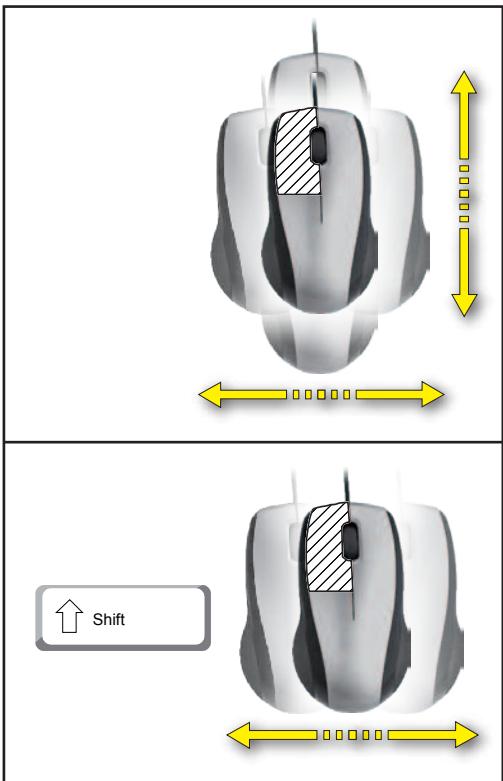
The workpiece zero point W lies on the left face of the raw part (inside the chuck).



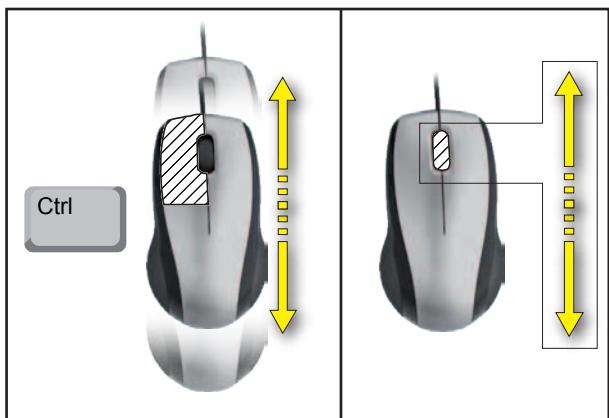
The workpiece zero point W lies 2mm inside the right face of the raw part (practical reason: for facing the front surface up to the workpiece zero point W).

Simulation

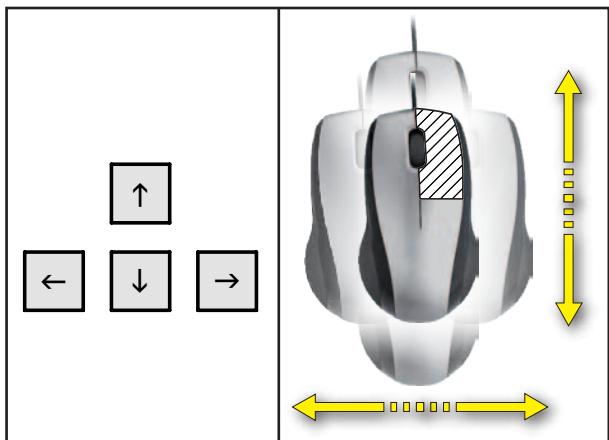
In addition to the 2D-mode functions described, the following are also possible in 3D-mode:



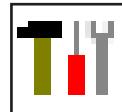
Rotate image



Zoom

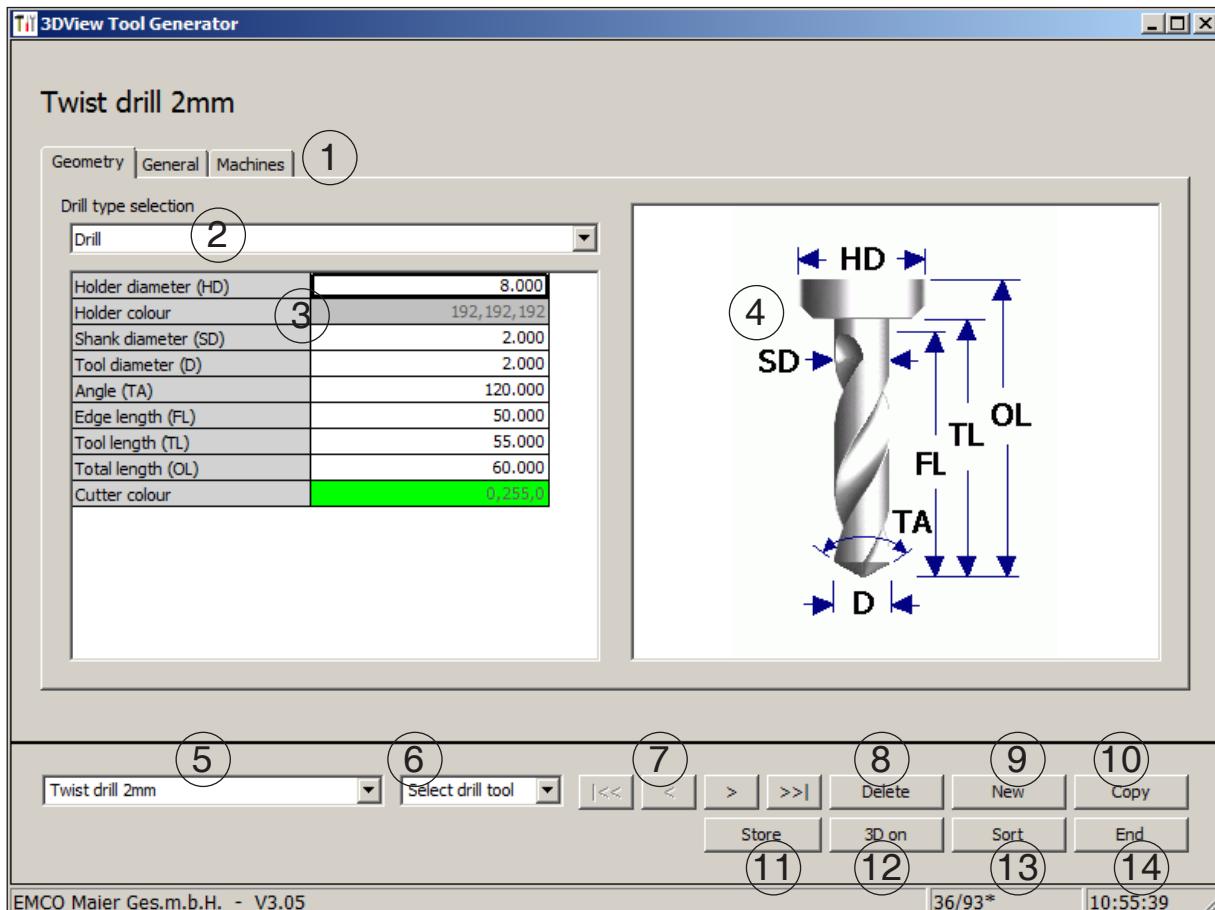


Shift



Tool modelling with the 3D-Tool Generator

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

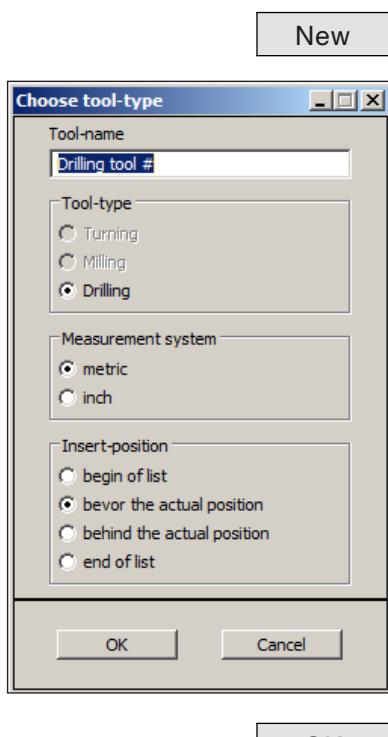


- 1 Register cards for "Geometry", "General" and "Machines" for drilling and milling tools and "Tip", "Holder", "General" and "Machines" for turning tools.
- 2 Selection of tool types
- 3 This window enables the input of tool dimensions.
- 4 Graphical support for the tool dimensioning
- 5 Choice of tools 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

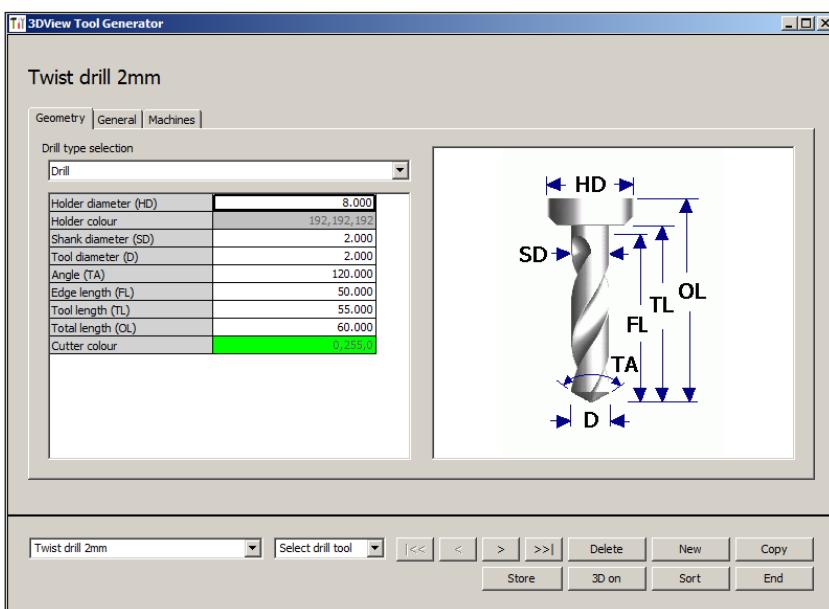
Generating a new tool

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



OK

- Confirm the entries with "OK".



Store

- Confirm the entries with "Store".

Copying a tool

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

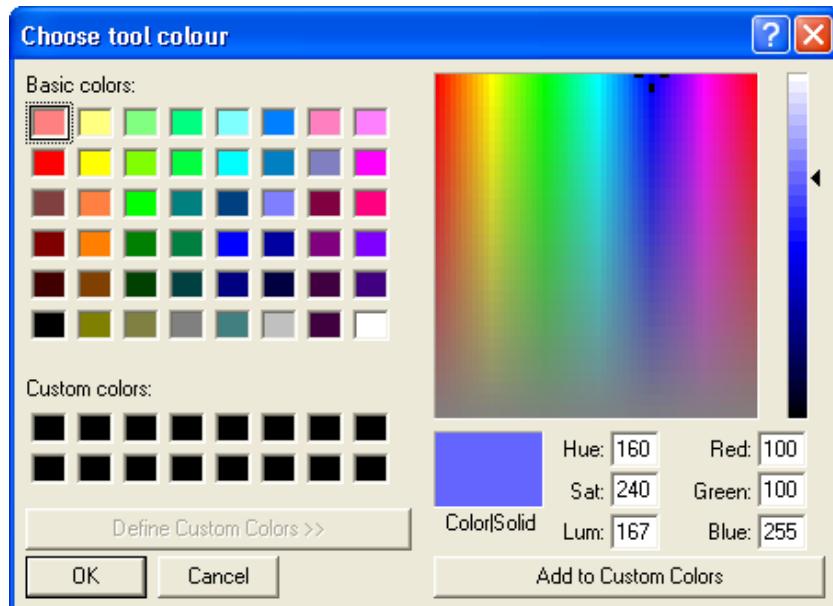
Copy

Changing an existing tool

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

Store

Selecting a tool colour



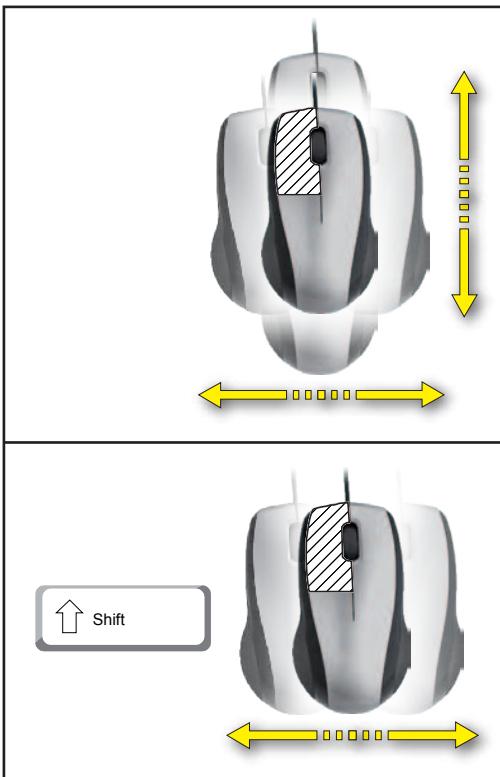
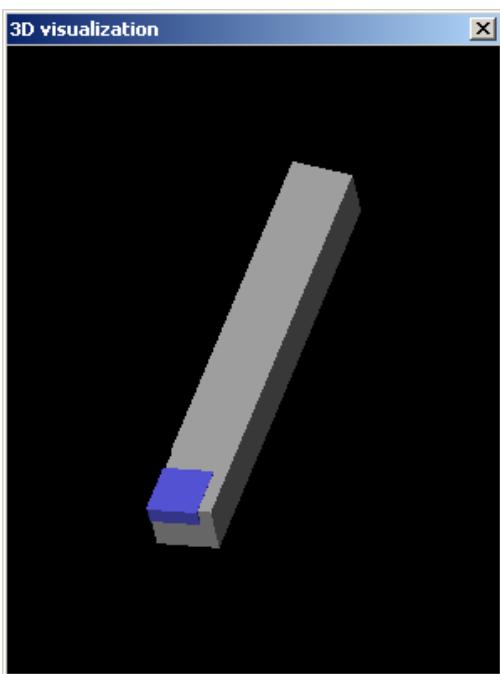
- Make a double click with the mouse pointer in the coloured box of the tool colour. The window "Select tool colour" will appear.
- Select the required colour.

- Confirm the entries with "OK".

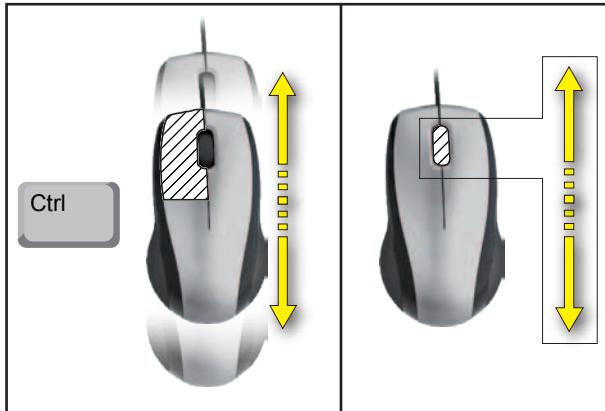
OK

Visualizing a tool

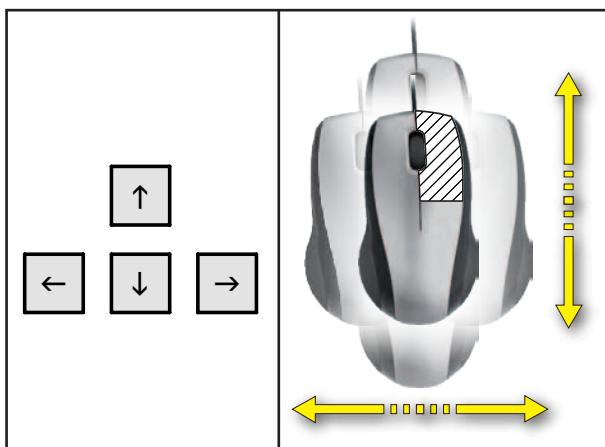
- Press the button for 3D visualization.



Rotating image



Zooming



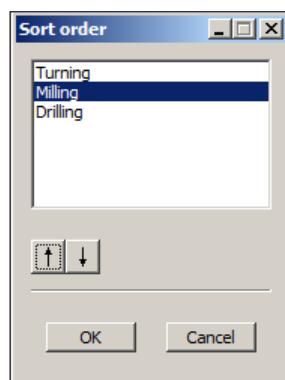
Shifting

Sorting function

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

- Press the button for sorting.

Sort



- Define new sorting sequence.

OK

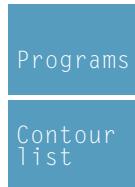
- Confirm the entries with "OK".

Contour management

When programming certain cycles, the definition of a contour to be machined is required.

Prior to this, a contour must be defined.

The contour definition serves to creating any contour with any number of points. Defined contours are saved in the contour list.



- Press the soft key to get into the program management.

- Press the soft key to switch over into the contour management.

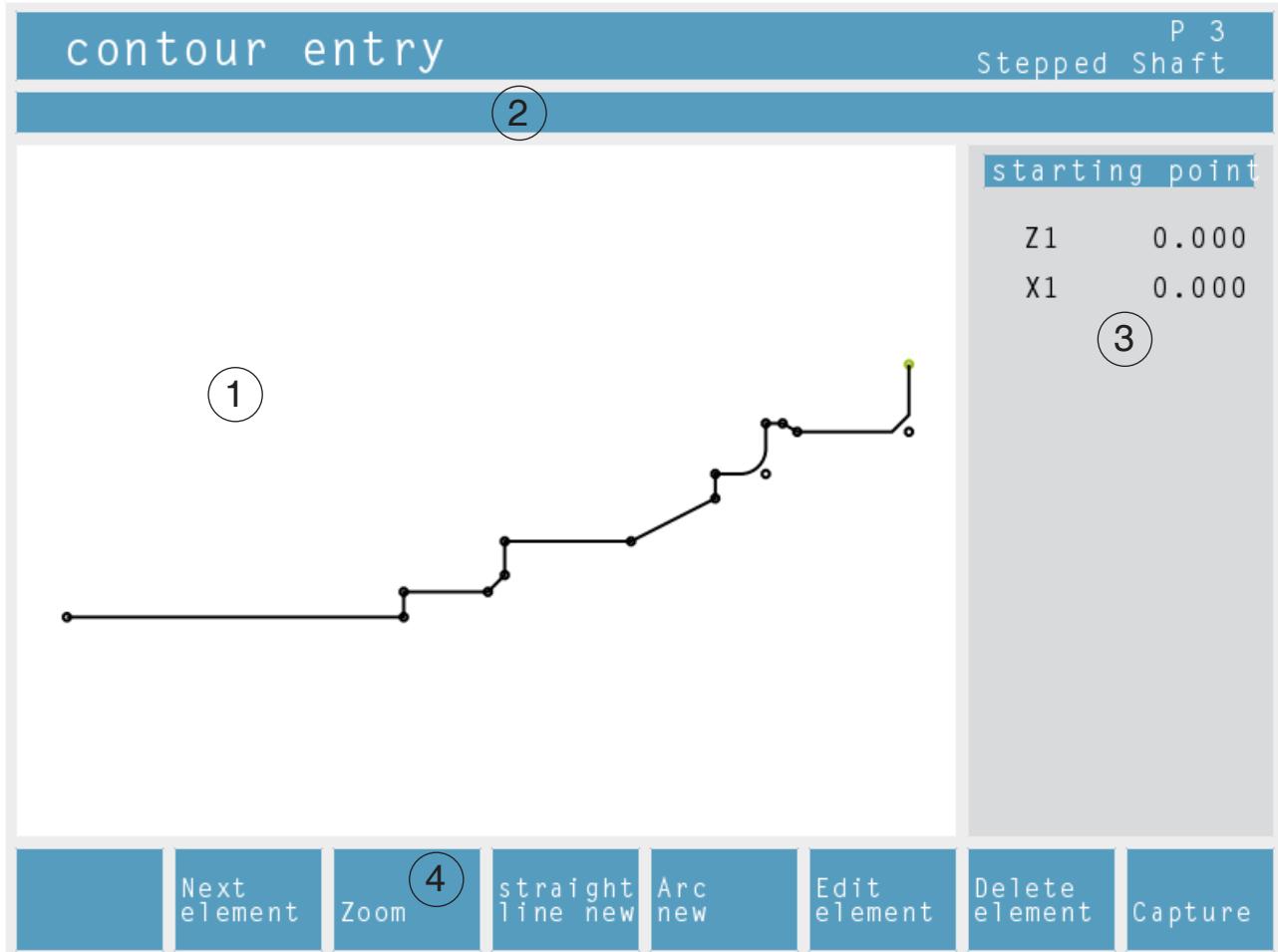


On the left side of the illustration, you see the contour programs that are saved in the EASY CYCLE control. On the right side of the illustration, the preview window displays a schematic drawing of the marked contour.

In the top right-hand corner, the program that has been selected at the machine is displayed (here: "P3 - Stepped Shaft").

If there are more contours available than the ones being displayed by the window, use the cursor keys to move through the list or press the soft key "Go to".

Screen layout



1 Graphic display of the profile that is currently created.

2 Alarm message line

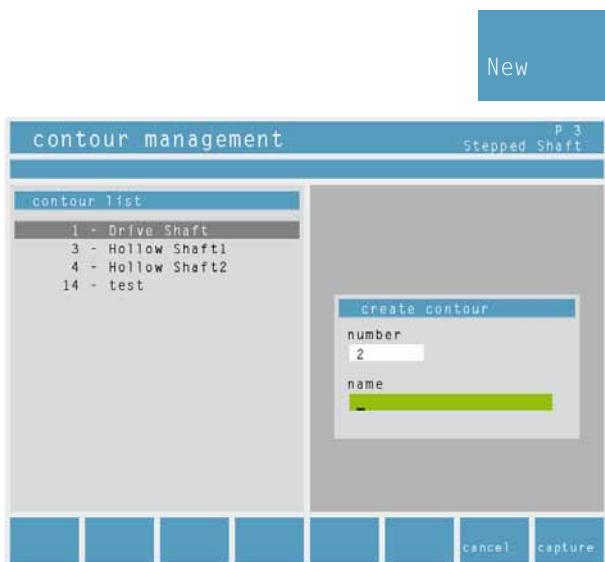
3 Window for creating or editing the selected profile section.

4 Soft-key row

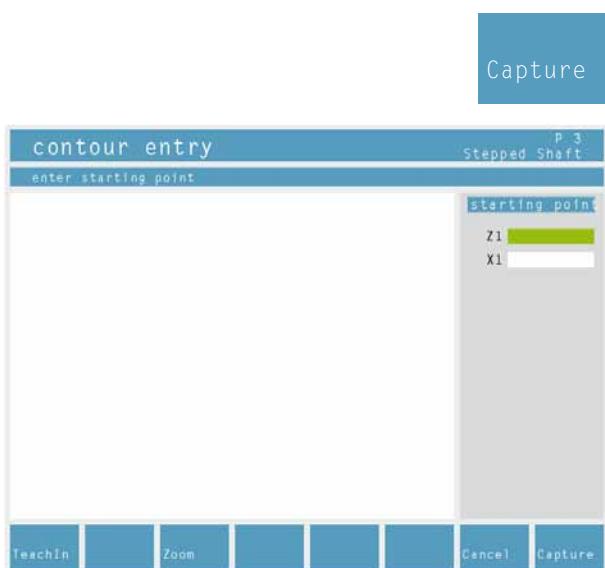
ESC

The contour is exited. If something has changed, you will be queried "Save Yes / No?"

Define / create a contour



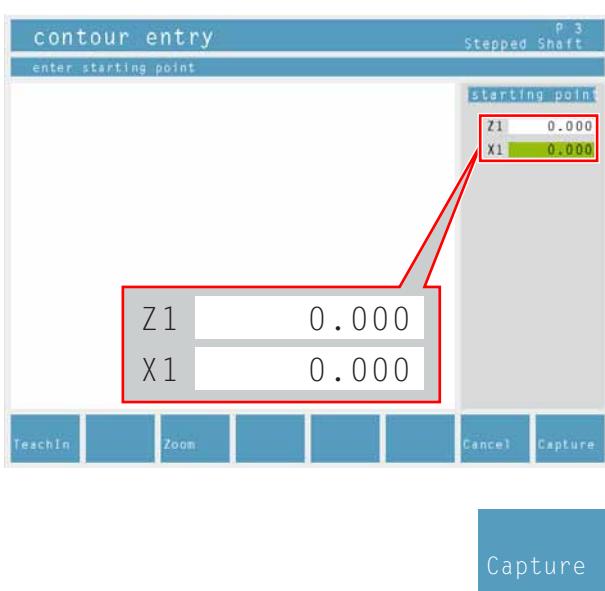
- Press the soft key.



- Enter the contour number and the contour name and confirm with the soft key.

Creating the element starting point

After having pressed "Capture", you will automatically get to contour entry. At the beginning of a contour definition, it is necessary to enter the coordinates for the starting point.



Example:

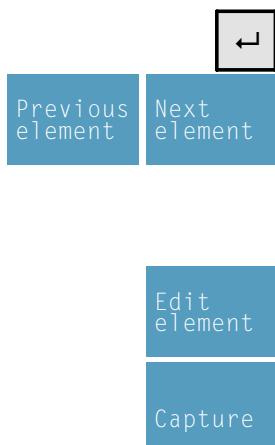
- Z1 = 0
- X1 = 0

- Confirm the definition of the starting point with the soft key.

Editing an element

For alteration of value inputs of an element definition.

- Move the highlight on the required contour.
- Press the "ENTER" key.
- Select the element in the contour. These soft keys serve to forward or backward movement in the contour, element by element. The current position in the contour is marked with green colour.
- Edit the element. Carry out the changes in the contour.
- Confirm the alterations with the soft key.

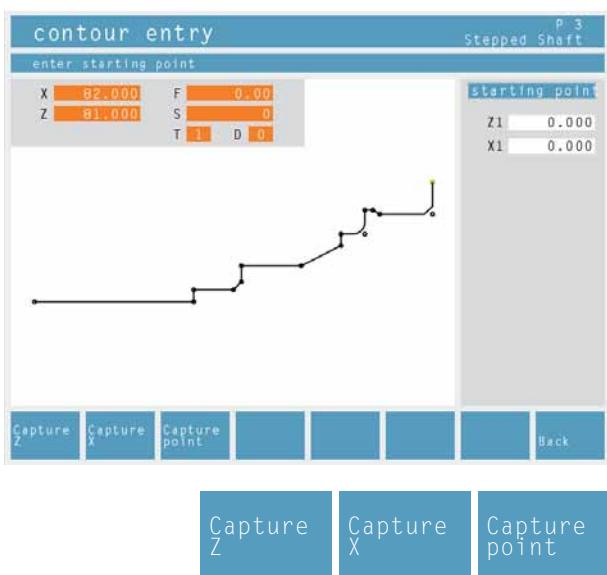


See chapter C Operation "Graphic illustration".

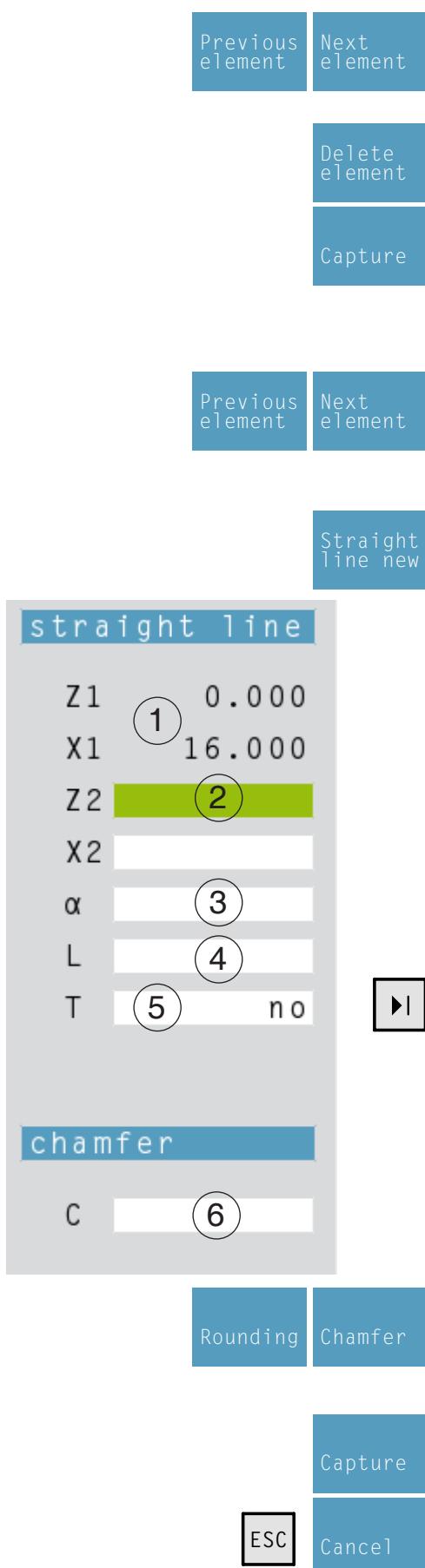


TeachIn

TeachIn enables you to directly capture the current tool position in the contour definition.



These soft keys serve to directly capture the respective axis positions.



Deleting an element

- By pressing the soft keys, move the highlight to the element you wish to delete. The element will be displayed in green colour.
- Press the soft key.
- Confirm the alterations with the soft key.

Creating the element straight line

- By pressing the soft keys, move the highlight to the element where the new straight line shall be inserted. The element will be displayed in green colour.

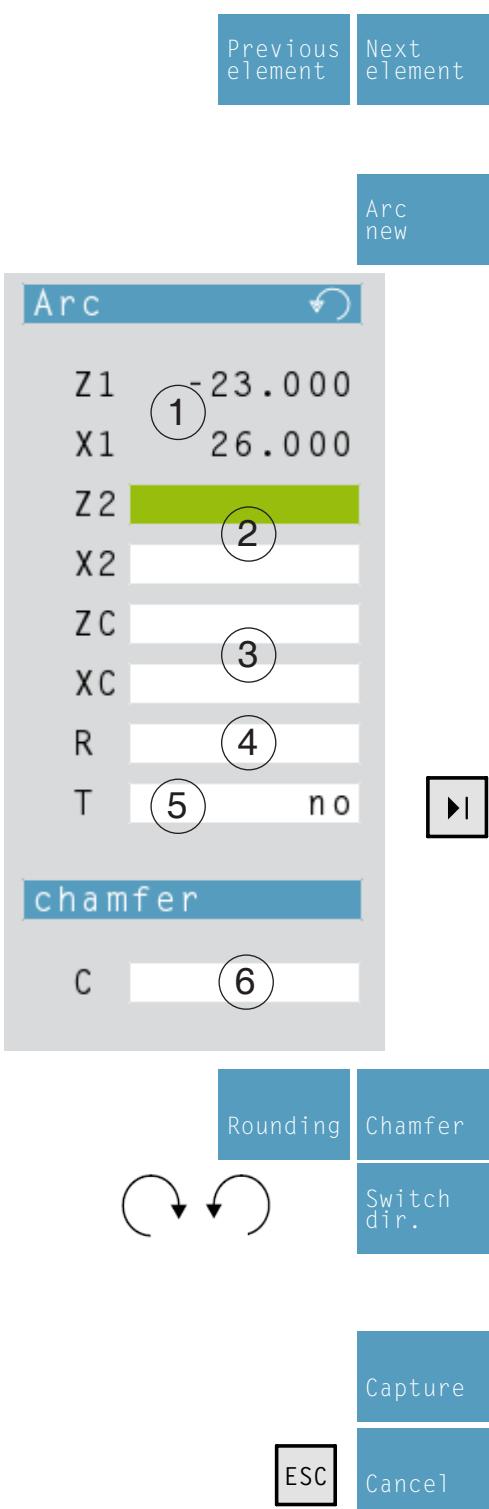
- Press the soft key.

- 1 Coordinate values of the starting point of the straight line.
These values cannot be changed because they correspond to the last point of the previous element.
- 2 Coordinate values of the end point of the straight line.
- 3 Angle between the straight line and the absciss.
- 4 Length of the straight line.
- 5 Indicates, whether the straight line to be drawn runs tangentially to the previous section. Use the selection key to make your choice.

- 6 Enter the radius for the rounding or the chamfer. Switchable by soft key.

- Confirm the alterations with the soft key.

- Alterations that have been carried out, will be cancelled.



Creating the arc element

- By pressing the soft keys, move the highlight to the element where the new arc shall be inserted.
The element will be displayed in green colour.
- Press the soft key.

1 Coordinate values of the starting point of the arc element.
These values cannot be changed because they correspond to the last point of the previous element.

2 Coordinate value of the end point of the arc element.

3 Coordinate value of the center of the arc element.

4 Radius of the arc element.

5 Indicates, whether the circular arc to be drawn runs tangentially to the previous section.

6 Enter the radius for the rounding or the chamfer. Switchable by soft key.

7 Circular arc in clockwise direction or in counter-clockwise direction. Switchable by soft key.

- Confirm the alterations with the soft key.

- Alterations that have been carried out, will be cancelled.

Next
solution

Defining contours clearly

If a contour cannot be clearly defined, you have to make your selection by means of the soft key.

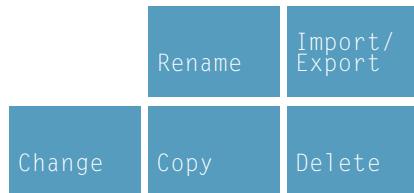
EASY CYCLE uses different colours to display profiles:

black The element is clearly defined.

green Marks the current element.

blue From several solutions, no element has been defined yet.

red The element has not been completely defined or it has been apparently defined contradictorily.



Copying, deleting, renaming, changing and importing / exporting a contour

- You may also copy, delete, rename, change, import or export a contour. The working principle of these soft keys is the same as for the soft keys in the program management.

System

In the system, you can readout important system data.

Diagnosis function PLC

With this soft key, the inputs and outputs of the PLC are displayed.

Diagnosis function USB-SPS

This soft key enables to carry out a SPS diagnosis via the USB interface.

Parameters

This Softkey shows parameters like language, measurement system (metric / anglo-american), reference point, continuous operation and backlash. They can also be changed.

If EASY CYCLE is being operated in conjunction with EMCO Concept machines, it is possible to change other parameters with the utility software EmConfig (see Chapter X EmConfig).



- With these soft keys, the parameters can be exported to an USB keychain or imported from an USB keychain into the EASY CYCLE control. Alterations due to an data import will only become active after the EASY CYCLE is started again.

Security

Use this softkey to protect parts of the controller interface (programs, cycles, tool tables and NC start) with a password.

Network

Use this softkey to configure the network settings for the machine.

Setup a FTP-connection to EASY CYCLE

- 1 Turn off the machine and open the cabinet.
- 2 Connect the patch cable to the ethernet jack on the ACC (above the USB ports).

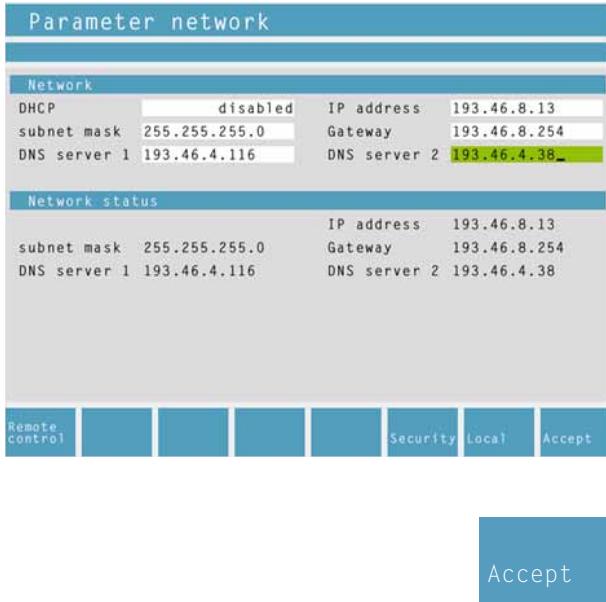


- 3 Close the cabinet and turn on the machine.

TCP/IP settings

- 1 Press the soft keys "System", "Parameter" and "Network".
- 2 If a DHCP server is running in your network, the control automatically assigns an IP address to it. The currently assigned IP address appears under "Network status".

Parameter network		
Network		
DHCP	enabled	IP address
subnet mask		Gateway
DNS server 1		DNS server 2
Network status		
subnet mask	255.255.255.0	IP address 193.46.8.13
DNS server 1	193.46.4.116	Gateway 193.46.8.254
		DNS server 2 193.46.4.38
<input type="button" value="Remote control"/> <input type="button" value="Security"/> <input type="button" value="Local"/> <input type="button" value="Accept"/>		



- To use a fixed IP address, DHCP has to be turned off and the corresponding values for IP address, subnet mask, gateway and DNS servers have to be listed.

- The changes are activated in the controller immediately by pressing the soft key "Accept".

FTP connection

- The connection to EASY CYCLE can be established with all popular FTP-clients (eg. FileZilla):

Username: ecuser

Password: emco

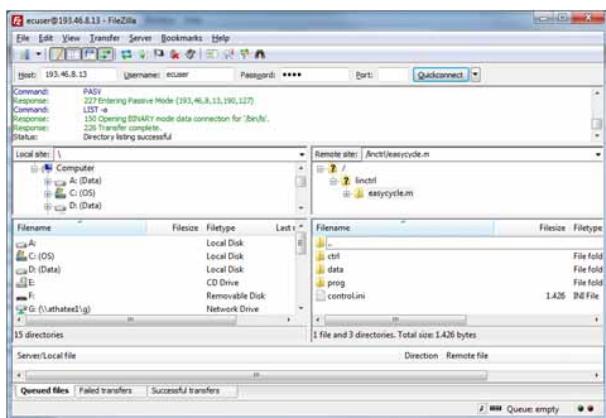
- Once the connection is established, you are in the directory / lctrl / easycycle.m (milling machine) respectively / lctrl / easycycle.t (lathe).

- In the subdirectory "data" you will find both files with the tool data: toolmill.tab and toolmill.tech (milling machine) or toolturn.tab and toolturn.tech (lathe).

- In the subdirectory "prog" you will find all programs, contours and hole patterns. To be recognized by the control system, all files have to have a name with exactly six digits and the file type pim (milling machine) respectively pit (lathe).

The program numbers are divided into the following areas:

- 000001-994999: NC-Program
- 995001-995999: hole pattern
- 996001-996999: contours (milling machine)
- 998001-998999: contours (lathe)



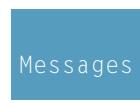
Note:

When transferring data via FTP, make sure that the machine side of the "position indicator" (main page) is displayed and no NC program is executed, only in this way can be guaranteed that only current data will be used.



Alarms

With this soft key, active alarms of the EASY CYCLE control are displayed.



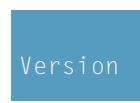
Messages

With this soft key, active messages of the EASY CYCLE control are displayed.



Date on/off

With this soft key the date can be turned on and off.



Version numbers

With this soft key, the version number of the installed software products are displayed.

Version numbers		P. 5 antriebswelle
EasyCycle :	1.14	
Axis controller :	10.16	
Converter :	1.22	
Cycle library :	2.22	
Geometry processor :	1.16	
3DView :	14.36	



Switching EASY CYCLE off

With this soft key, EASY CYCLE is shutdown.



Restarting EASY CYCLE

With this soft key, EASY CYCLE is restarted.

Reference

This soft key enables you to set a reference point.

Refer-
ence

Referencing the X/Z axis

With this soft key, the X/Z axis is referenced.

Ref.
X-axis

Ref.
Z-axis

Referencing all

With this soft key, the X and Z axis are referenced.

Ref.
all

Reference position / path to go

With this soft key, you can either have the reference position without offset (tool, datum shift, ...) or the path to go from the current tool position to the target position displayed.

Ref Pos
distance

Block scan

Press the soft key Programs to enter the program management and select a program.

With block scan, cycles can be skipped during program run.

Skipping cycles in the block scan

- Move the highlight to the required program.
- Press the soft key.
- Press the "ESC" key.
- Press the soft key.



- Use the cursor keys to select the cycle from which the NC program shall continue.

- Press the soft key. Wait until the EASY CYCLE control has calculated the remaining program. Press "NC-Start" only after EASY CYCLE has prompted you to do so in the status display. The skipped cycles will not be carried out.

D: Programming

Cycles

Overview Cycles

The cycle groups including the defined cycles of the EASY CYCLE are listed below.



Longitudinal turning

- Turning 1
- Turning 2



Facing

- Facing 1
- Facing 2



Thread cutting

- Thread cutting 1
- Thread cutting 2
- Thread cutting 3
- Multiple gear thread



Grooving

- Grooving 1
- Grooving 2
- Grooving 3
- Grooving 4
- Cut-off



Contour turning

- Contour turning



Bore

- Drilling
- Tapping



Positioning

- Positioning

Taper
turning

Taper turning

- Taper 1
- Taper 2
- Taper 3

Radius
turning

Radius turning

- Radius turning

Iso
edit

ISO Edit

- Input of the DIN/ISO code

Coord.
trans.

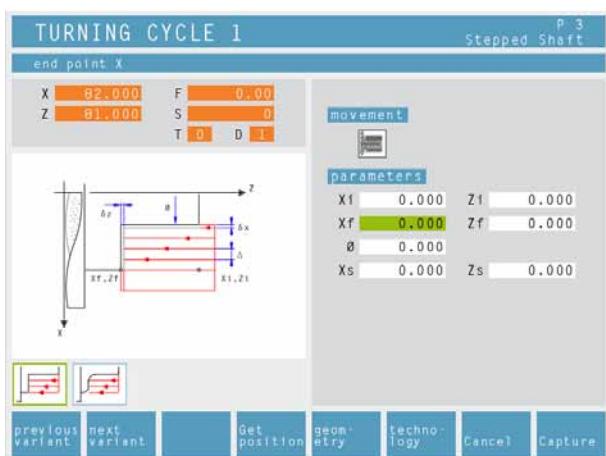
Coordinate transformation

- Absolute zero offset
- Incremental zero offset
- Cancel zero offset

Call
subprog.

Call subprogram

- Select a subprogram



Defining a cycle

The soft-key row displays the different cycle groups.

- Select the cycle group.
- Select the cycle.
- Enter all required parameters.

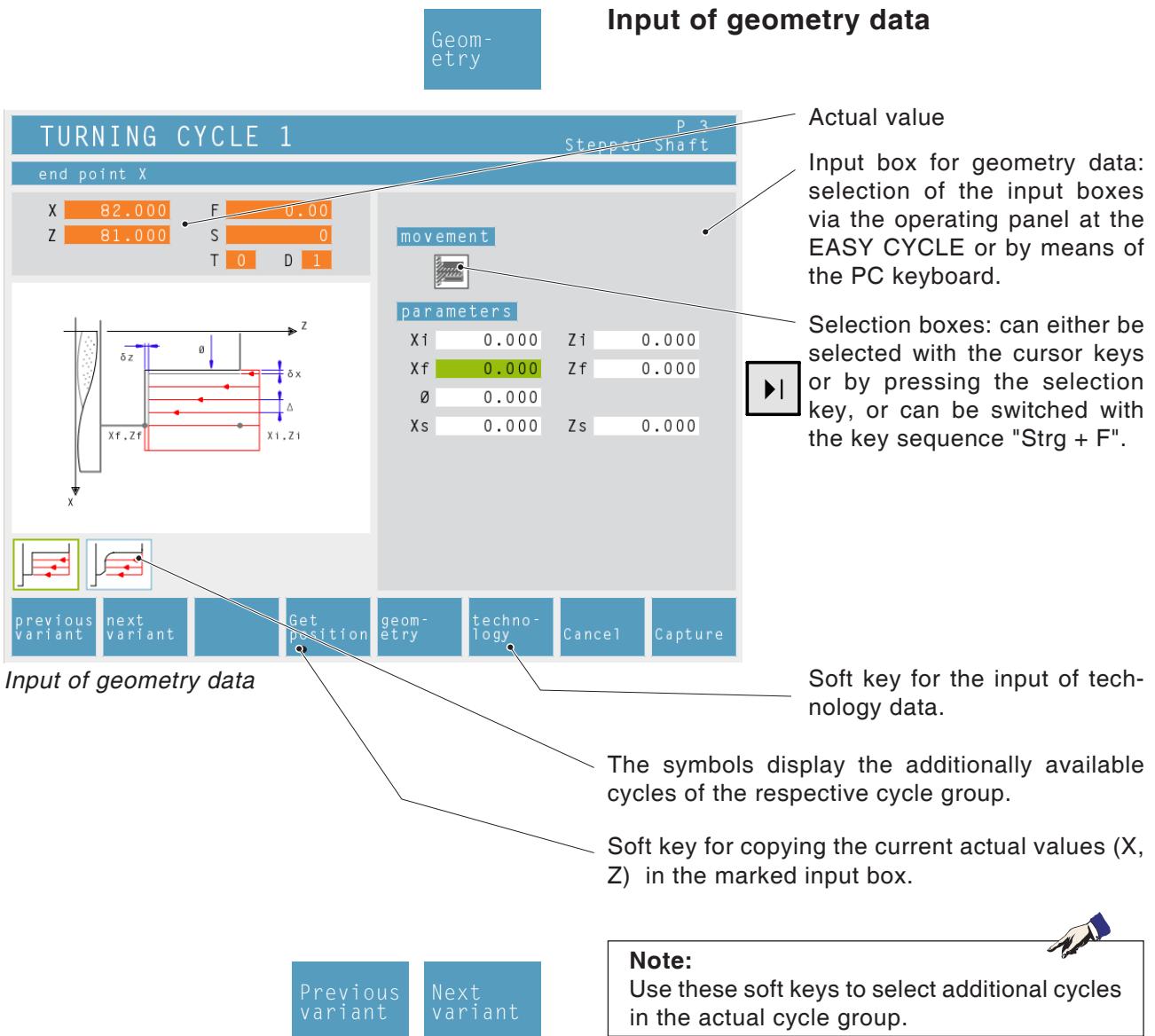
- Terminate the input with this soft key.

There are two possibilities to define a cycle:

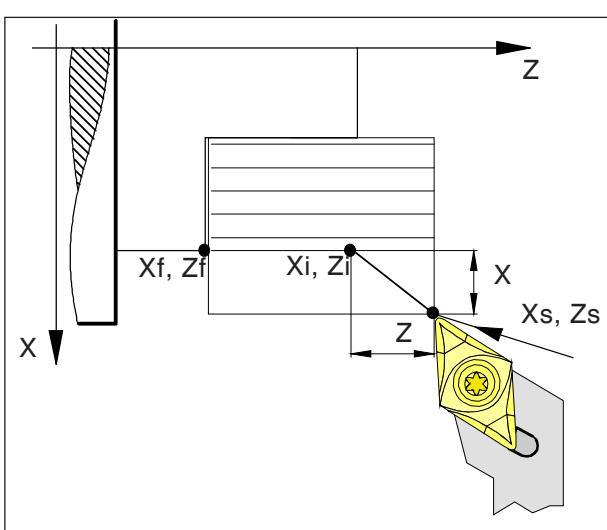
- In the program management: After having been defined, the cycle can be saved.
- In the cycle menu: After having been defined, the cycle is ready to be run at the machine.

Note:

The cycles „Coordinate transformation“ and „Call subprogram“ can only be defined in the program management in the cycle list (see chapter C, operation „Creating a program“).

**Note:**

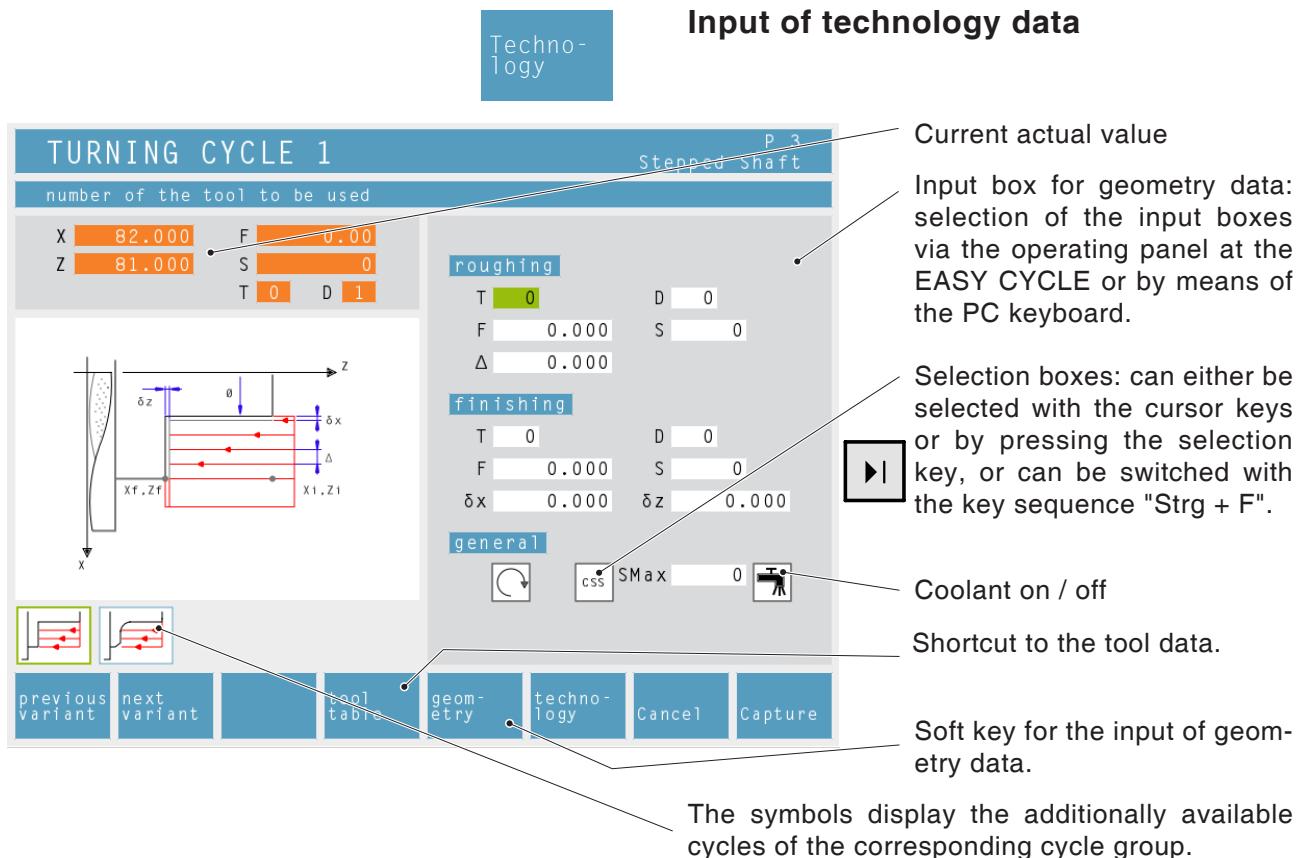
Use these soft keys to select additional cycles in the actual cycle group.

**Safety distance**

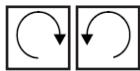
In order to avoid collisions with the workpiece during cycles, you can determine an approach point that will be traversed prior to the cycle starting point.

The safety distance Xs, Zs defines the position of this approach point in relation to the cycle starting point.

The safety distance Xs is always programmed as radius value. Safety distance Xs, Zs

**Spindle speed [rpm]**

The spindle speed is entered under the parameter S. You can program different spindle speeds for the roughing and the finishing pass.

Direction of rotation of spindle
clockwise / counter-clockwise**Constant spindle speed [rpm] / cutting speed [m/min]**

Set the constant spindle speed with RPM, the constant cutting speed with CSS and the maximum spindle speed with Smax.

**Feed rate [mm/rpm]**

The feed rate is entered under the parameter F. You can program different feed rates for the roughing and the finishing pass.

Tool

Enter the respective tool and the tool correction data under T and D (for every tool, several correction values are possible).

In working-off cycles (e.g. longitudinal turning), you can program different tools for the roughing and finishing pass (see chapter E, tool measurement). Use the soft key "Tool table" to read or edit the tool data. A tool can be accepted in the cycle automatically. When changing the tool, the feed and speed (rpm) are taken from the tool table into the corresponding fields.

Complete machining

Define a tool for roughing and finishing. Both cycles will be carried out one after the other with the respective settings and tools.

You can select different feed rates, spindle speeds and tools for both roughing and finishing.

For already defined tools, after being selected, the corresponding spindle speeds and feed rates will be entered as changeable suggestion.

When different tools have been defined for roughing and finishing, the toolholder will automatically traverse a tool changing point.

Roughing

Select the tool T0 as roughing tool. Then the rough cycle will not be carried out.

A defined finishing clearance will be considered during roughing.

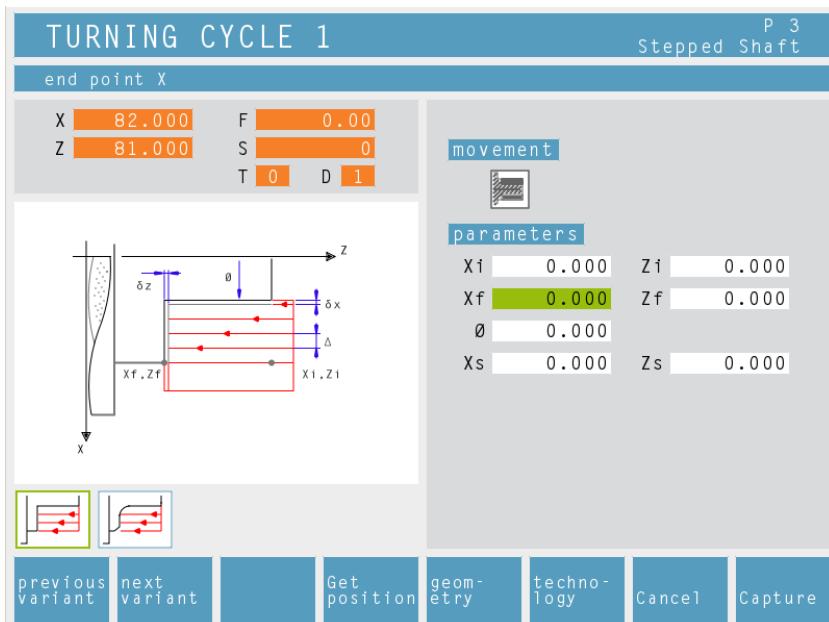
Finishing

Select the tool T0 as finishing tool. Then the finishing cycle will not be carried out.

Coolant

turning the coolant on / off





Turning Cycle 1

Previous
variant

Next
variant

Selection of the cycle from the cycle group by means of the soft keys "Previous variant" and "Next variant".

Moving direction:

outside turning
inside turning



Coordinates (X_i, Z_i)

Coordinates of the starting point

Coordinates (X_f, Z_f)

Coordinates of the end point

Remaining diameter (Ø)

Final turning diameter

Safety distance (X_s, Z_s)

Techno-
logy

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

Cutting depth for roughing (Δ)

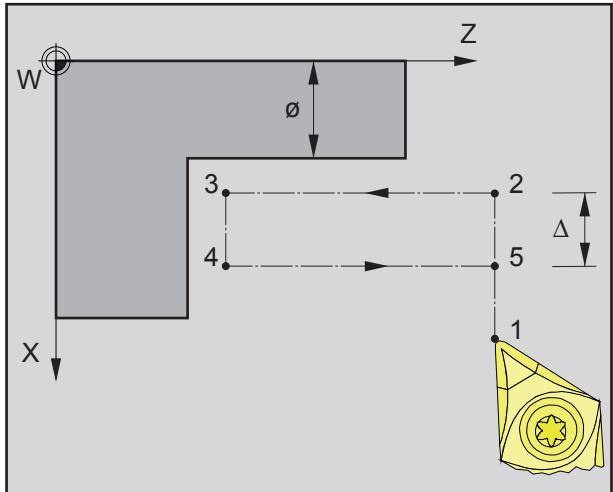
Lateral distance by which the tool is shifted during roughing.

Finishing offset in the X axis (δx)

Offset that should remain during roughing.

Finishing offset in the Z axis (δz)

Offset that should remain during roughing.



Sequence of an individual turning operation during roughing

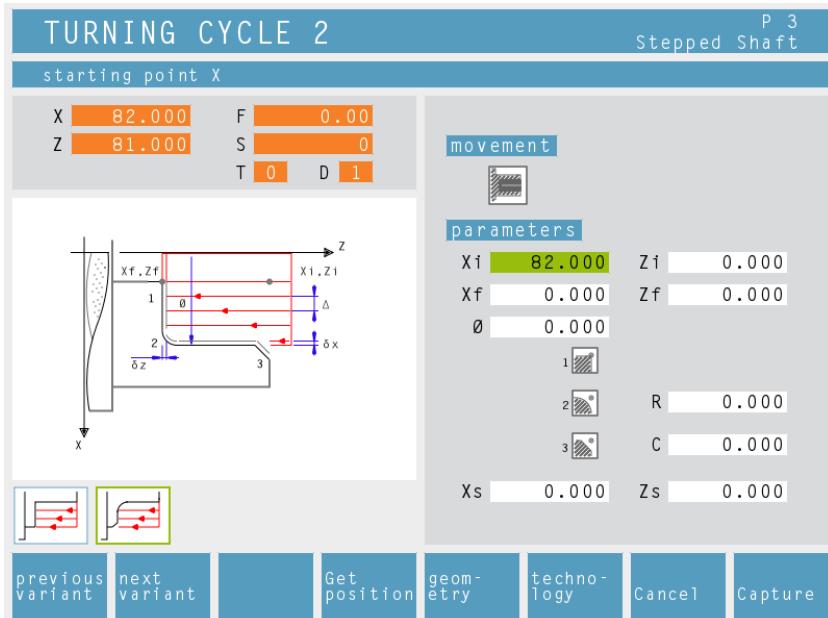
Cycle Description

Roughing

- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several turning operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ_x, δ_z) are reached.

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual turning operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

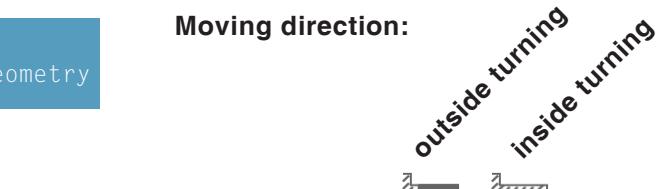


Turning Cycle 2

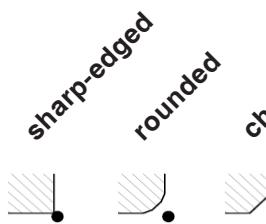
Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.



Corner rounding:



Geometry

Moving direction:

Coordinates (Xi, Zi)
Coordinates of the starting point

Coordinates (Xf, Zf)
Coordinates of the end point

Remaining diameter (\emptyset)
Final turning diameter

Safety distance (Xs, Zs)

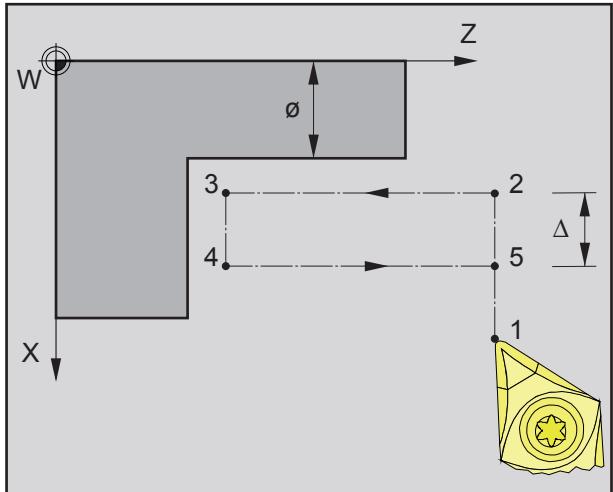
Techno-
logy

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

Cutting depth for roughing (Δ)
Lateral distance by which the tool is shifted during roughing.

Finishing offset in the X axis (δx)
Offset that should remain during roughing.

Finishing offset in the Z axis (δz)
Offset that should remain during roughing.



Sequence of an individual turning operation during roughing

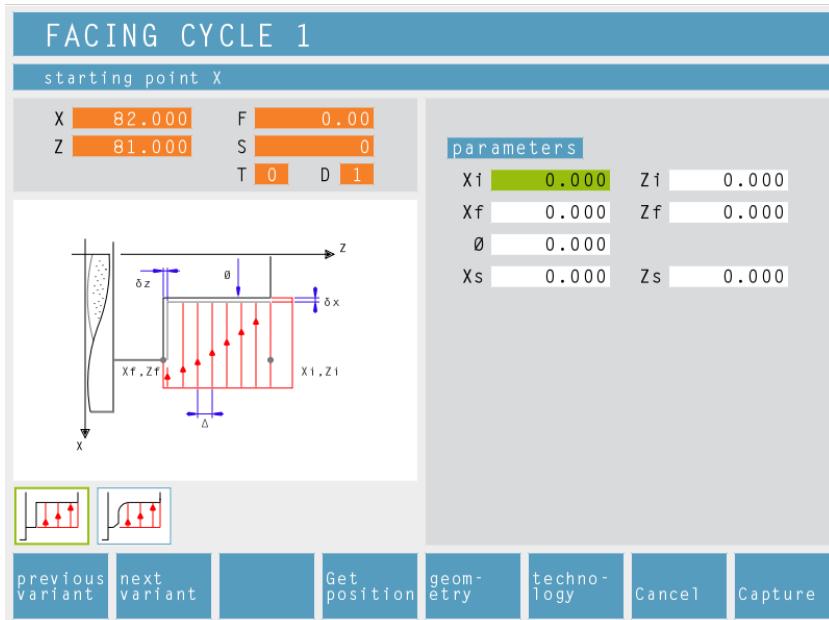
Cycle Description

Roughing

- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several turning operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ_x, δ_z) are reached.

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual turning operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Facing Cycle 1

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

Geom-
etry

Coordinates (X_i, Z_i)

Coordinates of the starting point

Coordinates (X_f, Z_f)

Coordinates of the end point

Diameter (\emptyset)

Turning diameter (negative values are allowed)

Safety distance (X_s, Z_s)

Techno-
logy

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

Cutting depth for roughing (Δ)

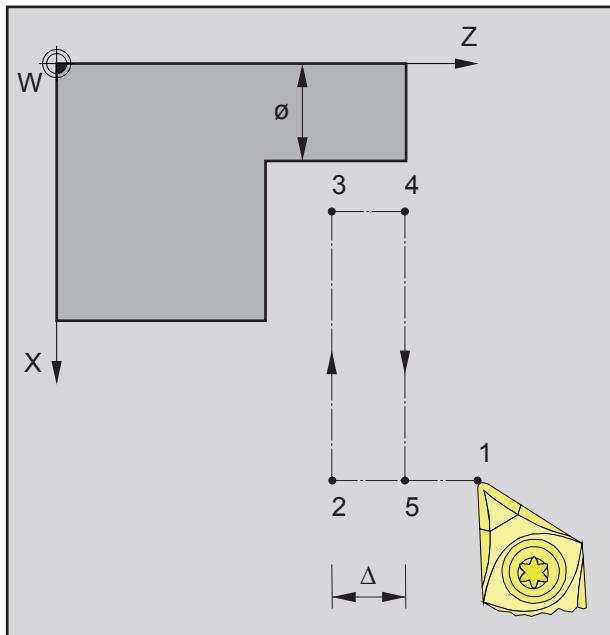
Lateral distance by which the tool is shifted during roughing.

Finishing offset in the X axis (δx)

Offset that should remain during roughing.

Finishing offset in the Z axis (δz)

Offset that should remain during roughing.



Sequence of an individual facing operation during roughing

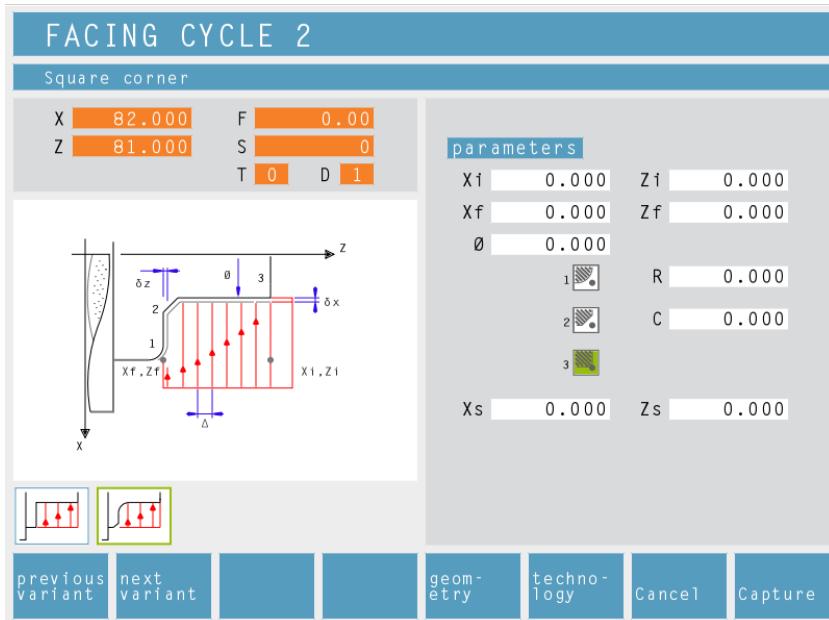
Cycle Description

Roughing

- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several facing operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ_x, δ_z) are reached.

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual facing operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Facing Cycle 2

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

Geometry

Coordinates (Xi, Zi)

Coordinates of the starting point

Coordinates (Xf, Zf)

Coordinates of the end point

Diameter (\emptyset)

Turning diameter (negative values are allowed)

Corner rounding:

sharp-edged
rounded
chamfered at 45°



Safety distance (Xs, Zs)

Technology

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

Cutting depth for roughing (Δ)

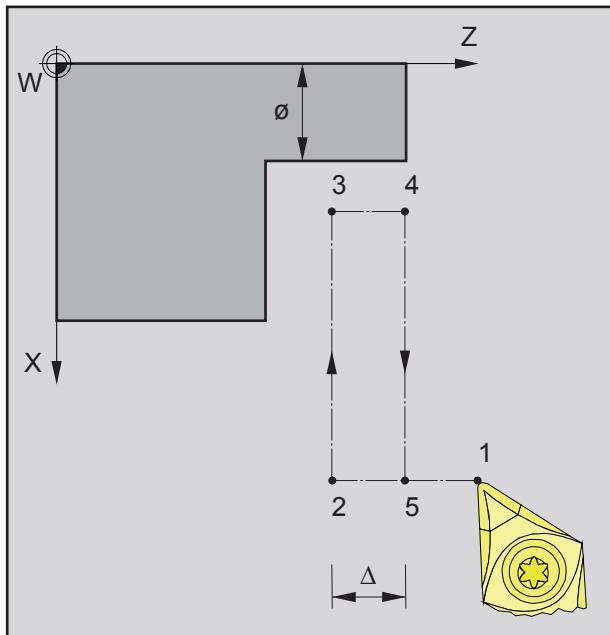
Lateral distance by which the tool is shifted during roughing.

Finishing offset in the X axis (δx)

Offset that should remain during roughing.

Finishing offset in the Z axis (δz)

Offset that should remain during roughing.



Sequence of an individual facing operation during roughing

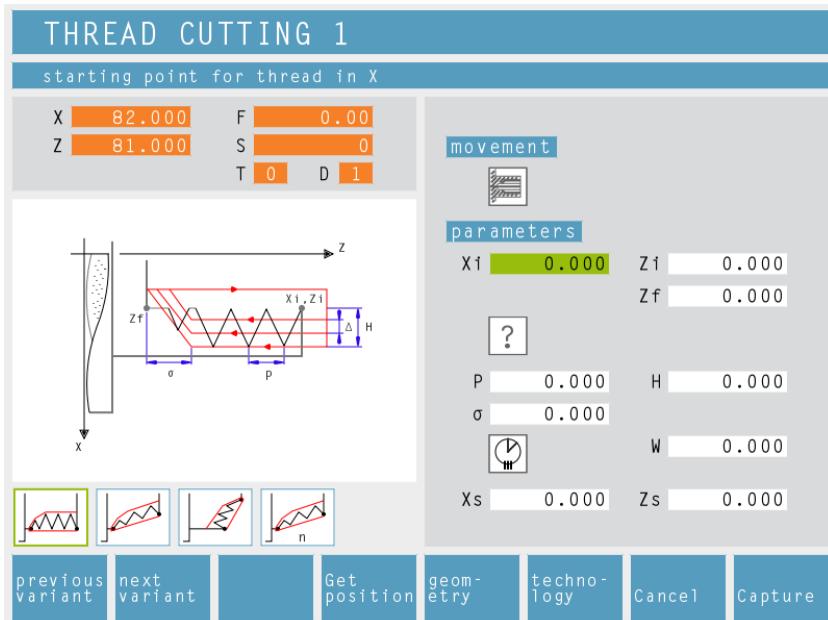
Cycle Description

Roughing

- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several facing operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ_x, δ_z) are reached.

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual facing operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Moving direction:

outside thread
inside thread



Note:

With the thread type „free pitch thread“ the thread pitch P and the thread depth H can be freely chosen.

With all other thread types the thread pitch P and the thread depth H will be automatically preset with standardized values after input of the thread diameter \varnothing .

In case a thread diameter is entered which is not standardized, EASY CYCLE will automatically set the next standardized diameter.



Thread Cutting 1

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine longitudinal threads (inside and outside).

Coordinates (X_i, Z_i)

Coordinates of the starting point of the thread

Coordinate (Z_f)

Coordinate of the end point of the thread

Thread type

You can choose among the following standardized thread types:

?

Free pitch thread

60 MM M (S.I.) Normal pitch metric thread

60 MM M (S.I.F.) Fine pitch metric thread

55 INCH B.S.W. (W) Normal pitch Withworth thread

55 INCH B.S.F. Fine pitch Withworth thread

60 INCH U.N.C. Normal pitch unified American thread

60 INCH U.N.F. Fine pitch unified American thread

Thread diameter (\varnothing)

Thread pitch (P)

Depth of thread (H)

Thread runout (σ)

Starting angle (W)

Position of the spindle

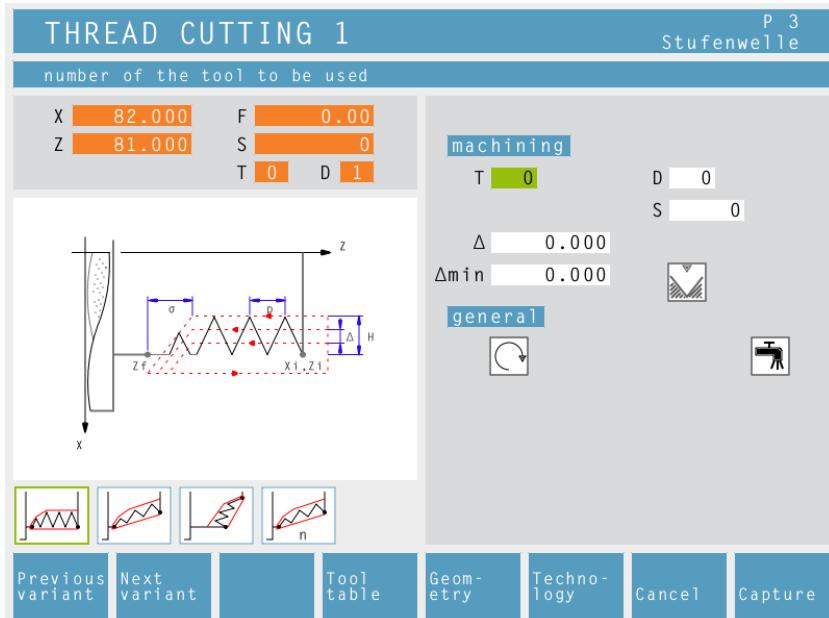
not active
active



Safety distance (X_s, Z_s)

Techno- logy

Then, please enter all necessary technological data (tool number, direction of rotation, spindle speed).



Maximum cutting depth (Δ)

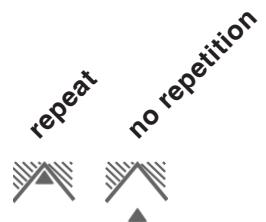
Maximum cutting depth by which the tool is shifted during roughing.

Minimum cutting depth (Δ_{min})

Minimum cutting depth by which the tool is shifted during roughing.

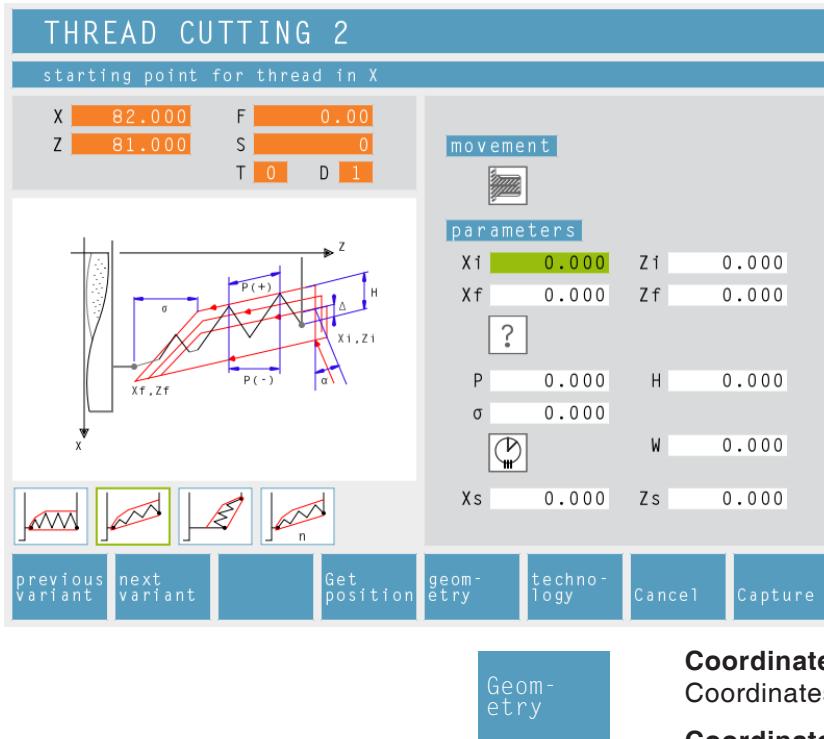
Repeat last threading pass

In order to observe tolerances or to remove a fin, it is possible to repeat the last threading pass without infeed.



Cycle Description

- 1 According to the spindle's chosen direction of rotation, a right or left thread will be turned.
- 2 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 3 The thread will be cut in several radial grooving operations until the thread depth (H) is reached. The infeed (Δ) of the individual operations depends on which operation it is (Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, ...). If a decreasing infeed is lower than the minimum infeed (Δ_{min}), at least (Δ_{min}) will be fed in.
- 4 If selected - the tool repeats the last threading operation.
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Moving direction:

outside thread
inside thread



Note:

With the thread type „free pitch thread“ the thread pitch P and the thread depth H can be freely chosen.

With all other thread types the thread pitch P and the thread depth H will be automatically preset with standardized values after input of the thread diameter \varnothing .

In case a thread diameter is entered which is not standardized, EASY CYCLE will automatically set the next standardized diameter.

Thread Cutting 2

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine tapered threads (inside and outside).

Coordinates (X_i, Z_i)

Coordinates of the starting point of the thread

Coordinates (X_f, Z_f)

Coordinates of the end point of the thread

Thread type

You can choose among the following standardized thread types:

• Free pitch thread

60 MM M (S.I.) Normal pitch metric thread

60 MM M (S.I.F.) Fine pitch metric thread

55 INCH B.S.W. (W) Normal pitch Withworth thread

55 INCH B.S.F. Fine pitch Withworth thread

60 INCH U.N.C. Normal pitch unified American thread

60 INCH U.N.F. Fine pitch unified American thread

Thread diameter (\varnothing)

Thread pitch (P)

+ pos. arithmetic sign: dimension in direction of the thread

- neg. arithmetic sign: dimension in direction of the Z axis

not active
active

Depth of thread (H)

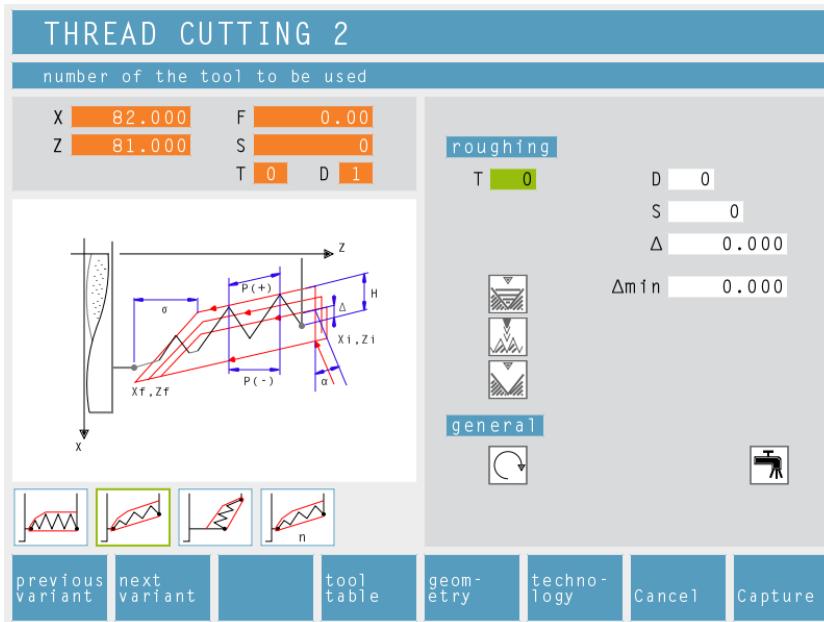
Thread runout (σ)

Starting angle (W)

Position of the spindle

Safety distance (X_s, Z_s)

Then, please enter all necessary technological data (tool number, direction of rotation, spindle speed).



Maximum cutting depth (Δ)

Maximum cutting depth by which the tool is shifted during roughing.

Minimum cutting depth (Δ_{min})

Minimum cutting depth by which the tool is shifted during roughing.

Angle of infeed (α)

Half flank angle (normally 30°)

Decreasing or constant cutting depth

constant
infeed
decreasing in-
feed



Types of infeed:

centric in-
feed infeed along the
flank infeed in zig-
zag



Repeat last threading pass

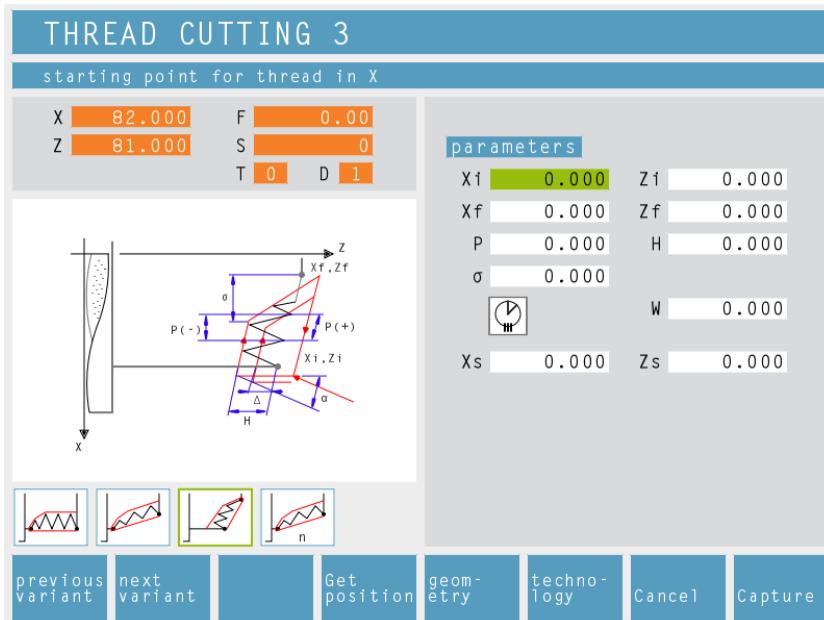
In order to observe tolerances or to remove a fin, it is possible to repeat the last threading pass without infeed.

repeat no repetition



Cycle Description

- 1 According to the spindle's chosen direction of rotation, a right or left thread will be turned.
- 2 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 3 The thread will be cut in several operations until the thread depth (H) is reached.
The icon-selected infeed is carried out:
 - at a constant infeed each time of (Δ),
 - at an infeed decreasing by (Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, ...). If a decreasing infeed is lower than the minimum infeed (Δ_{min}), at least (Δ_{min}) will be fed in.The icon-selected infeed type will be taken into account.
- 4 If selected - the tool repeats the last threading operation.
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

**Note:**

With the thread type „free pitch thread“ the thread pitch P and the thread depth H can be freely chosen.

With all other thread types the thread pitch P and the thread depth H will be automatically preset with standardized values after input of the thread diameter ϕ .

In case a thread diameter is entered which is not standardized, EASY CYCLE will automatically set the next standardized diameter.

Thread Cutting 3

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine face threads.

Coordinates (X_i, Z_i)

Coordinates of the starting point of the thread

Coordinates (X_f, Z_f)

Coordinates of the end point of the thread

Thread pitch (P)

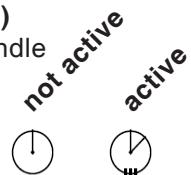
+ pos. arithmetic sign: dimension in direction of the thread
- neg. arithmetic sign: dimension in direction of the Z axis

Depth of thread (H)

Thread runout (σ)

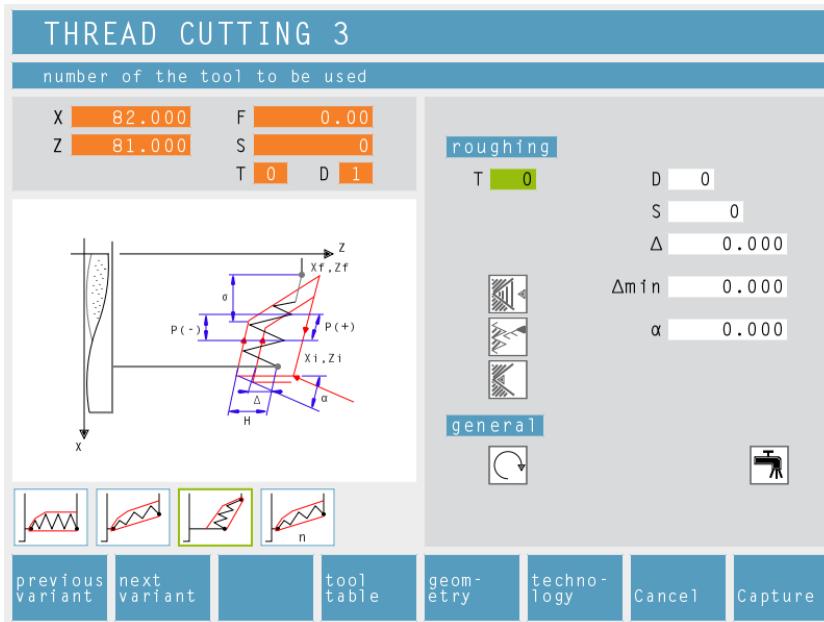
Starting angle (W)

Position of the spindle



Safety distance (X_s, Z_s)

Then, please enter all necessary technological data (tool number, direction of rotation, spindle speed).



Maximum cutting depth (Δ)

Maximum cutting depth by which the tool is shifted during roughing.

Minimum cutting depth (Δ_{min})

Minimum cutting depth by which the tool is shifted during roughing.

Angle of infeed (α)

Half flank angle (normally 30°)

Decreasing or constant cutting depth

constant
infeed
decreasing in-
feed



Types of infeed:

centric in-
feed infeed along the
flank infeed in zig-
zag



Repeat last threading pass

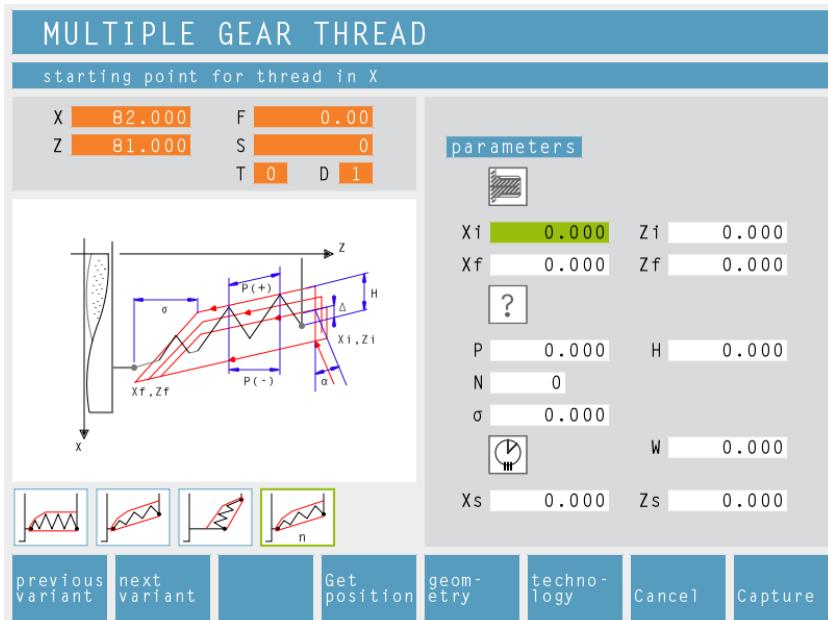
In order to observe tolerances or to remove a fin, it is possible to repeat the last threading pass without infeed.

repeat no repetition



Cycle Description

- 1 According to the spindle's chosen direction of rotation, a right or left thread will be turned.
- 2 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 3 The thread will be cut in several operations until the thread depth (H) is reached.
The icon-selected infeed is carried out:
 - at a constant infeed each time of (Δ),
 - at an infeed decreasing by (Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, ...). If a decreasing infeed is lower than the minimum infeed (Δ_{min}), at least (Δ_{min}) will be fed in.The icon-selected infeed type will be taken into account.
- 4 If selected - the tool repeats the last threading operation.
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Moving direction:

outside thread
inside thread



Note:

With the thread type „free pitch thread“ the thread pitch P and the thread depth H can be freely chosen.

With all other thread types the thread pitch P and the thread depth H will be automatically preset with standardized values after input of the thread diameter \varnothing .

In case a thread diameter is entered which is not standardized, EASY CYCLE will automatically set the next standardized diameter.

Multiple Gear Thread

Previous variant Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine outside, inside and tapered threads with any number of threads.

Coordinates (X_i, Z_i)

Coordinates of the starting point of the thread

Coordinates (X_f, Z_f)

Coordinates of the end point of the thread

Thread type

You can choose among the following standardized thread types:



Free pitch thread

60 MM M (S.I.) Normal pitch metric thread

60 MM M (S.I.F.) Fine pitch metric thread

55 INCH B.S.W. (W) Normal pitch Withworth thread

55 INCH B.S.F. Fine pitch Withworth thread

60 INCH U.N.C. Normal pitch unified American thread

60 INCH U.N.F. Fine pitch unified American thread

Thread diameter (\varnothing)

Thread pitch (P)

+ pos. arithmetic sign: dimension in direction of the thread

- neg. arithmetic sign: dimension in direction of the Z axis

Depth of thread (H)

Number of threads (N)

not active active

Thread runout (σ)

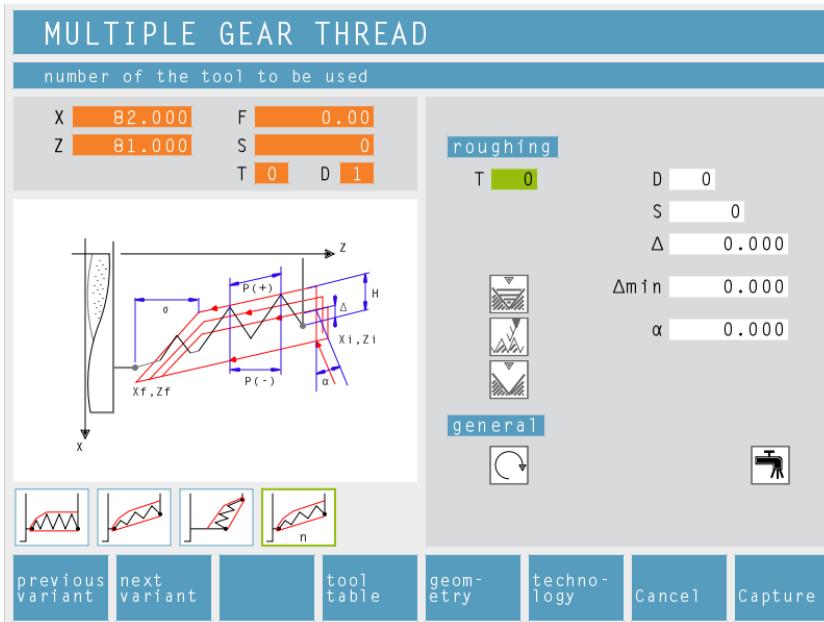


Starting angle (W)

Position of the spindle

Safety distance (X_s, Z_s)

Then, please enter all necessary technological data (tool number, direction of rotation, spindle speed).



Maximum cutting depth (Δ)

Maximum cutting depth by which the tool is shifted during roughing.

Minimum cutting depth (Δ_{min})

Minimum cutting depth by which the tool is shifted during roughing.

Angle of infeed (α)

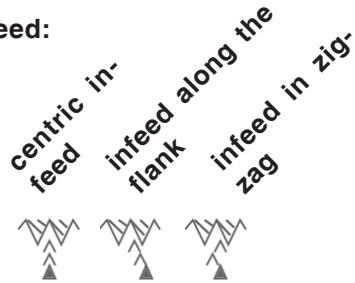
Half flank angle (normally 30°)

Decreasing or constant cutting depth

constant
infeed
decreasing in-
feed

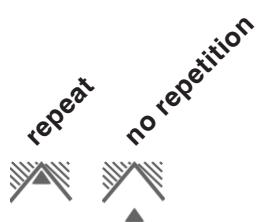


Types of infeed:



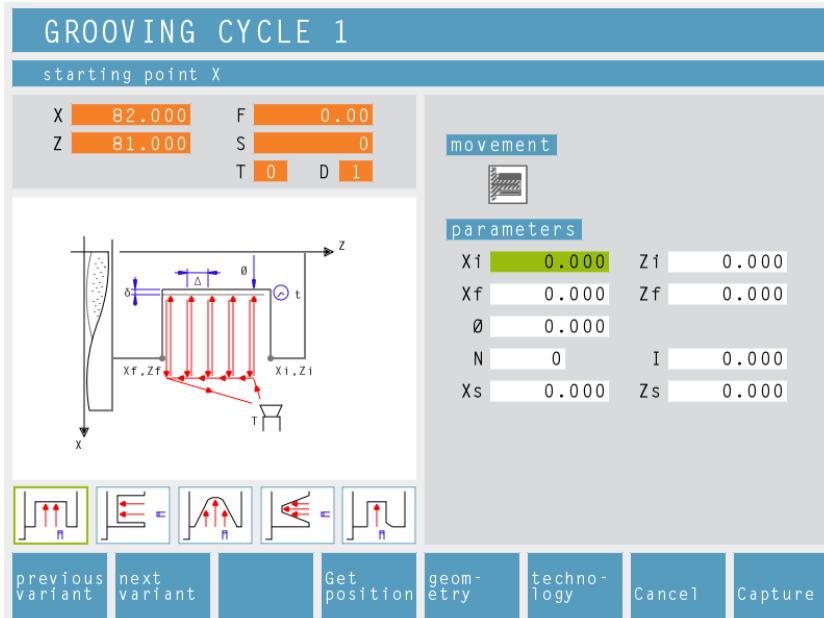
Repeat last threading pass

In order to observe tolerances or to remove a fin, it is possible to repeat the last threading pass without infeed.



Cycle Description

- 1 According to the spindle's chosen direction of rotation, a right or left thread will be turned.
- 2 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 3 The thread will be cut in several operations until the thread depth (H) is reached.
The icon-selected infeed is carried out:
 - at a constant infeed each time of (Δ),
 - at an infeed decreasing by (Δ , $\Delta\sqrt{2}$, $\Delta\sqrt{3}$, $\Delta\sqrt{4}$, ...). If a decreasing infeed is lower than the minimum infeed (Δ_{min}), at least (Δ_{min}) will be fed in.The icon-selected infeed type will be taken into account.
- 4 If selected - the tool repeats the last threading operation.
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

**Note:**

The groove has to **be the same size or** must be larger than the tool width.
It is absolutely necessary to enter the tool width (B) during tool calibration.
The cutting depth for roughing Δ should be a little bit smaller than the tool width.

**Techno-
logy**

Grooving Cycle 1

[Previous variant](#) [Next variant](#)

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine a straight, radial groove.

Moving direction:

outside turning
inside turning

**Coordinates (Xi, Zi)**

Coordinates of the starting point

Coordinates (Xf, Zf)

Coordinates of the end point

Remaining diameter (\emptyset)

Final turning diameter

Number of grooves (N)**Distance between grooves (I)**

Distance between two grooves (from $X_{i_1}Z_{i_1}$ - $X_{i_2}Z_{i_2}$).

For one groove only please enter $I=0$.

Safety distance (Xs, Zs)

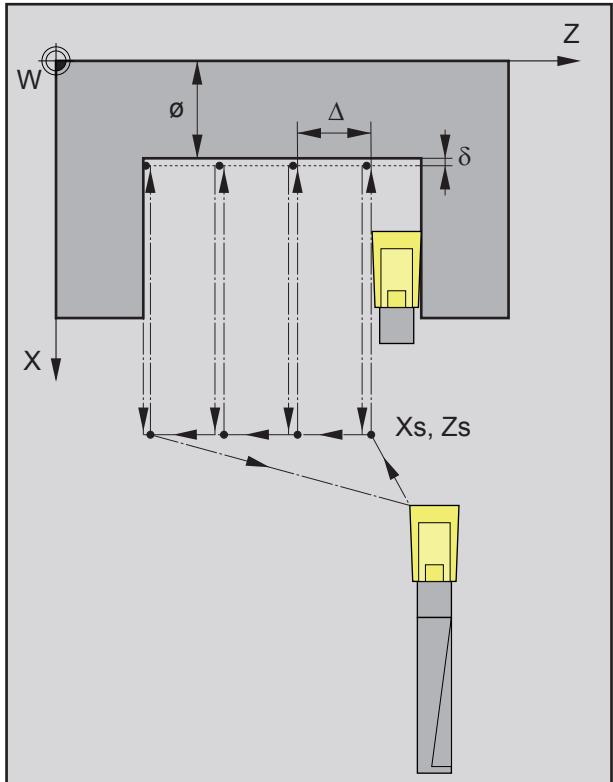
Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Dwell time (t)**Finishing offset (δ)**

Offset that should remain during roughing.

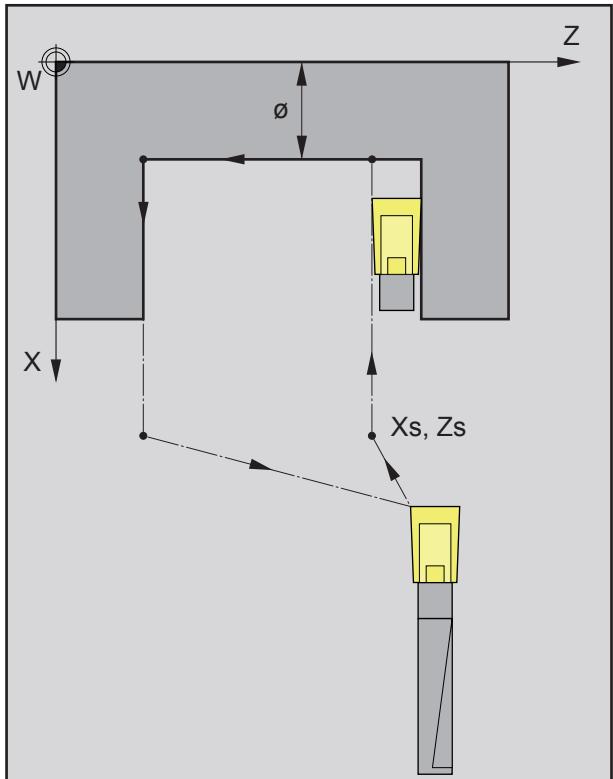


Sequence of grooving operations during roughing

Cycle Description

Roughing

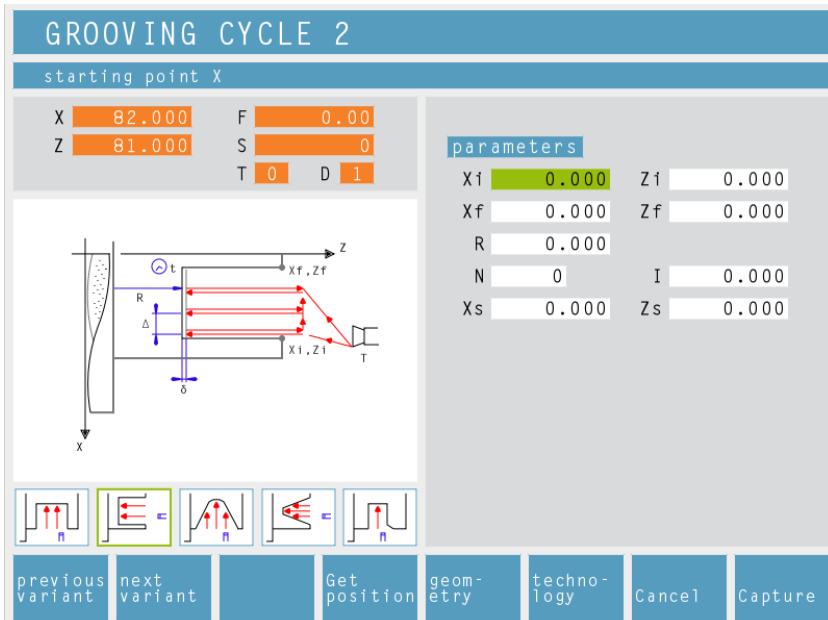
- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several turning operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowance (δ) is reached and lingers there for (t) - if entered.



Sequence of grooving operations during finishing

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual grooving operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Note:

The groove has to **be the same size or** must be larger than the tool width.
It is absolutely necessary to enter the tool width (B) during tool calibration.
The cutting depth for roughing Δ should be a little bit smaller than the tool width.

Techno-
logy

Grooving Cycle 2

Previous variant Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine a straight, axial groove.

Coordinates (X_i, Z_i)

Coordinates of the starting point

Coordinates (X_f, Z_f)

Coordinates of the end point

Depth of groove (R)

Z coordinate of the groove

Number of grooves (N)

Distance between grooves (I)

Distance between two grooves (from X_i, Z_i_1 - X_i, Z_i_2). For one groove only please enter I=0.

Safety distance (Xs, Zs)

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

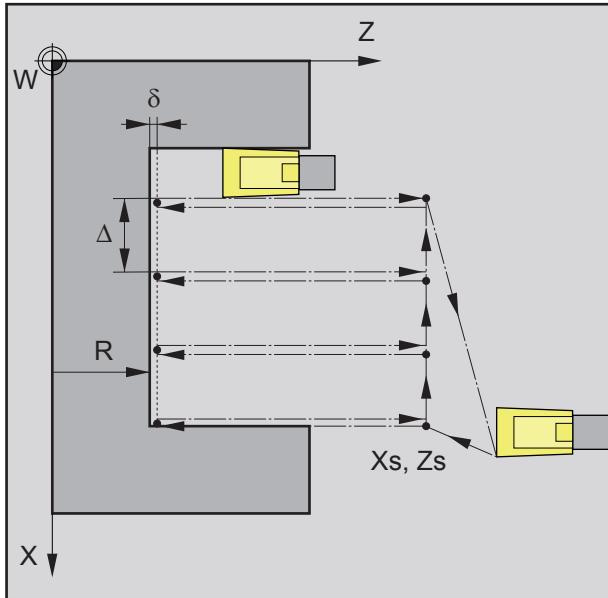
Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Dwell time (t) in seconds

Finishing offset (δ)

Offset that should remain during roughing.

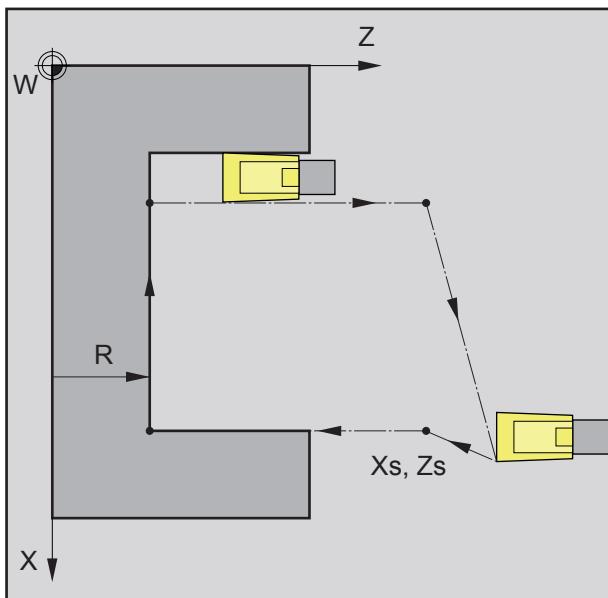


Sequence of grooving operations during roughing

Cycle Description

Roughing

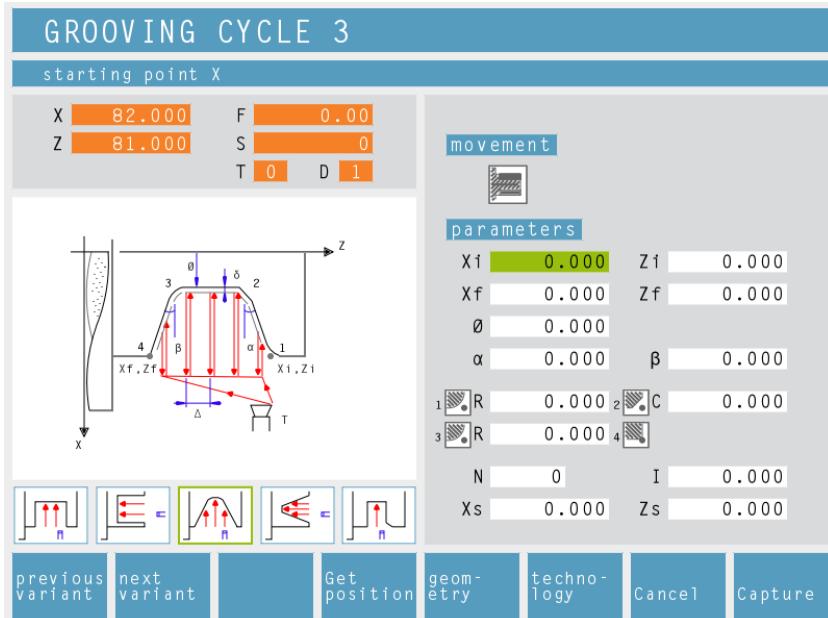
- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several grooving operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowance (δ) is reached and lingers there for (t) - if entered.



Sequence of grooving operations during finishing

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual grooving operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

Geom-
etry**Moving direction:**

outside turning
inside turning

**Coordinates (Xi, Zi)**

Coordinates of the starting point

Coordinates (Xf, Zf)

Coordinates of the end point

Remaining diameter (Ø)

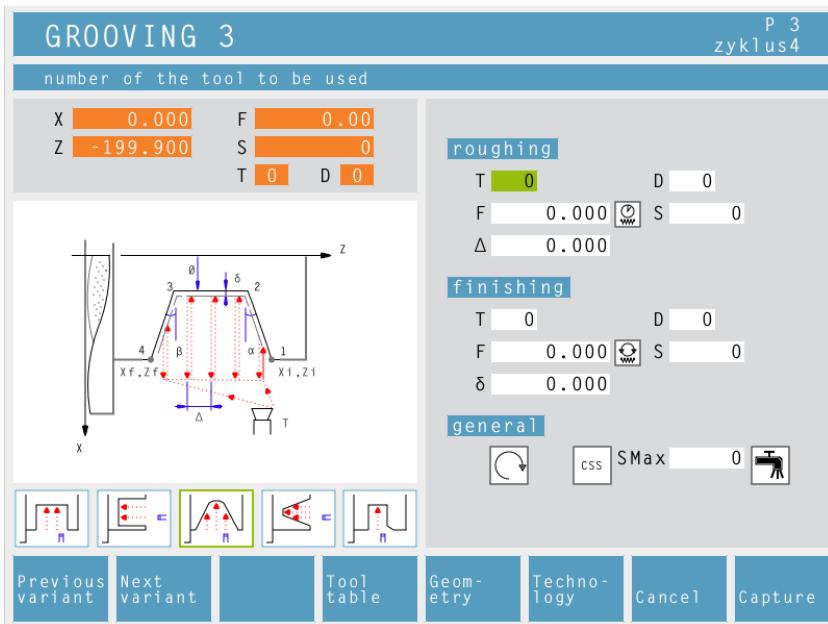
Final turning diameter

Flank angle (α)**Flank angle (β)****Corner rounding:**

sharp-edged
rounded
chamfered at 45°

**Number of grooves (N)****Distance between grooves (l)**Distance between two grooves (from $X_{i_1}Z_{i_1}$ - $X_{i_2}Z_{i_2}$). For one groove only please enter $l=0$.**Safety distance (Xs, Zs)**

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Finishing offset (δ)

Offset that should remain during roughing.



Feed setting

Roughing feed in mm/revolution

Feed setting

Roughing feed in mm/min

Note:

The groove has to **be the same size or** must be larger than the tool width.
It is absolutely necessary to enter the tool width (B) during tool calibration.

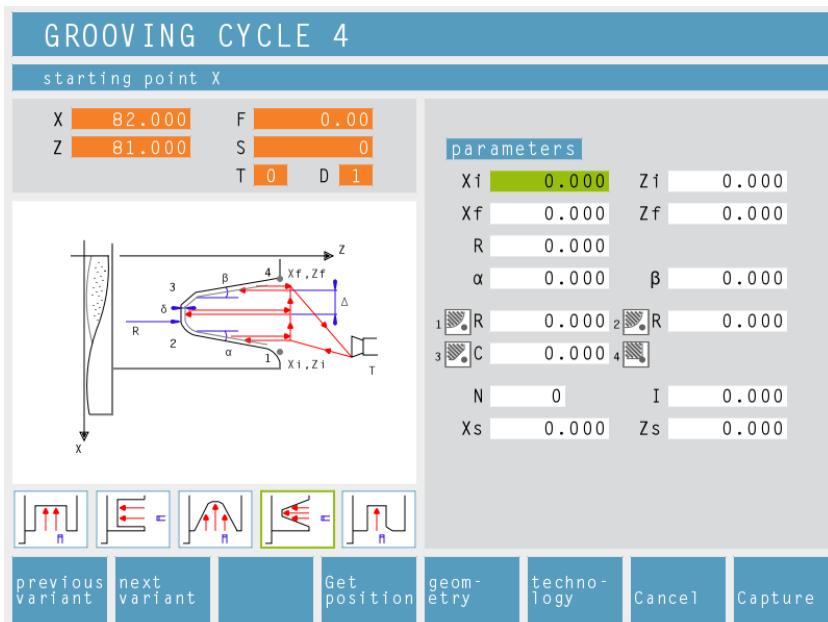
The cutting depth for roughing Δ should be a little bit smaller than the tool width.

Cycle Description**Roughing**

- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several grooving operations in succession. In this, each time the tool shifts the roughing infeed by (Δ), taking into account the thread angles(α , β) until the programmed slide allowance (δ) is reached and lingers there for (t) - if entered.

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual grooving operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

Geom-
etry

Grooving Cycle 4

Previous
variantNext
variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to machine a wedge-shaped, axial groove.

Coordinates (X_i, Z_i)

Coordinates of the starting point

Coordinates (X_f, Z_f)

Coordinates of the end point

Depth of groove (R)

Z coordinate of the groove

Flank angle (α)

Flank angle (β)

Corner rounding:

sharp-edged rounded chamfered at 45°



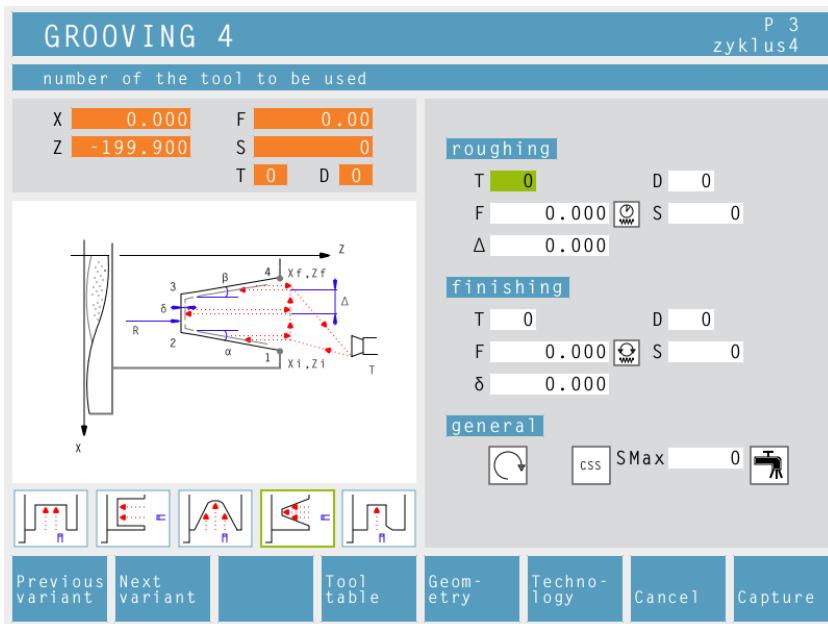
Number of grooves (N)

Distance between grooves (I)

Distance between two grooves (from $X_{i_1}Z_{i_1}$ - $X_{i_2}Z_{i_2}$). For one groove only please enter $I=0$.

Safety distance (Xs, Zs)

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Finishing offset (δ)

Offset that should remain during roughing.



Feed setting

Roughing feed in mm/revolution

Feed setting

Roughing feed in mm/min

Note:

The groove has to **be the same size or** must be larger than the tool width.
It is absolutely necessary to enter the tool width (B) during tool calibration.

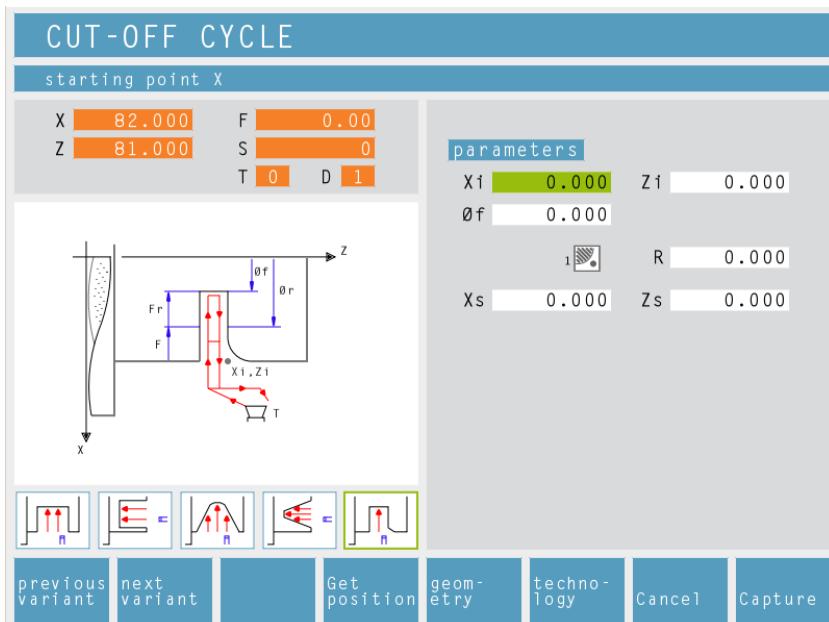
The cutting depth for roughing Δ should be a little bit smaller than the tool width.

Cycle Description**Roughing**

- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several grooving operations in succession. In this, each time the tool shifts the roughing infeed by (Δ), taking into account the thread angles(α, β) until the programmed slide allowance (δ) is reached and lingers there for (t) - if entered.

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 The tool with finishing feed then finish-machines an individual grooving operation on the remaining diameter (\emptyset).
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

**Note:**

Select $\text{of} = 0$ to cut off workpieces.
It is absolutely necessary to enter the tool width (B) during tool calibration.

Cut-Off Cycle

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

This cycle enables to quickly cut off turning components.

Coordinates (X_i, Z_i)

Coordinates of the starting point

Diameter (Of)

Final diameter

Corner rounding:

sharp-edged
rounded
chamfered at 45°

**Safety distance (X_s, Z_s)**Techno-
logy

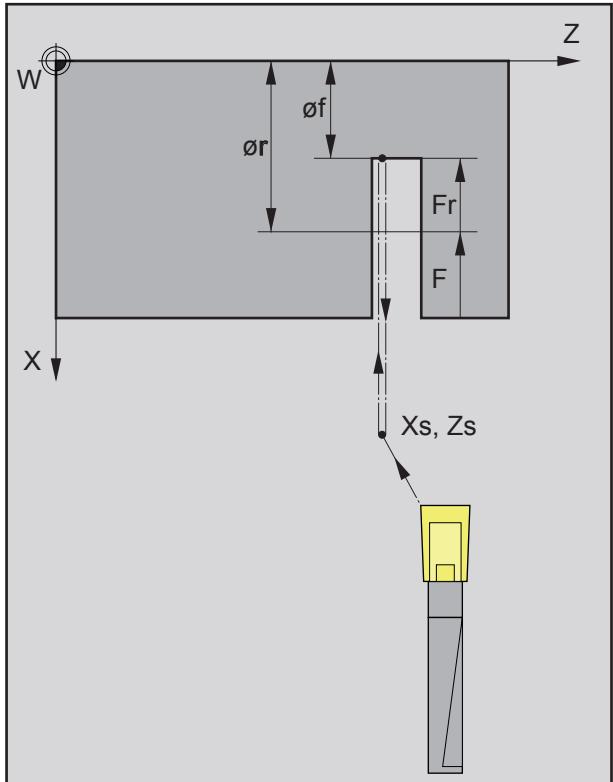
Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

Feed rate for cutting-off cycle (Fr)

Feed rate at which the tool moves from the intermediate diameter to the final diameter. In doing so, the feed rate F is gradually adjusted to the feed rate Fr.

Diameter (Or)

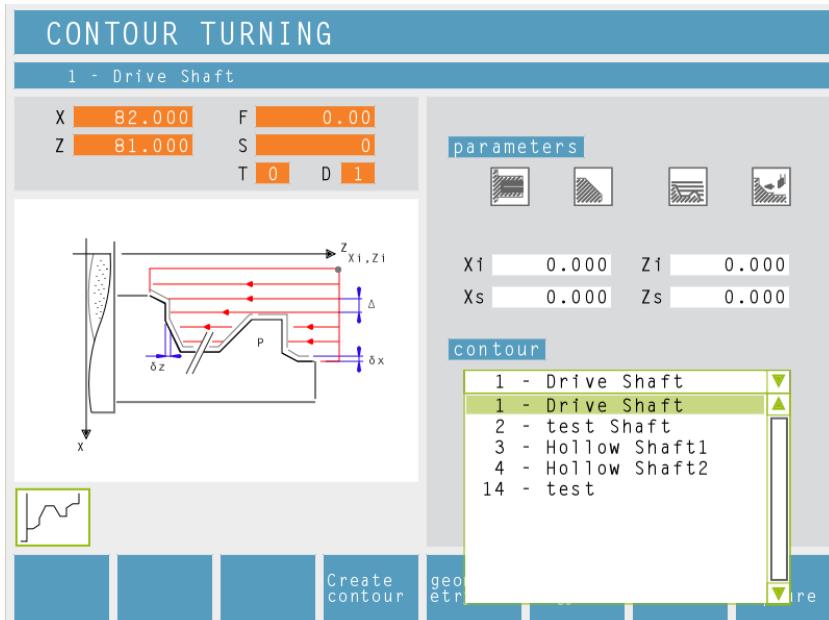
Intermediate diameter



Sequence of grooving operations during cutting

Cycle Description

- 1 The tool moves in rapid traverse to the safety distance (X_s , Z_s).
- 2 Then the tool with feed (F) grooves to the intermediate diameter ($\varnothing r$). The feed (F) is adjusted step by step to the feed (F_r) until the end diameter ($\varnothing f$). If selected - bevels and radii will be created if needed by multiple grooving.
- 3 At the end, the tool will be pulled back to the safety distance (X_s , Z_s) in rapid traverse.

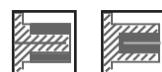


Contour Turning

This cycle enables to turn an already defined contour. Defined contours are saved in the window "machining steps".

Moving direction:

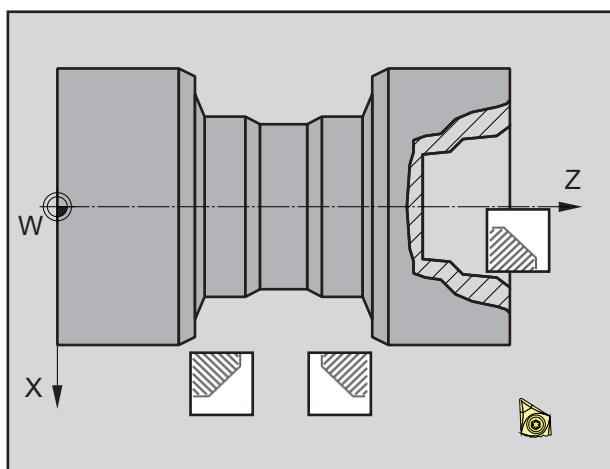
outside turning
Innendrehen

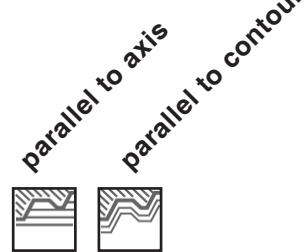
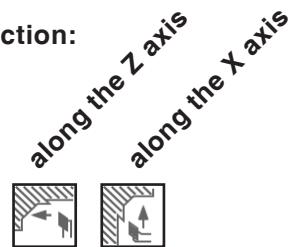


Taper position:

Please define the area in the quadrant that you wish to machine.

quadrant Z-
quadrant Z+



Cutting movement characteristic:**Machining direction:****Distance during 1st roughing pass (ε)**

Distance during parallel contour pre-roughing
With profile repetition, the material quantity that should be removed from the parent part must be defined.

Coordinates (Xi, Zi)

Coordinates of the starting point

Safety distance (Xs, Zs)**Contour**

Select a contour from the select list.

Create
contour

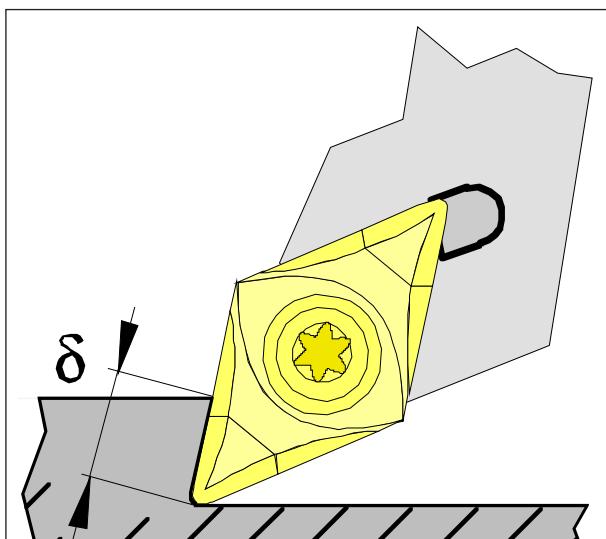
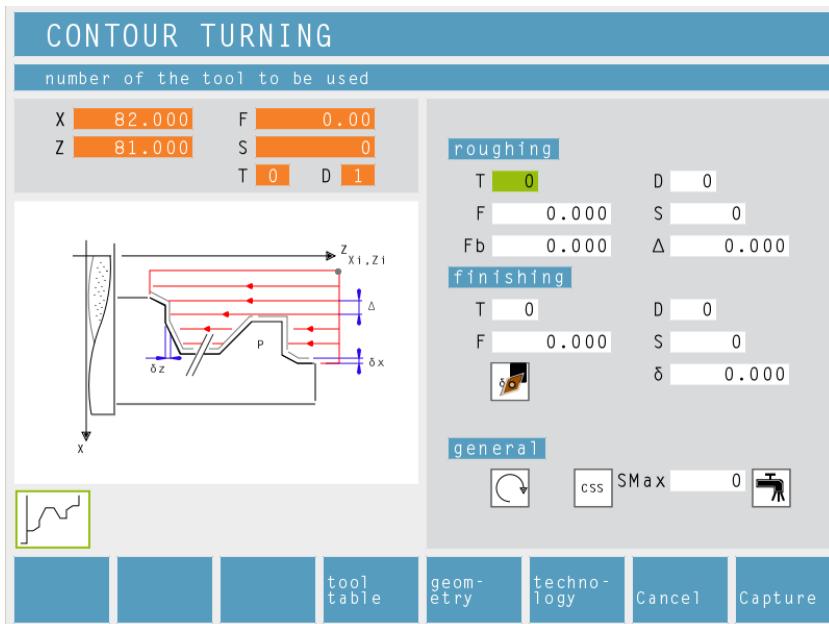
By pressing this soft key, you get back to the contour management to define a new contour. When going back, the new contour will be automatically selected in the list.

Modify
Contour

The softkey leads directly to contour ***editing***, in order to make changes to a contour here. See also Chapter C under "Contour Administration".

Techno-
logy

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Finishing offset δ

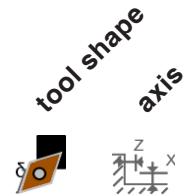
Feed for infeed at relief cut (Fb)

Feed rate during roughing parallel to axis

Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Finishing offset either by tool shape or axis:



Finishing offset (δ)

Offset that should remain during roughing.

Finishing offset in the X axis (δ_x)

Offset that should remain during roughing.

Finishing offset in the Z axis (δ_z)

Offset that should remain during roughing.

Cycle Description**Roughing**

1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).

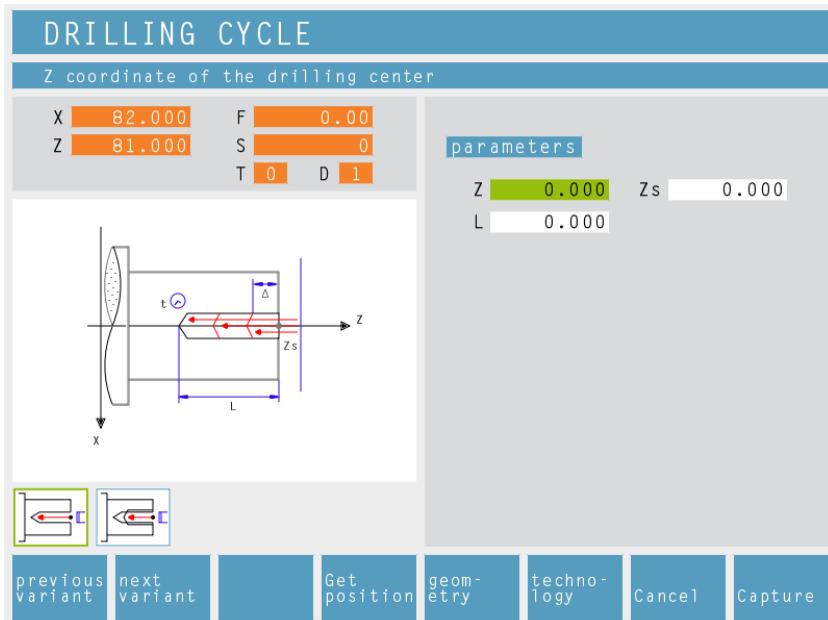
2 Then the tool with roughing feed moves to the first contour point and rough-machines axis/contour parallel several turning/facing operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ or δ_x , δ_z) are reached. With axis-parallel pre-roughing, the grooving feed (F_b) of the tool must be defined in the pockets.

Finishing

3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.

4 Then the tool with finishing infeed finish-machines the contour in an individual turning operation.

5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Drilling Cycle

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

Z coordinate of the drilling center

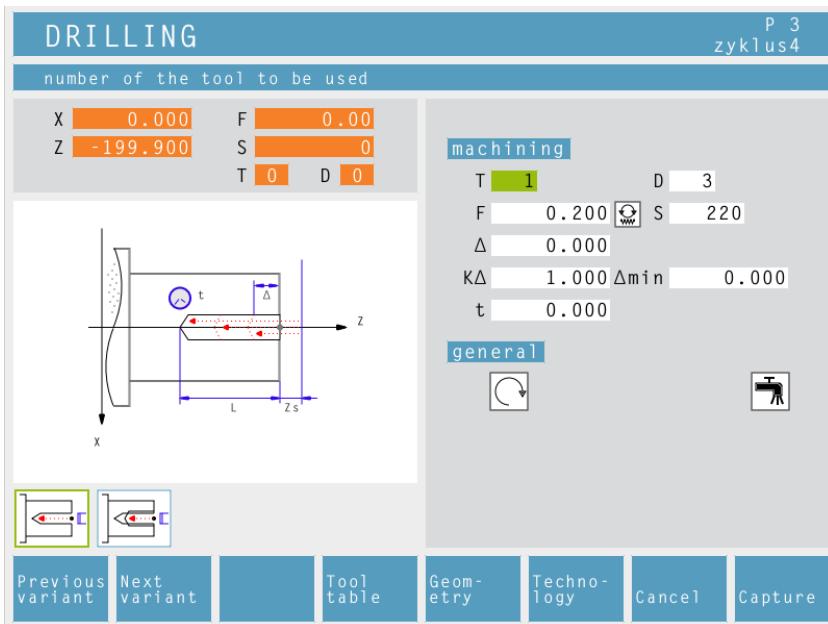
Safety distance Z (Zs)

Total depth (L)

Actual drilling depth in Z

Geom-
etry

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Drilling stroke for 1st cutting depth (Δ)
1st depth of a drilling operation

Reducing factor for drilling stroke (K Δ)
Factor by which the subsequent drilling strokes will be reduced.

Minimum drilling stroke (Δ_{min})
Smallest cutting depth required (only for decreasing cutting depth)

Dwell time (t)
in seconds



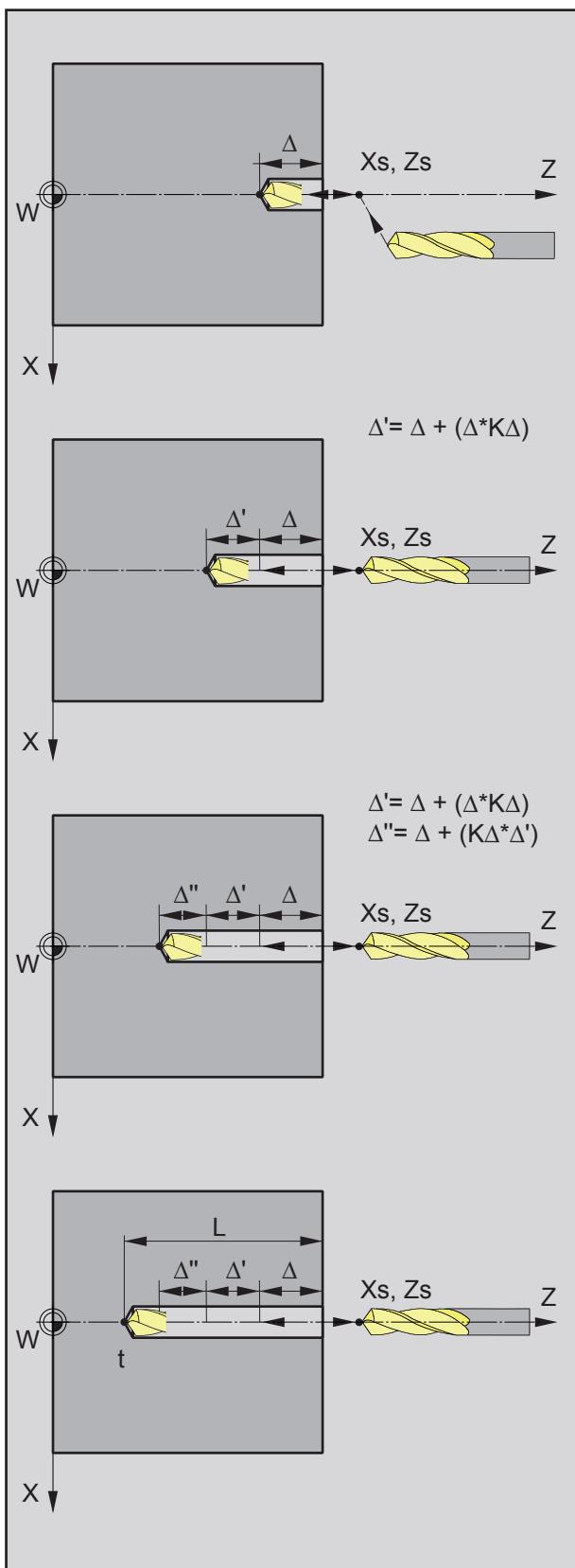
Feed setting

Roughing feed in mm/revolution



Feed setting

Roughing feed in mm/min



Example 2

Example 1:

Total drilling depth $L = 20$ mm,
drilling stroke $\Delta = 7$ mm, reducing factor $K\Delta = 1$

The control automatically calculates the number of the required infeeds from L and Δ :

$$20 : 7 = 2.85 \approx 3$$

This means that 3 drilling operations will be carried out until the total drilling depth of $L = 20$ mm is reached.

1. drilling depth = 7mm
2. drilling depth = 14mm
3. drilling depth = 20mm

Example 2:

Total drilling depth $L = 20$ mm,
drilling stroke $\Delta = 7$ mm, reducing factor $K\Delta = 0,8$

$$\begin{aligned} 1. \text{drilling depth} &= \Delta \\ &= 7 \text{mm} \end{aligned}$$

$$\begin{aligned} 2. \text{drilling depth} &= \Delta + \Delta' \\ &= \Delta + (\Delta * K\Delta) \\ &= 7 + (7 * 0,8) \\ &= 12,6 \text{mm} \end{aligned}$$

$$\begin{aligned} 3. \text{drilling depth} &= \Delta + \Delta' + \Delta'' \\ &= \Delta + (K\Delta * (\Delta + (\Delta * K\Delta))) \\ &= 7 + (0,8 * 12,6) \\ &= 17,08 \text{mm} \end{aligned}$$

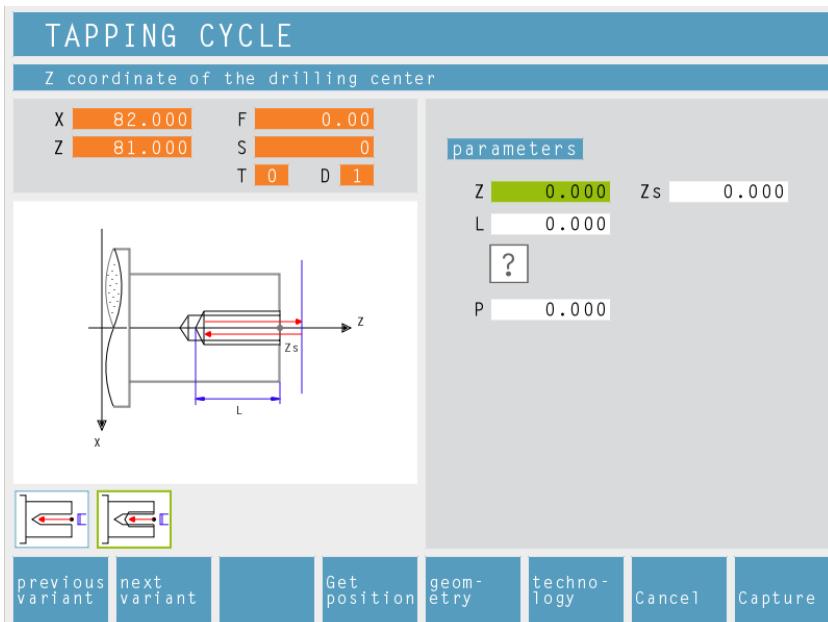
$$\begin{aligned} 4. \text{drilling depth} &= L \\ &= 20.000 \text{mm} \end{aligned}$$

$K\Delta = 0$ or 1 : no reducing factor (drilling stroke unchanged). The total depth is divided equally. The cutting depth is automatically calculated (see example 1).

$K\Delta \neq 1$: first drilling operation with depth $= \Delta$, second drilling operation with depth $= \Delta + (\Delta * K\Delta)$, third drilling operation with depth $= \Delta + (K\Delta * (\Delta + (\Delta * K\Delta)))$
(see example 2)

Cycle Description

- 1** The tool moves in rapid traverse to the safety distance (Xs, Zs).
- 2** The tool with roughing feed then taps several tapping operations in succession. In this, the tool moves to the safety distance and then advances by the calculated infeed in each case (see example). The tool dwells (t) at the hole depth - if entered.
- 3** At the end, the tool will be pulled back to the safety distance (Xs, Zs) in rapid traverse.



Tapping Cycle

Previous variant

Next variant

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

Geom-
etry

Z coordinate of the drilling center

Safety distance Z (Zs)

Total depth (L)

Actual drilling depth in Z

Thread type

You can choose among the following standardized thread types:



Free pitch thread

60 MM M (S.I.) Normal pitch metric thread

60 MM M (S.I.F.) Fine pitch metric thread
MM

55 INCH B.S.W. (W) Normal pitch Withworth thread

55 MM INCH B.S.F. Fine pitch Withworth thread

60 INCH U.N.C. Normal pitch unified American thread

60 MM INCH U.N.F. Fine pitch unified American thread

Note:

With the thread type „free pitch thread“ the thread pitch P and the thread depth H can be freely chosen.

With all other thread types the thread pitch P and the thread depth H will be automatically preset with standardized values after input of the thread diameter \varnothing .

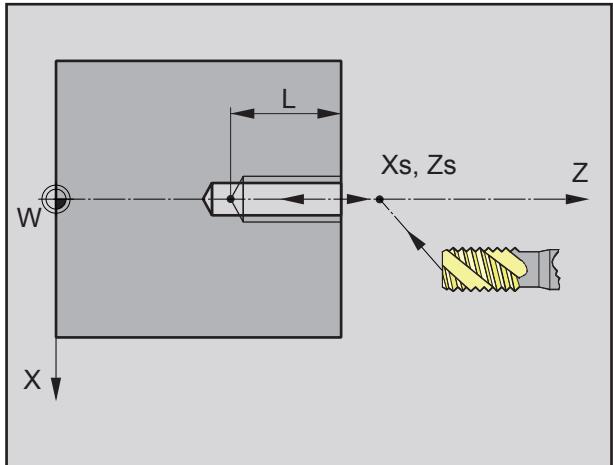
In case a thread diameter is entered which is not standardized, EASY CYCLE will automatically set the next standardized diameter.

Techno-
logy

Thread pitch (P)

Thread diameter (\varnothing)

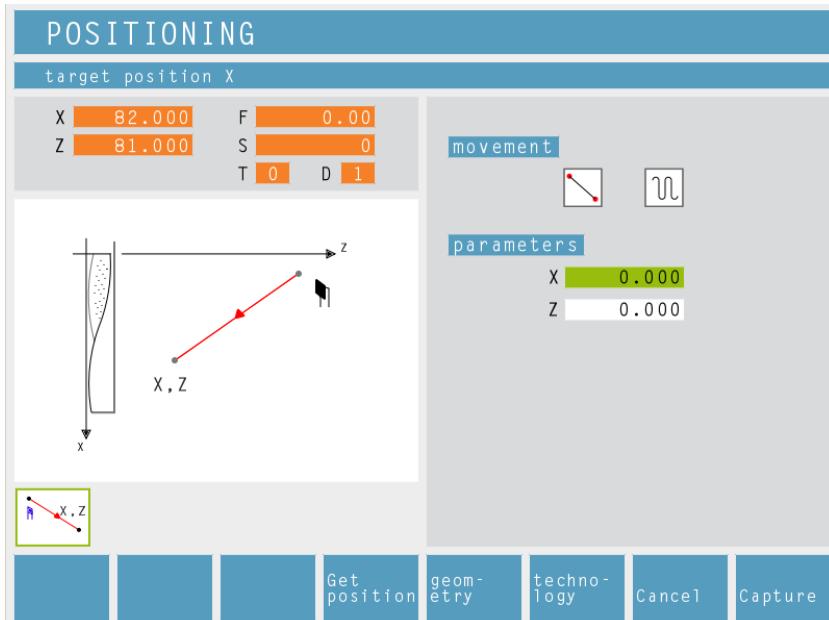
Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Sequence of the thread tapping

Cycle Description

- 1 The tool moves in rapid traverse to the safety distance (X_s , Z_s).
- 2 Then the tool with feed (F) taps the thread until the depth (L) is reached. The spindle stops and starts in the opposite direction.
- 3 At the end, the tool will be pulled back to the safety distance (X_s , Z_s) in rapid traverse.



Positioning

You can traverse or position the tool in a straight way.

Use:

- to position the tool prior to the machining itself
- to traverse the tool between two cycles



The tool traverses from its current position to its target position in a straight line.



The tool traverses from its current position first in X and then in Z to its target position.



The tool traverses from its current position first in Z and then in X to its target position.



The tool moves at rapid traverse.



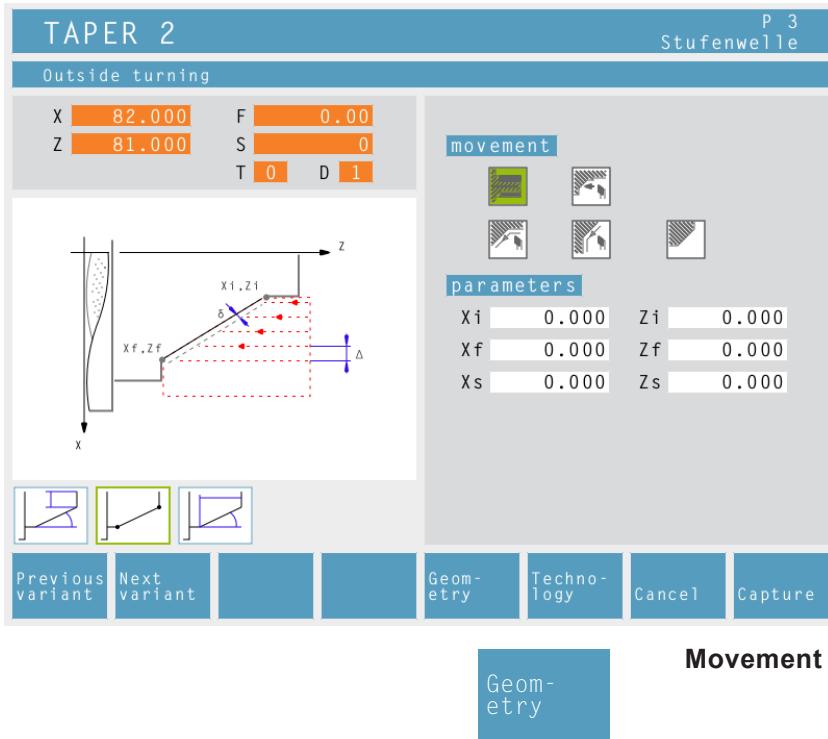
The tools moves at the programmed feed rate F.

Coordinates

The target position is defined by X and Z.

Techno-
logy

Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).

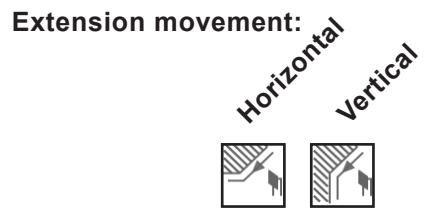
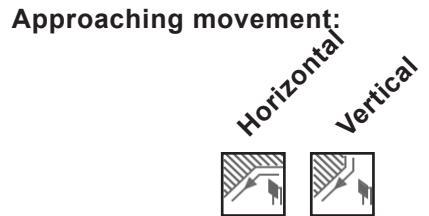
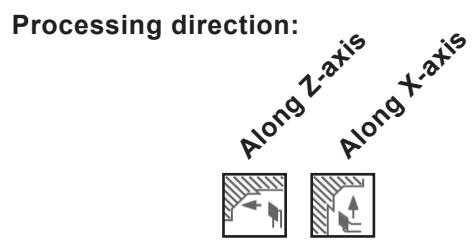
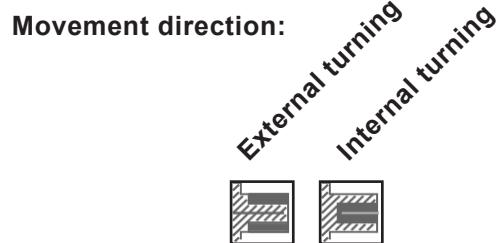


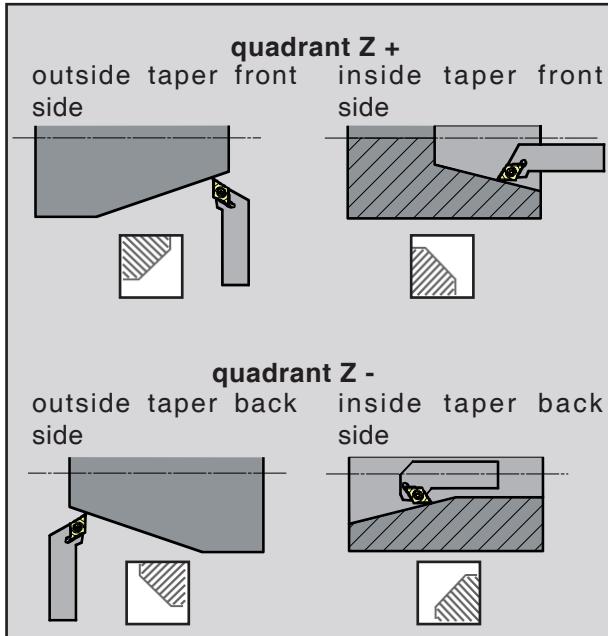
Taper Cycle 1 / 2 / 3

Vorige Variante Nächste Variante

Selection of the cycle from the cycle group by means of the soft keys „Previous variant“ and „Next variant“.

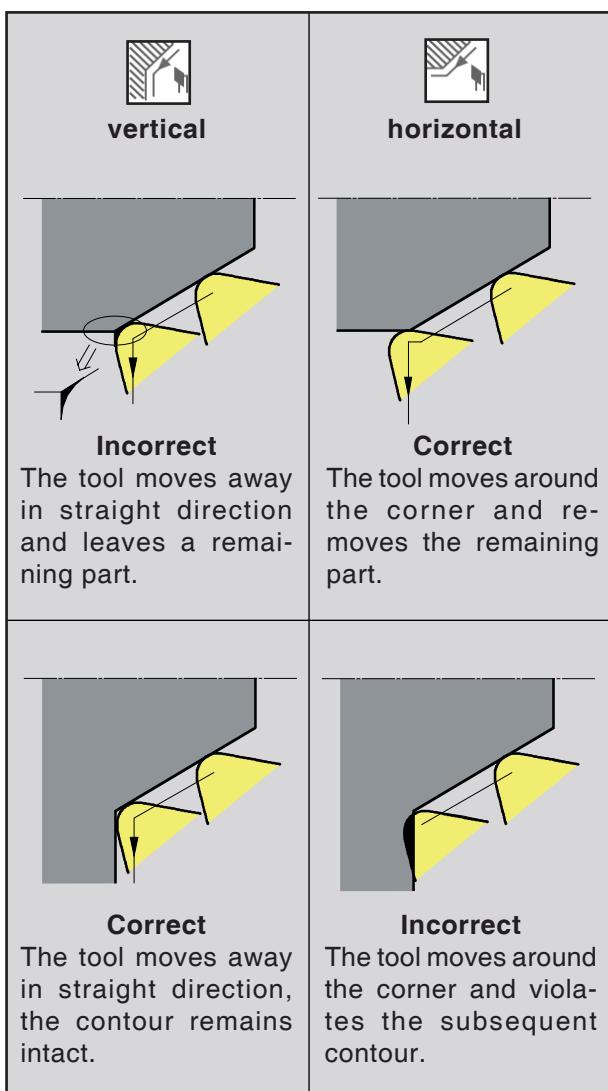
Taper turning 1, 2 and 3 only differ in the type of indication of measurement for the taper (point + angle + diameter, respectively 2 points, respectively point + angle + chamfer length).





Picture above: possible quadrants

Picture below: Example: Departing the contour



Taper position:

Define the area to be machined in the quadrant.

Approaching and departing the contour with tool radius compensation

To make sure that no corners will be violated and no remaining parts will stay behind during approaching or departing, the form of the corner must be defined.

Coordinates (X_i, Z_i)

Coordinates of the starting point.

Diameter (\emptyset)

Initial diameter of the taper (only for taper turning 1).

Taper length in Z direction (Z)

Lenght of the taper in direction of the Z axis.

Angle (α)

Taper angle (only for taper turning 1).

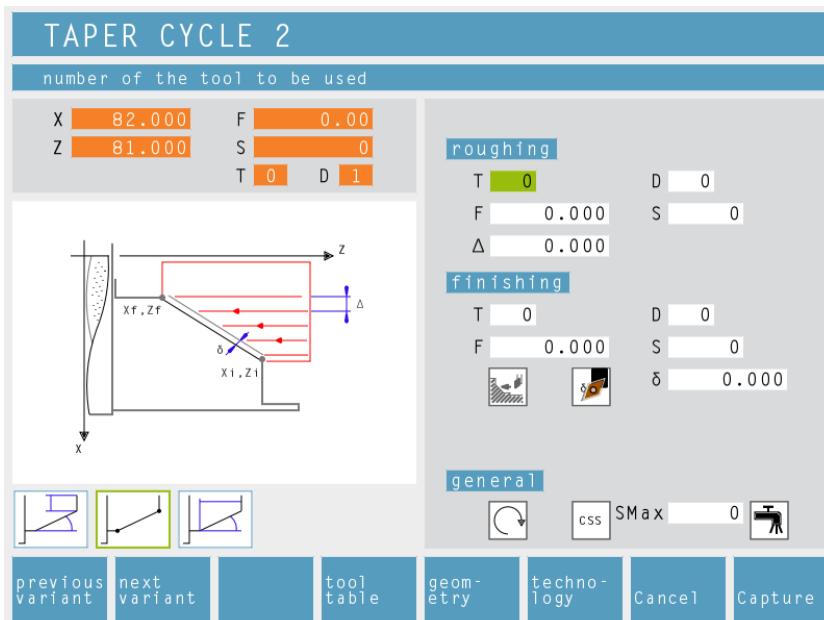
Coordinates (X_f, Z_f)

Coordinates of the end point (only for taper turning 2).

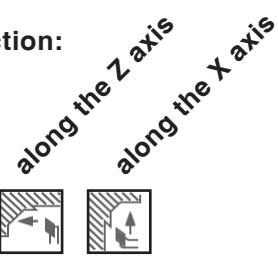
Safety distance (X_s, Z_s)

Techno- logy

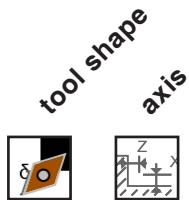
Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Machining direction:



Finishing clearance according to:



Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Finishing clearance (δ)

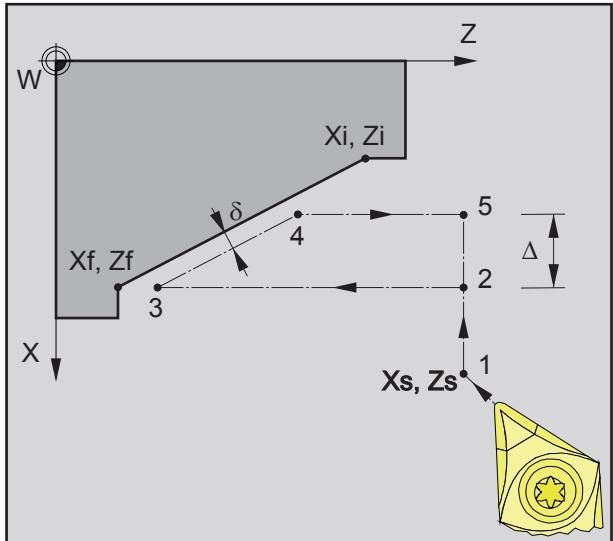
Clearance that should remain during roughing.

Finishing clearance in the X axis (δ_x)

Clearance that should remain during roughing.

Finishing clearance in the Z axis (δ_z)

Clearance that should remain during roughing.

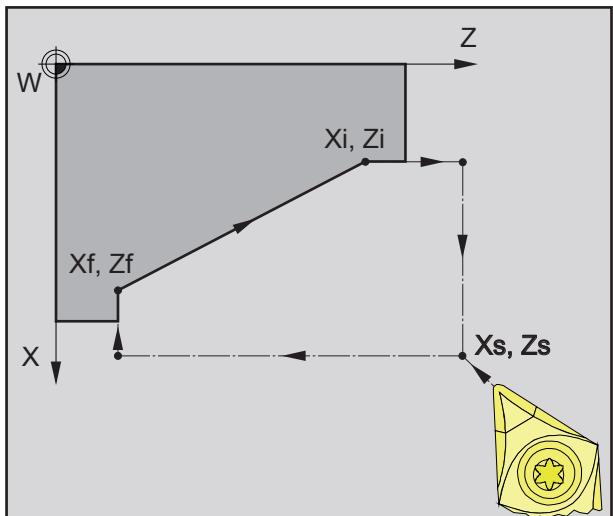


Sequence of an individual roughing operation with a processing direction along the Z-axis

Cycle Description

Roughing

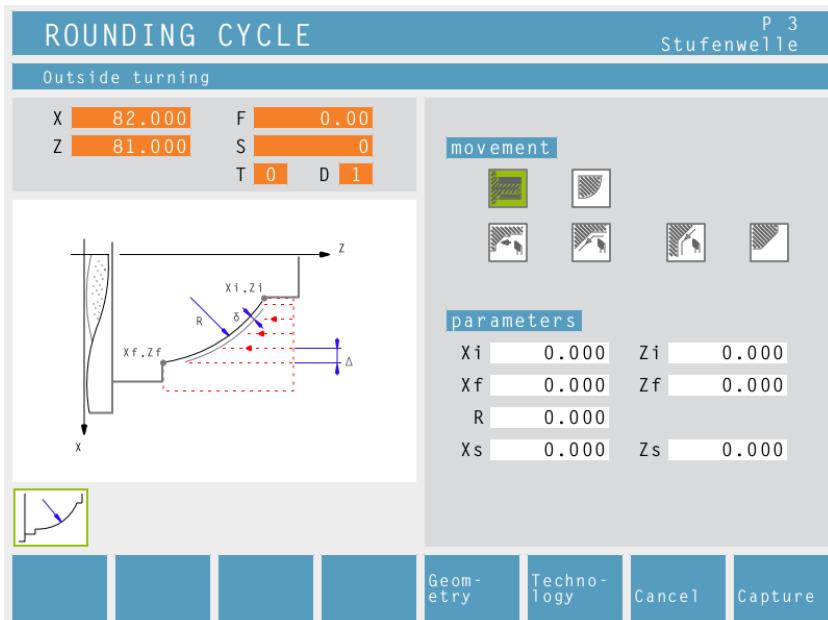
- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several grooving operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ or δ_x, δ_z) are reached.



Sequence of finishing operations with a processing direction along the Z-axis

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 Then the tool with finishing infeed finishes an individual operation.
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.



Rounding Cycle

This cycle serves to creating arc segments.

Moving direction:

outside turning
inside turning



Contour shape:

convex
concave



Machining direction:

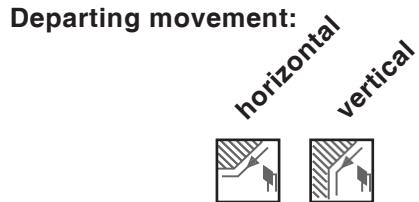
along the Z axis
along the X axis



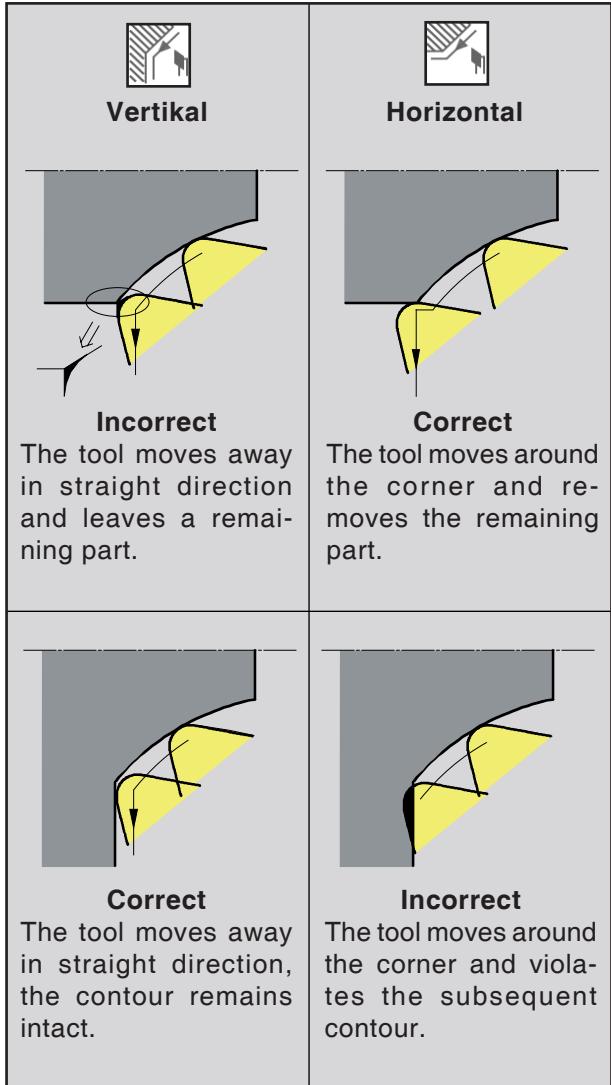
Approaching movement:

horizontal
vertical



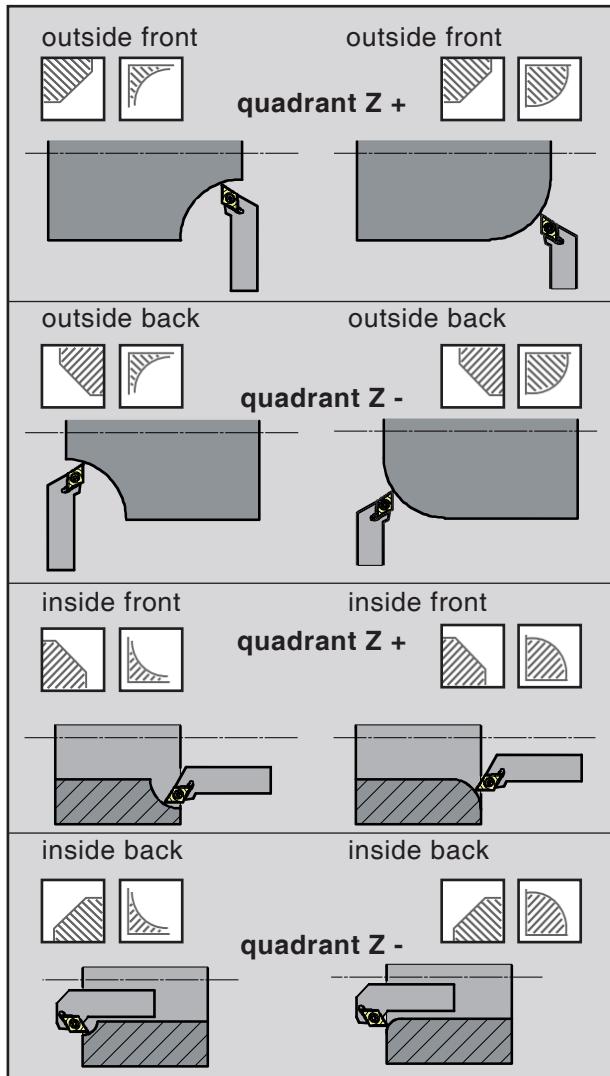


Example: Departing the contour



Approaching and departing the contour with tool radius compensation

To make sure that no corners will be violated and no remaining parts will stay behind during approaching or departing, the form of the corner must be defined.

**Radius position:**

Define the area to be machined in the quadrant.

Coordinates (X_i, Z_i)

Coordinates of the starting point

Coordinates (X_f, Z_f)

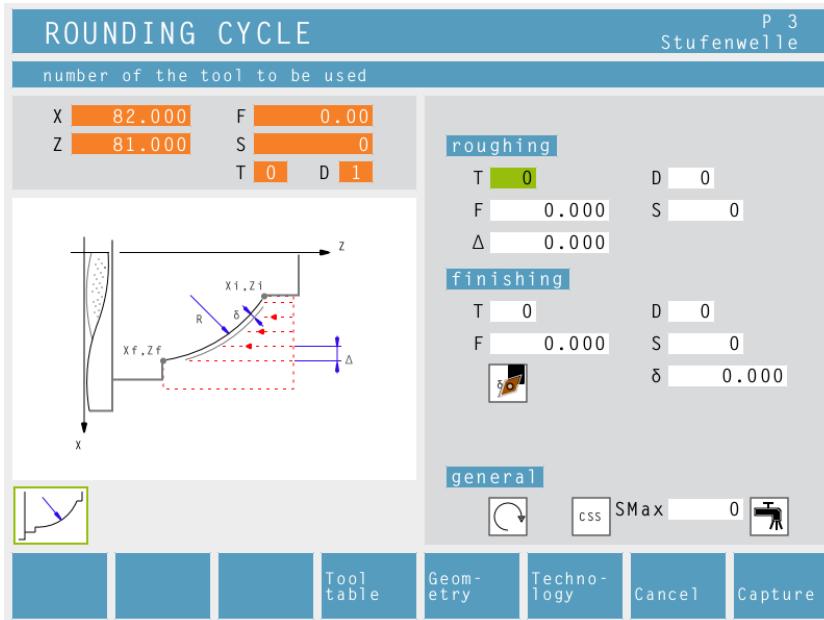
Coordinates of the end point

Radius R

Radius of the rounding

Safety distance (X_s, Z_s)

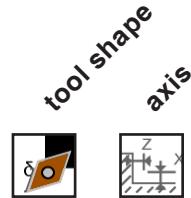
Then, please enter all necessary technological data (tool number, direction of rotation, feed rate, spindle speed).



Cutting depth for roughing (Δ)

Lateral distance by which the tool is shifted during roughing.

Finishing clearance according to:



Finishing clearance (δ)

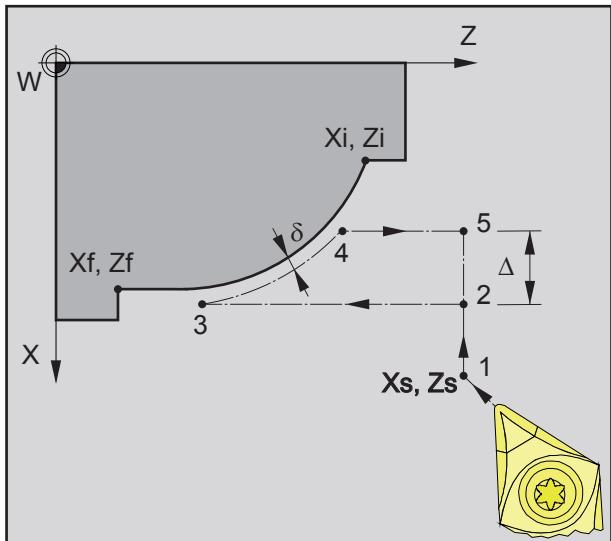
Clearance that should remain during roughing.

Finishing clearance in the X axis (δ_x)

Clearance that should remain during roughing.

Finishing clearance in the Z axis (δ_z)

Clearance that should remain during roughing.

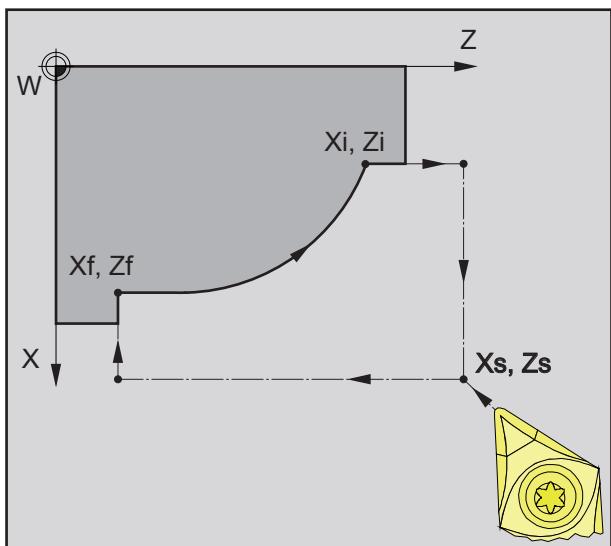


Sequence of an individual roughing operation with a processing direction along the Z-axis

Cycle Description

Roughing

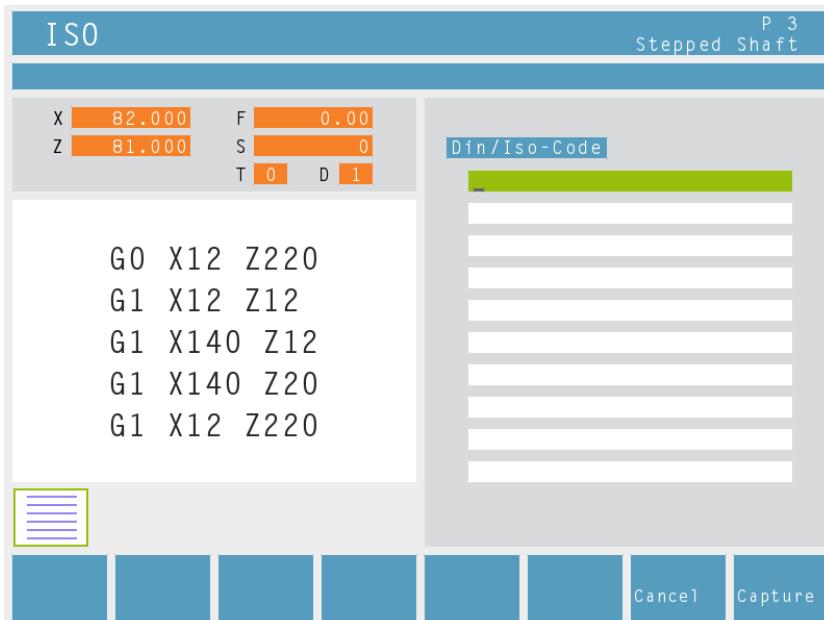
- 1 The tool moves in rapid traverse to the safety distance (X_s, Z_s).
- 2 The tool with roughing feed then rough-machines several operations in succession. In this, each time the tool shifts the roughing infeed by (Δ) until the programmed slide allowances (δ or δ_x , δ_z) are reached.



Sequence of finishing operations with a processing direction along the Z-axis

Finishing

- 3 If another tool was programmed for the finishing, a tool change will take place at the tool change point.
- 4 Then the tool with finishing infeed finishes an individual operation.
- 5 At the end, the tool will be pulled back to the safety distance (X_s, Z_s) in rapid traverse.

**Iso Edit**

This cycle serves to limited DIN/ISO-Code programming.

Overview M-Commands

Command	Description
M0	Programmed stop
M1	Optional stop (program stop only with OPT. STOP)
M2	End of program
M7	Minimum lubrication ON
M8	Coolant ON
M9	Coolant OFF, Minimum lubrication OFF
M20	Tailstock quill BACKWARDS
M21	Tailstock quill FORWARDS
M25	Clamping device OPEN
M26	Clamping device CLOSE
M30	End of main program
M71	Blow out ON
M72	Blow out OFF
M90	Manual chuck
M91	Collet chuck
M92	Power chuck

Overview G-Commands

COMMAND	DESCRIPTION	FORMAT
G0	Rapid traverse	G0 X... Y... Z...
G1	Straight-line interpolation	G1 X... Y... Z...
G2	Circular interpolation in clockwise direction	G2 X... Y... Z... I... J... K... G2 X... Y... Z... R...
G3	Circular interpolation in counter-clockwise direction	G3 X... Y... Z... I... J... K... G3 X... Y... Z... R...
G4	Dwell time (in seconds)	G4 F...
G33	Thread cutting	G33 X... Y... Z... L-pitch
G40	Cutter radius compensation OFF	G40
G41	Cutter radius compensation ON left	G41
G42	Cutter radius compensation ON right	G42
G94	Feed per minute	G94
G95	Feed per rotation	G95
G96	Constant cutting speed	G96
G97	Constant spindle speed	G97

I, J, K circle centre point in cartesian coordinates, related to start point.

R=+ angle smaller or equal 180°.

R=- angle larger 180°.

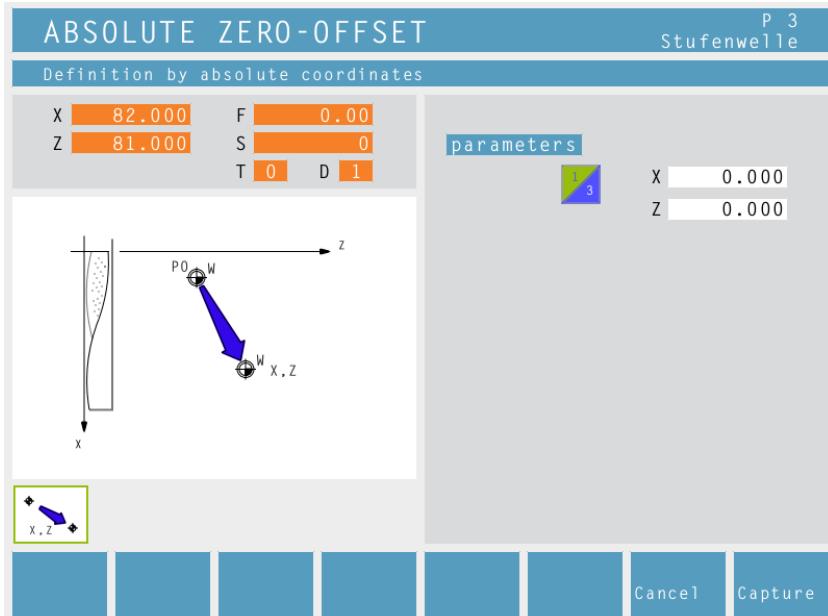
Miscellaneous commands

COMMAND	DESCRIPTION	FORMAT
T	Tool selection	T...
D	Selection of the cutting edge	D...
S	Spindle speed, respectively constant cutting speed	S...
F	Feed	F...

Coord. Trans.

Coordinate transformation

The cycle „Coordinate transformation“ serves to zero offset the workpiece zero point (W). The following possibilities exist:



Absolute Zero-Off-set

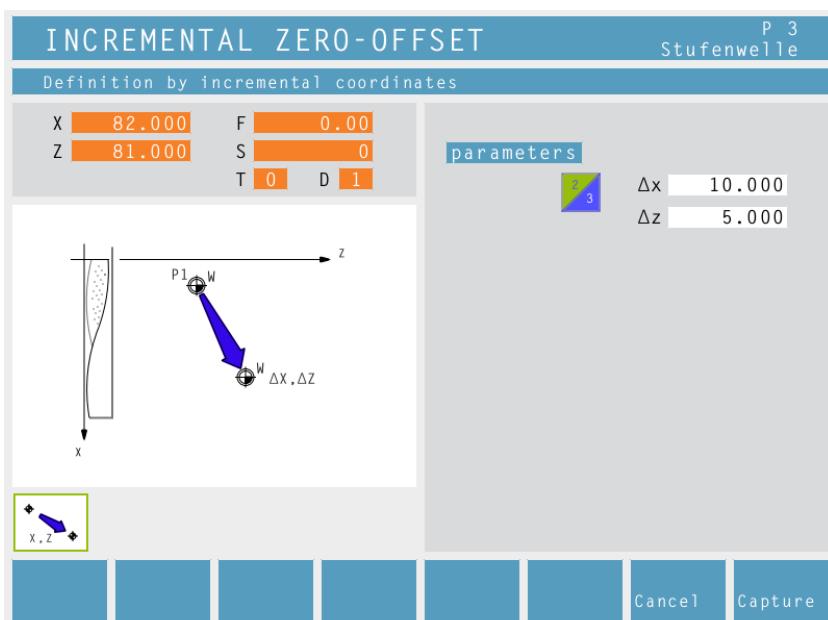


Coordinates (X, Z)

Absolute zero-offset of the workpiece zero point (W):

X...absolute zero-offset on the X axis

Z...absolute zero-offset on the Z axis



Incremental Zero-Offset

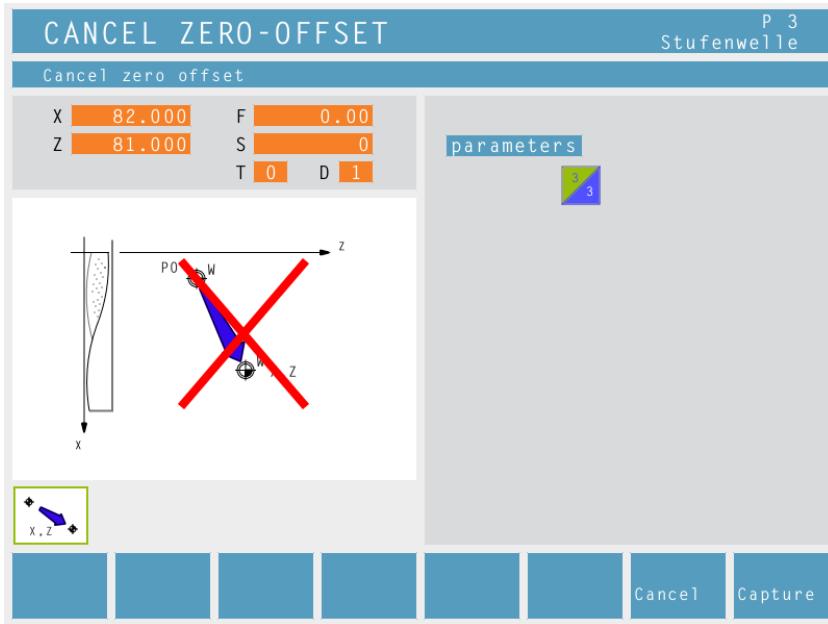


Coordinates (Δx , Δz)

Incremental zero-offset of the workpiece zero point (W):

Δx ...incremental zero-offset on the X axis

Δz ...incremental zero-offset on the Z axis



Cancel Zero-Offset

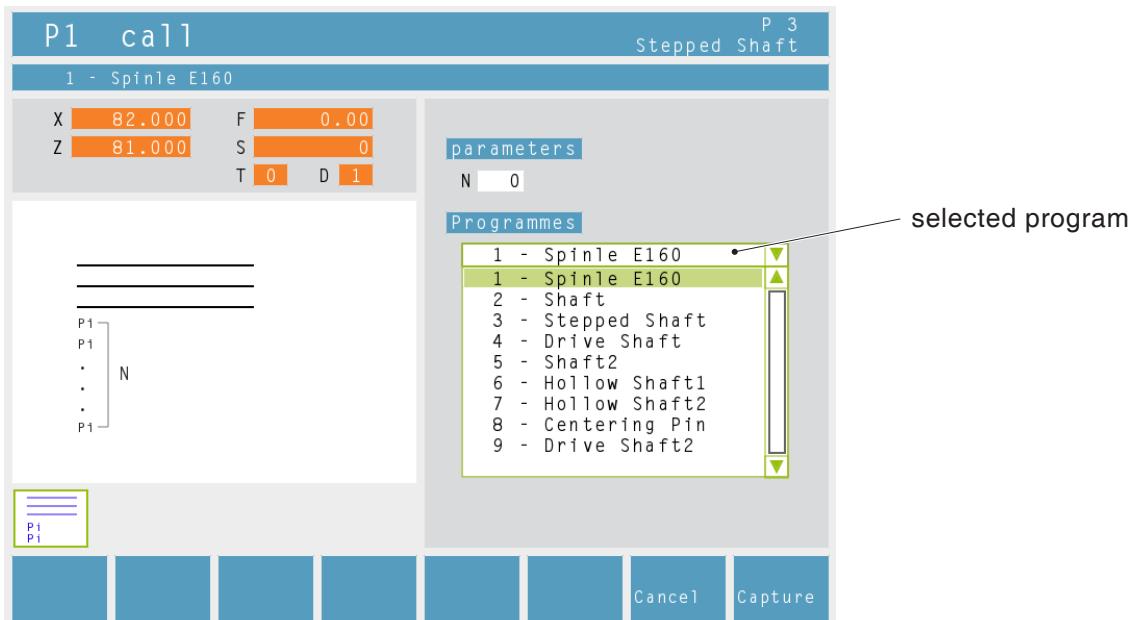


The zero offset of the work-piece zero point (W) is cancelled.

Call
Subprog.

Call subprogram

In the cycle „Call subprogram“, you can select a program from the program list that will be used as subprogram in the future.

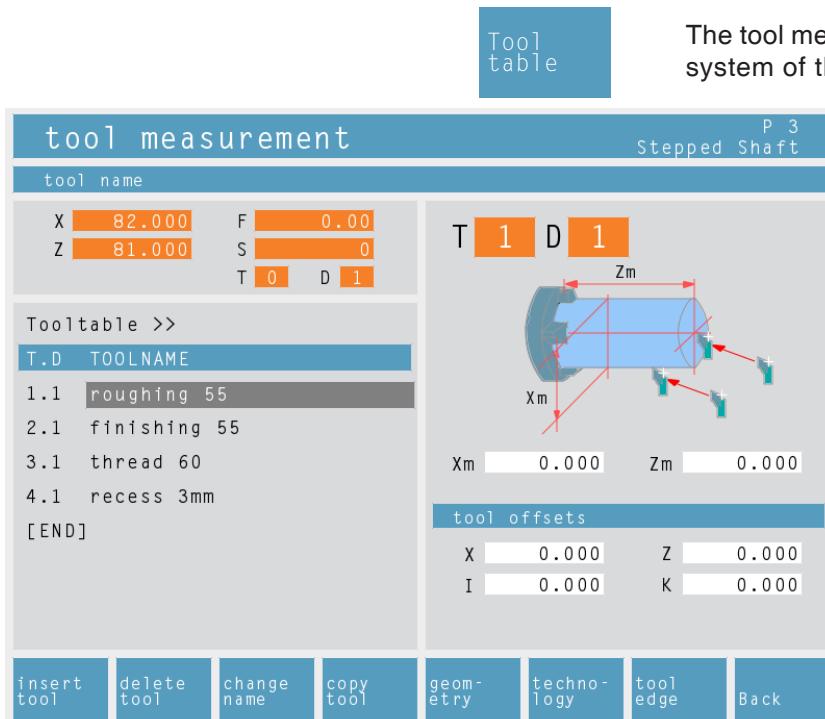


Number of repetitions (N)

Programmes

Select a program from the program list.

E: Tool measurement



The tool measurement enables you to edit the tool system of the NC machine:

- Deleting an existing tool.
- Renaming an existing tool.
- Copying an existing tool to the last position of the tool table.
- Measuring a tool at the machine.
- Entering a tool or a new cutting edge in the tool table.

Geometry

Defining tool data

- Fasten a workpiece of known height in the toolholder.
- Press the soft key.
- Define the following parameters:
 - Xm...X-coordinate of the reference part
 - Zm...Z-coordinate of the reference part
 - X...tool length in X
 - Z...tool length in Z
 - I...wear in X
 - K...wear in Z

Editing tool data



- You create blades for the tools here.



- You delete tool data for the tool here.



- You delete tools here.



- You change the name for the tool here.

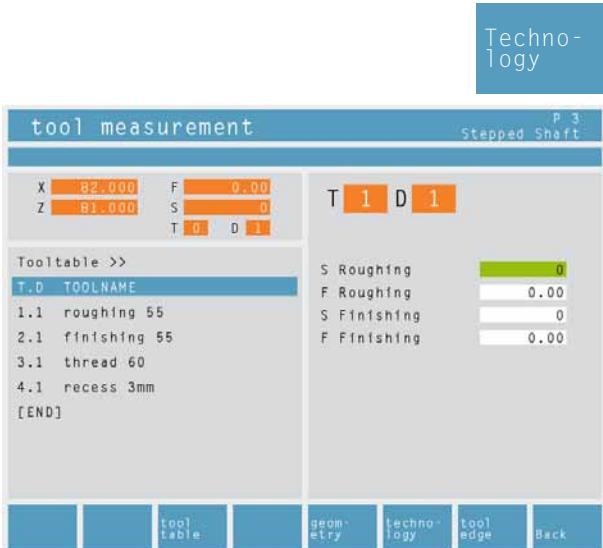


- You load tools for 3D View here.

Instructions:

The tool database is only active if EMCO Win 3D-View is installed in the same directory as WinNC EASY CYCLE.





Technology data for tools

The standard values of the machine for the feed and for the spindle speed can be further limited by means of the soft key.

After selection of the tool the respective spindle speeds and feeds will be already entered in the cycles as changeable proposal.



Tool family / Tool edge position

- Press the soft key.

The following families are possible:



- rhombic carbide tip



- carbide tip for threading



- carbide tip for grooving



- circular carbide tip



- drill or driven tool

Note:

In order to define the type, look at the tool, as if it were clamped in the machine.

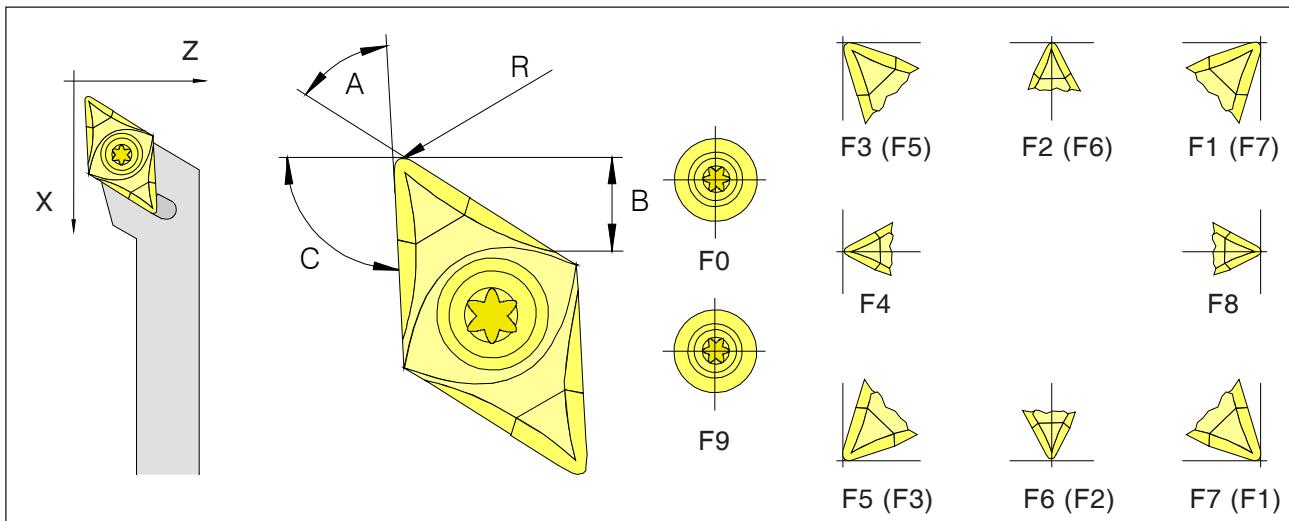
For machines where the tool is above (behind) the turning center, the values enclosed in brackets have to be used, due to the reversion of the +X direction.



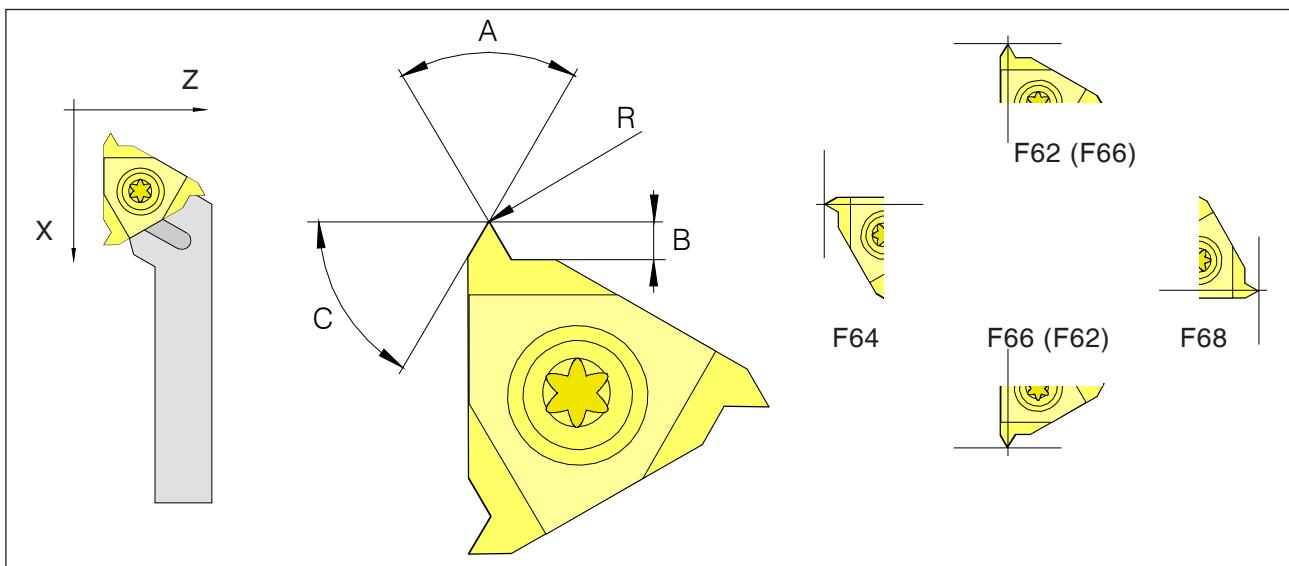
The following tool edge positions or tool tip positions are possible:



Available tool tip positions for the type:

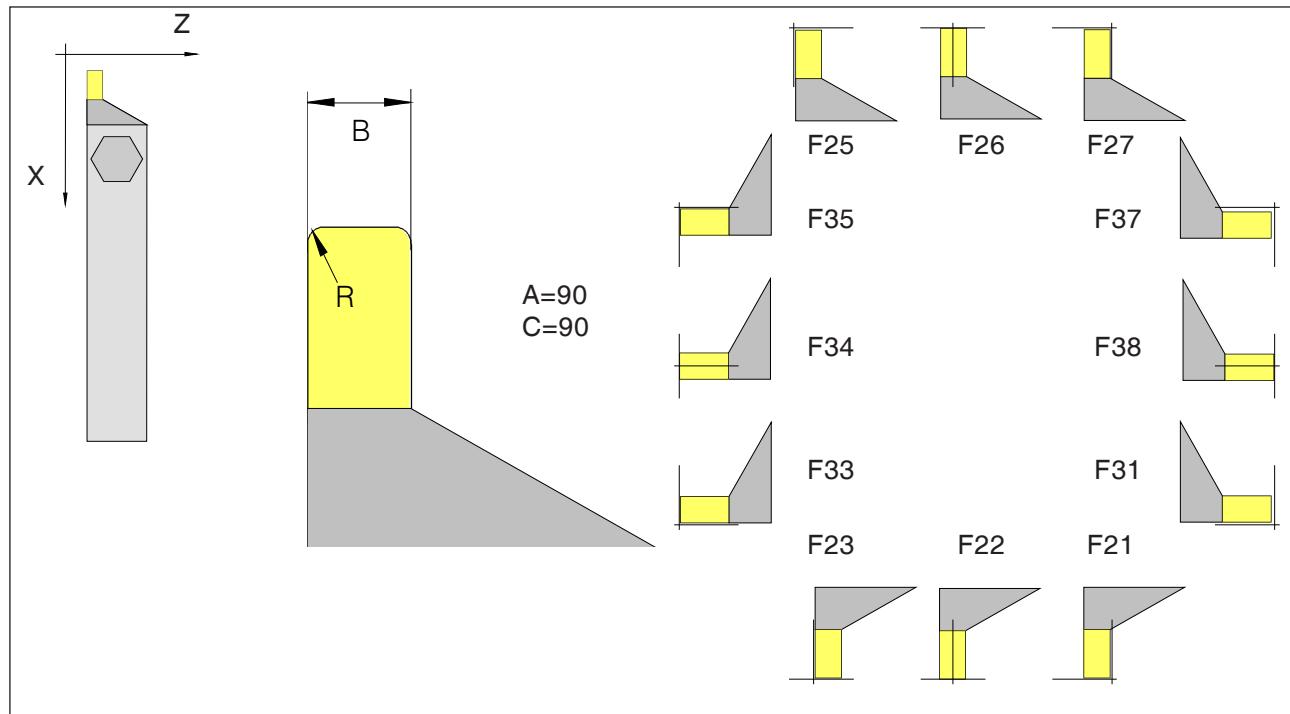


Available tool tip positions for the type:

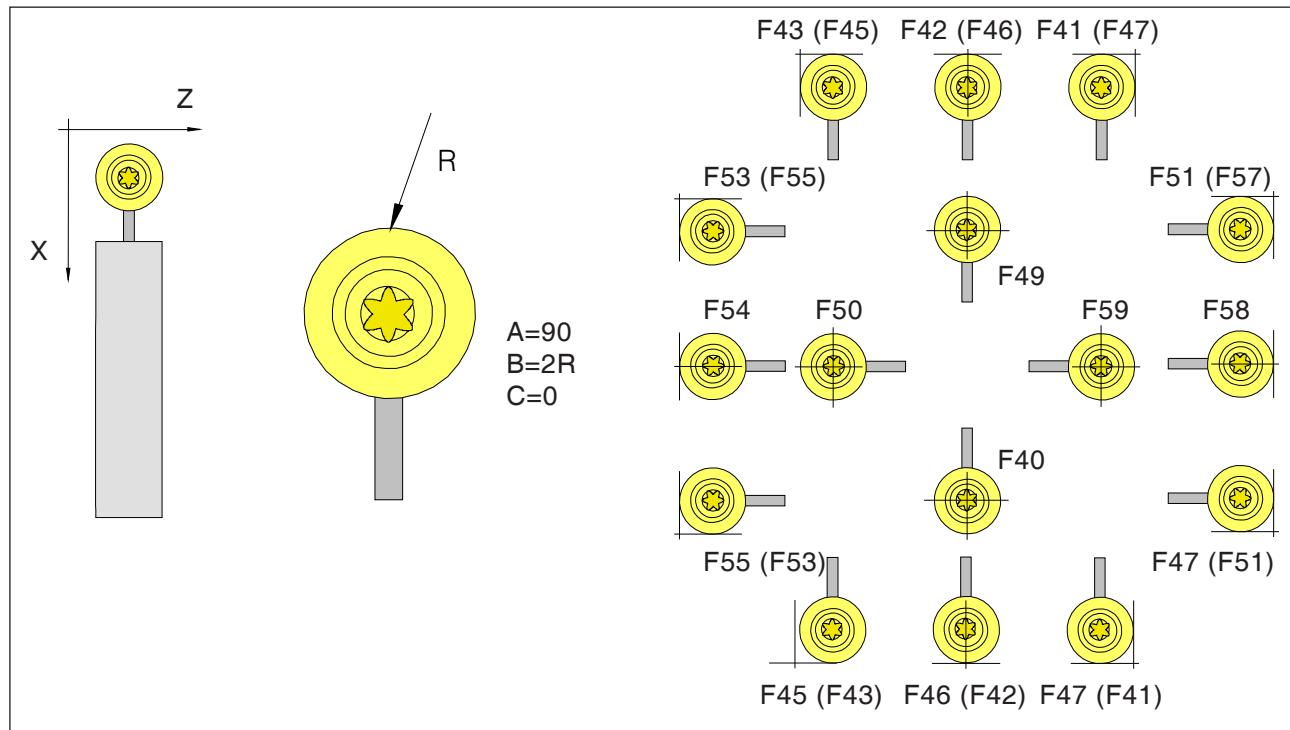


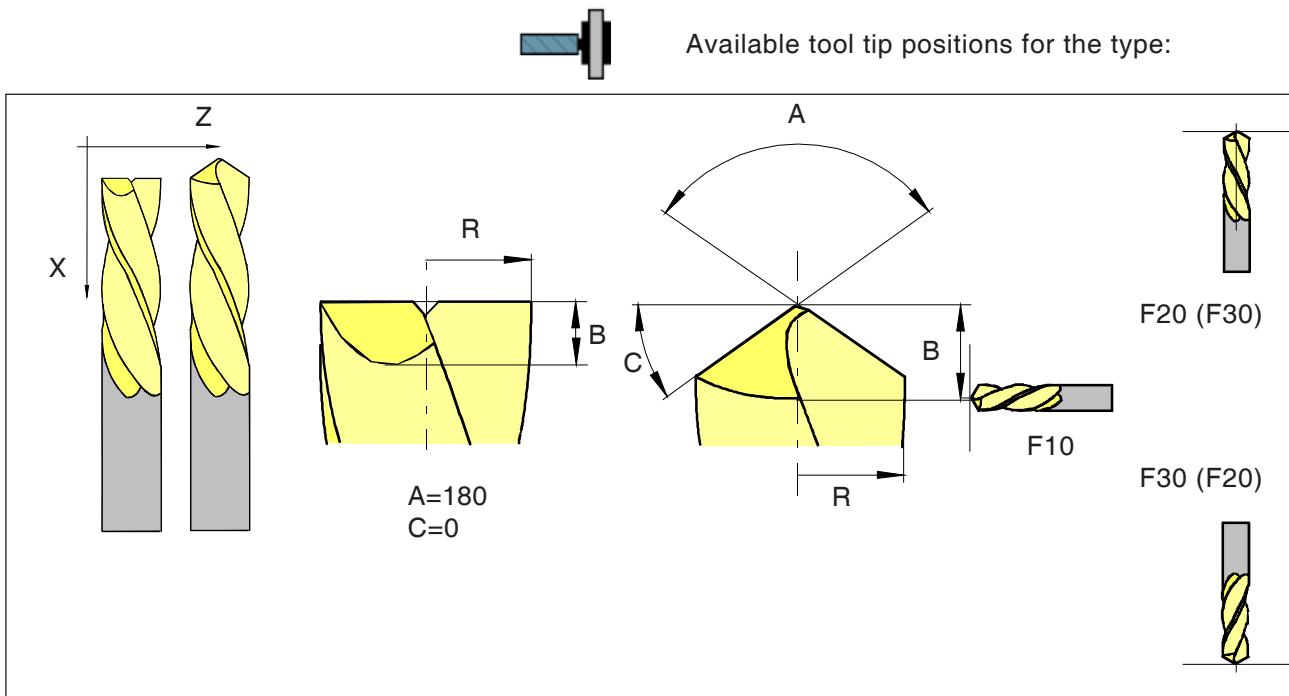


Available tool tip positions for the type:



Available tool tip positions for the type:



**Note:**

For drills it is necessary to define the length of the cutting edge (see illustration).

For standard drills with a nose angle of 120° applies:

$$B = 0,5774 \times \phi$$

Dimensions of the carbide tip:

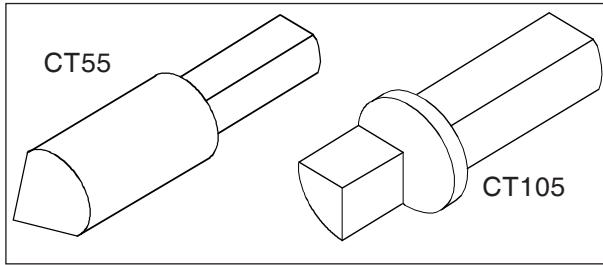
- A Angle of the tool tip
- B Cutting width
- C Cutting angle
- R Tool radius

Normally, the tool tip has no sharp edges, but rounded ones.

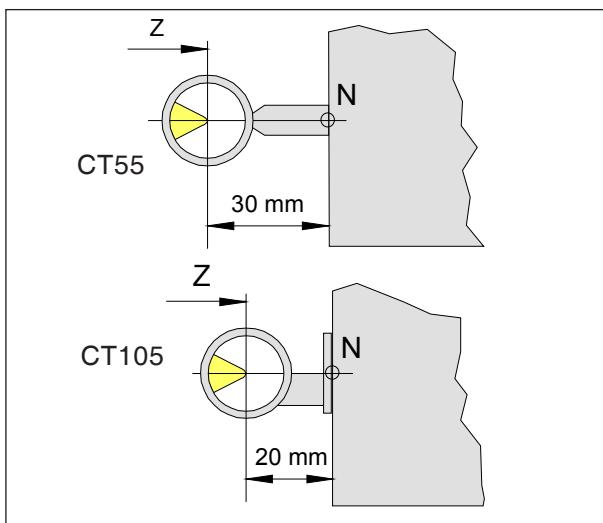
During tool calibration the tool is calibrated at two points (tangential towards the X and Z axis).

This leads to a dimensional error of approximately half the size of the tool radius during machining operations, that are not carried out parallelly to the X or Z axis.

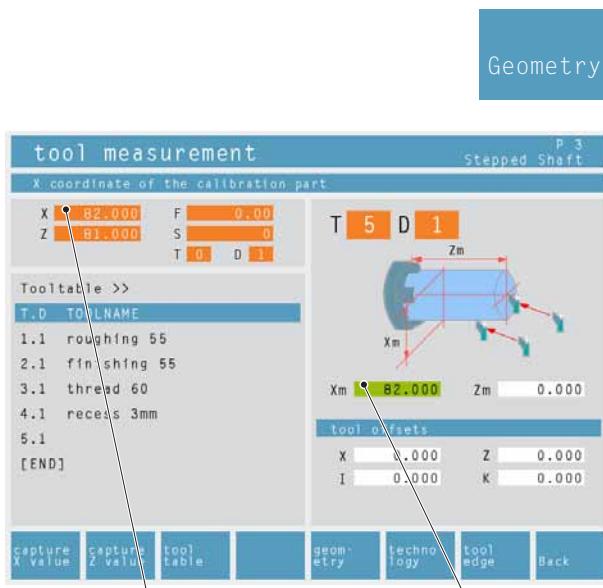
This dimensional error is automatically corrected by the control.



Reference tool Concept Turn 55 / 105



Calibration of the reference tool



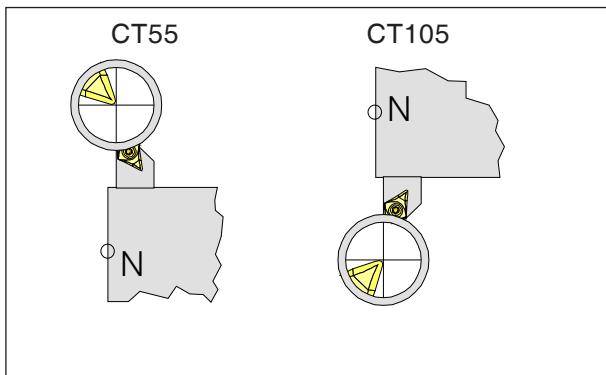
Tool calibration with optical presetting device

1.) Gauging the optical tool presetting device:

- Install the optical tool presetting device.
- Clamp the reference tool together with the tool-holder into the tool turret disc.
- Create the reference tool as drill or cutter.
- Enter the correction values X=0 and Z=20 (CT105), respectively Z=30 (CT55).
- Swivel the reference tool in and traverse the tool tip into the cross hairs.
- Define Z=0 and confirm with the enter key. The question "Setting reference point?" will appear. Please confirm with the enter key.
- Call the tool table and select the reference tool position.

- Enter Xm= (enter the displayed position value of the X axis) and Zm=0.

Now, the presettings for the tool calibration are completed.



Move the tool tip into the cross-hair

Insert
tool

Capture
X value Capture
Z value

Techno-
logy Tool
edge

2.) Tool measurement:

- Swivel the first tool to be measured. Move the tool tip into the cross-hair.

- Select the tool position in the tool table or create a new one by means of the soft key „Insert tool“.

Use the mouse button or the tab key to select any input box on the right side of the screen window.

Press the soft keys, so that the measured values will be captured.

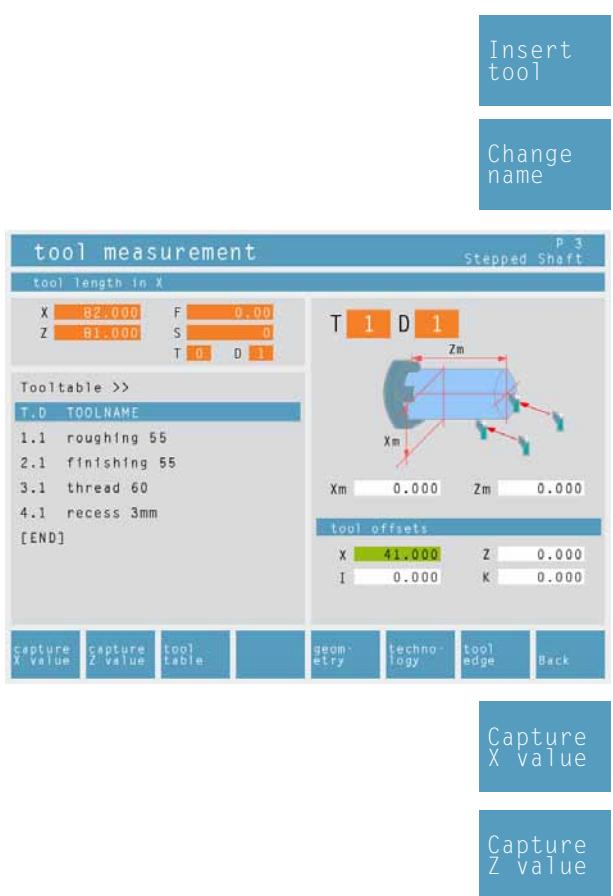
- Edit or complete the cutting data and the tool radius.

Measure all further tools as described under point 2.).

Then set the workpiece zero point (using a measured tool).

Instructions:

Only if machines with tool changers are used can a tool be deleted or added. To do this, the cursor must be on END in the tool list.



Tool measurement with touching method

- Press the soft key.
- Enter the tool name.
- Enter the Xm value of the reference part.
- Define the Zm value (length between the machine zero point and the plane surface) of the reference part with „0“.
- Touch the tool at the circumference of the workpiece and press the soft key.
- Touch the tool at the face of the workpiece and press the soft key.
- Now, the tool is measured. EASY CYCLE refreshes the data X and Z and sets the values for I and K to 0.
- If necessary, define the cutting data for the respective tool types.

Note:

The data X and Z define the dimensions of the tool in X and Z. The data I and K define the correction value that EASY CYCLE has to consider in order to compensate for the tool wear.

EASY CYCLE adds the value of the correction value (I,K) to the length (X, Z) in order to get the actual tool length (X+I) and (Z+K), it has to use.

The value "I" (correction value of the longitudinal wear in X direction) must be defined in diameters.

F: Program run

Requirements

Datum setting

The used datums must be measured and entered.

Tools

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



Note:

It is not allowed to change tool settings of used tools during program run.

Reference point

The reference point must be traversed in all axes.

Machine

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

Alarms

There must not be any alarms activated.



NC-Start

With this key, you switch from "JOG" mode into the operating mode "AUTO" and the NC program run will be started.

An EASY CYCLE program must be open in order to be able to start the NC program run. The file name of the currently openend EASY CYCLE program is displayed in the top right-hand corner of the simulation window.



NC-Reset

With this key, you switch from the "AUTO" operating mode into "JOG" mode and the NC program run will be aborted and reset into its original state.



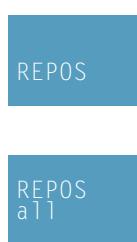
NC-Stop

With this key, the NC program run is stopped. The simulation can be continued with the "NC-Start" key.

Program start, Program stop



- Select a program for machining.
- Press the "NC-Start" key.
- Stop the program by means of "NC-Stop" and continue it with "NC-Start".
- Abort the program by means of "NC-Reset".



Repositioning

You require this function when you stop a running program with "NC-Stop" and traverse the axes with the handwheel or via jog. In order to be able to continue with "NC-Start", the starting point of the current element has to be traversed again using the soft keys "REPOS X", "REPOS Z" and "REPOS all".

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

Defective step motor system.
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.

I 2014-04

6022: CLAMPING DEVICE BOARD DEFECTIVE

The signal "clamping device clamped" is constantly released, although no command has been given.
Replace the board.

6024: MACHINE DOOR OPEN

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

6027: DOOR LIMIT SWITCH DEFECTIVE

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

6028: DOOR TIMEOUT

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

6030: NO PART CLAMPED

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

6040: TOOL TURRET INDEX FAILURE

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

6041: TOOL CHANGE TIMEOUT

Tool drum sticks (collision?), main drive not ready, fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6043-6046: TOOL DISK POSITION FAULT

Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.
The Z axis could have been slipped out of the toothings while the machine was switched off.
A running CNC program will be stopped.
Contact EMCO service.

6047: TOOL DISK UNLOCKED

Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.
A running CNC program will be interrupted.
Contact EMCO service.

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

Turn the drum into locking position manually
Change into MANUAL (JOG) mode.
Turn the key switch. Traverse the Z slide upwards, until the alarm disappears.

6048: DIVIDING TIME EXCEEDED

Dividing head stuck, insufficient pressured air supply, hardware defective.
Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED

see alarm 6048

6050: M25 AT RUNNING MAIN SPINDLE

Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY

Cause: pressure failure automatic door
automatic door sticks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN

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

6070: PRESSURE SWITCH FOR TANI MISSING

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

6071: DIVIDING DEVICE NOT READY

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

6072: VICE NOT READY

Attempt to start the spindle with an open vice or without clamped workpiece.
Vice sticks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.
Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY

Cause: locking switch defective
cabling defective
fuses defective

A running program will be aborted.

The 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

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 auxilliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxilliary 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 auxilliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.

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

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

7021: INITIALIZE TOOL TURRET

The tool turret operating was interrupted.

No traversing operation is possible.

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

7022: INITIALIZE TOOL TURRET !

see 7021

7023: WAITING TIME MAIN DRIVE!

The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

7038: LUBRICATION SYSTEM FAULT

The pressure switch is defective or gagged.

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

Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT

Not enough lubricant, the pressure switch is defective.

NC-Start is locked.

Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN

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

Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR

Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED

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

7050: NO PART CLAMPED

After switching on or after an the vice is neither at the open position nor at the closed position.
NC-Start is locked.
Traverse the vice manually on a valid end position.

7051: DIVIDING HEAD NOT LOCKED!

Either the dividing head is in an undefined position after the machine has been switched on, or the locking signal after a dividing process is missing.
Initiate the dividing process, check, respectively adjust the proximity switch for locking.

7054: VICE OPEN

Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6072 (vice not ready) will be released.
Remedy: Clamp

7055: OPEN TOOL CLAMPING SYSTEM

A tool is clamped in the main spindle and the control does not recognize the corresponding T number.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7056: SETTING DATA INCORRECT

An invalid tool number is stored in the setting data.
Delete the setting data in the machine directory xxxx.pls.

7057: TOOLHOLDER OCCUPIED

The clamped tool cannot be positioned in the tool turret since the position is occupied.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7058: RETRACTING THE AXES

The position of the tool turret arm cannot be clearly defined during the tool change.
Open the machine door, push the tool turret magazine backwards to the stop. Move the milling head in the JOG mode upwards to the Z reference switch and then traverse the reference point.

7087: MOTOR PROTECTION HYDRAULIC CLAMPING RELEASED!

Hydraulic motor is defective, stiff, circuit breaker is set incorrectly.
Replace motor or check circuit breaker and replace if necessary.

7090: ELECTRICAL CABINET OVERRIDE SWITCH ACTIVE

The cabinet door can only be opened when the key switch is switched on without raising an alarm.
Switch off key switch.

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

Concept MILL 250

EMCOMAT E160

EMCOMAT E200

EMCOMILL C40

EMCOMAT FB-450 / FB-600

6000: EMERGENCY OFF

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

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

Check fuses or contact EMCO service.

6001: PLC-CYCLE TIME EXCEEDING

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

6011: DRIVE Z-AXIS NOT READY

see 6010.

6012: DRIVE Z-AXIS NOT READY

see 6010.

6002: PLC - NO PROGRAM CHARGED

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

6013: MAIN DRIVE NOT READY

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

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

Check fuses or contact EMCO Service.

6003: PLC - NO DATA UNIT

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

6014: NO MAIN SPINDLE SPEED

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

The CNC program will be aborted, the auxiliary drives will be switched off.

6004: PLC - RAM MEMORY FAILURE

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

6015: NO DRIVEN TOOL SPINDLE SPEED

see 6014.

6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING

6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING

In the tool turret that can be coupled, the position of the coupling and uncoupling magnet is monitored by means of two proximity switches. It has to be made sure that the coupling is in the rear stop position so that the tool turret can get to the next tool position. Equally, during operation with driven tools the coupling has to be safe in the front stop position.

Check and adjust the cables, the magnet and the stop position proximity switches.

6005: K2 OR K3 NOT DE-ENERGIZED

Turn machine on/off. Defective security board.

6006 EMERGENCY-OFF RELAY K1 NOT DE-ENERGIZED

Turn machine on/off. Defective security board.

6007 SAFETY CIRCUIT FAULT

6008: MISSING CAN SUBSCRIBER

The PLC-CAN board is not identified by the control.

Check the interface cable and the power supply of the CAN board.

6009: SAFETY CIRCUIT FAULT

6010: DRIVE X-AXIS NOT READY

The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.

6018: AS SIGNALS, K4 OR K5 NOT DE-ENERGIZED

Turn machine on/off. Defective security board.

6019: POWER SUPPLY MODULE NOT READY

Turn machine on/off. Power supply module, defective axis controller 6020 AWZ drive failure turn machine on/off, defective axis controller.

6021: COLLET TIME OUT

During closing of the clamping device the pressure switch has not reacted within one second.

6022: CLAMPING DEVICE BOARD DEFECTIVE

The signal "clamping device clamped" is constantly released, even though no command has been given. Replace the board.

6023: COLLET PRESSURE MONITORING

The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

6024: MACHINE DOOR OPEN

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

6025: GEARBOX COVER NOT CLOSED

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

Close the cover to continue.

6026: MOTOR PROTECTION COOLANT PUMP RELEASED**6027: DOOR LIMIT SWITCH DEFECTIVE**

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

Contact EMCO service.

6028: DOOR TIMEOUT

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

6029: TAILSTOCK QUILL TIME EXCEED

The tailstock quill does not reach a final position within 10 seconds.

Adjust the control and the stop position proximity switches, or the tailstock quill is stuck.

6030: NO PART CLAMPED

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

6031: QUILL FAILURE**6032: TOOL CHANGE TIMEOUT**
see alarm 6041.**6033: TOOL TURRET SYNC ERROR**

Hardware defective.
Contact EMCO service.

6037: CHUCK TIMEOUT

The pressure switch does not react within one second when the clamping device is closed.

6039: CHUCK PRESSURE FAILURE

The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

6040: TOOL TURRET INDEX FAILURE

The tool turret is in no locked position, tool turret sensor board defective, cabling defective, fuse defective.

A running CNC program will be stopped.
Swivel the tool turret with the tool turret key, check fuses or contact EMCO service.

6041: TOOL CHANGE TIMEOUT

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

A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6042: TOOL TURRET OVERHEAT

Tool turret motor too hot.

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

6043: TOOL CHANGE TIMEOUT

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

A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6044: BRAKING RESISTANCE - MAIN DRIVE OVERLOADED

Reduce number of speed changes in the program.

6045: TOOL TURRET SYNC MISSING

Hardware defective.

Contact EMCO service.

6046: TOOL TURRET ENCODER FAULT

Fuse defective, hardware defective.

Check fuses or contact EMCO service.

6048: CHUCK NOT READY

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

Chuck sticks mechanically, insufficient pressured air supply, fuse defective, hardware defective.

Check fuses or contact EMCO service.

6049: COLLET NOT READY

see 6048

6050: M25 DURING SPINDLE ROTATION

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

6055: NO PART CLAMPED

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

6056: QUILL NOT READY

Attempt to start the spindle or to move an axis or to swivel the tool turret with undefined tailstock position.

Tailstock is locked mechanically (collision), insufficient pressured air supply, fuse defective, magnetic switch defective.

Check for collisions, check fuses or contact EMCO service.

6057: M20/M21 DURING SPINDLE ROTATION

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

6058: M25/M26 DURING QUILL FORWARD

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

6059: C-AXIS SWING IN TIMEOUT

C-axis does not swivel in within 4 seconds.

Reason: not sufficient air pressure, and/or mechanics stuck.

6060: C-AXIS INDEX FAILURE

When swivelling in the C-axis the limit switch does not respond.

Check pneumatics, mechanics and limit switch.

6064: AUTOMATIC DOOR NOT READY

Door sticks mechanically (collision), insufficient pressured air supply, limit switch defective, fuse defective.

Check for collisions, check fuses or contact EMCO service.

6065: LOADER MAGAZINE FAILURE

Loader not ready.

Check if the loader is switched on, correctly connected and ready for operation and/or disable loader (WinConfig).

6066: CLAMPING DEVICE FAILURE

No compressed air at the clamping device

Check pneumatics and position of the clamping device proximity detectors.

6067: NO COMPRESSED AIR

Turn the compressed air on, check the setting of the pressure switch.

6068: MAINDRIVE OVERTEMPERATURE**6070: LIMIT SWITCH TAILSTOCK SLEEVE ACTIVE**

Cause: The axis arrived in the tailstock sleeve.

Remedy: Drive the travel off the tailstock sleeve.

6071: LIMIT SWITCH X AXIS ACTIVE

Cause: The axis arrived to the end switch.

Remedy: Drive the axis off the end switch again.

6072: LIMIT SWITCH Z AXIS ACTIVE

see 6071

6073: CHUCK GUARD OPEN

Cause: The chuck guard is open.

Remedy: Close the chuck guard.

6074: NO FEEDBACK FROM USB-PLC

Turn machine on/off. Check cabling, defective USB board.

6075: AXIS LIMIT SWITCH TRIGGERED

see 6071

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.

6900 USBPLC not available

Cause: USB communication with the safety board could not be established.
Remedy: Switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6901 Error emergency-off relay USBPLC

Cause: USBPLC EMERGENCY-OFF relay error.
 Remedy: Switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6902 Error standstill monitoring X

Cause: Unauthorized movement of the X axis in the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6903 Error standstill monitoring Z

Cause: Unauthorized movement of the Z axis in the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6904 Error alive circuit PLC

Cause: Error in the connection (Watchdog) of the safety board with the PLC.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6906 Error overspeed spindle

Cause: The main spindle speed exceeds the maximum permissible value for the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6907 Error enable pulses ER-module

Cause: ACC-PLC did not shutdown the input/negative feeder-module.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6908 Error standstill monitoring main drive

Cause: Unexpecd warm up of the main spindle in the operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6909 Error main drive enable without spindle start

Cause: The release of the control unit of the main spindle was given by the ACC-PLC without the spindle-start key being pressed.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6910 Error standstill monitoring Y

Cause: Unauthorized movement of the Y axis in the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6911 Error standstill axes

Cause: Unauthorized movement of the axis in the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6912 Error overspeed axis

Cause: The feed of the axes exceeds the maximum permissible value for the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6913 Error overspeed X

Cause: The feed of the X axis exceeds the maximum permissible value for the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6914 Error overspeed Y

Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.
 Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6915 Error overspeed Z

Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.

Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6916 ERROR: X-INDUCTIVE PROXIMITY SWITCH DEFECT

Cause: No signal is delivered by X axis Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6917 ERROR: Y-INDUCTIVE PROXIMITY SWITCH DEFECT

Cause: No signal is delivered by Y axis Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6918 ERROR: Z-INDUCTIVE PROXIMITY SWITCH DEFECT

Cause: No signal is delivered by Z axis Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6919 ERROR: SPINDLE-INDUCTIVE PROXIMITY SWITCH DEFECT

Cause: No signal is delivered by main spindle Bero.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6920 INVERSION OF X-DIRECTION TOO LONG "1"

Cause: The change in direction of X axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6921 INVERSION OF Y-DIRECTION TOO LONG "1"

Cause: The change in direction of Y axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6922 INVERSION OF Z-DIRECTION TOO LONG "1"

Cause: The change in direction of Z axis was being sent to USBSPS for more than three seconds.

Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6923 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC

Cause: ACC-PLC and USBSPS receive different door status reports.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6924 ERROR ENABLE PULSES MAIN DRIVE

Cause: The pulse release on the main spindle control element was interrupted by the USBSPS, as the PLC did not shut it down in a timely fashion.

Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

6925 GRID PROTECTION ERROR!

Cause: Grid protection does not drop out in current operating state, or does not engage.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6926 MOTOR PROTECTION ERROR!

Cause: Motor protection drops out in current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6927 EMERGENCY OFF ACTIVE ERROR!

Cause: Emergency off button was pressed.

Remedy: Restart the machine.

6928 TOOL CHANGER SHUTDOWN MONITORING ERROR

Cause: Unauthorised tool changer movement in the current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6929 MACHINE DOOR CLOSING/LOCKING ERROR

Cause: State of the door lock not plausible or door closure unserviceable.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6930 BEROS MAIN SPINDLE PLAUSIBILITY ERROR

Cause: Beros main spindle signal different.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6931 MAIN DRIVE QUICK STOP FUNCTION PLAUSIBILITY ERROR

Cause: Main drive actuator does not confirm the quick stop function in the current operating state.

Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6988 USB-EXTENSION FOR ROBOTIK NOT AVAILABLE

Cause: The USB extension for robotics cannot be addressed by ACC.

Remedy: Contact EMCO.

7000: INVALID TOOL NUMBER PROGRAMMED

The tool position was programmed larger than 8. The CNC program will be stopped.

Interrupt program with RESET and correct the program.

7007: FEED HOLD

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

7016: SWITCH ON AUXILIARY DRIVES

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

7017: REFERENCE MACHINE

Approach the reference point.

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

7018: TURN KEY SWITCH

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

NC-Start is locked.

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

7019: PNEUMATIC LUBRICATION MONITORING!

Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE

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

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

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

7021: TOOL TURRET NOT LOCKED

The tool turret operating was interrupted.

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

7022: COLLECTION DEVICE MONITORING

Time exceed of the swivelling movement.

Check the pneumatics, respectively whether the mechanical system is jammed (possibly a work-piece is jammed).

7023: ADJUST PRESSURE SWITCH !

During opening and closing of the clamping device the pressure switch has to turn off and on once.

Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !

When the clamping device is open and the position stop control is active, the respective proximity switch has to feed back that the clamping device is "Open".

Check and adjust the clamping device proximity switch, check the cables.

7025 WAITING TIME MAIN DRIVE !

The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

7026 PROTECTION MAIN MOTOR FAN RELEASER!

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.

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.

Inputunit alarms 1700 - 1899

These alarms and messages are raised by the control keyboard.

1701 Error in RS232

Cause: Serial port settings are invalid or the connection to the serial keyboard were interrupted.

Remedy: Check the settings of the serial interface and/or turn keyboard off/on and check the control cable connection.

1703 Ext. keyboard not available

Cause: Connection with the external keyboard can not be made.

Remedy: Check the settings of the external keyboard and/or check the cable connection.

1704 Ext. keyboard: checksum error

Cause: Error in the transmission.

Remedy: The connection to the keyboard is automatically restored. If this fails, turn off or on the keyboard.

1705 Ext. keyboard: general error

Cause: The attached keyboard reported an error.

Remedy: Plug the keyboard off and on again. Contact EMCO Customer Service if the error occurs on several occasions.

1706 General USB error

Cause: Error in the USB communication.

Remedy: Plug the keyboard off and on again. Contact EMCO Customer Service if the error occurs on several occasions

1707 Ext. Keyboard: no LEDs

Cause: Fehlerhaftes LED-Kommando wurde an die Tastatur gesandt.

Remedy: EMCO-Service kontaktieren.

1708 Ext. Keyboard: unknown command

Cause: Unknown command was sent to the keyboard.

Remedy: Contact EMCO Customer Service

1710 Installation of Easy2control is damaged!

Cause: Incorrect installation of Easy2control

Remedy: Reinstall software and/or contact EMCO Customer Service

1711 Initialization of Easy2Control failed!

Cause: Configuration file onscreen.ini for Easy2control is missing.

Remedy: Reinstall software and/or contact EMCO Customer Service.

1712 USB-Dongle for Easy2control could not be found!

Cause: USB-Dongle for Easy2control is not connected. Easy2control is displayed but can not be operated.

Remedy: Connect USB-Dongle for Easy2control.

1801 Keytable not found!

Cause: The file with the keytable couldn't be found.

Remedy: Reinstall software and/or contact EMCO Customer Service.

1802 Connection to keyboard lost

Cause: Connection to the serial keyboard was interrupted.

Remedy: Turn keyboard off/on and check the cable connection.

Axis Controller Alarms

8000 Fatal Error AC**8100 Fatal init error AC**

Cause: Internal error

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

8101 Fatal init error AC

see 8101.

8102 Fatal init error AC

see 8101.

8103 Fatal init error AC

see 8101.

8104 Fatal system error AC

see 8101.

8105 Fatal init error AC

see 8101.

8106 No PC-COM card found

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

Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working

see 8106.

8108 Fatal error on PC-COM card

see 8106.

8109 Fatal error on PC-COM card

see 8106.

8110 PC-COM init message missing

Cause: Internal error

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

8111 Wrong configuration of PC-COM

see 8110.

8113 Invalid data (pccom.hex)

see 8110.

8114 Programming error on PC-COM

see 8110.

8115 PC-COM packet acknowledge missing

see 8110.

8116 PC-COM startup error

see 8110.

8117 Fatal init data error (pccom.hex)

see 8110.

8118 Fatal init error AC

see 8110, ev. insufficient RAM memory

8119 PC interrupt no. not valid

Cause: The PC interrupt number can not be used.

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

8120 PC interrupt no. unmaskable

see 8119

8121 Invalid command to PC-COM

Cause: Internal error or defective cable

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

8122 Internal AC mailbox overrun

Cause: Internal error

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

8123 Open error on record file

Cause: Internal error

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

8124 Write error on record file

Cause: Internal error

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

8125 Invalid memory for record buffer

Cause: Insufficient RAM, record time exceeding.

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

8126 AC Interpolation overrun

Cause: Ev. insufficient computer performance.

Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

8127 Insufficient memory

Cause: Insufficient RAM

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

8128 Invalid message to AC

Cause: Internal error

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

8129 Invalid MSD data - axisconfig.

see 8128.

8130 Internal init error AC

see 8128.

8130 Internal init error AC

see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified

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

Remedy: Program correction.

8140 Maschine (ACIF) not responding

Cause: Machine off or not connected.

Remedy: Switch on machine or connect.

8141 Internal PC-COM error

Cause: Internal error

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

8142 ACIF Program error

Cause: Internal error

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

8143 ACIF packet acknowledge missing

see 8142.

8144 ACIF startup error

see 8142.

8145 Fatal init data error (acif.hex)

see 8142.

8146 Multiple request for axis

see 8142.

8147 Invalid PC-COM state (DPRAM)

see 8142.

8148 Invalid PC-COM command (CNo)

see 8142.

8149 Invalid PC-COM command (Len)

see 8142.

8150 Fatal ACIF error

see 8142.

8151 AC Init Error (missing RPG file)

see 8142.

8152 AC Init Error (RPG file format)

see 8142.

8153 FPGA program timeout on ACIF

see 8142.

8154 Invalid Command to PC-COM

see 8142.

8155 Invalid FPGA packet acknowledge

see 8142 or hardware error on ACIF board (contact EMCO Service).

8156 Sync within 1.5 revol. not found

see 8142 or Bero hardware error (contact EMCO Service).

8157 Data record done

see 8142.

8158 Bero width too large (referencing)

see 8142 or Bero hardware error (contact EMCO Service).

8159 Function not implemented

Bedeutung: In normal operation this function can not be executed

8160 Axis synchronization lost axis 3..7

Cause: Axis spins or slide is locked, axis synchronisation was lost

Remedy: Approach reference point

8161 X-Axis synchronization lost

Step loss of the step motor. Causes:

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

8162 Y-Axis synchronization lost

see 8161

8163 Z-Axis synchronization lost

see 8161

8164 Software limit switch max axis 3..7

Cause: Axis is at traverse area end

Remedy: Retract axis

8168 Software limit overtravel axis 3..7

Cause: Axis is at traverse area end

Remedy: Retract axis

8172 Communication error to machine

Cause: Internal error

Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.
Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running

Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

8174 INC not allowed

Cause: At the moment the axis is in motion.

Remedy: Wait until the axis stops and then traverse the axis.

8175 MSD file could not be opened

Cause: Internal error

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

8176 PLS file could not be opened

see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

8183 Gear too high
Cause: The selected gear step is not allowed at the machine.

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message
see 8175.

8191 Invalid jog feed unit
Cause: The machine does not support the rotation feed in the JOG operating mode.
Remedy: Order a software update from EMCO.

8192 Invalid axis in command
see 8175.

8193 Fatal PLC error
see 8175.

8194 Thread without length
Cause: The programmed target coordinates are identical to the starting coordinates.
Remedy: Correct the target coordinates.

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

8196 Too manny axis for thread
Remedy: Program max. 2 axes for thread.

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

8198 Internal error (to manny threads)
see 8175.

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

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

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

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

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8212 Rotation axis not allowed
see 8199.

8213 Circle and rotation axis can't be interpolated

8214 Thread and rotation axis can't be interpolated

8215 Invalid state
see 8199.

8216 No rotation axis for rotation axis switch
see 8199.

8217 Axis type not valid!
Cause: Switching during the rotary axis operating mode when the spindle is running.
Remedy: Stop the spindle and switch over to the rotary axis operating mode.

8218 Referencing round axis without selected round axis!
see 8199.

8219 Thread not allowed without spindle encoder!
Cause: Thread cutting, respectively tapping is only possible with spindles with encoders.

8220 Buffer length exceeded in PC send message!
see 8199.

8221 Spindle release although axis is no spindle!
see 8199.

8222 New master spindle is not valid

Cause: The indicated master spindle is not valid when switching over to the master spindle.

Remedy: Correct the spindle number.

8224 Invalid stop mode

see 8199.

8225 Invalid parameter for BC_MOVE_TO_IO!

Cause: The machine is not configurated for touch probes. A traversing movement with rotary axis is not allowed during touch probe operating mode.

Remedy: Remove the rotary axis movement from the traversing movement.

8226 Rotary axis switch not valid (MSD data)!

Cause: The indicated spindle does not have a rotary axis.

8228 Rotary axis switch not allowed while axis move!

Cause: The rotary axis has moved during switching over to the spindle operating mode.

Remedy: Stop the rotary axis before switching.

8229 Spindle on not allowed while rotary axis is active!**8230 Program start not allowed due to active spindle rotation axis!****8231 Axis configuration (MSD) for TRANSMIT not valid!**

Cause: Transmit is not possible at this machine.

8232 Axis configuration (MSD) for TRACYL not valid!

Cause: Tracyl is not possible at this machine.

8233 Axis not available while TRANSMIT/TRACYL is active!

Cause: Programming of the rotary axis is not allowed during Transmit/ Tracyl.

8234 Axis control grant removed by PLC while axis interpolates!

Cause: Internal error

Remedy: Delete error with reset and inform EMCO.

8235 Interpolation invalid while axis control grant is off by PLC!

see 8234.

8236 TRANSMIT/TRACYL activated while axis or spindle moves!

see 8234.

8237 Motion through pole in TRANSMIT!

Cause: It is not allowed to move through the coordinates X0 Y0 inTransmit.

Remedy: Alter the traversing movement.

8238 Speed limit in TRANSMIT exceeded!

Cause: The traversing movement gets too close to the coordinates X0 Y0. In order to observe the programmed feed rate, the maximum speed of the rotary axis would have to be exceeded.

Remedy: Reduce the feed rate. Set the value of the C-axis feed limitation in WinConfig, machine data settings / general machine data/ to 0.2. Thus, the feed rate will be automatically reduced near the coordinates X0 Y0.

The distance to the center is calculated with the following formula:

for CT155/CT325/CT450:

$F[\text{mm/min}] * 0.0016 = \text{distance} [\text{mm}]$

for CT250:

$F[\text{mm/min}] * 0.00016 = \text{distance} [\text{mm}]$

This applies for rapid traverse in transmit:

CT155/250/325: 4200 mm/min

CT450: 3,500 mm/min

8239 DAU exceeded 10V limit!

Cause: Internal error

Remedy: Start the software again or install it anew. Report the error to EMCO.

8240 Function not valid during active transformation (TRANSMIT/TRACYL)!

Cause: The Jog and INC operating mode are not possible during Transmit in X/C and during Tracyl in the rotary axis.

8241 TRANSMIT not enabled (MSD)!

Cause: Transmit is not possible at this machine.

8242 TRACYL not enabled (MSD)!

Cause: Tracyl is not possible at this machine.

8243 Round axis invalid during active transformation!

Cause: It is not allowed to program the rotary axis during Transmit/Tracyl.

8245 TRACYL radius = 0!

Cause: When selecting Tracyl, a radius of 0 was used.

Remedy: Correct the radius.

8246 Offset alignment not valid for this state!

see 8239.

8247 Offset alignment: MSD file write protected!**8248 Cyclic supervision failed!**

Cause: The communication with the machine keyboard is interrupted.

Remedy: Start the software again or install it anew. Report the error to EMCO.

8249 Axis motion check alarm!

see 8239

8250 Spindle must be rotation axis !

see 8239

8251 Lead for G331/G332 missing !

Cause: The threading pitch is missing or the starting coordinates are identical to the target coordinates.

Remedy: Program the threading pitch.
Correct the target coordinates.

8252 Multiple or no linear axis programmed for G331/G332 !

Remedy: Program exactly one linear axis.

8253 Speed value for G331/G332 and G96 missing !

Cause: No cutting speed has been programmed.

Remedy: Program the cutting speed.

8254 Value for thread starting point offset not valid!

Cause: The thread starting point offset is not within the range of 0 to 360°.

Remedy: Correct the thread starting point offset.

8255 Reference point not in valid software limits!

Cause: The reference point has been defined outside the software limit switches.

Remedy: Correct the reference points in WinConfig.

8256 Spindle speed too low while executing G331/G332!

Cause: During tapping the spindle speed has decreased. Perhaps the incorrect threading pitch was used or the core drilling is not correct.

Remedy: Correct the threading pitch. Adapt the diameter to the core drilling.

8257 Real Time Module not active or PCI card not found!

Cause: ACC could not be started correctly or the PCI card in the ACC was not recognized.

Remedy: Report the error to EMCO.

8258 Error allocating Linux data!

see 8239.

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.

8261 Missing thread in sequence !

Cause: A successive thread has not been programmed for a thread in sequence, the number has to be in accordance with the SETTHREADCOUNT () that has been defined before.

Remedy: Correct the number of threads in the thread in sequence and add a thread.

8262 Reference marks are not close enough !

Cause: The settings of the linear scale have been changed or the linear scale is defective.

Remedy: Correct the settings. Contact EMCO.

8263 Reference marks are too close together!

see 8262.

8265 No or wrong axis in axis switch command!

Cause: Internal error.

Remedy: Please contact the EMCO after-sales service.

8266 Invalid tool

Cause: Programmed tool is not set in magazine.

Remedy: Correct tool number and/or load tool in magazine.

8267 Speed difference to high

Cause: Die Soll- und Istgeschwindigkeit der Achse weichen zu stark voneinander ab.

Remedy: Run the program again with reduced feed. If this does not remedy the problem, contact EMCO.

8269 USBSPS and ACC speed values or override are different

Cause: USBSPS and ACC have different rotations saved.

Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

8270 Reference switch defective

Cause: The reference switch did not switch within the specified range.

Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

8271 Tool load in locked place not possible

Cause: There was an attempt to swing a tool into a locked place in the magazine.

Remedy: Choose a free, unlocked place in the magazine and then swing the tool into the magazine.

8272 Old PLC version, update necessary

Cause: The PLC version is too old to fully support randomised tool management.

Remedy: Update the PLC.

8273 Spindle overload

Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).

Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

8274 Define tool before loading

Cause: The tool must be defined in the tool list before it is possible to transfer the tool into the spindle.

Remedy: Create the tool in the tool list, then load.

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.

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

Axis Controller Messages

8700 Execute REPOS in all axes before program start

Cause: After the program was stopped, the axes were manipulated with the hand wheel or with the jog keys and then a restart of the program was attempted.

Remedy: Before starting the program again, one should reposition the axes along the contour by executing "REPOS".

8701 No NCStop during offset align

Cause: The machine is currently executing an automatic offset adjustment. NC stop is not possible at this time.

Remedy: Wait until the offset adjustment is finished and then stop the program with NC stop.

8702 No NCStop during positioning after block search

Cause: The machine is currently finishing the block search operation and then it starts to go back to the last programmed position. No NC stop is possible in the meantime.

Remedy: Wait until positioning is finished and then stop the program with NC stop.

8703 Data record done

The recording of data is finished and the file record.acp has been copied to the installation folder.

8705 Feed-override missing, REPOS will not be executed

Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).

Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

8706 Tool sorting active

Cause: The tools were resorted during randomised tool management in order to enable non-randomised operation (tool 1 to place 1, tool 2 to place 2, etc.).

Remedy: Wait till the sorting is finished. The message will be deleted automatically by the control system.

8707 New control - please check tool table

Cause: The control system was changed when randomised tool management was active.

Remedy: Check the tool or place table to cancel the alarm.

8708 Switch off auxiliary drives for shutdown

Cause: There was an attempt to shut down the control system although the auxiliary drives are still switched on.

Remedy: Switch off the auxiliary drives and then shut down the control system.

8709 Insert tool in spindle for loading

Cause: During loading a tool must be physically available in the spindle.

Remedy: Clamp tool in the spindle. The message disappears.

Control alarms 2000 - 5999

The alarms are released by the software.

Fagor 8055 TC/MC
Heidenhain TNC 426
CAMConcept
EASY CYCLE
Sinumerik for OPERATE

2200 Syntax error in line %s, column %s

Cause: Syntax error in the program code.

2001 Circle end point invalid

Cause: The start-middle point and middle-end point distances differ by more than 3 µm.

Remedy: Correct circular arc point.

2300 tracyl without corresponding round-axis invalid

Cause: Maybe the machine has no rotary axis.

3000 Traverse feed axis manually to position %s

Remedy: Move the axis manually to the required position.

3001 Tool T.. change!

Cause: A new tool was programmed in the NC program.

Remedy: Clamp the required tool in the machine.

4001 slot width too small

Cause: The tool radius is too large for the slot to be milled.

4002 slot length to small

Cause: The slot length is too small for the slot to be milled.

4003 length equal zero

Cause: Pocket length, pocket width, stud length, stud width are zero.

4004 slot width too big

Cause: The programmed slot width is larger than the slot length.

4005 depth equal zero

Cause: No machining takes place since no effective cutting depth has been defined.

4006 corner radius too big

Cause: The corner radius is too large for the size of the pocket.

4007 diameter too big

Cause: The remaining material (nominal diameter - diameter of the prebore) /2 is larger than the tool diameter.

4008 diameter too small

Cause: The tool diameter is too large for the intended bore.

Remedy: Enlarge the nominal diameter and use a smaller milling cutter.

4009 length too small

Cause: Width and length must be larger than the double tool radius.

4010 diameter less equal zero

Cause: The pocket diameter, the stud diameter, etc. must not be zero.

4011 blank diameter too big

Cause: The diameter of the machined pocket must be larger than the diameter of the premachined pocket.

4012 blank diameter too small

Cause: The diameter of the machined stud must be smaller than the diameter of the premachined stud.

4013 start angle equal to end angle

Cause: Start angle and end angle for hole pattern are identical.

4014 tool radius 0 not permitted

Cause: Tool radius zero is not permitted.

Remedy: Select a valid tool.

4015 no outer contour defined

Cause: The contour file indicated in the cycle was not found.

4017 tool radius too big

Cause: For the programmed machining, a tool being too large was selected. Therefore, machining is not possible.

4018 allowance must not be 0

Cause: There were programmed finishing operations without finishing offset.

4019 too many iterations

Cause: The contour definitions are too complex for the roughing-out cycle.

Remedy: Simplify the contour.

4020 illegal radian correction

Cause: An error has occurred during the programming of the radius compensation.

Remedy: Check the cycle parameters.

4021 can't calculate parallel contour

Cause: The control was not able to calculate the tool radius compensation.

Remedy: Check the programmed contour for plausibility. Maybe contact EMCO.

4022 illegal contour definition

Cause: The programmed contour is not suited to the selected machining.

Remedy: Check the programmed contour.

4024 no contour definition

Cause: The contour file being defined in the cycle has not been found.

4025 internal calculation error

Cause: An unexpected error has occurred during calculation of the cycle movements.

Remedy: Please inform the EMCO after-sales service.

4026 allowance too big

Cause: A part of the finishing offset (for several finishing passes) is larger than the total finishing offset.

Remedy: Correct the finishing offsets.

4028 pitch 0 not permitted

Cause: The thread was programmed with pitch zero.

4029 undefined working mode

Cause: Internal error (invalid machining type for the thread).

4030 function not yet supported

Cause: Roughing out with pockets is not implemented yet.

Remedy: Please inform the EMCO after-sales service.

4031 value not permitted

Cause: An invalid retracting direction was programmed during inside turning.

4032 plunging must be defined

Cause: For the programmed cycle no cutting depth has been programmed.

4033 radius/chamfer too big

Cause: The radius, respectively the chamfer, cannot be inserted in the programmed contour.

Remedy: Reduce the radius, respectively the chamfer.

4034 diameter too big

Cause: The programmed starting point and the machining diameter are contradictory.

4035 diameter too small

Cause: The programmed starting point and the machining diameter are contradictory.

4036 unknown working direction

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

4037 unknown working type

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

4038 unknown sub cycle

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

4039 rounding not possible

Cause: The programmed radius contradicts the rest of the cycle parameters.

4042 illegal tool width

Cause: The tool width for the cutting-off cycle must be defined.

4043 groove width too small

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

4044 distance not defined

Cause: The distance for the multiple grooving cycle must not be zero.

4045 illegal allowance type

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

4046 invalid speed

Cause: The spindle speed must be nonzero.

4047 invalid end point

Cause: The programmed end point contradicts the rest of the cycle definition.

4048 tool cut width too small

Cause: The cutting edge is too small for the programmed cutting depth.

4050 invalid distance

Cause: The hole patterns do not tally with the selected distance.

4052 working pattern not possible

Cause: Error in the definition of the hole pattern. The number of bores is contradictory.

4053 invalid start point

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

4055 illegal working direction

Cause: The machining direction is contradictory to the rest of the cycle definition.

4057 plunging angle less equal zero

Cause: The plunging angle must be between 0 and 90 degree.

4058 chamfer too large

Cause: The programmed chamfer is too large for the pocket cycle.

4062 radius/chamfer too small

Cause: The radius, respectively the chamfer, cannot be machined with the current tool radius.

4066 invalid mill step

Cause: The mill step must be greater than zero.

4069 invalid angle

Cause: An angle of zero degree is not permitted.

4072 plunging too small

Cause: For the cycle, a cutting depth has been selected that leads to extra-long machining time.

4073 invalid clearance angle

Cause: The clearance angle indicated for the tool cannot be machined.

Remedy: Correct the clearance angle for the tool.

4074 contour-file not found

Cause: The contour file indicated in the cycle has not been found.

Remedy: Please select the contour file for the cycle.

4075 not machinable with selected tool

Cause: The tool is too wide for the programmed groove.

4076 reciprocating plunge cut impossible

(initial move too short)

Cause: The first movement of the contour is shorter than the double tool radius and cannot be therefore used for the swinging delivery.

Remedy: Extend the first movement of the contour.

4077 wrong tool type in grooving or cut-off cycle

Cause: The wrong tool type was used in the cutting cycle.

Remedy: Use only grooving and punch tools in the cutting cycles.

4078 radius of helix too small

Cause: The pitch of the helix is ≤ 0 .

Remedy: Program the radius > 0 .

4079 pitch of helix too small

Cause: The radius of the helix is ≤ 0 .

Remedy: Program the pitch > 0 .

4080 radius of helix or tool too large

Cause: The helical approach cannot be executed with the selected data for the helix and the current tool radius without a contour breach.

Remedy: Use a tool with a smaller radius or reduce the radius of the helix.

4200 leaving movement is missing

Cause: No movement after the tool radius compensation was deactivated in the current plane.

Remedy: Insert the departing movement in the current plane after having deactivated the tool radius compensation.

4201 TPC off missing

Cause: The tool radius compensation has not been deactivated.

Remedy: Deactivate the tool radius compensation.

4202 TPC requires at least three movements

Cause: The tool radius compensation requires at least 3 movements in the current plane in order to calculate the tool radius compensation.

4203 approaching movement not possible

Cause: It was not possible to calculate the approaching movement.

4205 leaving movement not possible

Cause: It was not possible to calculate the departing movement.

4208 TPC curve could not be calculated

Cause: It was not possible to calculate the tool radius compensation for the programmed contour.

4209 switching the plane is not allowed when TPC is switched on

Cause: The programmed plane must not be changed during the tool radius compensation.

Remedy: Remove the change of planes during the tool radius compensation.

4210 tool path compensation already activated

Cause: G41 is active and G42 was programmed or G42 is active and G41 was programmed.

Remedy: Switch tool radius compensation off with G40 before programming the radius compensation again.

4211 Bottleneck detected

Cause: In the radius correction calculation some parts of the contour were omitted, as too large a milling cutter tool was used.

Remedy: Use a smaller milling cutting tool to process the contour completely.

4212 Infeed has been programmed twice during approach

Cause: After the approach movement a second infeed has been programmed, without previously moving to the work plane.

Remedy: First program a movement to the work plane before programming a second infeed.

5000 drill manually now**5001 contour has been adjusted to the programmed clearance angle**

Cause: The programmed contour was adapted to the programmed clearance angle. Maybe there will remain rest material that cannot be machined with this tool.

5500 3D simulation: Internal error

Cause: Internal error inside the 3D simulation.

Remedy: Restart the software or, if necessary, report the error to EMCO customer service.

5502 3D simulation: Tool place invalid

Cause: Tool place unavailable on the machine used.

Remedy: Correct tool call-up.

**5503 3D simulation: Chuck invalid owing to
the unmachined part definition**

Cause: The distance from the front of the unmachined part is > the unmachined part length.

Remedy: Change the distance.

5505 3D simulation: Unmachined part definition invalid

Cause: Implausibility in the unmachined part geometry (e.g. expansion in one axis ≤ 0 , inside diameter > outside diameter, unmachined part contour not closed, etc.).

Remedy: Correct unmachined part geometry.

5506 3D simulation: STL chuck file has auto-overcuts

Cause: Error in the chuck description.

Remedy: Correct file.

5507 3D simulation: Pole transit on TRANS-MIT!

Cause: Travel comes too close to the X0 Y0 coordinates.

Remedy: Change travel.

X: EMConfig

Note:

The settings which are available in EMConfig are depending on the machine and the control that is used.



General

EMConfig is a configuration software for WinNC. EMConfig helps you to alter the settings of WinNC.

The most important settings are:

- Control language
- System of measurement mm - inch
- Activate accessories
- Selection of interface for control keyboard

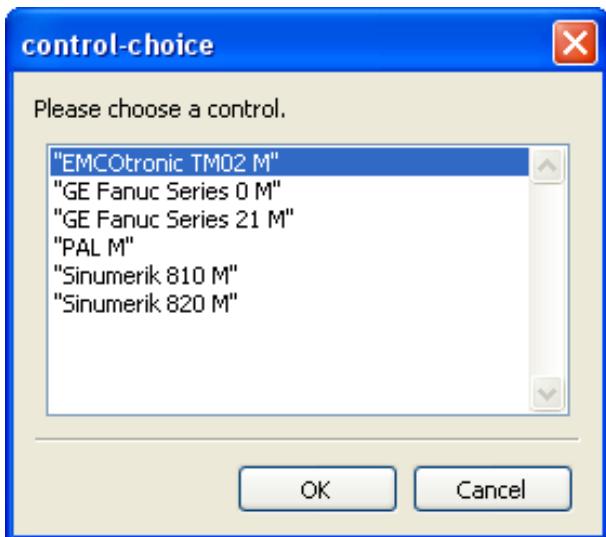
Using EMConfig you can also activate diagnostic functions in case of troubles - that way you get help immediately.



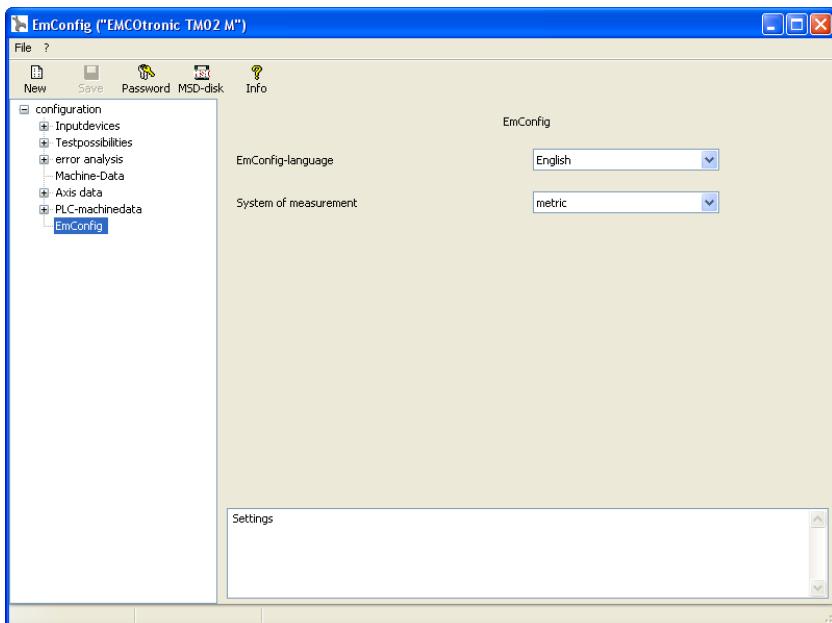
Safety-related parameters are protected by a password. They can only be activated by set-up technicians or by technical support representatives.



Icon for EMConfig



Selection box for control type



Change the language of EMConfig

How to start EMConfig

Open EMConfig.

In case several control types are installed, a selection box will appear on the screen.

Select the required control type and click OK.

The following settings are only valid for the selected control type.

The window for EMConfig appears on the screen.

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

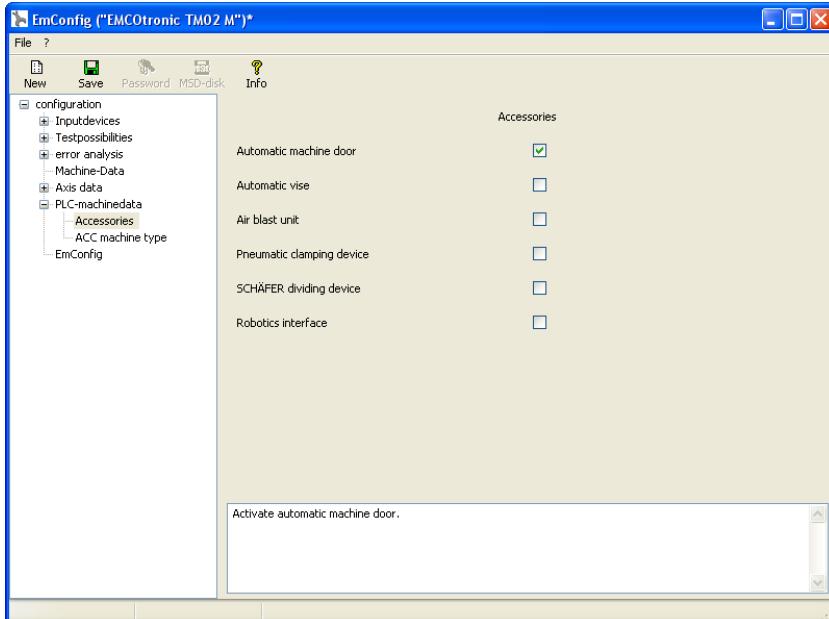


Note:

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

How to activate accessories

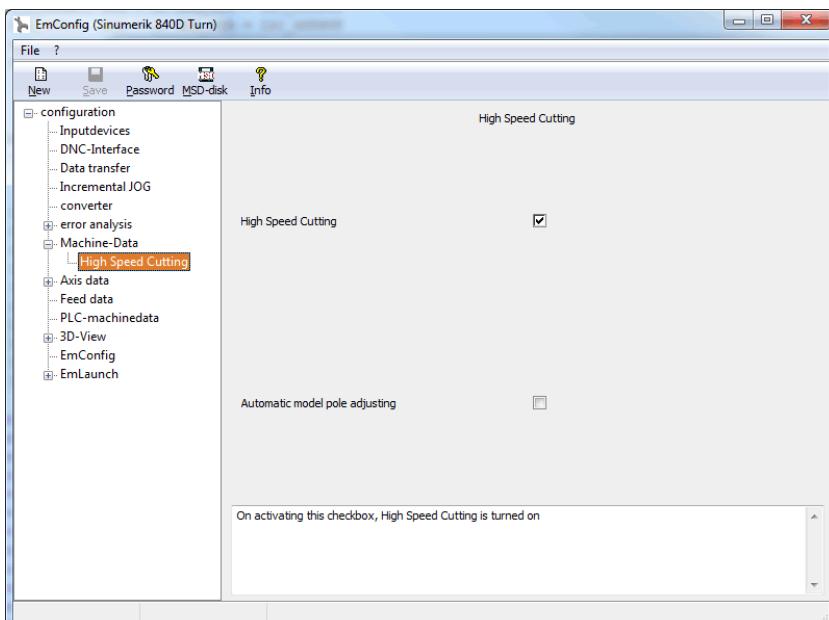
When you install accessories on your machine, you need to activate them here.



Activate accessories

High Speed Cutting

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



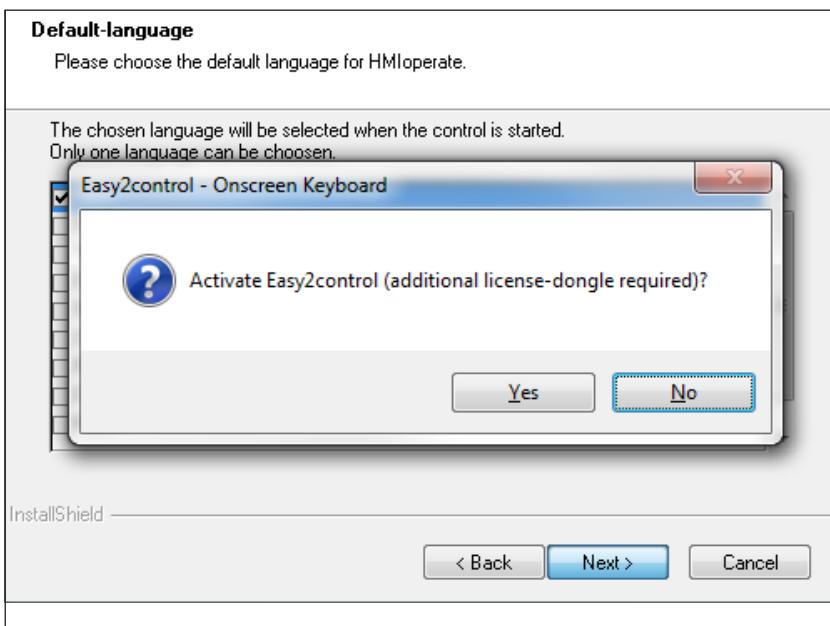
Activate High Speed Cutting

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

If the feed is set up to higher than 2500 mm/min, it is automatically reset to the normal operating mode and sanded and rounded edges are created.

Easy2control on screen operation

Installation and activation using the example of WinNC for Sinumerik Operate.

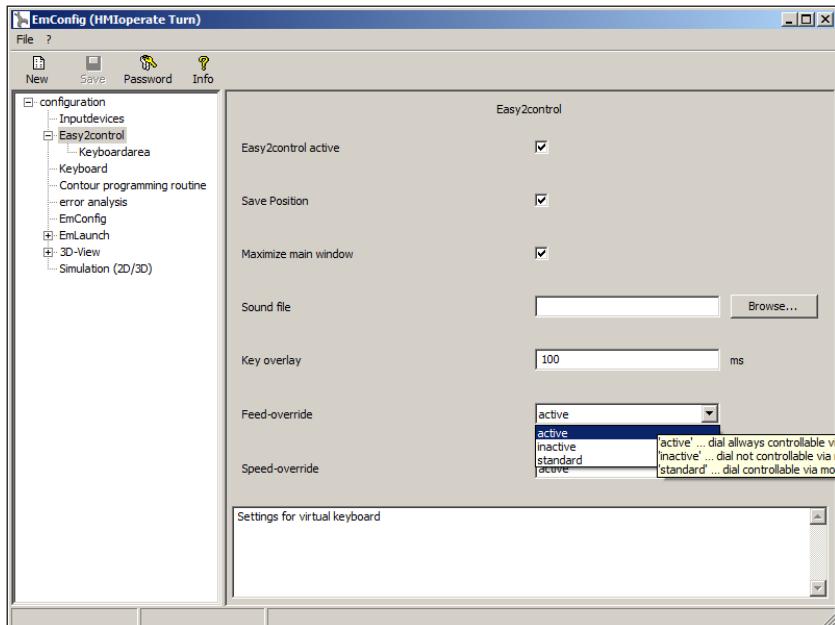


Activating Easy2control

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:

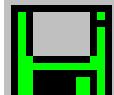
- **Aktive:** dial always controllable via mouse/touchscreen (even with available hardware-version).
- **Inaktive:** dial not controllable via mouse/touchscreen.
- **Standard:** dial controllable via mouse/touchscreen when no hardware-version is available.

Note:

If Easy2control is used without the hardware-dongle, the controls are deactivated and an appropriate alarm is output by the controller. However, the virtual keyboard is displayed completely.

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.

Z: Software Installation Windows

System prerequisites

Machine with integrated control PC:

- All Concept machines
- Machines that were converted to ACC
- MOC with Windows XP SP3 or higher (32 / 64 Bit)

Machines with included control PC and programming stations:

- PC 1000 Mhz
- Windows XP SP2 or higher (32 / 64 Bit)
- Working memory min. 256 MB RAM
- free hard drive space 400 MB
- Programming station: 1*USB, machine version: 2*USB
- TCP/IP-capable network card for machine version)

Software installation

- Start Windows XP SP3 or higher
- 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
- SINUMERIK 810D/840D T and M
- HEIDENHAIN TNC 426
- FANUC Series 0-TC and 0-MC
- FANUC Series 21 TB and MB
- FAGOR 8055 TC and MC
- CAMConcept T and M
- EMCO EASY CYCLE T and M (except machine licence)

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:

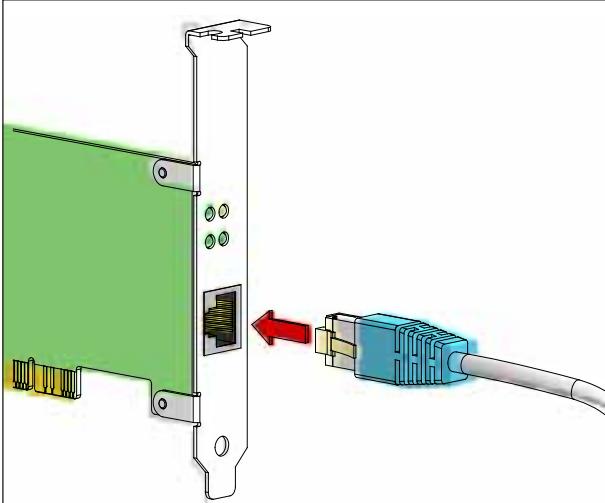
- Demo licence:
The demo licence is valid for 30 days after the first use. 5 days before the demo licence expires, you can enter another valid licence key (see licence manager)
- Programming station:
Programming and operation of the appropriate CNC control type is simulated by WinNC on your PC.
- Single user licence:
Authorizes to external programming of CNC-controlled machine tools on one PC workstation (machine-independent).
- 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 (machine-independent).
- Educational licence version:
Is a time-limited multi-liscence especially for schools and educational institutes.
- Machine licence:
This licence allows to directly operate a PC-controlled machine (PC TURN, Concept TURN, PC MILL, Concept MILL) of WinNC as if it was operated by an ordinary CNC control.

**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

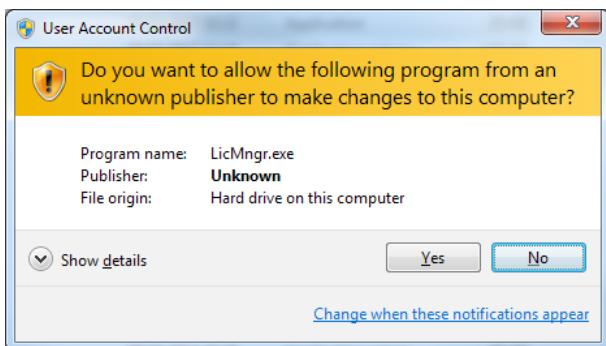
- 1 Switch off auxiliary drive with AUX OFF. Only for machine places, not for programming stations.
- 2 By simultaneously pressing these buttons WinNC for Sinumerik Operate will be terminated specifically. This corresponds to Alt+F4 on the PC keyboard.



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).

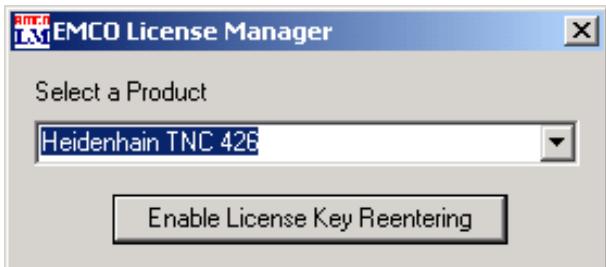
The **EMCO Licence Manager** (see picture on the bottom on the left) enables the input of further new licence keys. For this purpose select the new product in the selection window and confirm the input.

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.



EMCO Licence Manager