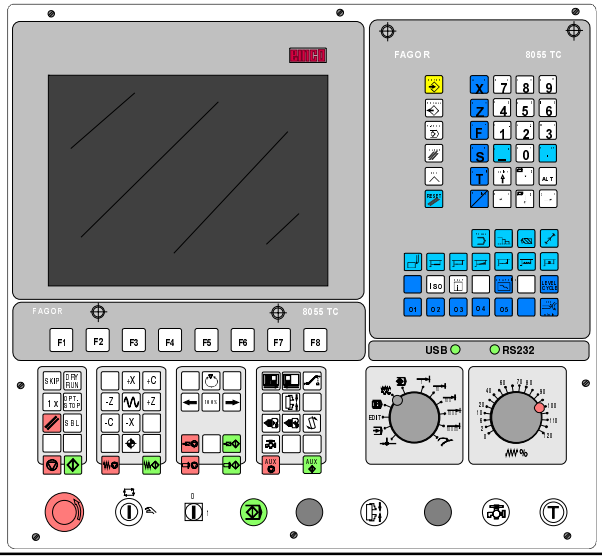
EMCO WinNC Fagor 8055 TC

Software Description/ Software version from 1.11



Software Description
EMCO WinNC
Fagor 8055 TC Turning

Ref.No. EN 1819 Edition A2004-10

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

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

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

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

Internet: www.emco.at

E-Mail: service@emco.at

emco
industrial training systems

Notice

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

Preface

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

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

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

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

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

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

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

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

Contents

A: Fundamentals

Reference Points of the EMCO 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

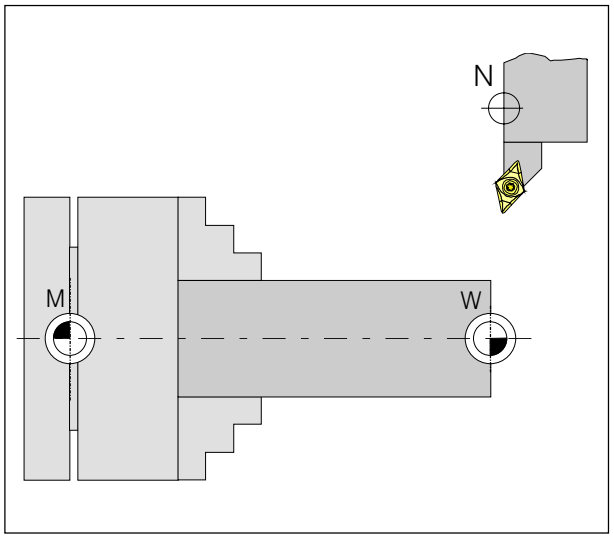
A position in the machine working area which is determined exactly by limit switches. The slide positions are reported to the control by the slides approaching the „R“.
Required after every power failure.

N = Tool mount reference point

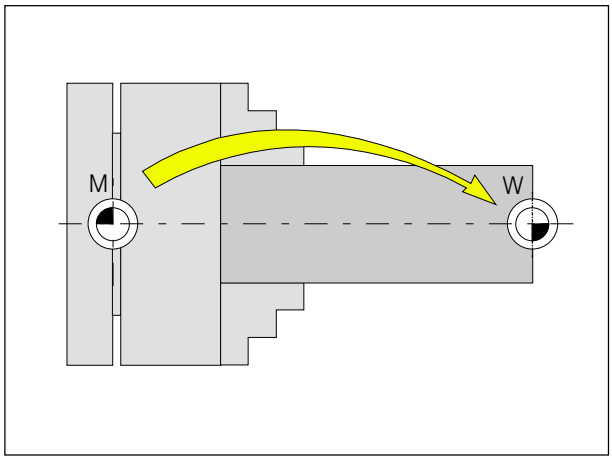
Starting point for the measurement of the tools. „N“ lies at a suitable point on the tool holder system and is established by the machine manufacturer.

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.



Reference points in the working area



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

Definition of the axis value in Z-direction

Zero Offset

For EMCO lathes the machine zero point "M" is on the turning axis on the face of the spindle flange. This position is unsuitable as a starting point for dimensioning. With the so-called zero offset the coordinate system can be moved to a suitable point in the working area of the machine.

The zero offset is carried out axis by axis.

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

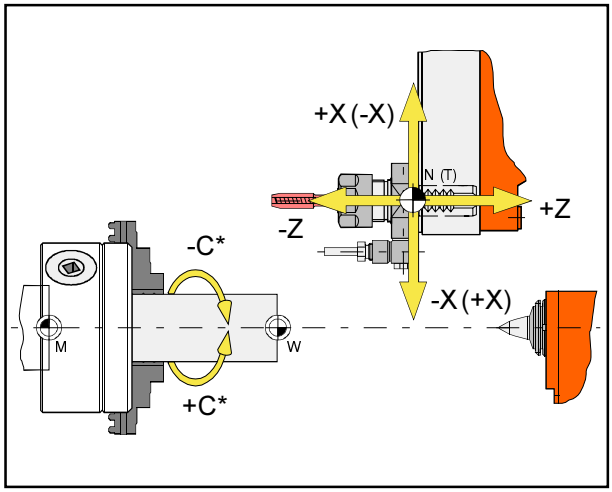
Example:
The workpiece zero point should be defined at the face of a turning part.
Execution: Use a tool to scratch at the face of a workpiece.

Press Z.

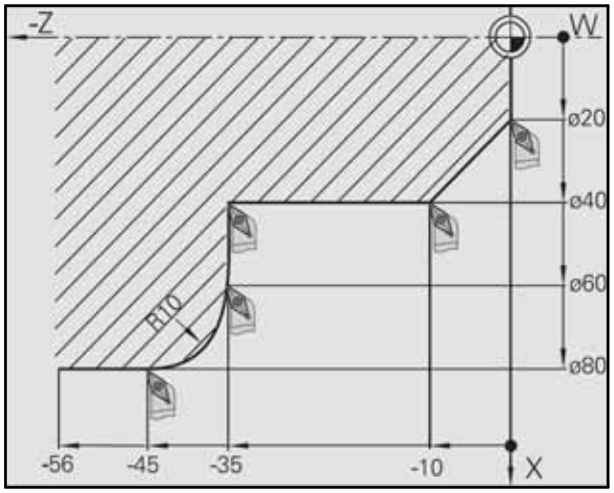
Enter the value "0" and confirm with ENTER.

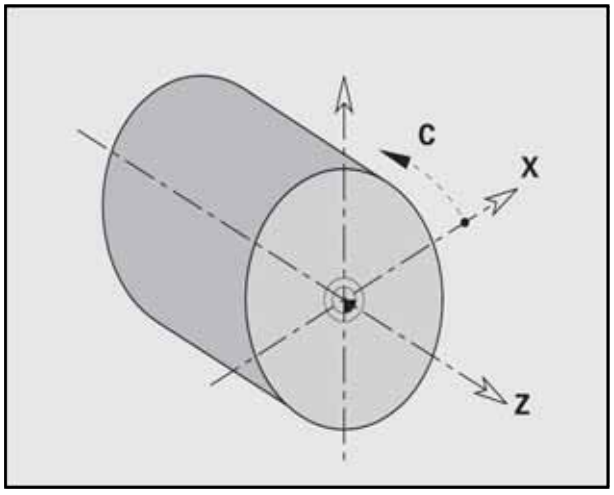
Press ENTER again.

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



Absolute coordinates refer to a fixed point, incremental coordinates to the tool position.
The directions in brackest for X, -X are valid for the PC TURN 50/55, because on thiese machines the tool is in front of the turning axis.
** only for driven tools*





Reference system for turning machines

A reference system is required to define positions in a plane or in space. The position data always refer to a determined point and are described through coordinates.
In the Cartesian coordinate system (a rectangular coordinate system) three directions are defined as the axes X, Y and Z. The axes are mutually perpendicular and intersect at one point, called the datum. A coordinate indicates the distance from the datum in one of these directions. A position in a plane is thus described by two coordinates, a position in space by three coordinates. The X-coordinate is oriented towards the horizontal slide, the Z-coordinate is oriented towards the longitudinal slide.
Data on coordinates in minus direction describe the movements of the tool system towards the workpiece, whereas data in plus direction describe movements that lead away from the tool.

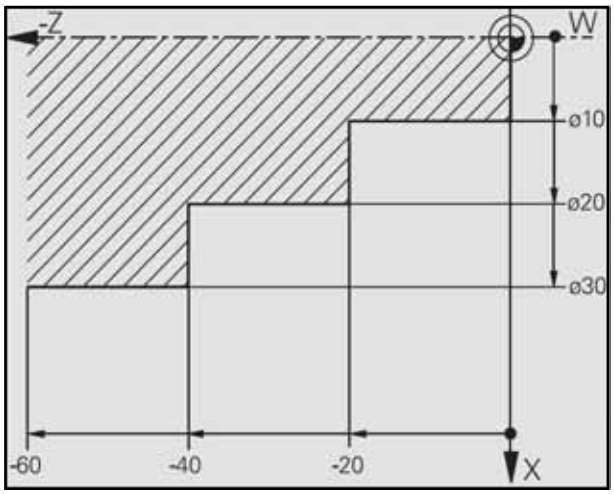
Coordinates that refer to the datum, are called **absolute coordinates**.
Relative coordinates refer to any other position (reference point) within the coordinate system. Relative coordinate values are also called **incremental coordinate** values.

The WinNC knows straight or circular traversing movements between the programmed points. Through definition of successive coordinates and linear or circular traversing movements you can program the machining of a workpiece.

The coordinate data of the axes X and Z refer to the workpiece datum (picture in the center on the left).
Data on angles for the C-axis relate to the "datum of the C-axis" (picture on the bottom left).

Note:
The machines of the EMCO PC-family do not have additional axes.

Absolute and incremental workpiece positions



Absolute workpiece positions

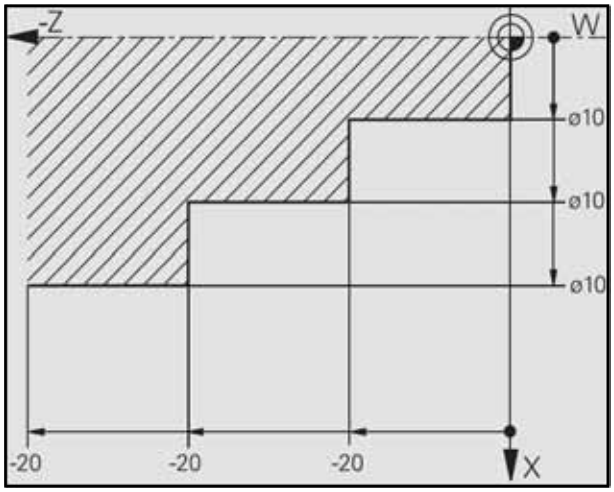
Position coordinates that refer to the workpiece datum are called absolute coordinates.

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).



Incremental workpiece positions

Incremental coordinates refer to the latest programmed position of the tool, that serves as relative zero point. Incremental coordinates describe the actual travels of the tool. That is why they are also called incremental dimensions.

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

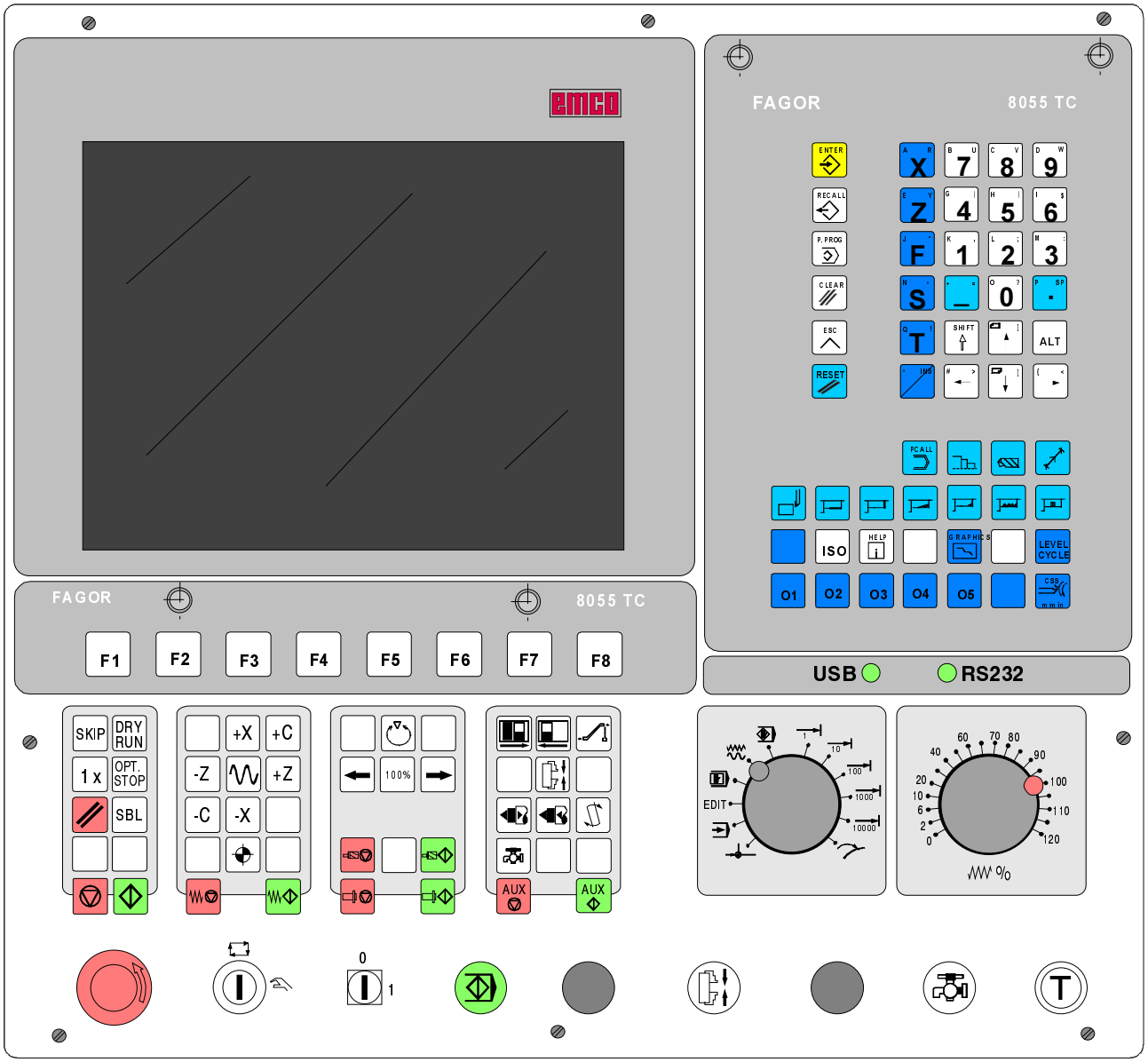
The origin of the coordinate system lies in the tool mount reference point "N", respectively in the tool tip after a tool call.

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

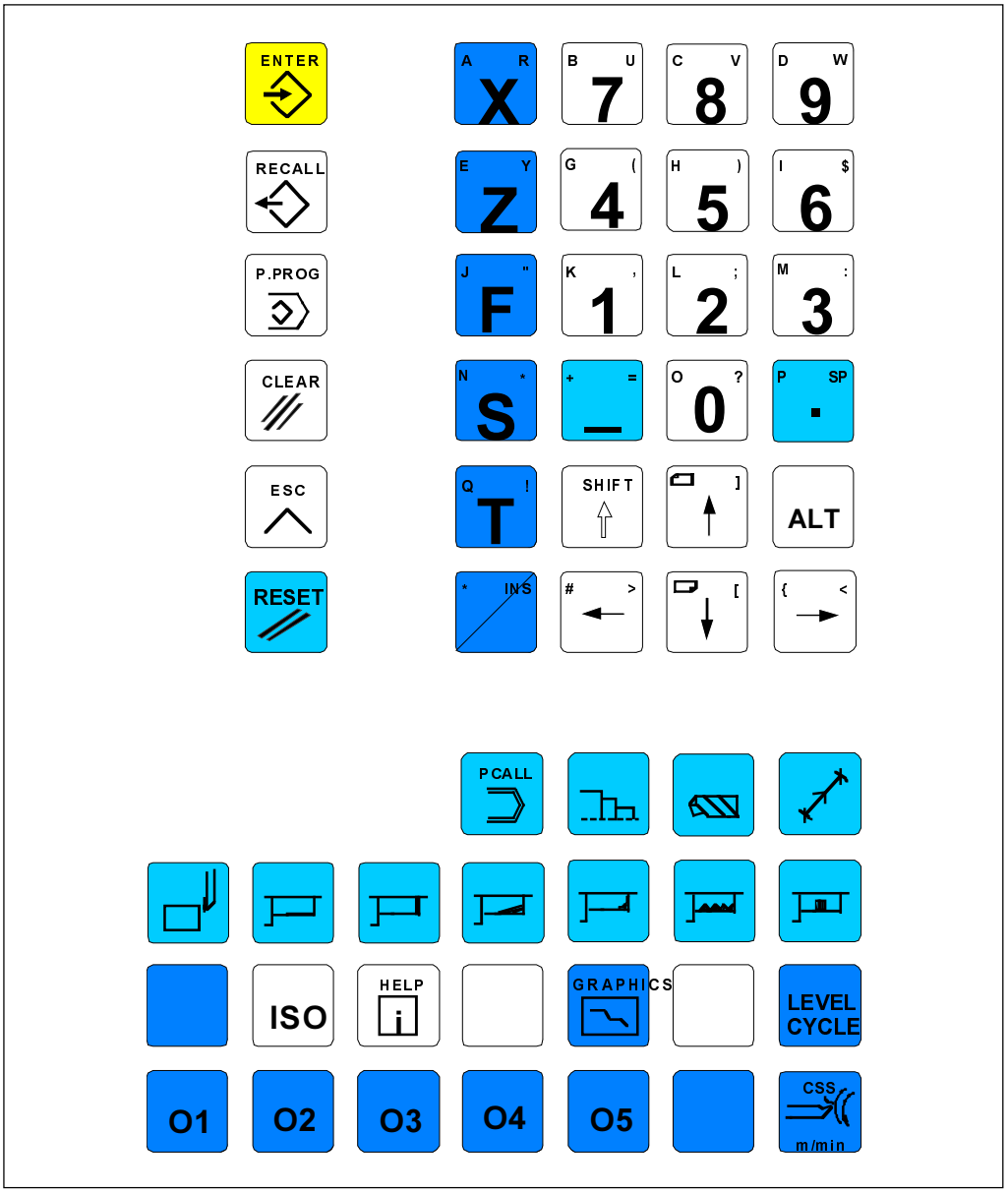
X is entered as radius dimension.

B: Key Description

Control Keyboard, Digitizer Overlay



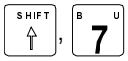
Address and Numeric Keyboard



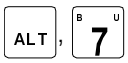
Key Functions

Input by means of alphanumeric keyboard

= 7

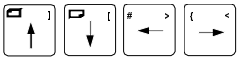


= B

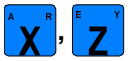


= U

Move highlight


Cursor down / up / left / right

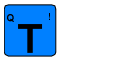
Coordinate axes, feed rate, spindle speed, input of tool number



Selecting coordinate axes

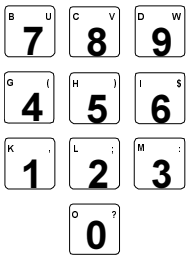
Programming the feed rate

Programming the spindle speed



Programming the tool number

Enter numbers



Numbers

Decimal point

Editing

Selection key

Confirm input and and continue the dialogue

Take over coordinate values or call current cycle

Delete input

Abort dialogue or leave menu

Delete error message in the graphics mode


Delete error message in the machine mode

Managing programs, files, WinNC functions

Create programs, copy or call file management

ISO-programming

Show help

Graphic illustration

Switch to cycle menu

Constant cutting speed (CSS) / (RPM)

Cycles

Calling cycle selection


Profiling cycle


Drilling / Milling cycle


Positioning cycle


Tool calibration cycle


Turning cycle


Facing cycle

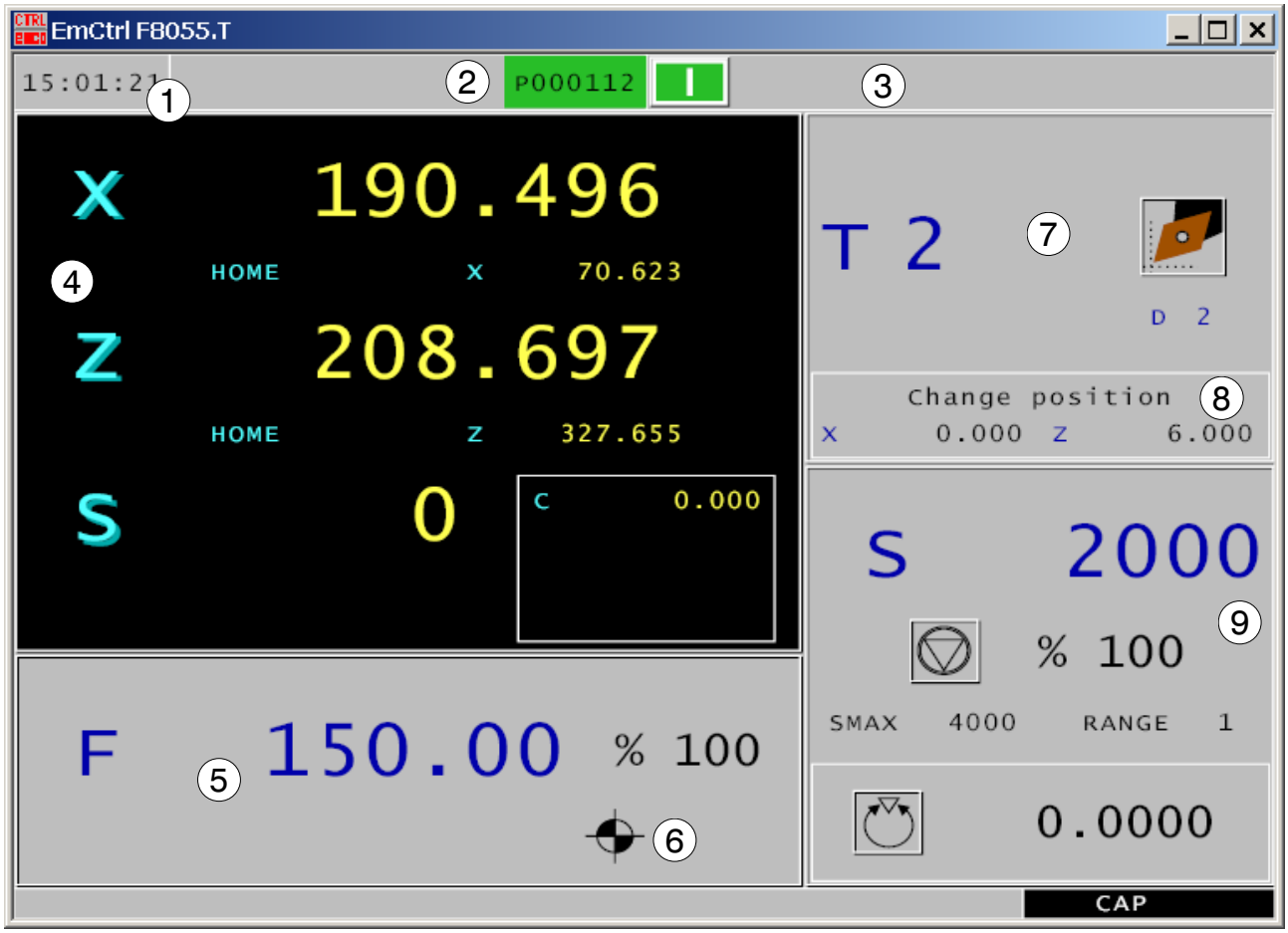

Taper cycle


Rounding cycle


Threading cycle


Grooving cycle

Screen layout of standard screen



1 Time

2 This window can show the following data: SBK in case the mode blockwise execution has been selected.

P.... number of the selected program
Text display: "Positioning", "Execution", "Interruption", "RESET"

3 The text displays of the WinNC appear in this window.

4 Working window, NC-displays

5 Display of the feed rate

6 Display of the operating mode

7 This window shows the number of the selected tool "T" as well as the number of the correction value "D".

8 Coordinates of the tool changing point related to the workpiece zero point.

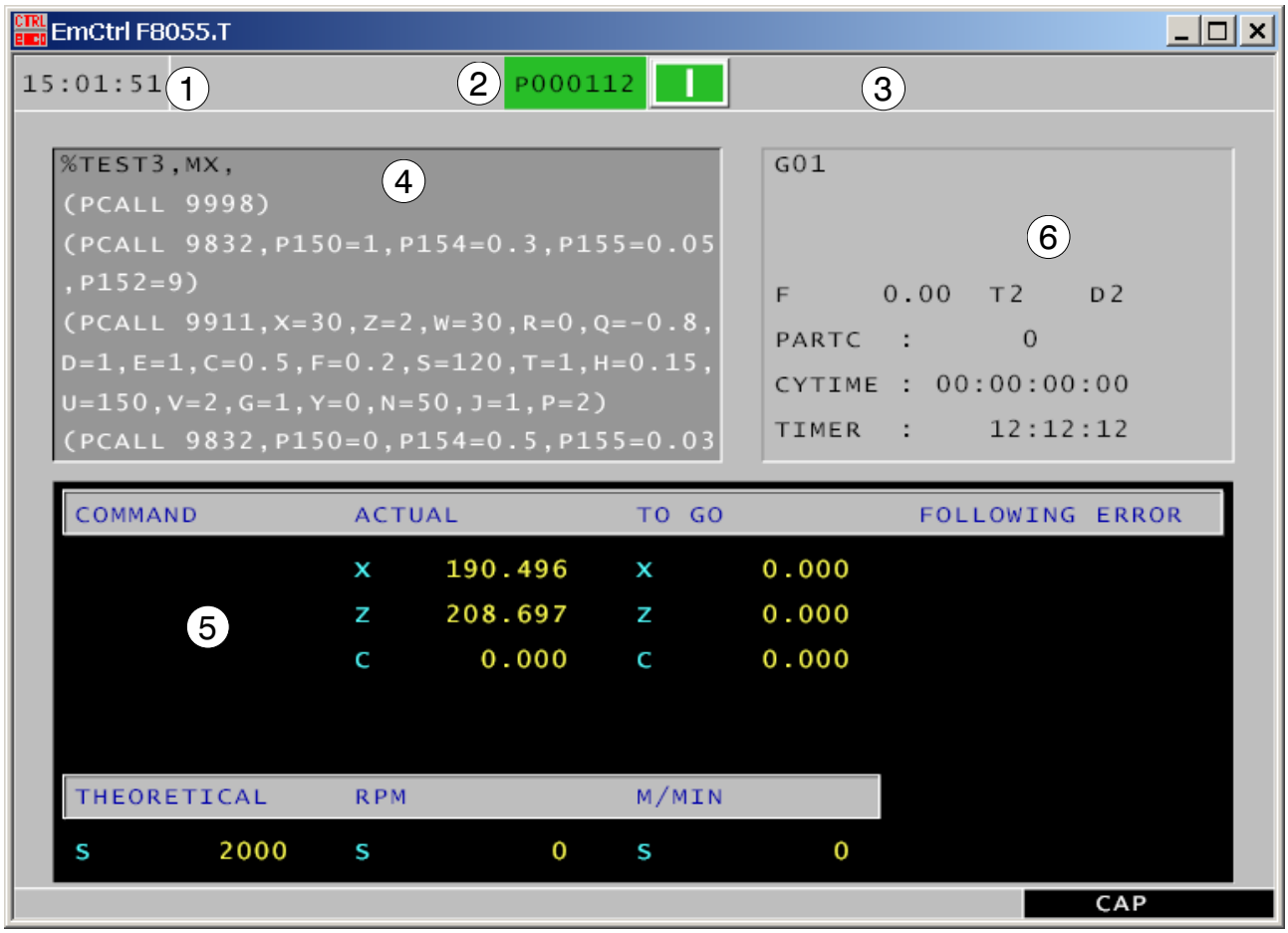
9 This window shows all information regarding the spindle:

The selected nominal spindle speed "S", during operation in RPM/min.
Symbol for spindel condition (clockwise rotation, counterclockwise rotation or standstill).
Used % of the spindle speed
Maximum spindle rotations
Active gear position
Spindle positioning

Constant cutting speed 

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

Screen layout of special screen



1 Time

2 This window can show the following data: SBK in case the mode blockwise execution has been selected.

P.... number of the selected program
Text display: "Positioning", "Execution", "Interruption", "RESET"

3 The text displays of the WinNC appear in this window.

4 This window shows the command lines of the selected program.

5 Each axis disposes of the following fields:

ACTUAL POSITION shows the actual or current axis position.
PATH TO GO shows the remaining distance by which the axis has to be moved in order to reach the programmed coordinate value.
The spindle disposes of the following fields:
NOMINAL VALUE programmed nominal spindle speed S.
RPM spindle speed in rotations per minute.
M/MIN cutting speed in meters/ minute.

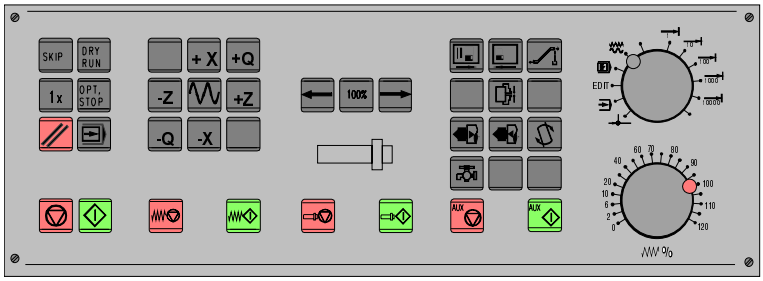
6 This window shows the status of the activated G-functions and of the auxiliary functions M.

PARTC shows the number of workpieces that have been machined in sequence with the same programm.
CYTIME shows the time that has gone by during the machining of the workpiece.

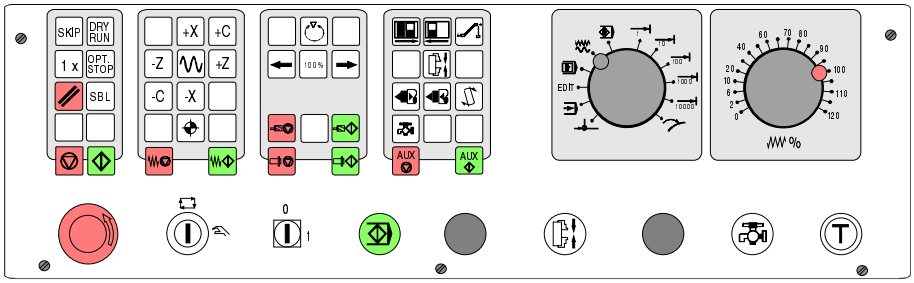
You can switch between standard and special screen by means of the  key.

Machine Control Keys

The machine keys are in the lower part of the control keyboard or digitizer overlay.
Depending on the used machine and accessory not all of these functions are active.



Machine control keyboard of the EMCO control keyboard



Machine control keyboard of the EMCO Concept-Turn Series

Description of keys

SKIP (skip blocks will not be executed)

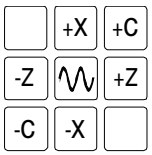
DRY RUN (test run of programs)

OPT STOP (program stop at M01)

RESET

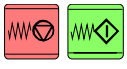
Single block machining

Program stop / program start



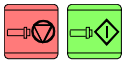
Manual axis movement

Approaching the reference point in all axes



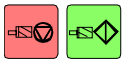
Feed stop / feed start

Spindle override lower / 100% / higher



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

Spindle positioning



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

Permissive button open / close door

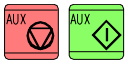
Open / close door

Clamping device open / close

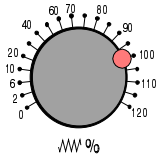
Tailstock quill forwards / backwards

Swivelling toolholder

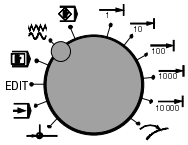
Coolant switch (coolant off / on)



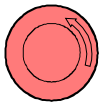
AUX OFF / AUX ON (auxiliary drives off / on)



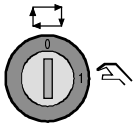
Feed override switch (rapid traverse)



Mode selection switch (for detailed description please see machine description)



EMERGENCY OFF (to unlock pull and turn the switch)



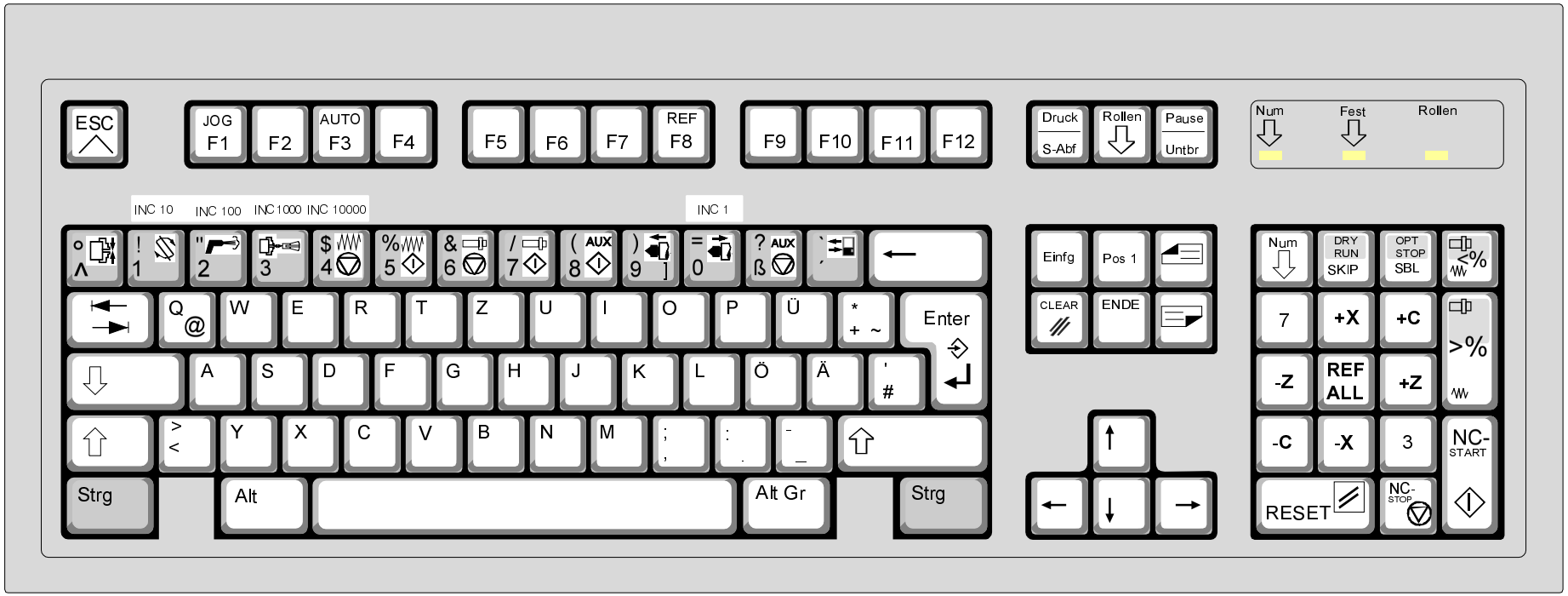
Key switch special operating mode (see machine description)

Additional NC-start key

Key for additional clamping device

Without function

German PC keyboard



$ 4 = 4 ⇧ $ 4 = $ Strg $ 4 = Alt $ 4 = INC 1 000

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

Several alarms are confirmed with the ESC key.

The meaning of the key combination STRG 2 depends on the machine:
MILL 55: Puff blowing device ON/OFF
MILL 105: Coolant ON/OFF
MILL 125: Coolant ON/OFF
The assignment of the accessories` functions is described in the chapter "functions of accessories".

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

Description of keys for German PC keyboard

JOG F1	Tool calibration cycle
F2	Turning cycle
AUTO F3	Facing cycle
F4	Taper cycle
F5	Rounding cycle
F6	Threading cycle
F7	Grooving cycle
⇧ HELP F1	HELP
⇧ F4	PCALL
⇧ F5	Profiling cycle
⇧ F6	Drilling cycle

Strg C	CSS m/min
Strg G	GRAPHICS
Strg I	ISO
Strg L	Switch cycle menu
Strg O	Spindle positioning
Strg P	P. PROG
Strg R	RECALL
Strg S	SWITCH
Strg T	Reset during graphic simulation
↑ ↓ ← →	Move highlight

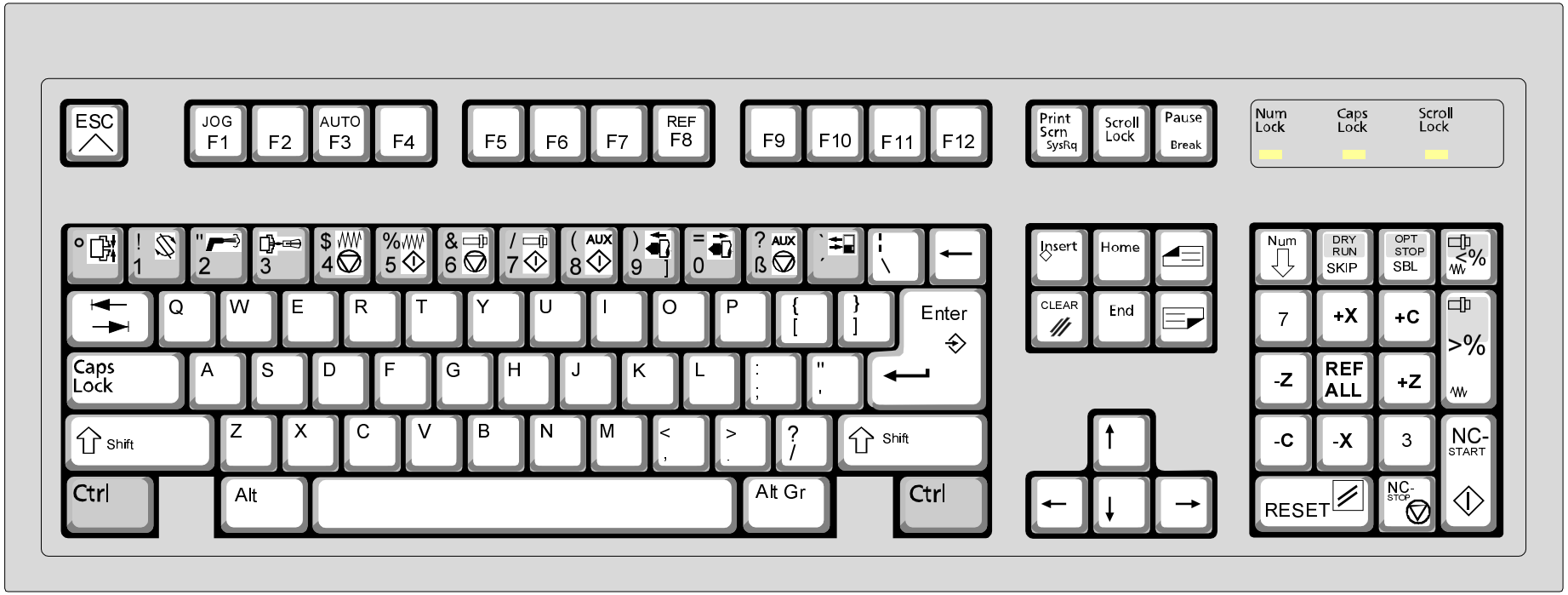
Notice:
Selecting machine keys via the PC keyboard:

1.) Press and hold Strg key.

2.) Press machine key and then release it.

3.) Release Strg key.

English PC keyboard



$4 = 4 Shift $4 = $ Ctrl $4 = ⚙

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

Several alarms are confirmed with the ESC key.

The meaning of the key combination STRG 2 depends on the machine:
MILL 55: Puff blowing device ON/OFF
MILL 105: Coolant ON/OFF
MILL 125: Coolant ON/OFF.
The assignment of the accessories` functions is described in the chapter "functions of accessories".

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

Descripton of keys for English PC keyboard

JOG F1	Tool calibration cycle
F2	Turning cycle
AUTO F3	Facing cycle
F4	Taper cycle
F5	Rounding cycle
F6	Threading cycle
F7	Grooving cycle
Shift + HELP F1	HELP
Shift + F4	PCALL
Shift + F5	Profiling cycle
Shift + F6	Drilling cycle

Ctrl + C	CSS m/min
Ctrl + G	GRAPHICS
Ctrl + I	ISO
Ctrl + L	Switch cycle menu
Ctrl + O	Spindle positioning
Ctrl + P	P. PROG
Ctrl + R	RECALL
Ctrl + S	SWITCH
Ctrl + T	Reset during graphic simulation
↑ ↓ ← →	Move highlight

Notice:
Selecting machine keys via the PC keyboard:

1.) Press and hold Ctrl key.

2.) Press machine key and then release it.

3.) Release Ctrl key.

C: Operation

Operation area of the machine

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

Two modes of operation are distinguished:

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

  Traversing the reference point (Ref)

  Traversing in incremental steps 1 ... 10000

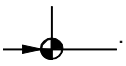
- AUTOMATIC MODE
  Part programs are worked off fully automatic.

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

Manual JOG mode

Traversing the reference point

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

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

Danger of collision

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

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

Traversing the slide manually

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

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

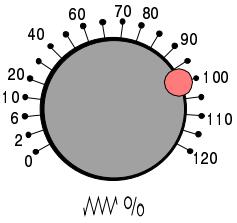
Traversing the slide with presetted coordinates

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

Traversing the slide in incremental steps

During positioning by increments the WinNC moves one machine axis in incremental steps, that you have set before.
You can traverse the machine axes in increments by using the direction keys.

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



- Set the mode selection switch to INC (1 ... 10000 or Alt+0 ... Alt+4 at the PC keyboard, in order to adjust an individual incremental step).
- With each depression of the keys +X, -X, +Z, -Z, +C, -C, and so on, the axes are moved in the corresponding direction by the set incremental step.
- The feed rate is set by means of the override switch.
- If the key is pressed simultaneously, the slides move at rapid traverse (only for PC MILL 300).

AUTOMATIC MODE

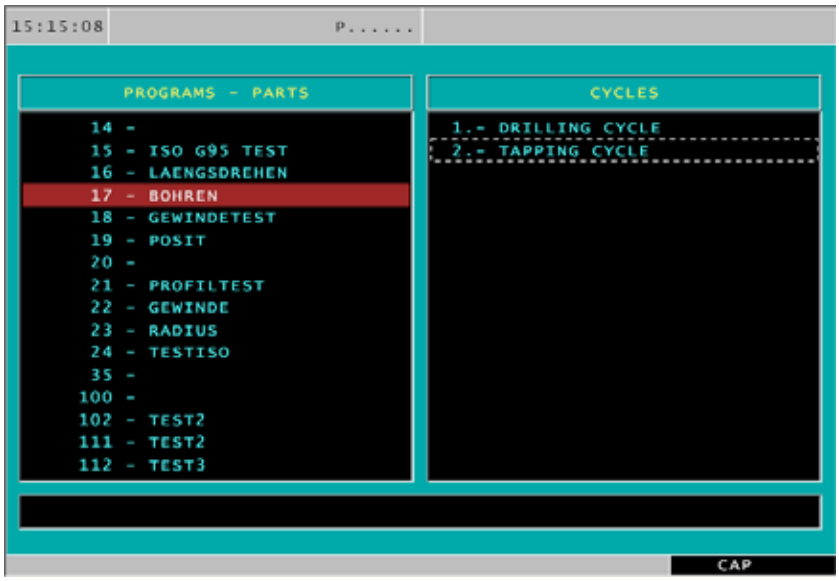
In the AUTOMATIC mode (program run - block mode)
you can execute part programs in a fully automatic way.

Preconditions for working off part programs:

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

see chapter F - program run

Program management



A program consists of a sequence of cycles.

Press the P.PROG key to get into the program management.

On the left you see the directory of the workpiece programs that are stored in the WinNC. If there exist more programs than the ones displayed in the window, use the keys] and [to leaf through the list. Press the keys SHIFT] or the keys SHIFT [simultaneously to move one page up or down. If one of these programs consists of cycles from the TC mode, these cycles are displayed in the right column.

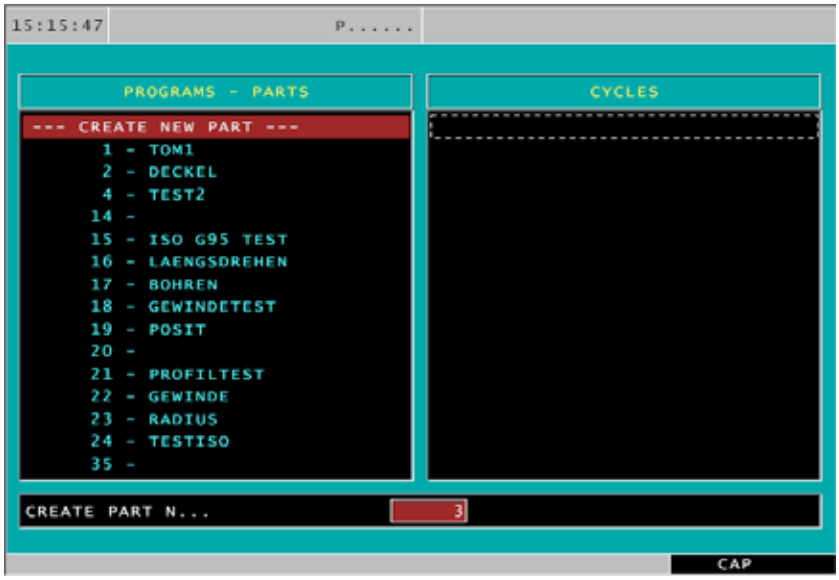
Note:

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

The program managment enables you to:

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

Creating a workpiece program



- Press the P.PROG key.
- Use the highlight to select the option "-CREATE NEW PART-" from the left column.
- Press P.PROG. Enter the program number and press ENTER. If there exists already a program that has the same number, the message "PART NUMBER ALREADY EXISTS." appears.

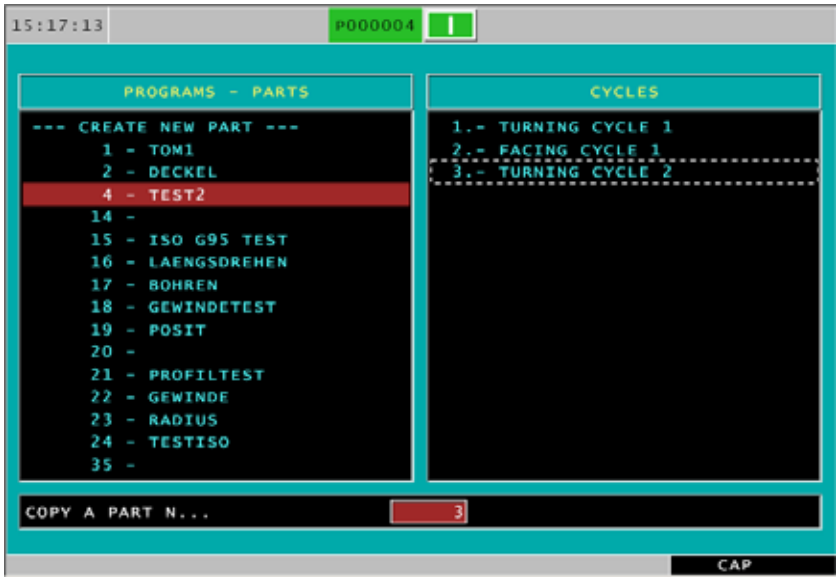
  With ENTER the existing program is overwritten , whereas ESC enables you to select a new program number.

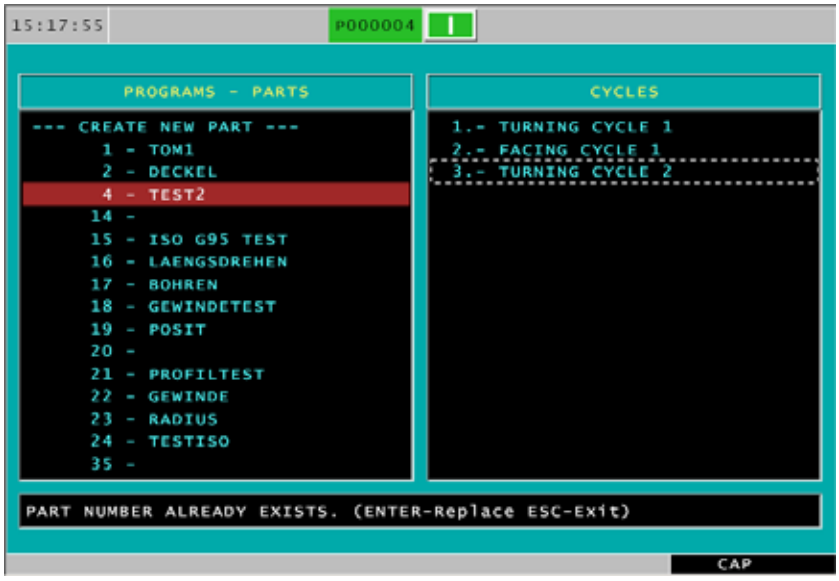
- Enter the program name and press ENTER.
- Use the direction keys to switch to the cycle field and define all values of a cycle.
- Press P.PROG to take the cycle over into the workpiece program.
- Move the highlight to that position within the program where the cycle should be stored and confirm with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

Deleting a workpiece program

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

Copying a workpiece program





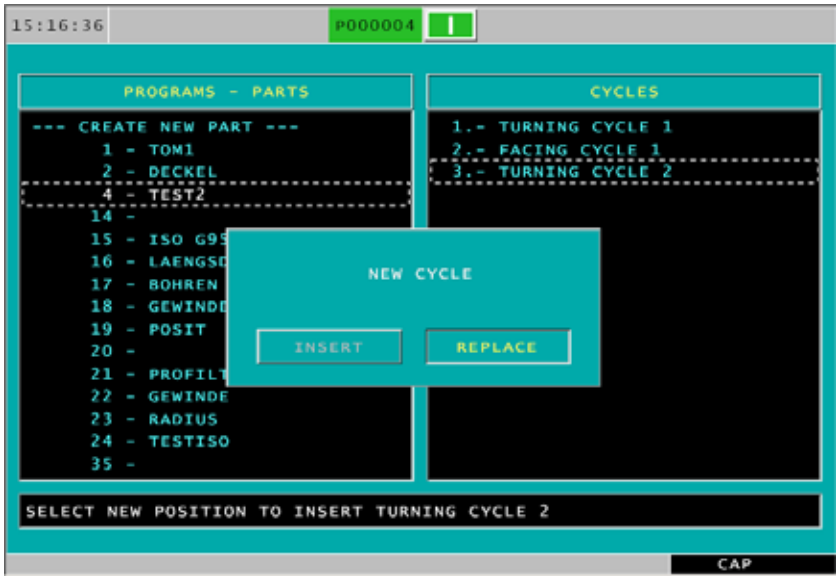
- Press the P.PROG key. Move the highlight to the program you wish to copy.
- Press P.PROG. The WinNC opens the dialogue box: "COPY A PART N..." (see picture on the left). Enter the program number and confirm with ENTER.
  The program is now stored under the new number and also remains stored under the old program number.
- If there exists already a program that has the same number, the WinNC opens the dialogue box "PART NUMBER ALREADY EXISTS." (see picture on the bottom left). You can abort by pressing ESC. To overwrite an existing program, press the ENTER key, enter a new program name and confirm with ENTER.

Changing a workpiece program

Shifting a cycle

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

Changing a cycle



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

"INSERT" adds the changed cycle additionaly. The old cycle remains stored.
Through "REPLACE" the old cycle is overwritten by the new, changed cycle.

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

Deleting a cycle

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

Graphic simulation

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

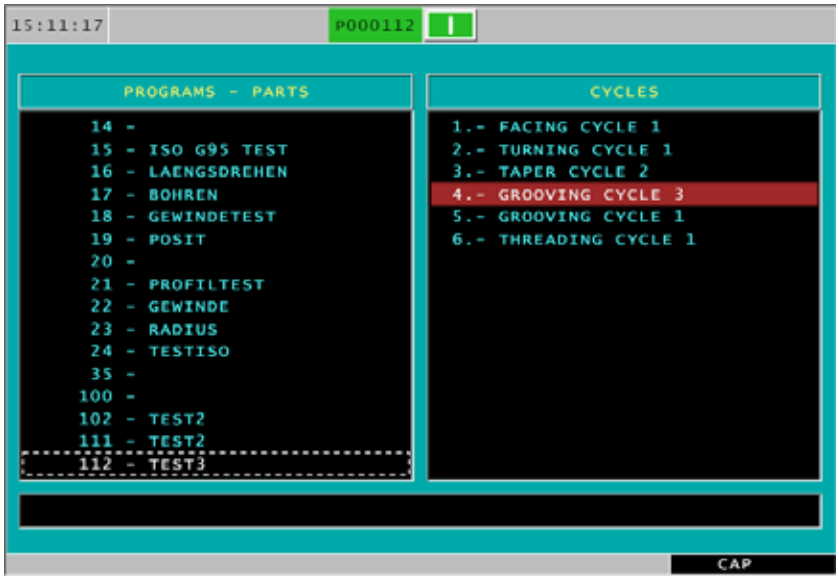
The graphic simulation enables to detect geometric faults like e.g. wrong positions, contour violations, wrong tools, etc..
Technological errors like, e.g. a wrong spindle speed or wrong feed rates are not detected.

Simulating a working cycle

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

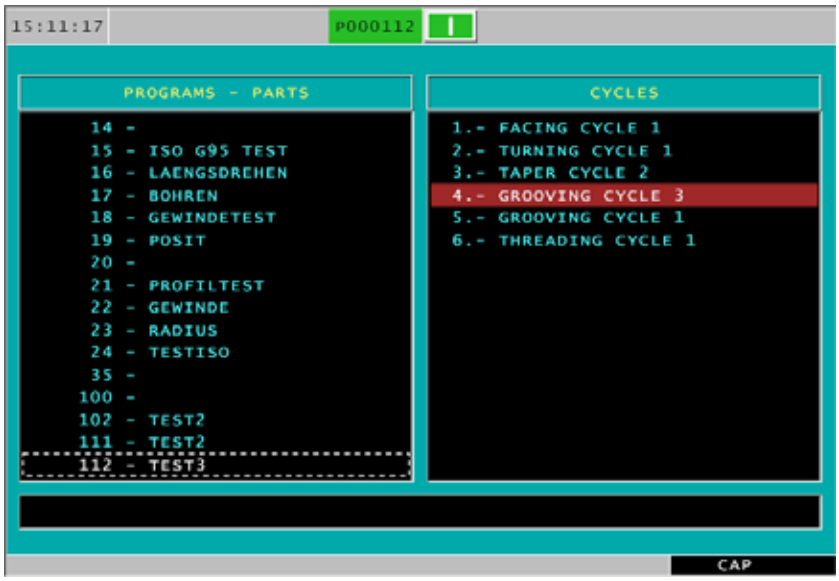
Simulating the whole workpiece program

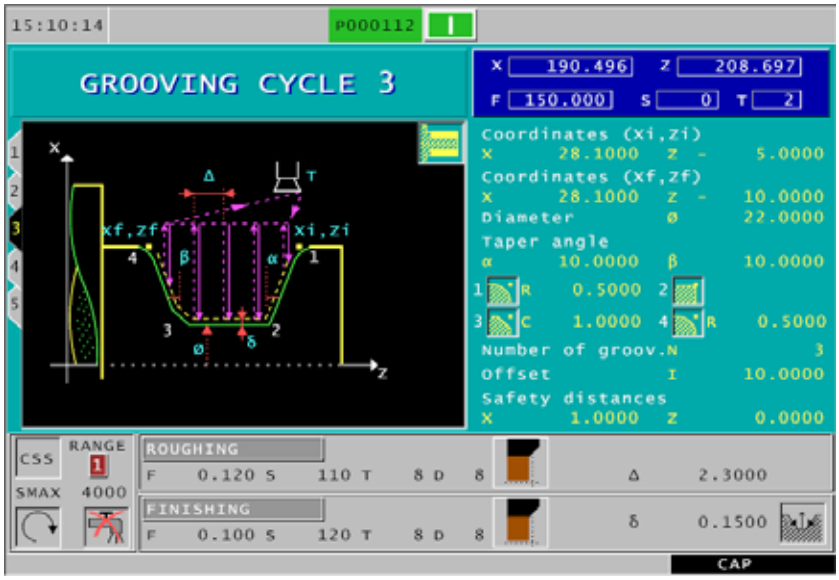
- Press the P.PROG key to call the directory of the stored workpiece programs.
- Use the cursor to select the workpiece program, that should be simulated, from the left column (see illustration in the center left).
- Press the GRAPHICS key.



Simulating a part of a workpiece program

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





Simulating a stored working cycle

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

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

Graphic illustration

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

The WinNC displays the following soft keys:

TYPE OF GRAPHICS F1	Type of graphic illustration
ZOOM F3	Zoom
START F4	Starts the simulation
STOP F5	Stops the simulation
CLEAR SCREEN F6	Clears the screen
+ F7	Switch between the soft-key rows
CANCEL GRAPHICS F6	Switch to the cycle menu, program menu or standard menu that has been selected before
SBL F8	Single block

Type of graphics TYPE OF GRAPHICS F1

X-Z F1
X-C F2
Z-C F3
X-Z SOLID F5

X-Z, XC, ZC
This type of graphic illustration uses colourful lines to display the tool movements in the selected planes (XZ, XC, ZC).
The screen only displays the machined side of the part (+X up to the rotation axis).

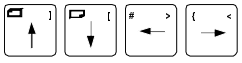
X-Z SOLID
This type of illustration can only be selected with the 3D-graphic simulation ***3D-View*** (optional accessory).

The simulation can be started by pressing the START F4 key.

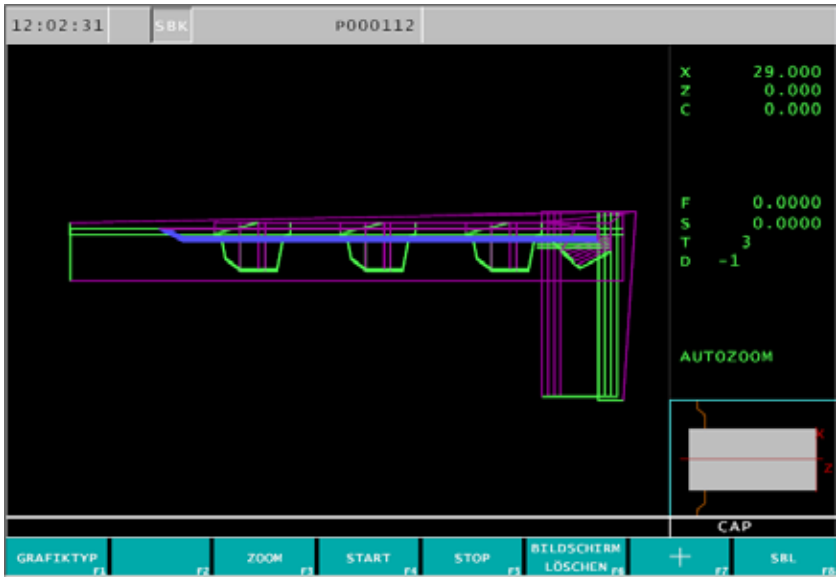
Zoom ZOOM F3

With the soft key ZOOM + F1 the illustration can be manually zoomed in, with ZOOM - F2 it is manually zoomed out.

With the soft key AUTOZOOM F3 the illustration is automatically magnified or reduced to the size of the window.

The simulation window can be shifted by means of the direction keys  .

Single block mode SBL F8



By means of this soft key the simulation is interrupted after every block. To continue the simulation, press the soft key START F4.

The single block mode is activated, when the symbol SBK appears in the simulation window.
To deactivate the single block mode, press SBL F8 again.

D: Programming

Notice:

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

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

Overview M-commands

Command	Description
M0	Programmed stop
M1	Optional stop (program stop only with OPT. STOP)
M2	End of program
M3	Spindle ON clockwise
M4	Spindle ON counterclockwise
M5	Spindle OFF
M6	Tool change
M7	Minimum lubrication ON
M8	Coolant ON
M9	Coolant OFF
M10	Spindle brake ON
M11	Spindle brake OFF
M20	Tailstock quill BACKWARDS
M21	Tailstock quill FORWARDS
M23	Collection device BACKWARDS
M24	Collection device FORWARDS to catch parts
M25	Clamping device OPEN
M26	Clamping device CLOSE
M30	End of main program
M32	End of program with new start for bar loader
M52	Circular C axis ON
M53	Circular C axis OFF
M57	Spindle oscillation ON
M58	Spindle oscillation OFF
M67	Bar feed / loading magazine feed ON
M68	Bar feed / loading magazine feed OFF
M69	Bar change
M71	Blowing out ON
M72	Blowing out OFF
M90	Manual chuck
M91	Collet chuck
M92	Power chuck
M93	Position stop OFF
M94	Bar machining ON
M95	Bar machining OFF

Overview Cycles

Calling cycle selection


Profiling cycles


Drilling / Milling cycles


Positioning cycles


Turning cycles


Facing cycles

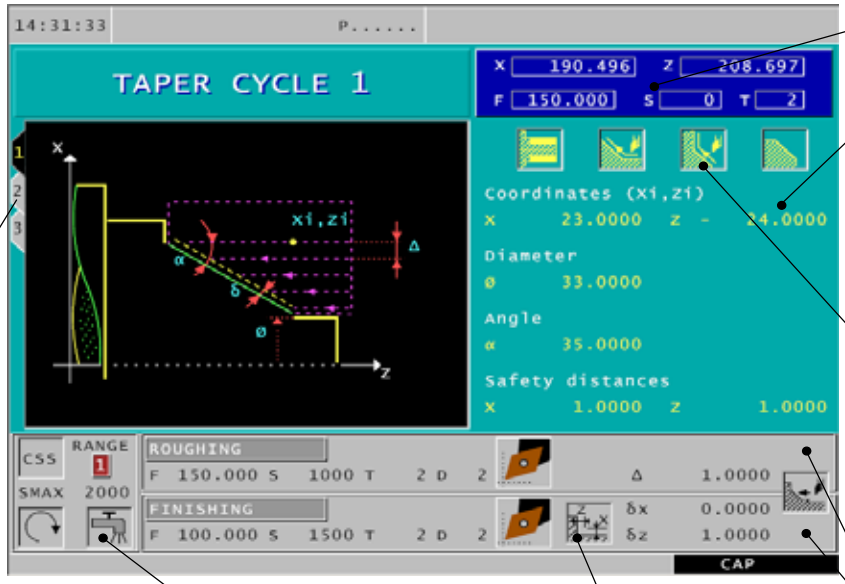

Taper cycles


Rounding cycles


Threading cycles


Grooving cycles

Input of cycle data



Actual values

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

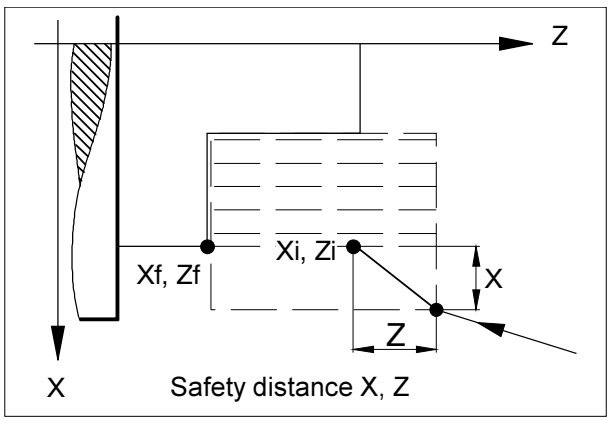
Use the direction keys , , , , to select the boxes. The INS key enables you to switch between the fields (e.g. inside / outside machining).

Fields for technological data

Finishing clearance: You can either select a finishing clearance for a tool shape or for an axis .

Cooling pump ON / OFF: to select, respectively deselect the cooling pump during the respective cycle (the cooling pump is automatically switched off "after" the cycle).

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

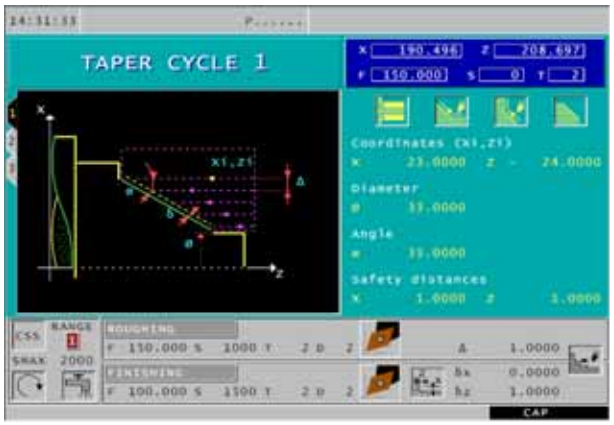


Safety distance

In order to prevent collisions with the workpiece during the machining cycles, a part approaching point can be defined that will be approached prior to the cycle starting point.

The safety distance X,Z indicates the position of this part approaching point in relation to the cycle starting point.

The safety distance in X is always programmed as radius value.



- **Feed rate**
  The working feed rate is entered under the parameter F.
  Place the cursor on the field F, key in the new value and confirm with ENTER.
  For the roughing and finishing pass of processing cycles (e.g. taper cycle) you can program various feed rates.

- **Tool**
  Enter the respective tool and the tool correction data under T and D (several tool correction data are possible per tool).
  For the roughing and finishing pass of processing cycles (e.g. taper cycle) you can program various tools (see chapter E tool programming).

- **Cutting depth**
  The cutting depth per cut for roughing is entered under Δ. The total depth is distributed equally, so that the actual cutting depth is less or equals Δ.

  The finishing clearance (= cutting depth for finishing), that should remain during roughing, is entered with δ.

- **Processing direction**
  There are some cycles, where you can select the processing direction.

  Along the Z axis, along the X axis..

  The setting is adjusted by means of the INS key.

Technological data input for turning cycles

- **Spindle speed / cutting speed**
  The spindle speed /cutting speed is entered under the parameter S.
  Place the cursor on the field S, key in the new value and confirm with ENTER.
  Enter the suitable gear position under GSTUFE.
  For the roughing and finishing pass of processing cycles (e.g. taper cycle) you can programm various spindle speeds.

- **Direction of rotation**
  The direction of rotation is displayed by the field , respectively by the field .
  Place the cursor on the field that shows the direction of rotation and adjust the direction by means of INS.

- **Constant spindle speed / cutting speed**

  The constant spindle speed is set with RPM.

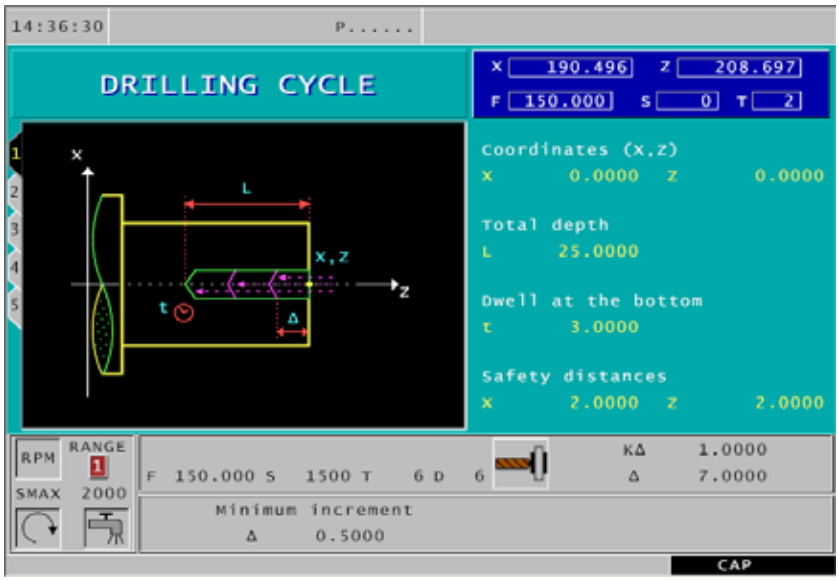
  The constant cutting speed is set with CSS.

  The setting is changed by means of the INS key.

- **Coolant**
  The setting of the cooling pump is displayed in the field . Place the cursor on the field that shows the direction of rotation and activate or deactivate the cooling pump by means of the INS key.

Caution
When you work with constant cutting speed (CSS), you have to enter a maximum spindle speed (SMAX), that is in accordance with the permissible spindle speed for the respective clamping situation.

Technological data input for drilling and threading cycles



Δ Drilling stroke

1. Depth of a drilling operation

KΔ Reducing factor

Factor by which the following drilling strokes are reduced.

Minimum increment Δ
Smallest required cutting depth (only for decreasing cutting depth).

KΔ = 0 or 1: no reducing factor (drilling stroke unchanged). The total depth is equally distributed. The cutting depth is automatically calculated (see example 1).

KΔ ≠ 1: first drilling operation at depth =Δ, second drilling operation at depth =Δ + (Δ*KΔ), third drilling operation at depth =Δ + (KΔ*(Δ + (Δ*KΔ)))
(see example 2)

Example 1:

Total drilling depth L = 20mm,
Drilling stroke Δ = 7mm, reducing factor KΔ = 1

The control automatically calculates the number of the required cutting depths from L and Δ.
20 : 7 = 2,85 ≈ 3
Therefore, 3 drilling operations are carried out until the total drilling depth of L = 20mm is reached.
1. Drilling depth = 7mm
2. Drilling depth = 14mm
3. Drilling depth = 20mm

Example 2:

Total drilling depth L = 20mm,
Drilling stroke Δ = 7mm, reducing factor KΔ = 0,8

1. Drilling depth = Δ
   = 7mm
2. Drilling depth = Δ + (Δ*KΔ)
   = 7 + (7*0,8)
   = 12,6mm
3. Drilling depth = Δ + (KΔ*(Δ + (Δ*KΔ)))
   = 7 + (0,8*12,6)
   = 17,08mm
4. Drilling depth = L
   = 20,000mm

Roughing, finishing, complete machining

The cycles
- turning cycle, facing cycle
- taper cycle 1, 2
- rounding cycle 1, 2
- grooving cycle 1 - 4
- profiling cycle 1, 2

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

For roughing and finishing you can select various feed rates, spindle speeds and tools.
In case different tools are selected for roughing and finishing, the toolholder automatically approaches a tool changing point.

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

Roughing
Select the tool T0 as finishing tool, which means that the finishing cycle will not be executed. A defined finishing clearance is taken into consideration during roughing.
The roughing process is executed in several passes. The total depth is divided into several cutting depths of the same size. The individual cutting depth is less or equals the defined cutting depth for roughing Δ.

Finishing
Select the tool T0 as roughing tool, which means that the roughing cycle will not be executed. The finishing process is carried out along the contour in one cutting depth with a continous cutting movement.

δ ... Finishing clearance

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

Cycles

Machining cycles

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

Calling cycle selection


Profiling cycles


Drilling / Milling cycles


Positioning cycles


Turning cycles


Facing cycles


Taper cycles


Rounding cycles

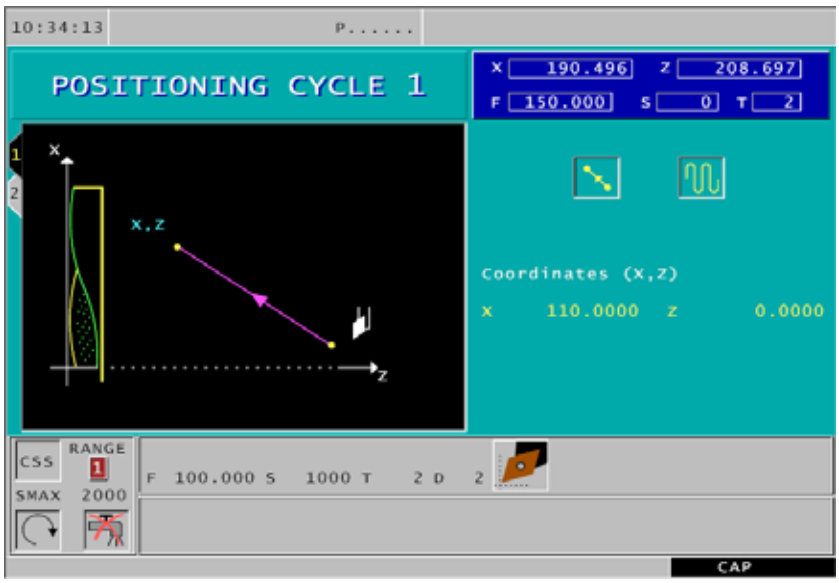

Threading cycles


Grooving cycles

You can leaf through the individual variants of cycles being available in several variants (e.g. Taper Cycle 1, Taper Cycle 2) by means of the 

key.

Positioning Cycle 1



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

Application:

- to position the tool before the the actual machining starts
- to traverse the tool between two cycles
- for linear turning (longitudinal turning, facing, taper turning) in one cutting depth

Call the positioning cycles with the key, then select Positioning Cycles 1-2 with LEVEL CYCLE.

, X-Z, Z-X, , ,

Selection with INS,

confirmation with ENTER.

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

X-Z

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

Z-X

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

The tool moves at rapid traverse.

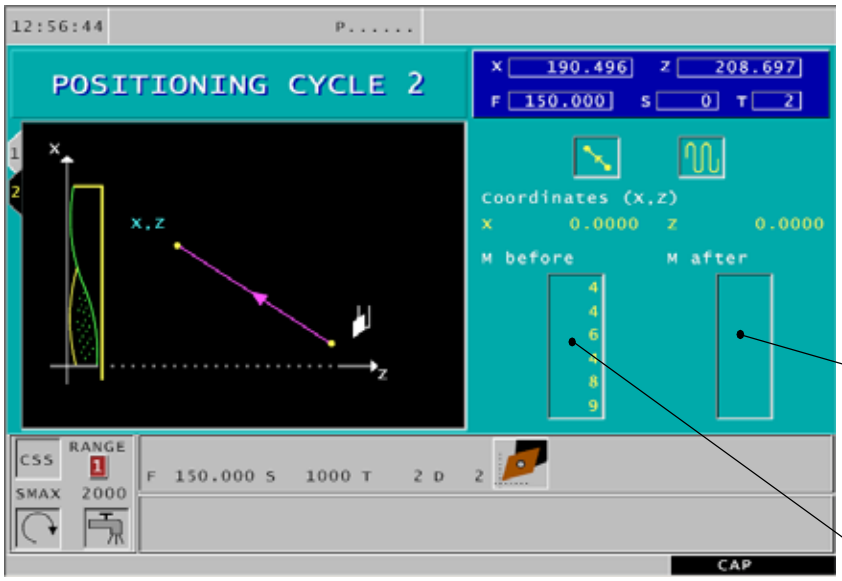
The tool traverses at a programmed feed rate F.

Coordinates

The target position is defined by X and Z.

Positioning Cycle 2

For input refer to Positioning Cycle 1.

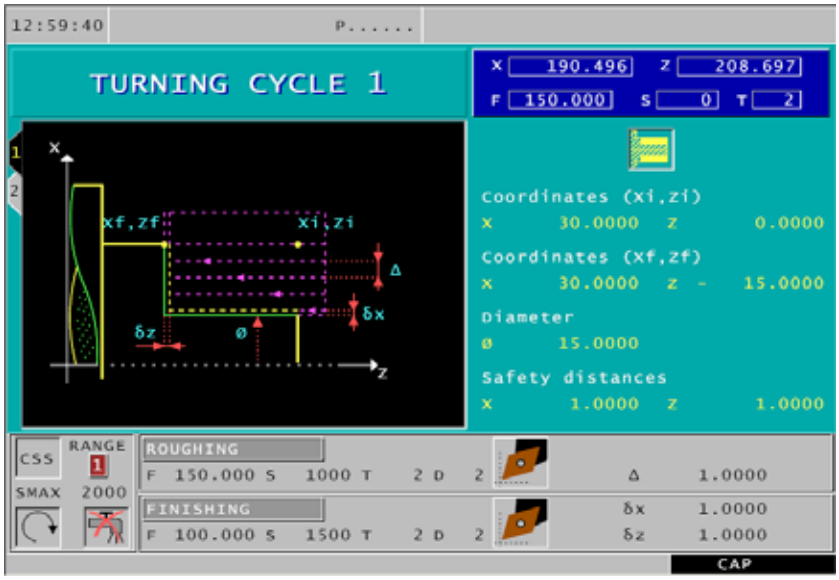


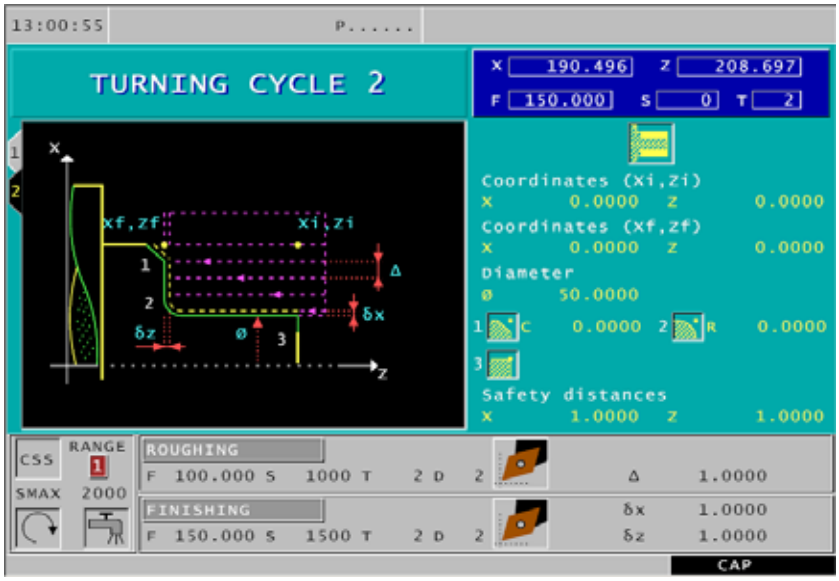
M-commands (auxiliary functions, switch functions), that are executed before and after the cycle, can be entered additionally in the Positioning Cycle 2 (e.g. coolant on / off).
(see overview M-commands in chapter D)

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

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

Turning Cycle 1 / 2





Call the turning cycles with the key, then select Turning Cycle 1 / 2 with LEVEL CYCLE.

Outside turning / inside turning, selection with .

Coordinates (Xi, Zi)
Coordinates of the starting point

Coordinates (Xf, Zf)
Coordinates of the end point

Diameter ø
Turning diameter

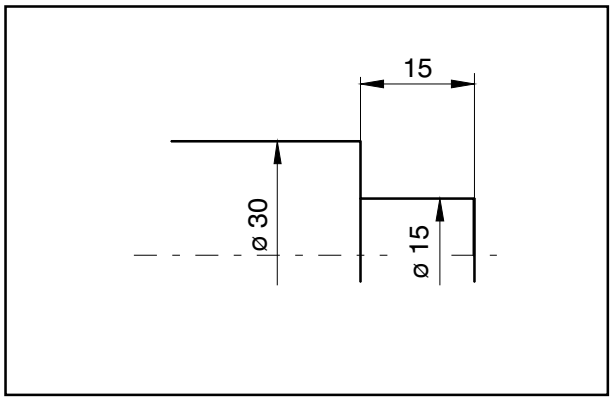
Corner point options 1,2,3
The following options can be selected for all points:

square corner

rounded (with radius indication)

chamfered at 45° (with size indication C)

Safety distance
In order to prevent collisions with the workpiece, an approach point can be defined, that will be approached prior to the cycle starting point.
The safety distance X, Z indicates the position of this part approaching point in relation to the cycle starting point.
The safety distance in X is always programmed as a radius value.



Note:

All value inputs have to be confirmed with ENTER.

Programming example Turning Cycle 1

For roughing and finishing various tools can be used.

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

- Call the turning cycles with the key, then select Turning Cycle 1 with LEVEL CYCLE.

Program input

- Select outside turning with the INS key.

Coordinates (Xi,Zi)

X 30.0000
Z 0.0000

Coordinates (Xf,Zf)

X 30.0000
Z -15.0000

Diameter

ø 15.0000

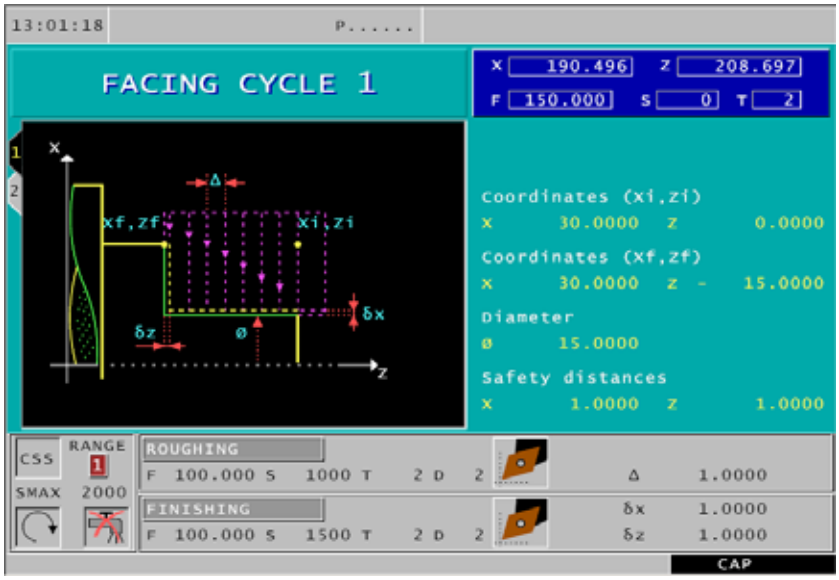
Safety distance

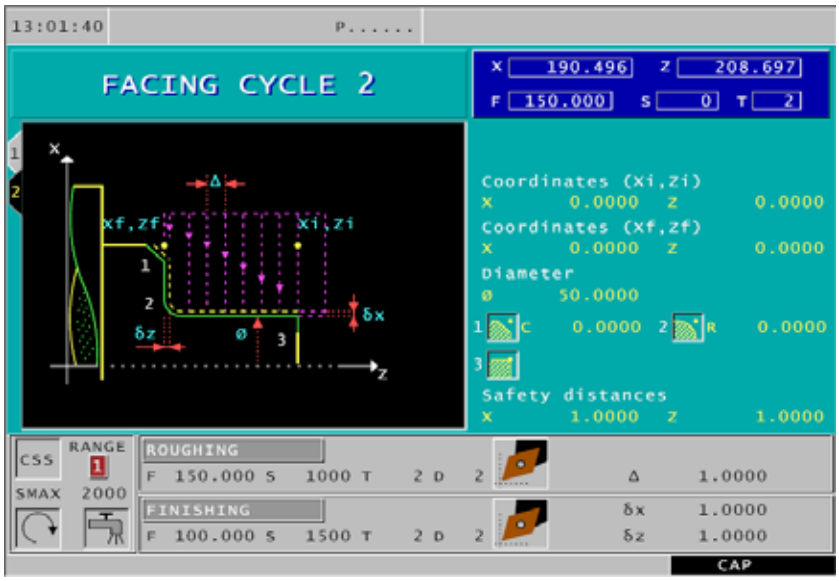
X 1.0000
Z 1.0000

Then please enter all necessary technological data (CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for the roughing and finishing function).

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

Facing Cycle 1 / 2





Call the facing cycles with the key, then select Facing Cycle 1 with LEVEL CYCLE.

Coordinates (Xi, Zi)
Coordinates of the starting point

Coordinates (Xf, Zf)
Coordinates of the end point

Diamter ø
Turning diameter

Corner turning points 1,2,3
The following options can be selected for all points:

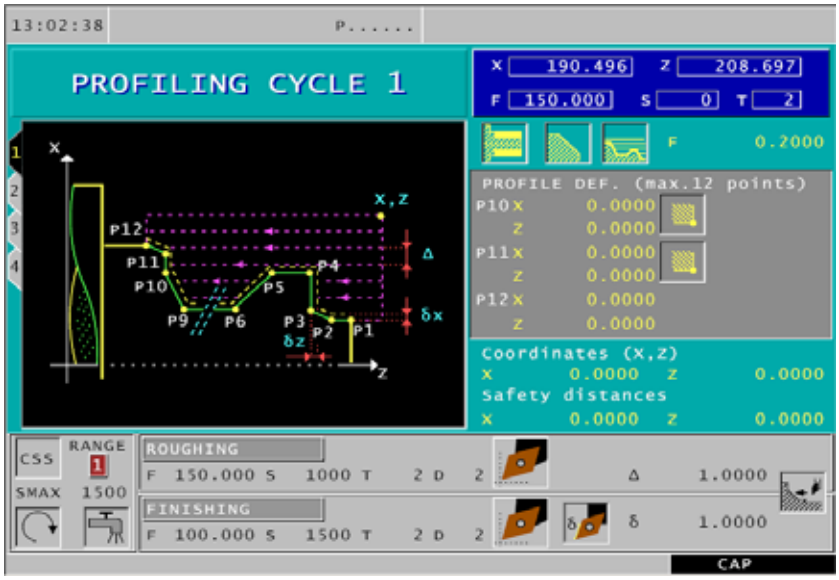
square corner

rounded (with radius indication)

chamfered at 45°(with size indication C)

Safety distance
In order to prevent collisions with the workpiece an approach point can be defined, that will be approached prior to the cycle starting point.
The safety distance X, Z indicates the position of this approach point in relation to the cycle starting point.
The safety distance in X is always programmed as a radius value.

Profiling Cycle 1



With the Profiling Cycle 1 you can manufacture any contour you wish. This contour must not consist of more than 12 positioning points.

Call the profiling cycles with the key, then select Profiling Cycle 1 with LEVEL CYCLE.

, Outside turning / inside turning, selection with INS.

, , , Definition of the taper position.
You can only select positions that are possible for the respective inside and outside machining.

, Definition of the cutting movement characteristic.
You can choose between 2 positions (roughing parallel to the contour and roughing parallel to the axis).

Definition of F, respectively E
Feed rate for undercut / offset on blank contour (pressed part, cast part).

Note:
The definition of F, respectively E changes when you switch from axis-parallel to contour-parallel.

Note:
Contour-parallel profiling is not available at the moment.

Coordinates of the point P
Coordinates in X and Z

- The following options can be selected for all positioning points:

square corner

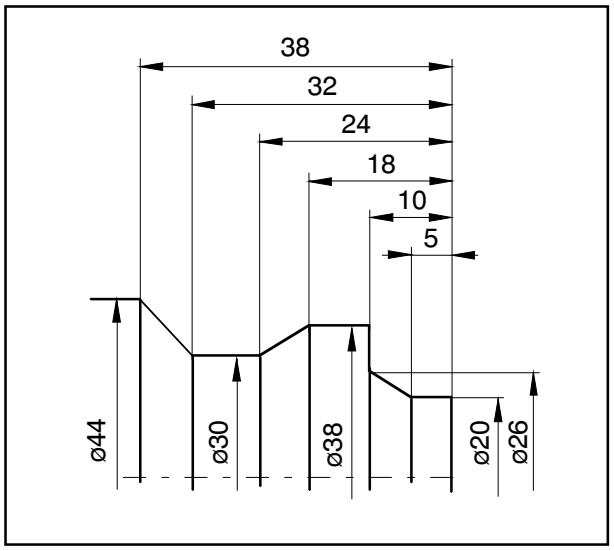
rounded (with radius indication)

chamfered at 45° (with size indication C)

Coordinates (X, Z)
Coordinates of the starting point (dimension of the blank workpiece)

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

, Longitudinal infeed, transversal infeed.



Note:

All value inputs have to be confirmed with ENTER.

If less than 12 contour points are defined, the last program point has to be programmed twice.

Programming example Profiling Cycle 1

For roughing and finishing various tools can be used.

Call the profiling cycles with the key, then select Profiling Cycle 1 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.
- Select the contour position with INS.
- Select the cutting movement characteristic with INS.

Cutting depth F F 0.200

Contour definition (12 points at maximum)

P1	X	20.0000	
	Z	0.0000	
P2	X	20.0000	
	Z	-5.0000	R 0.500
P3	X	26.0000	
	Z	-10.0000	R 0.500
P4	X	38.0000	
	Z	-10.0000	C 0.500
P5	X	38.0000	
	Z	-18.0000	
P6	X	30.0000	
	Z	-24.0000	
P7	X	30.0000	
	Z	-32.0000	
P8	X	44.0000	
	Z	-38.0000	
P9	X	44.0000	
	Z	-38.0000	

Coordinates (X,Z)

X 50.0000
Z 5.0000

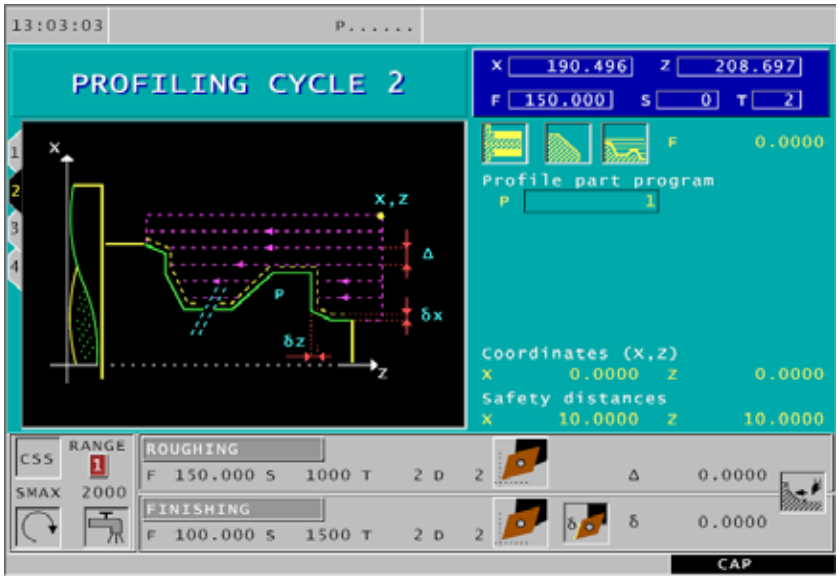
Safety distance

X 1.0000
Z 1.0000

Then please enter all necessary technological data (longitudinal and transversal infeed, CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for the roughing and finishing function).

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

Profiling Cycle 2



With the Profiling Cycle 2 you can manufacture any contour with any number of points. The contour is saved in a subroutine.

Call the profiling cycles with the key, then select Profiling Cycle 2 with LEVEL CYCLE.

Outside turning / inside turning, selection with .

Definition of the taper position.
You can only select positions that are possible for the respective inside and outside machining.

Definition of the cutting movement characteristic.
You can choose between 2 positions (roughing parallel to the contour and roughing parallel to the axis).

Definition of F, respectively E
Feed rate for undercut / offset on blank contour (pressed part, cast part)

Note:
The definition of F, respectively E changes when you switch from axis-parallel to contour-parallel.

In Profiling Cycle 2 the subroutine P1 is stored under the program number **998**001.
You can program 1000 subroutines for the Profiling cycle 2.

Program number for the contour P
Subroutine that contains the description of the contour to be machined.

Coordinates (X, Z)
Coordinates of the starting point (dimension of the blank workpiece)

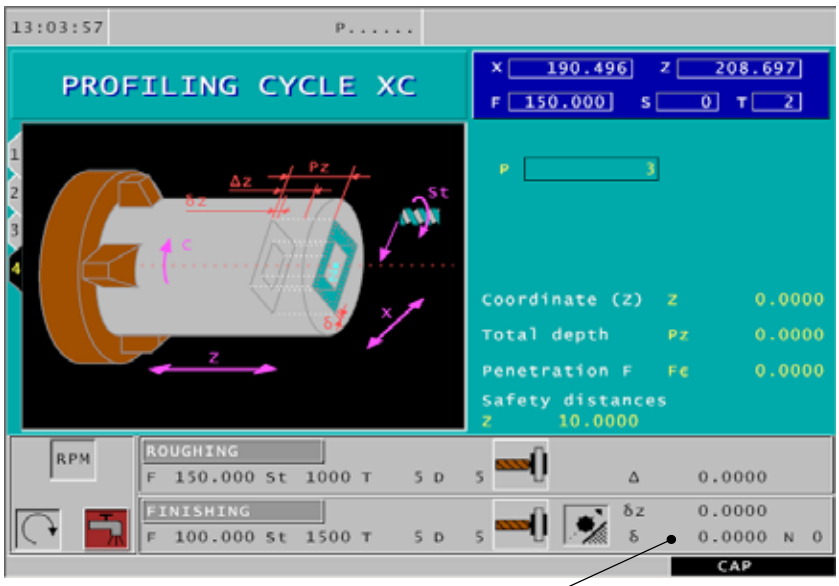
Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Longitudinal infeed, transversal infeed.

Note:
At the moment it is not possible to carry out this cycle.

Profiling Cycle XC

With the Profiling Cycle XC you can manufacture any contour with any number of points.
The contour is saved in a subroutine.



Call the profiling cycles with the key, then select the Profiling Cycle XC with LEVEL CYCLE.

Tool radius compensation:

without compensation,

with tool radius compensation to the left,

with tool radius compensation to the right,

selection with .

δ_z Finishing clearance in Z
δ Finishing clearance in X
N Number of finishing infeeds in Z

Program number for the contour P
Subroutine that contains the description of the contour to be machined.

Coordinate (Z)
Coordinates of the starting point

Total depth Pz
Milling depth in mm

F Penetration F_E
Penetration feed rate of the tool in mm/min

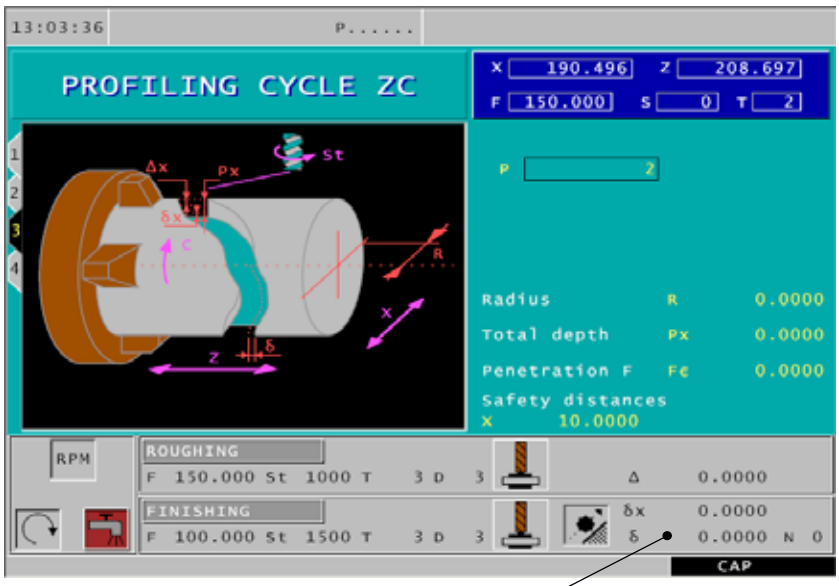
Safety distance Z
Indicates the position of the approach point in relation to the cycle starting point.

Note:
In the Profiling Cycle XC the subroutine P3 is stored under the program number **996003**.
You can program 1000 subroutines for the Profiling Cycle XC.

This cycle can only be programmed in connection with driven tools.

Profiling Cycle ZC

With the Profiling Cycle XC you can manufacture any contour with any number of points.
The contour is saved in a subroutine.



Call the profiling cycles with the key, then select the Profiling Cycle ZC with LEVEL CYCLE.

Tool radius compensation:

without compensation,

with tool radius compensation to the left,

with tool radius compensation to the right,

selection with INS.

δ_x Finishing clearance in X
δ Finishing clearance in Z
N Number of finishing infeeds in X

Program number for the contour P
Subroutine that contains the description of the contour to be machined.

Radius R
Turning radius

Total depth Px
Total milling depth in mm

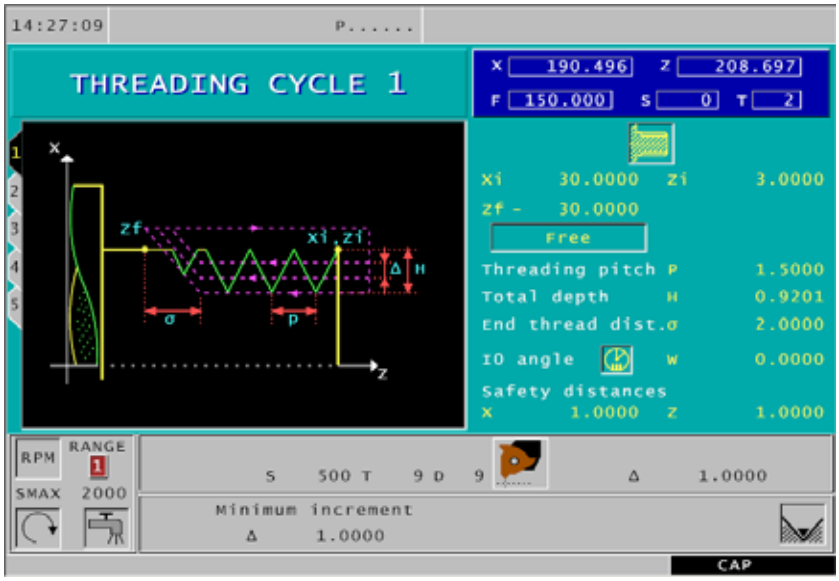
F Penetration F_E
Penetration feed rate of the tool in mm/min

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Note:
Input and contour programming are the same as for Profiling Cycle XC.
In the Profiling Cycle ZC the subroutine P2 is stored under the program number **997**002.
You can program 1000 subroutines for the Profiling Cycle ZC.

This cycle can only be programmed in connection with driven tools.

Threading Cycle 1



With the Threading Cycle 1 you can manufacture longitudinal threads (inside and outside).

Call the threading cycles with the key, then select Threading Cycle 1 with LEVEL CYCLE.

Outside turning / inside turning, selection with INS.

Coordinates (Xi,Zi)
Coordinates of the starting point of the thread.

Coordinate (Zf)
End coordinate in Z of the thread.

Thread type
You can choose among the following standard thread types:

- Free Free pitch thread
- M (S.I.) Normal pitch metric thread
- M (S.I.F.) Fine pitch metric thread
- B.S.W. (W) Normal pitch whitworth thread
- B.S.F. Fine pitch whitworth thread
- U.N.C. Normal pitch unified American thread
- 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 selected as desired.
For all other thread types the thread pitch P and the thread depth H are automatically preset with standardized values.

Threading pitch P
Pitch of the thread.

Depth of thread H
Depth of the thread.

End thread distance σ
Length of thread run-out.

IO angel W
Position value of the spindle.

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

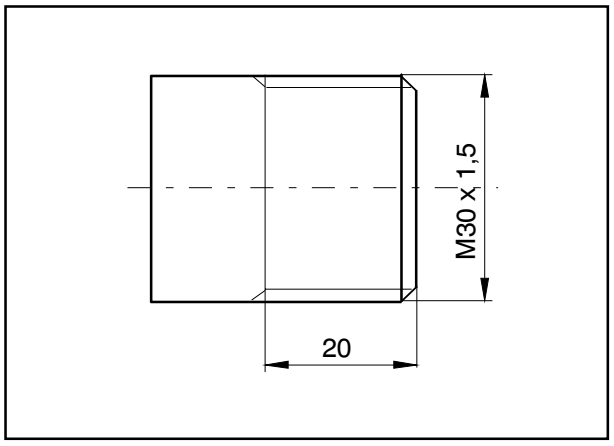
Minimum increment Δ
Smallest required cutting depth.

Repeating the last threading pass
It is possible to repeat the last threading pass without cutting depth in case you want to observe tolerances or to remove a fin.

....repeat the last threading pass

....do not repeat the last threading pass

Selection with INS.



Programming example Threading Cycle 1

Call the threading cycles with the key, then select Threading Cycle 1 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.

Coordinates (Xi,Zi)

X 30.0000
Z 3.0000

Coordinate (Zf)

Z -30.0000

Thread type

Free

Threading pitch P

P 1.5000

Depth of thread H

H 0.9201

End thread distance σ

σ 2.0000

IO- angle W

W 0,0000

Safety distance

X 1.000 Z 1.0000

Then please enter all necessary technological data (gear position, direction of rotation, spindle speed, coolant function, turning values, minimum increment Δ, last threading cycle).

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

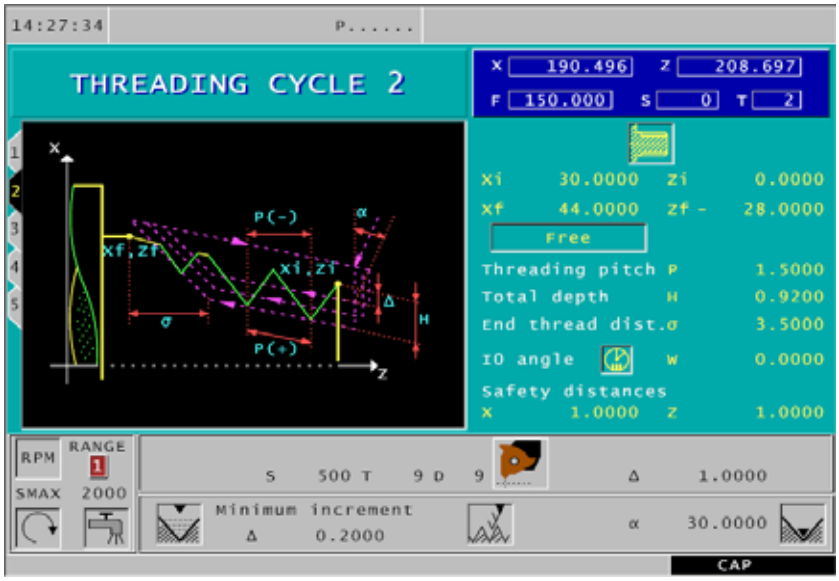
Note:

All value inputs have to be confirmed with ENTER.

The thread run-in should be approximately 2x the threading pitch → the start Z is 3 mm in front of the tool.

For metric threads applies:
Outside thread:
Depth of thread = 0,61343 x threading pitch
Inside thread:
Depth of thread = 0,5413 x threading pitch

Threading Cycle 2



With Threading Cycle 2 you can manufacture tapered threads (inside and outside).

Call the threading cycles with the key, then select Threading Cycle 2 with LEVEL CYCLE.

, Outside turning / inside turning, selection with INS.

Coordinates (Xi,Zi)
Coordinates of the starting point of the thread.

Coordinates (Xf,Zf)
End coordinates of the thread.

Thread type
For thread types please see Threading Cycle 1.

Threading pitch P
Pitch of the thread.

Note:
Algebraic signs of threading pitch **P**:

Tapered thread: **+** positive sign
Parallel thread: **-** negative sign

Depth of thread H
Depth of the thread.

End thread distance σ
Length of thread run-out.

IO angle W
Position value of the spindle.

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Minimum increment Δ
Smallest required cutting depth (only for decreasing cutting depth).

Decreasing or constant cutting depth
Select the type of cutting depth by means of the INS key (, , , cutting depth along the flanks, cutting depth in zig-zag or centric cutting depth)

....Decreasing cutting depth (4 lines)
(constant chip cross section)

....Constant cutting depth (3 lines)

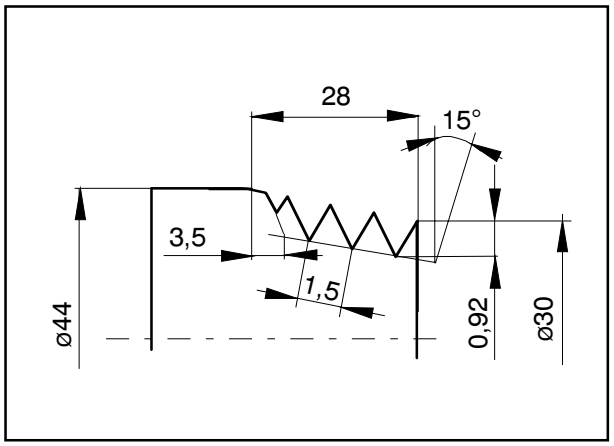
Cutting depth angle α
Half flank angle (normally 30°)

Repeating the last threading pass
It is possible to repeat the last threading pass without cutting depth in case you want to observe tolerances or to remove a fin.

....repeat the last threading pass

....do not repeat the last threading pass

Selection with INS.



Note:

All value inputs have to be confirmed with ENTER.

Programming example Threading Cycle 2

Call the threading cycles with the key, then select Threading Cycle 2 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.

Coordinates (Xi,Zi)

X 30.0000
Z 0.0000

Coordinates (Xf,Zf)

X 44,0000
Z -28.0000

Type of thread

Free

Threading pitch P

P 1.5000

Depth of thread H

H 0.9200

End thread distance σ

σ 3.5000

IO- angle W

W 0.0000

Safety distance

X 1.000 Z 1.0000

Then please enter all necessary technological data (gear position, direction of rotation, spindle speed, coolant function, turning values).

Cutting depth angle α

α 30

Decreasing or constant cutting depth

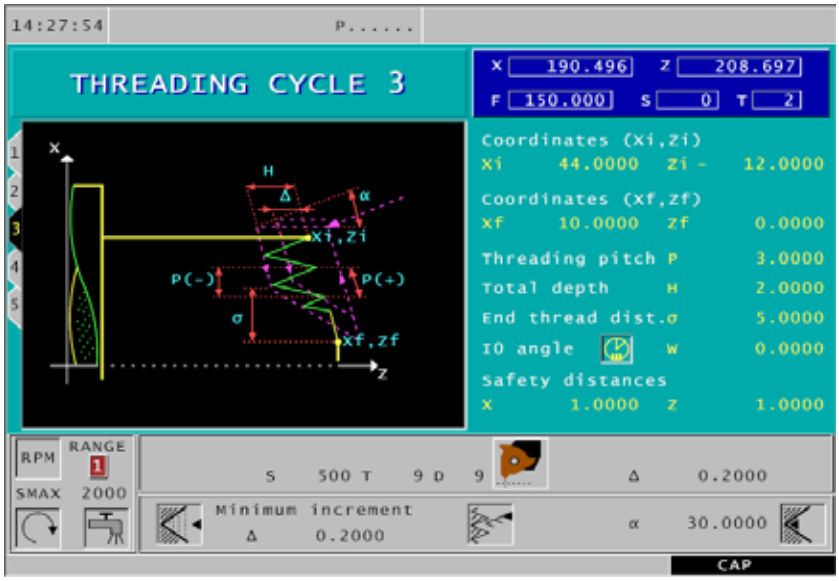
Minimum increment Δ

Δ 0,2

Type of cutting depth (zig-zag, flanks or centric)

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

Threading Cycle 3



With Threading Cycle 3 you can manufacture face threads.

Call the threading cycles with the key, then select Threading Cycle 3 with LEVEL CYCLE.

Coordinates (Xi,Zi)
Coordinates of the starting point of the thread.

Coordinates (Xf,Zf)
End coordinates of the thread.

Threading pitch P
Pitch of the thread.

Note:
Algebraic sign of the threading pitch **P**:

Tapered thread: **+** positive sign
Parallel thread: **-** negative sign

Depth of thread H
Depth of the thread.

End thread distance σ
Length of thread run-out.

IO angle W
Position value of the spindle.

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Minimum increment Δ
Smallest required cutting depth (only for decreasing cutting depth).

Decreasing or constant cutting depth
Select the type of cutting depth by means of the key. (, , , cutting depth along the flanks, cutting depth in zig-zag or centric cutting depth)

....Decreasing cutting depth (4 lines)
(constant chip cross section)

... Constant cutting depth (3 lines)

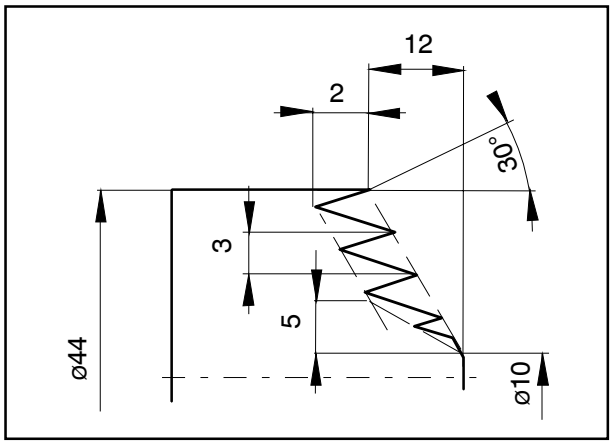
Cutting depth angle α
Half flank angle (normally 30°)

Repeating the last threading pass
It is possible to repeat the last threading pass without cutting depth in case you want to observe tolerances or to remove a fin.

....repeat the last threading pass

....do not repeat the last threading pass

Selection with .



Note:

All value inputs have to be confirmed with ENTER.

Programming example Threading Cycle 3

Call the threading cycles with the key, then select Threading Cycle 3 with LEVEL CYCLE.

Program input

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

Coordinates (Xi,Zi)

X 44.0000
Z -12.0000

Coordinates (Xf,Zf)

X 10,0000
Z 0.0000

Threading pitch P

P -3.0000

Depth of thread H

H 2.0000

End thread distance σ

σ 5.0000

IO- angle W

W 0.0000

Safety distance

X 1.000 Z 1.0000

Then please enter all necessary technological data (gear position, direction of rotation, spindle speed, coolant function, turning values).

Cutting depth angle α

α 30

Decreasing or constant cutting depth

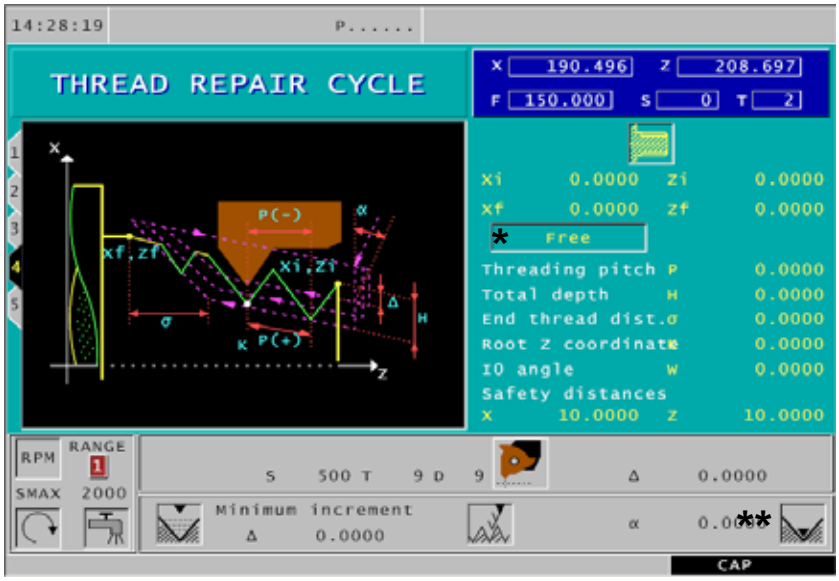
Minimum increment Δ

Δ 0,2

Type of cutting depth (zig-zag, flanks or centric)

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

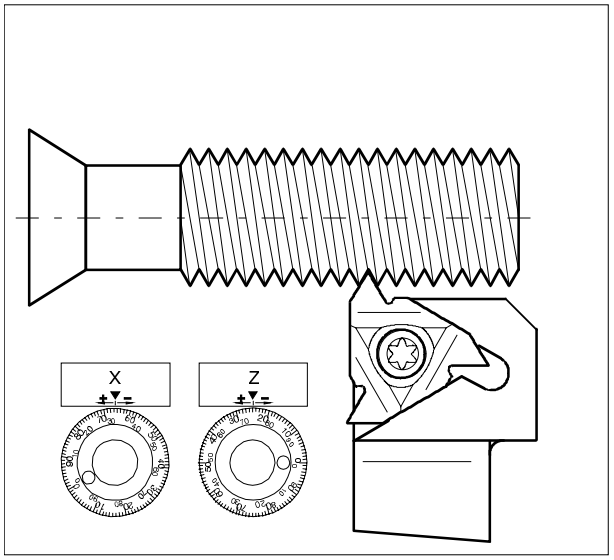
Thread Repair Cycle



The Thread Repair Cycle enables you to re-cut outside, inside and tapered threads.
Call the threading cycles with the key, then select the Thread Repair Cycle with LEVEL CYCLE.

You have to enter the same data as for Threading Cycle 2.

* see thread type under Threading Cycle 1
** see threading pass under Threading Cycle 1



Touching the thread

First the thread must be touched, so as to make it possible for the tool to hit the thread during machining.

- Use the handwheels to position the threading tool in an undamaged root of thread, if possible.

Root Z coordinate K

- Place the cursor on the input box for the root Z coordinate.
- Press RECALL. The position value of the tool is written into the input box.
- Press the ENTER key.

IO- angle W

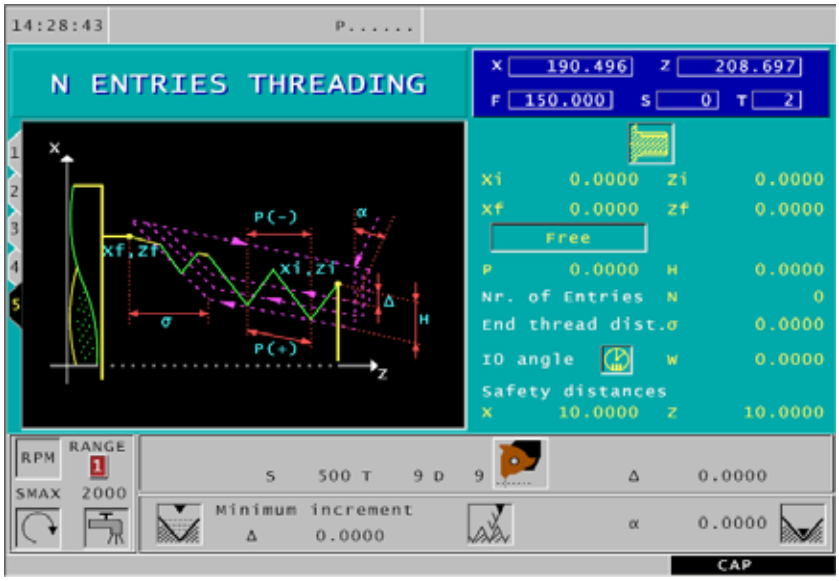
- Place the cursor on the input box for the IO angle.
- Press RECALL. The position value of the spindle is written into the input box.
- Press the ENTER key.

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

Note:
For thread repairs always enter the end thread distance σ, since otherwise it could result in contour violations at the already existing thread.

Note:
At the moment this cycle cannot be machined.

N Entries Threading



The cycle N Entries Threading enables to thread outside, inside and tapered threads with as many entries as desired.
Call the threading cycles with the key, then select N Entries Threading with LEVEL CYCLE.

You have to enter the same data as for Threading Cycle 2.

Coordinates (Xi,Zi)
Coordinates of the starting point of the thread.

Coordinates (Xf,Zf)
End coordinates of the thread.

Type of threads
For thread types please see Threading Cycle 1.

Threading pitch P
Pitch of the thread.

Note:
Algebraic signs for threading pitch **P**:

Tapered thread: **+** positive sign
Parallel thread: **-** negative sign

Depth of thread H
Depth of the thread

Number of entries
Number of threads

End thread distance σ
Length of thread run-out.

IO angle W
Position value of the spindle.

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Minimum increment Δ
Smallest required cutting depth (only for decreasing cutting depth).

Decreasing or constant cutting depth

Select the type of cutting depth with the INS key.

(, , , cutting depth along the flanks, cutting depth in zig-zag, or centric cutting depth)

....Decreasing cutting depth (4 lines)
(constant chip cross section)

....Constant cutting depth (3 lines)

Cutting depth angle α
Half flank angle (normally 30°)

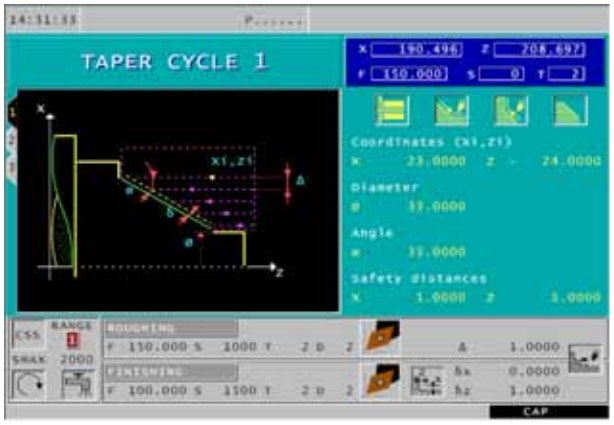
Repeating the last threading pass
It is possible to repeat the last threading pass without cutting depth in case you want to observe tolerances or to remove a fin.

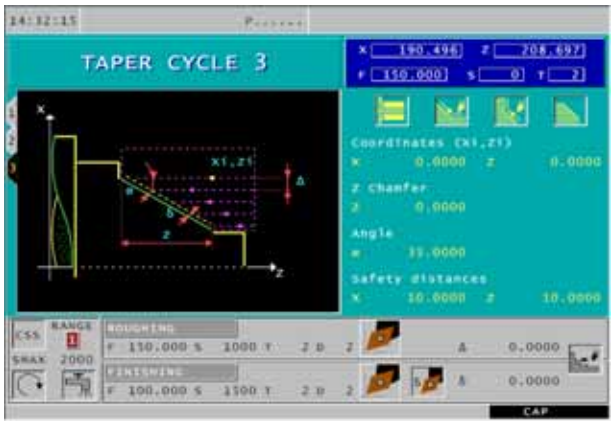
....repeat the last threading pass

....do not repeat the last threading pass.

Selection with INS.

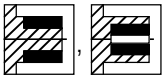
Taper Cycle 1 / 2 / 3





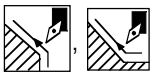
Call the taper cycles with the key, then select Taper Cycle 1 / 2 / 3 with LEVEL CYCLE.

The only difference between the Taper Cycles 1 / 2 / 3 is the way of entering the dimensions for the taper (point +angle + diameter, respectively 2 points, respectively point + angle + chamfer length).

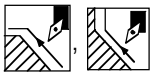


Outside turning / inside turning, selection with INS.

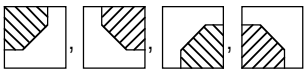
When switching between inside and outside the screen display and the following input symbols change according to the setting.



Approaching the contour



Departing the contour



Taper position
You can only select positions that are possible for the respective inside and outside machining.

Coordinates (Xi, Zi)
Coordinates of the starting point

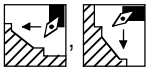
Diameter ø
Start diameter of the taper (only for taper cycle 1)

Z chamfer
Length of the chamfer in direction of the Z axis.

Angle α
Taper angle (only for taper cycle 1)

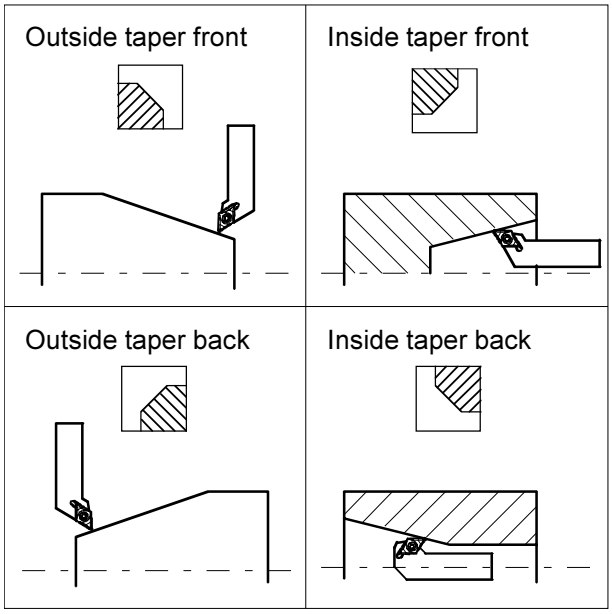
Coordinates (Xf, Zf)
Coordinates of the end point (only for taper cycle 2)

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.



Longitudinal infeed, transversal infeed.

Taper position

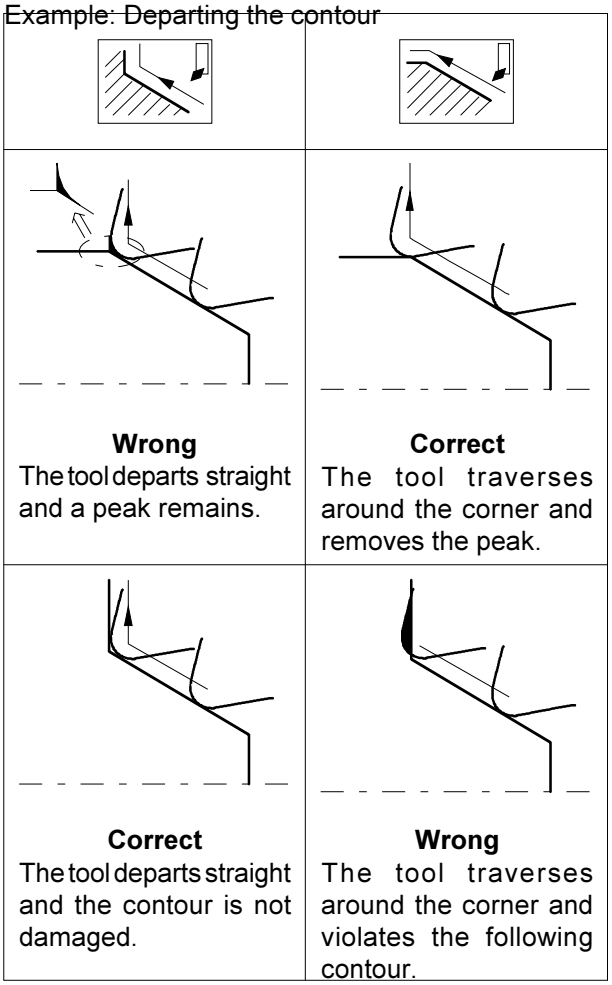


Approaching and departing the contour with tool radius compensation

The control automatically compensates the influence of the radius at the tool tip.

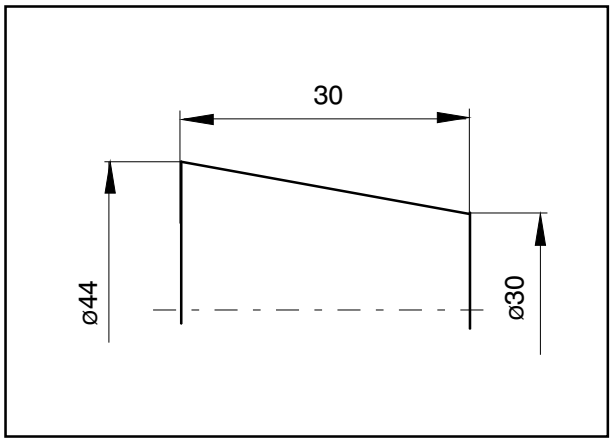
The shape of the corner has to be indicated so as to avoid violation of corners or remaining peaks during approaching (e.g. ) , respectively departing (e.g. ) .

Example: Departing the contour



Wrong The tool departs straight and a peak remains.	**Correct** The tool traverses around the corner and removes the peak.
Correct The tool departs straight and the contour is not damaged.	**Wrong** The tool traverses around the corner and violates the following contour.

Choose the selection window and select the required type of corner for approaching / departing by means of the  key and confirm with  .



Note:

All value inputs have to be confirmed with ENTER.

Programming example taper turning outside

For roughing and finishing various tools can be used.

Call the taper cycles with the key, then select Taper Cycle 2 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.
- Select contour approach with INS.
- Select contour departure with INS.
- Select the taper position with INS.

Coordinates (Xi,Zi)

X 30.0000
Z 0.0000

Coordinates (Xf,Zf)

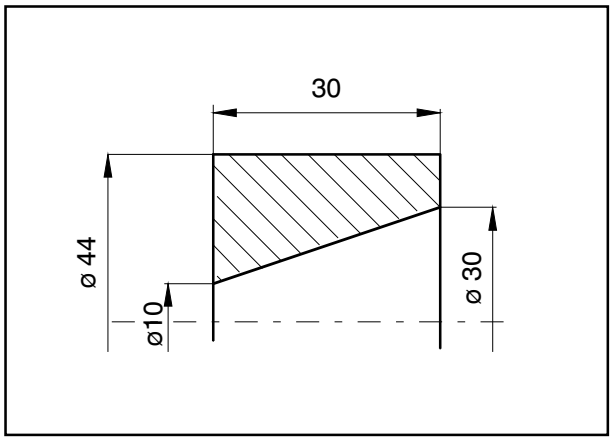
X 44.0000
Z -30.0000

Safety distance

X 1.0000
Z 1.0000

Then please enter all necessary technological data (longitudinal or transversal infeed, CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for roughing and finishing).

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



Note:

All value inputs have to be confirmed with ENTER.

Programming example taper turning inside

For roughing and finishing various tools can be used.

Call the taper cycles with the key, then select Taper Cycle 2 with LEVEL CYCLE.

Program input

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

- Select inside turning with the INS key.
- Select contour approach with INS.
- Select contour departure with INS.
- Select the taper position with INS.

Coordinates (Xi,Zi)

X 30.0000
Z 0.0000

Coordinates (Xf,Zf)

X 10.0000
Z -30.0000

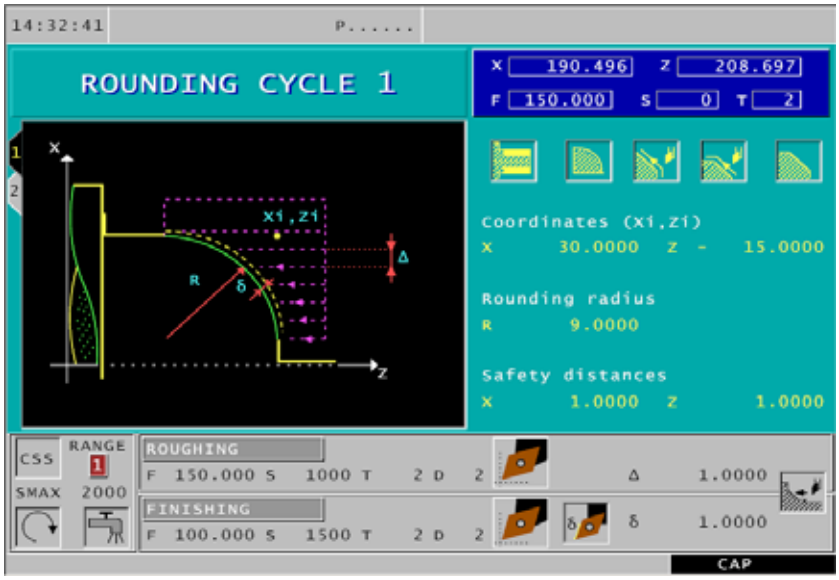
Safety distance

X 1.0000
Z 1.0000

Then please enter all necessary technological data (longitudinal or transversal infeed, CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for roughing and finishing).

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

Rounding Cycle 1



Rounding Cycle 1 enables you to manufacture quarter circles (simplified input).

Call the rounding cycles with the key, then select Rounding Cycle 1 with LEVEL CYCLE.

Outside turning / inside turning, selection with INS.

When switching between inside and outside the screen display and the following input symbols change according to the setting.

Convex / concave rounding

Approaching the contour

Departing the contour

Radius positions

You can only select positions that are possible for the respective inside / outside machining.

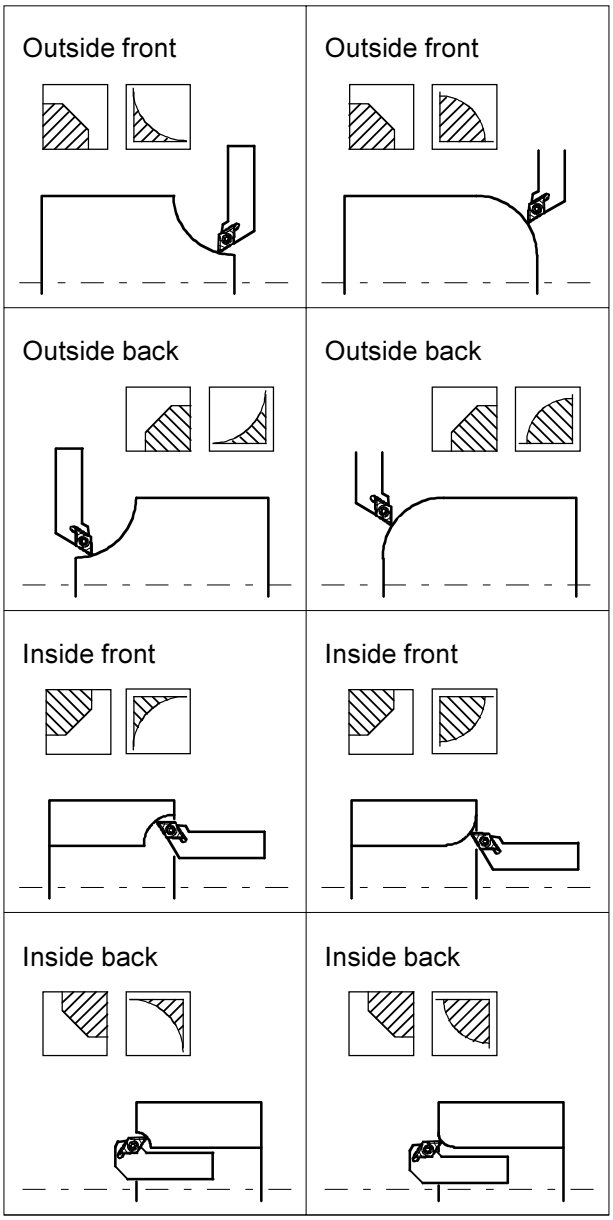
Coordinates (Xi, Zi)
Coordinates of the corner point that will be rounded.

Radius R
Radius of the rounding

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Longitudinal infeed, transversal infeed.

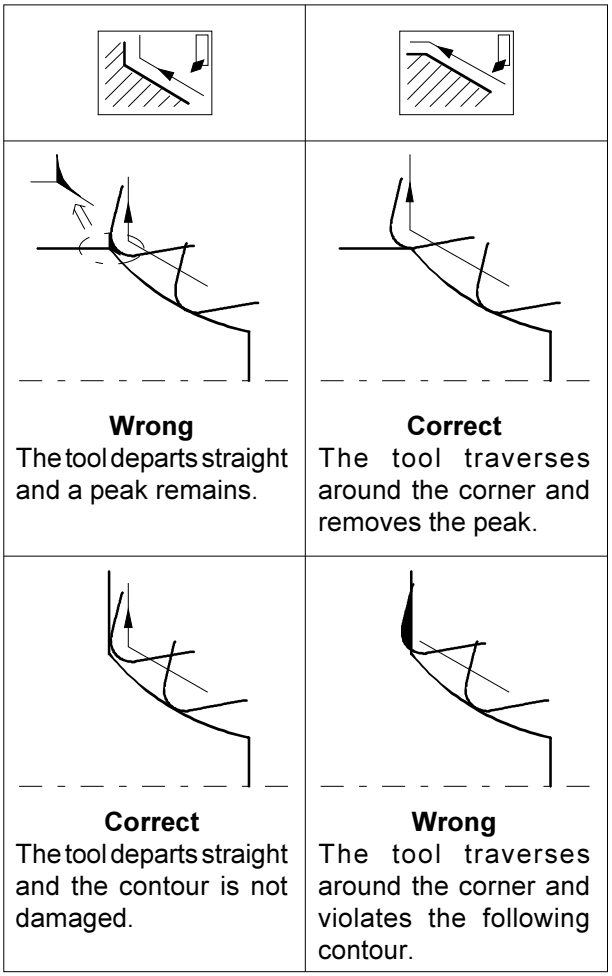
Position of the rounding



Approaching and departing the contour with tool radius compensation

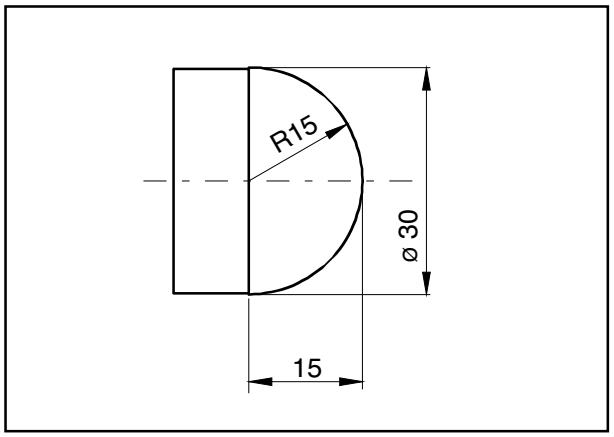
The control automatically compensates the influence of the radius at the tool tip.
The shape of the corner has to be indicated so as to avoid violation of corners or remaining peaks during approaching (e.g. ) , respectively departing (e.g. ) .

Example: Departing the contour



Wrong The tool departs straight and a peak remains.	**Correct** The tool traverses around the corner and removes the peak.
Correct The tool departs straight and the contour is not damaged.	**Wrong** The tool traverses around the corner and violates the following contour.

Choose the selection window and select the required type of corner for approaching / departing by means of the  key and confirm with  .



Note:

All value inputs have to be confirmed with ENTER.

Programming example Rounding Cycle 1

For roughing and finishing you can use various tools.

Call the rounding cycles with the key, then select Rounding Cycle 1 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.
- Select convex rounding .
- Select contour approach with INS .
- Select contour departure with INS .
- Select radius position with INS .

Coordinates (Xi,Zi)

X 30.0000
Z 0.0000

Radius

R 15.0000

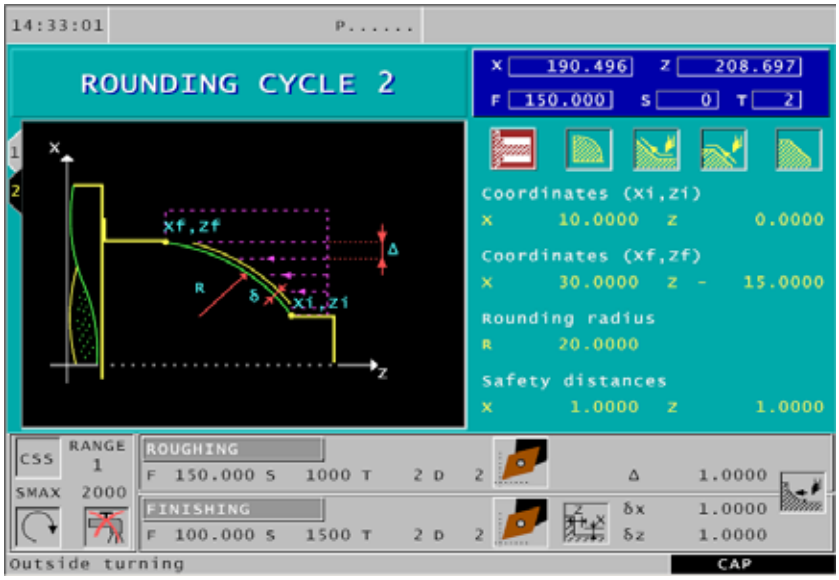
Safety distance

X 1.0000
Z 1.0000

Then please enter all necessary technological data (longitudinal and transversal infeed, CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for roughing and finishing function).

After having finished the input, the cycle can be checked by means of the graphic functions GRAPHICS (see graphic simulation).

Rounding Cycle 2



Rounding Cycle 2 enables you to machine any arc segment.

Call the rounding cycles with the key, then select Rounding Cycle 2 with LEVEL CYCLE.

Outside turning / inside turning, selection with INS.

When switching between inside and outside the screen display and the following input symbols change according to the setting.

Convex / concave rounding

Approaching the contour

Departing the contour

Radius position
You can only select positions that are possible for the respective inside and outside machining.

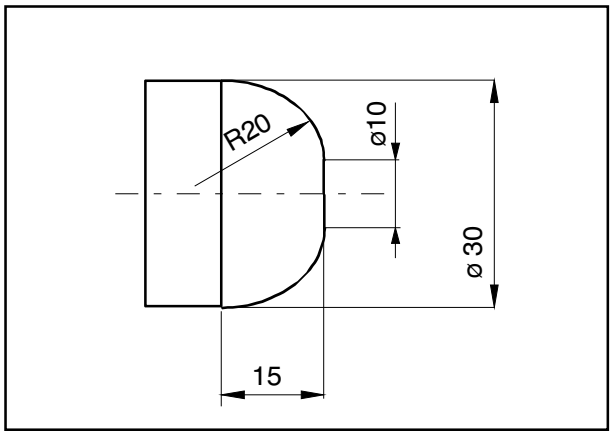
Coordinates (Xi, Zi)
Coordinates of the starting point of the rounding

Coordinates (Xf, Zf)
Coordinates of the end point of the rounding

Radius R
Radius of the rounding

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Longitudinal infeed, transversal infeed.



Note:

All value inputs have to be confirmed with ENTER.

Programming example Rounding Cycle 2

For roughing and finishing you can use various tools.

Call the rounding cycles with the key, then select Rounding Cycle 2 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.
- Select convex rounding .
- Select contour approach with INS.
- Select contour departure with INS.
- Select the radius position with INS.

Coordinates (Xi,Zi)

X 10.0000
Z 0.0000

Coordinates (Xf,Zf)

X 30.0000
Z -15.0000

Radius

R 20.0000

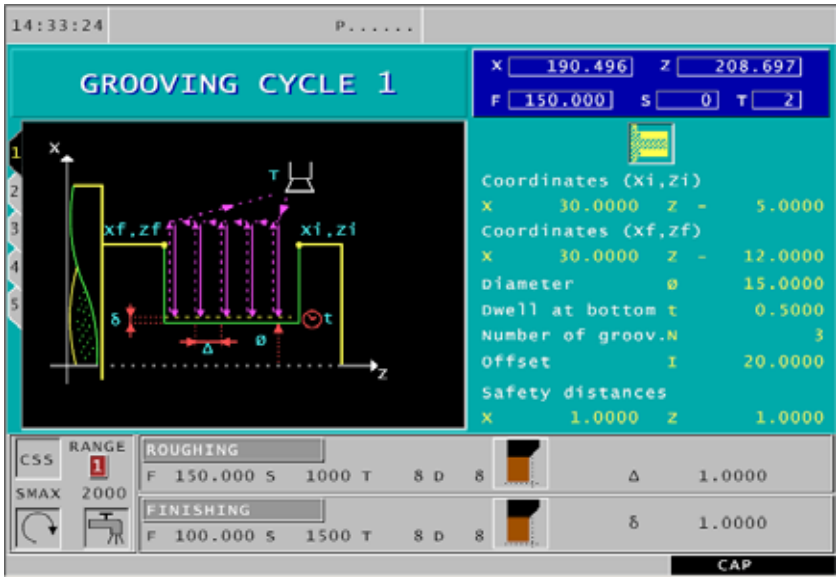
Safety distance

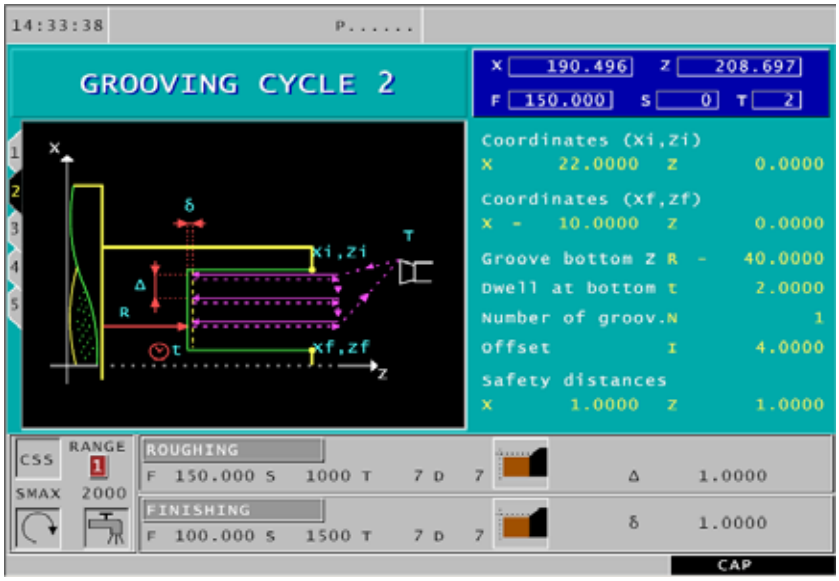
X 1.0000
Z 1.0000

Then please enter all necessary technological data (longitudinal and transversal infeed, CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for roughing and finishing function).

After having finished the input, the cycle can be checked by means of the graphic functions GRAPHICS (see graphic simulation).

Grooving Cycle 1 / 2





Note:
The groove must be **equal or larger** than the tool width.
The tool width (B) must be entered for the tool calibration by all means.

The lateral shift (input : Δ) should be a little smaller than the tool width.

Grooving Cycle 1 produces a straight, radial groove, whereas Grooving Cycle 2 produces a straight, axial groove.

Call the grooving cycles with the key, then select Grooving Cycle 1-4 with LEVEL CYCLE.

Outside turning / inside turning, selection with ING.

Coordinates (Xi, Zi)
Coordinates of the starting point of the groove

Coordinates (Xf, Zf)
Coordinates of the end point of the groove

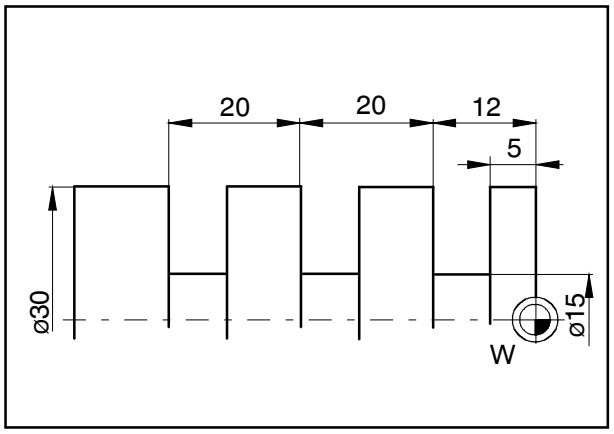
Diameter ø/ Z coordinate of the groove
Diameter of the groove bottom, respectively Z-value R (groove depth).

Dwell time t
Dwell time at the groove bottom.

Number of grooves N
Number of grooves to be produced.

Offset I
Distance between two grooves (from Xi_1Zi_1 to Xi_2Zi_2). Enter 0 for one groove.

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.



Programming example Grooving Cycle 1

For roughing and finishing you can use various tools.

Call the grooving cycles with the key, then select Grooving Cycle 1 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.

Coordinates (Xi,Zi)

X 30.0000
Z -5.0000

Coordinates (Xf,Zf)

X 30.0000
Z -12.0000

Diameter

ø 15.0000

Dwell time

t 0.5000

Number of grooves

N 3

Offset

I 20.0000

Safety distance

X 1.0000
Z 1.0000

Note:

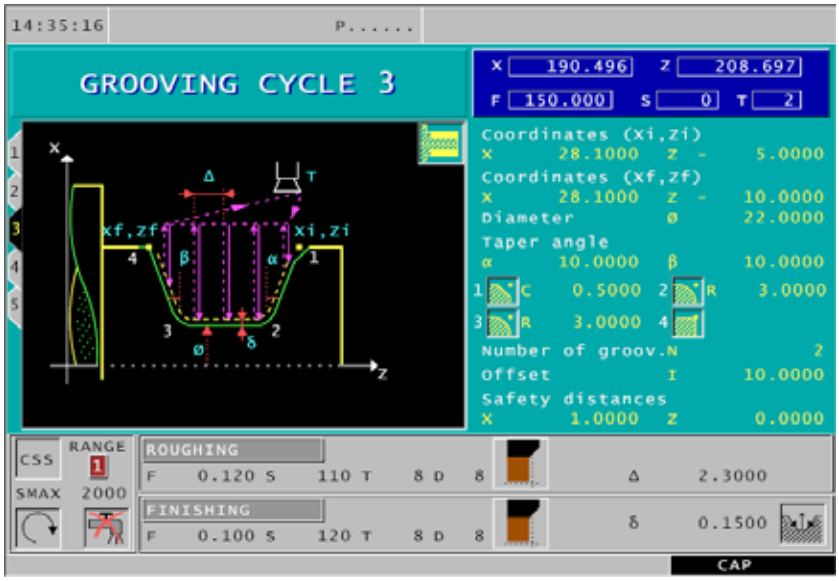
All value inputs have to be confirmed with ENTER.

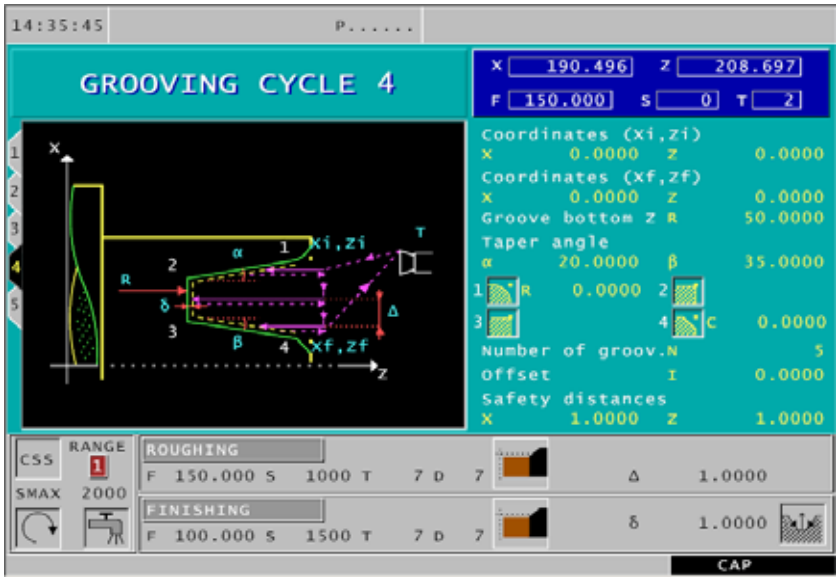
The lateral shift (input: Δ) should be a little smaller than the tool width.

Then please enter all necessary technological data (CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for roughing and finishing function).

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

Grooving Cycle 3 / 4





Note:
The groove must be **equal or larger** than the tool width.
The tool width (B) must be entered for the tool calibration by all means.

The lateral shift (input : Δ) should be a little smaller than the tool width.

Grooving Cycle 3 produces a wedge-shaped, radial groove, whereas Grooving Cycle 4 produces a wedge-shaped, axial groove.

Call the grooving cycles with the key, then select Grooving Cycle 1-4 with LEVEL CYCLE.

Inputs for:
- Outside / inside turning
- Coordinates (Xi,Zi)
- Coordinates (Xf,Xi)
- Diameter ø, respectively Z coordinate (R) of the groove (depth of groove)
- Taper angle ($\alpha + \beta$)

Corner point options 1, 2, 3, 4
For all corner points you can select the following options:

square corner

rounded (with radius input)

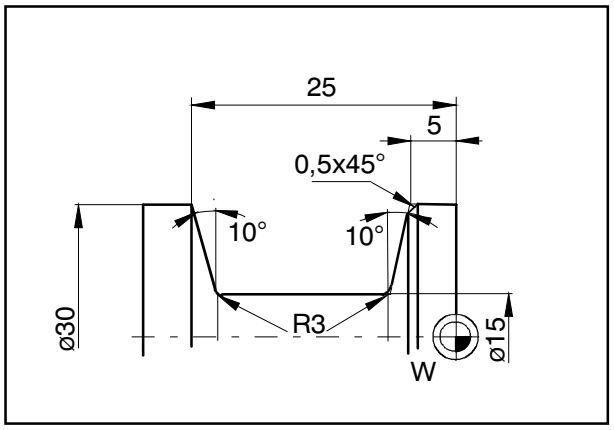
chamfered at 45° (with size indication C)

Number of grooves N
Number of grooves to be produced.

Offset I
Distance between two grooves (from Xi_1Zi_1 to Xi_2Zi_2). Enter 0 for one groove.

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Select the cutting type (,), with the INS key and confirm with ENTER.



Note:

All value inputs have to be confirmed with ENTER.

The lateral shift (input Δ) should be a little smaller than the tool width.

Programming example Grooving Cycle 3

You can use various tools for roughing and finishing.

Call the grooving cycles with the key, then select Grooving Cycle 3 with LEVEL CYCLE.

Program input

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

- Select outside turning with the INS key.

Coordinates (Xi,Zi)

X 30.0000
Z -5.0000

Coordinates (Xf,Zf)

X 30.0000
Z -25.0000

Diameter ø 15.0000

Taper angle

α 10.0000 β 10.0000

1 C 0.5000 2 R 3.0000

3 R 3.0000 4

Number of grooves N 1

Offset I
0.0000

Safety distance

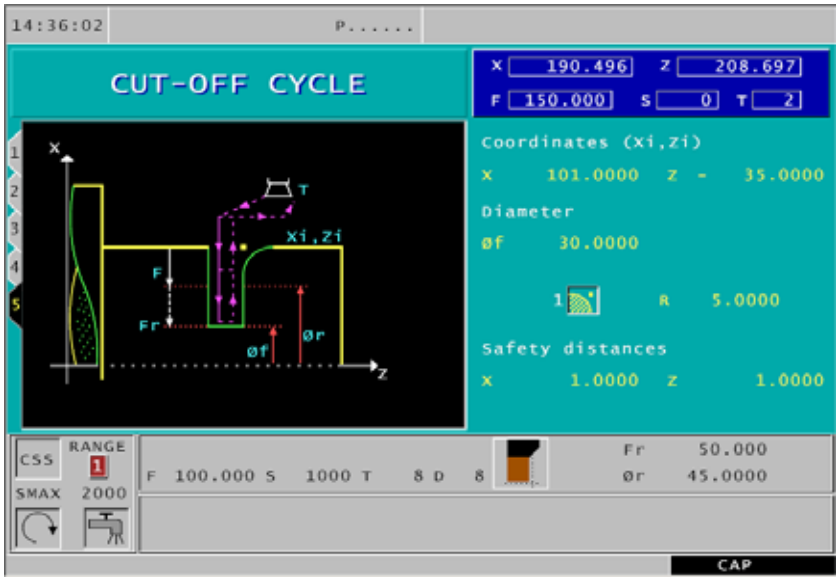
X 1.0000 Z 1.0000

Select with INS and confirm with ENTER.

Then please enter all necessary technological data (CSS, gear position, direction of rotation, maximum spindle speed, coolant function, data for roughing and finishing function).

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

Cut-Off Cycle



The Cut-Off Cycle enables the fast cutting off of turning components.

Call the grooving cycles with the key, then select the Cut-off Cycle with LEVEL CYCLE.

Note:
To cut off a workpiece øf = select 0.
The tool width (B) has to be entered for the tool calibration by all means.

Coordinates (Xi,Zi)
Coordinates of the starting point of the groove.

Diameter øf
End diameter

Corner point options
You can select the following options for the corner point:

square corner

rounded (with radius indication)

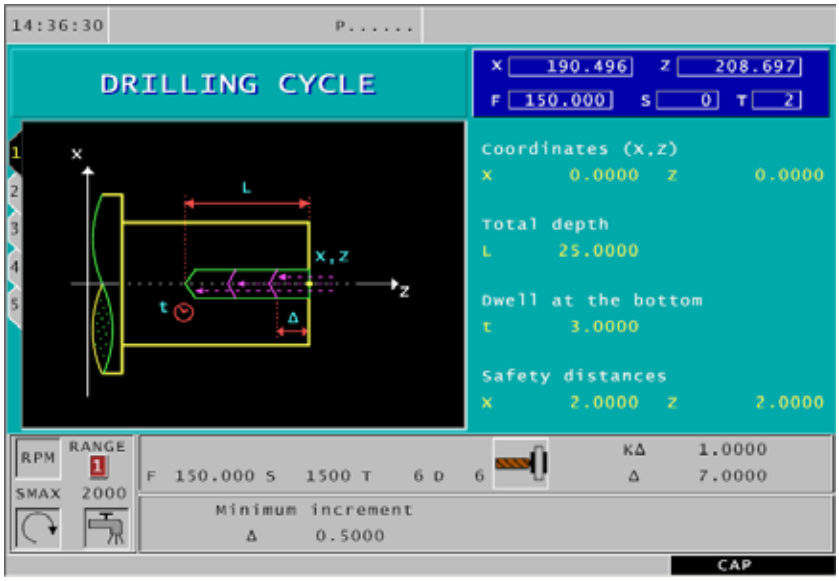
chamfered at 45° (with size indication C)

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.

Diameter ør
Intermediate diameter

Cut-off feed rate Fr
Feed rate at which the tool moves from the intermediate diameter to the end diameter. The feed rate F is hereby adjusted step by step to the feed rate Fr.

Drilling Cycle



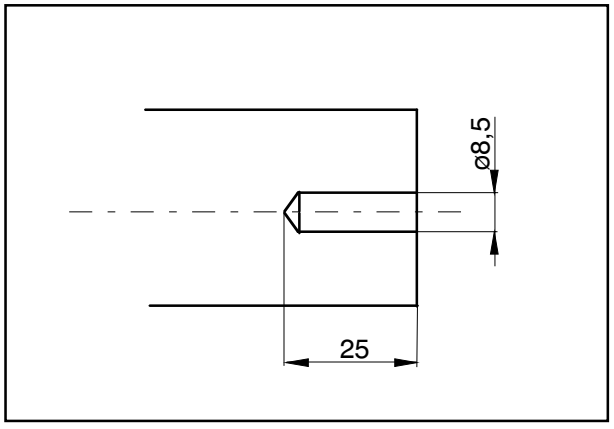
Call the drilling/milling cycles with the key, then select the Drilling Cycle with LEVEL CYCLE.

Coordinates (X,Z)
Coordinates of the starting point

Total depth L
Effective drilling depth in Z

Dwell time at the bottom t
Dwell time at the bottom in seconds

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.



Programming example Drilling Cycle

Call the drilling/milling cycles with the key, then select the Drilling Cycle with LEVEL CYCLE.

Program input
Use the direction keys or the enter key to move within the cycle program.

Coordinatens (X,Z)

X 0.0000
Z 0.0000

Total depth

L 25.0000

Dwell time at the bottom

t 3.0000

Safety distance

X 2.0000
Z 2.0000

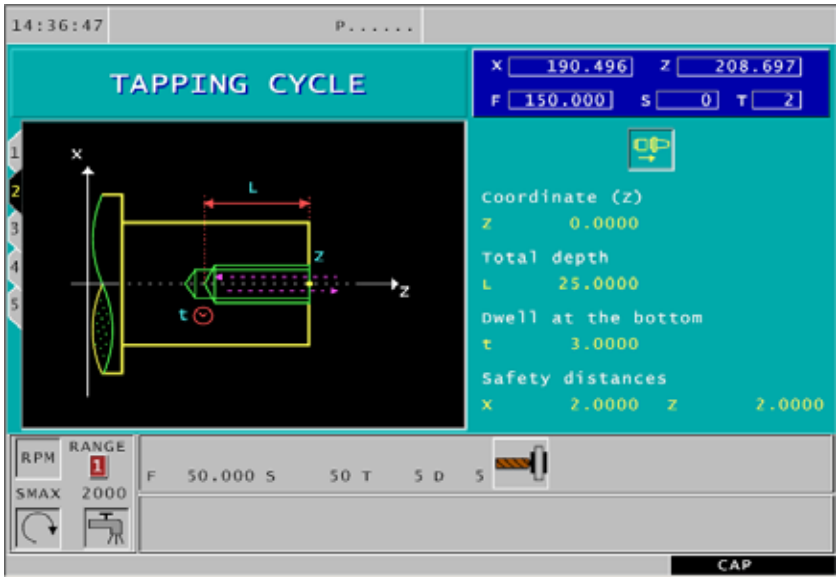
Then please enter all necessary technological data (RPM, gear position, direction of rotation, maximum spindle speed, coolant function, turning values).

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

Note:
For the X value "always" enter 0, otherwise the workpiece contour will be violated and the tool could break.

All value inputs have to be confirmed with ENTER.

Tapping Cycle



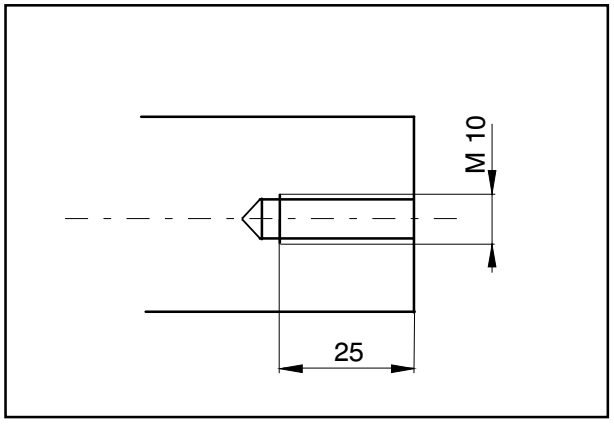
Call the drilling/milling cycles with the key, then select the Tapping Cycle with LEVEL CYCLE.

Coordinates (Z)
Coordinates of the starting point

Total depth L
Depth of thread in Z

Dwell time at the bottom t
Dwell time at the bottom in seconds

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.



Length compensation

....Tapping with length compensation

....Tapping without length compensation

Selection with the key.

Note:
Tapping with length compensation toolholder only.

All value inputs have to be confirmed with ENTER.

Programming example Tapping Cycle

Call the drilling/milling cycles with the key, then select the Tapping Cycle with LEVEL CYCLE.

Program input

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

Coordinates (X,Z)

Z 0.0000

Total depth

L 25.0000

Dwell time at the bottom

t 3.0000

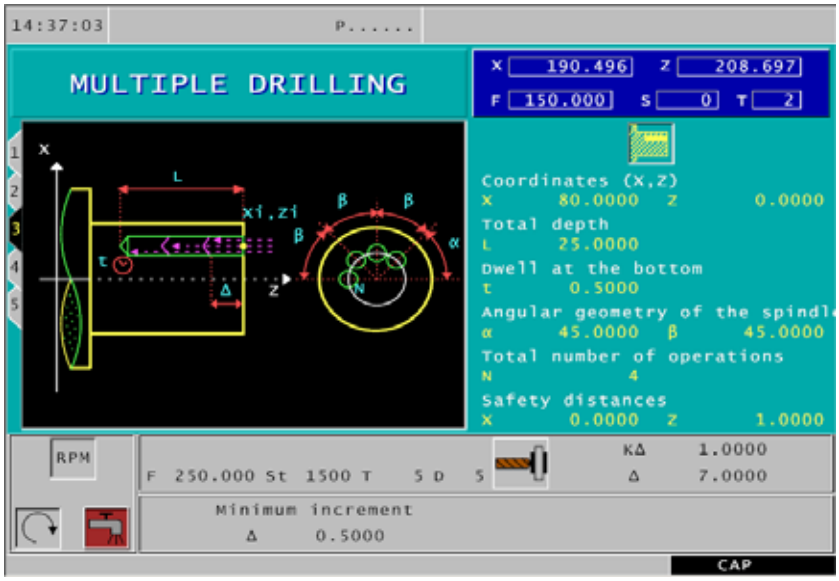
Safety distance

X 2.0000
Z 2.0000

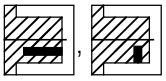
Then please enter all necessary technological data (RPM, gear position, direction of rotation, maximum spindle speed, coolant function, turning values).

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

Multiple Drilling



Call the drilling/milling cycles with the key, then select Multiple Drilling with LEVEL CYCLE.



Transversal and longitudinal drilling,

selection with the INS key.

Note:
The feed rate always has to be programmed in mm/min.


.....clockwise rotation


.....counterclockwise rotation

This cycle can only be programmed in connection with driven tools.

Coordinates (X,Z)
Coordinates of the starting point (diameter of circular hole pattern)

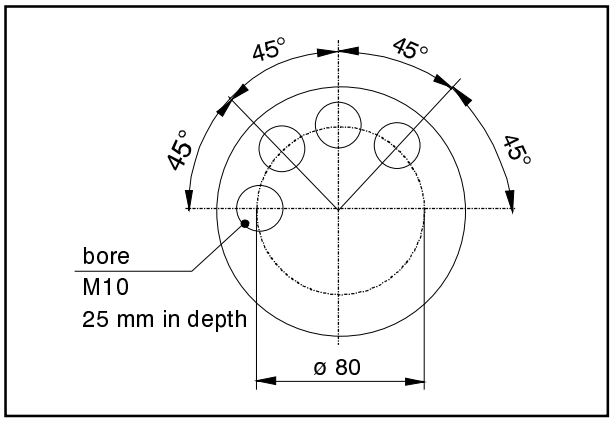
Total depth L
Effective drilling depth in Z

Dwell time at the bottom t
Dwell time at the bottom in seconds

Angular geometry of the spindle α, β
Angular offset of the bores

Total number of operations N
Number of drilling operations

Safety distance
Indicates the position of the approach point in relation to the cycle starting point.



Note:

All value inputs have to be confirmed with ENTER.

Programming example Multiple Drilling

Call the drilling/milling cycles with the key, then select Multiple Drilling with LEVEL CYCLE.

Program input

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

Select transversal drilling with the INS key.

Coordinates (X,Z)

X 80.0000
Z 0.0000

Total depth

L 25.0000

Dwell time at the bottom

t 0.5000

Angular geometry of the spindle

α 45.0000 β 45.000

Total number of operations

N 4

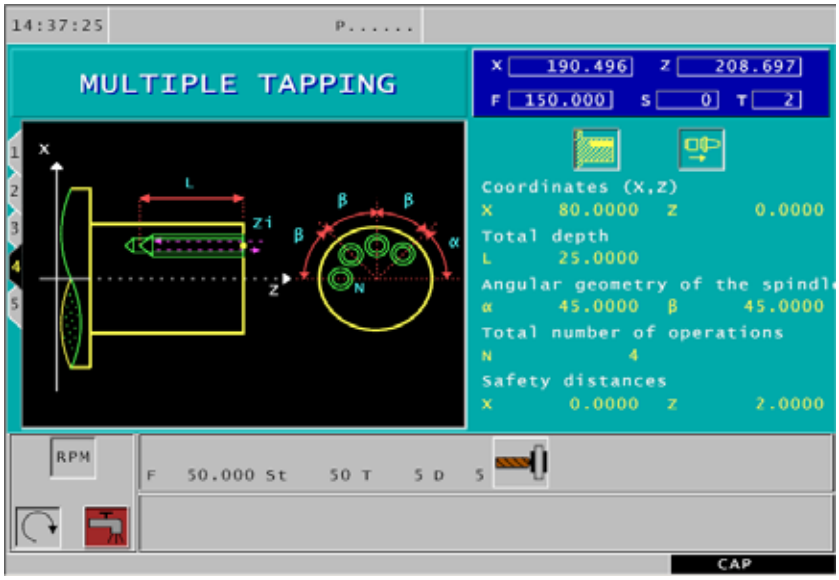
Safety distance

X 0.0000
Z 2.0000

Then please enter all necessary technological data (coolant function, maximum spindle speed, turning values).

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

Multiple Tapping



Call the drilling/milling cycles with the key, then select Multiple Tapping with LEVEL CYCLE.

, Transversal and longitudinal tapping,

selection with the INS key.

Length compensation

....Tapping with length compensation

....Tapping without length compensation

Selection with the INS key.

Note:
You get the feed rate F by means of the formula:
F(feed rate) = threading pitch X spindle speed

This cycle can only be programmed in connection with driven tools.

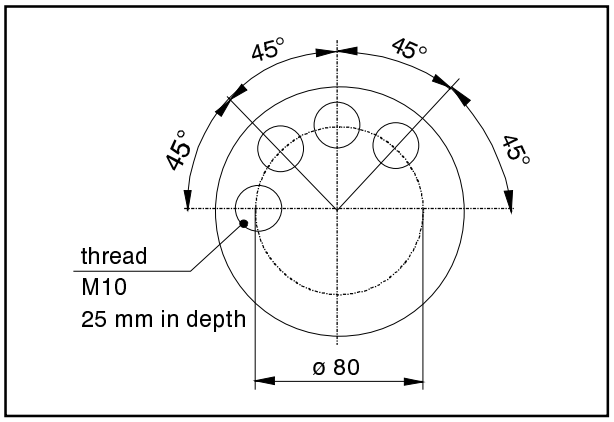
Coordinates (X,Z)
Coordinates of the starting point (diameter of the pitch circle)

Total depth L
Effective tapping depth in Z

Angular geometry of the spindle α, β
Angular offset of the bores

Total number of drilling operations N
Number of tapping operations

Safety distance X, Z
Indicates the position of the approach point in relation to the cycle starting point.



Note:

All value entries have to be confirmed with ENTER.

Programming example Multiple Tapping

Call the drilling/milling cycles with the key, then select Multiple Tapping with LEVEL CYCLE.

Program input

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

Selection of transversal tapping with the INS key .

Coordinates (X,Z)

X 80.0000
Z 0.0000

Total depth

L 25.0000

Angular geometry of the spindle

α -45.0000 β -45.000

Total number of operations

N 4

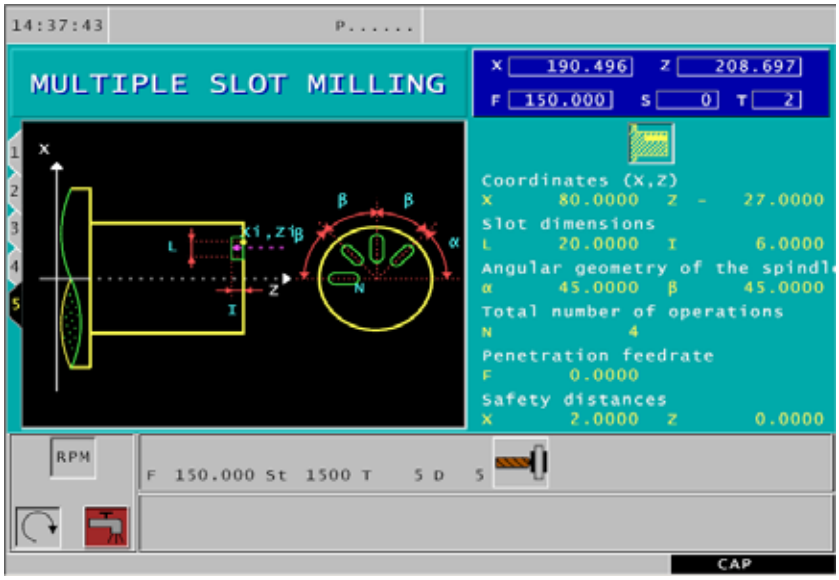
Safety distance

X 0.0000
Z 2.0000

Then please enter all necessary technological data (coolant function, maximum spindle speed, turning values).

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

Multiple Slot Milling



Call the drilling/milling cycles with the key, then select Multiple Slot Milling with LEVEL CYCLE.

Select transversal / longitudinal slot milling with the INS key.

Note:
The feed rate always has to be programmed in mm/min.

This cycle can only be programmed in connection with driven tools.

Coordinates (X,Z)
Coordinates of the starting point (diameter of circular hole pattern)

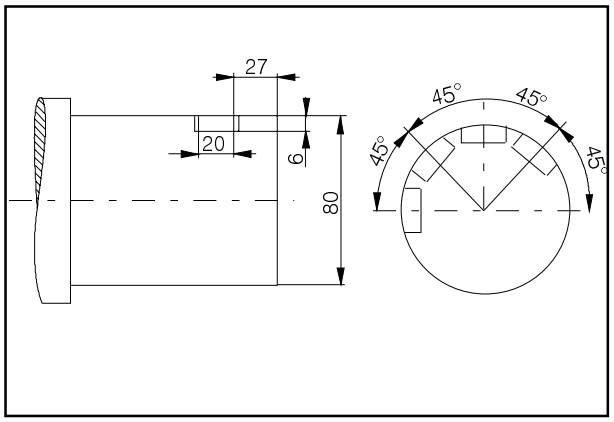
Dimensions for slot milling L, I
Milling length and depth

Angular geometry of the spindle α, β
Angular offset of the bores

Total number of operations N
Number of milling operations

Penetration feed rate F
Penetration feed rate in mm/min

Safety distance X, Z
Indicates the position of the approach point in relation to the cycle starting point.



Note:

All value inputs have to be confirmed with ENTER.

Programming example Multiple Slot Milling

Call the drilling/milling cycles with the key, then select Multiple Slot Milling with LEVEL CYCLE.

Program input

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

Selection of longitudinal milling with the INS key.

Coordinates (X,Z)

X 80.0000 Z -27.0000

Dimensions for slot milling

L 20.0000 I 6.0000

Angular geometry of the spindle

α 45.0000 β 45.000

Total number of operations

N 4

Penetration feed rate

F 100.0000

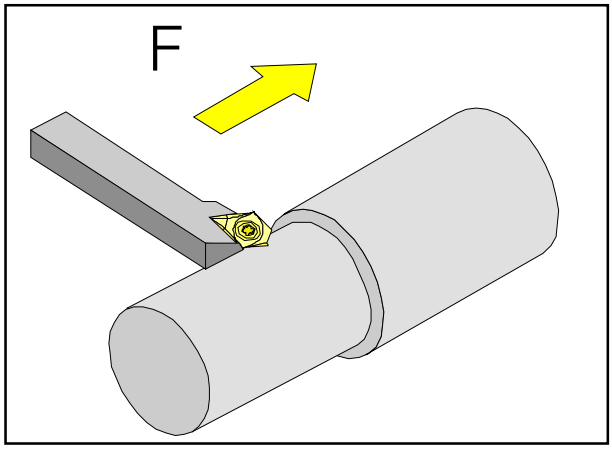
Safety distance

X 2.0000 Z 0.0000

Then please enter all necessary technological data (coolant function, maximum spindle speed, turning values).

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

E: Tool programming



Tool related data

Feed rate F

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

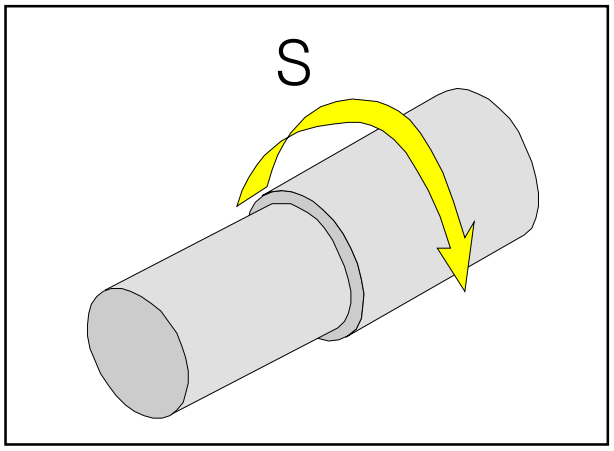
Input

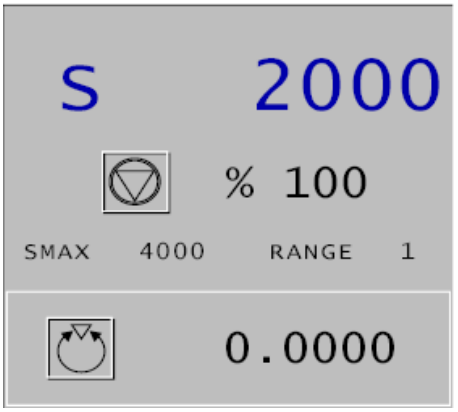
- Press the F key.
- Enter the required feed rate.
- Press ENTER so that the WinNC captures the new value for the axis feed.

  To abort, press ESC.

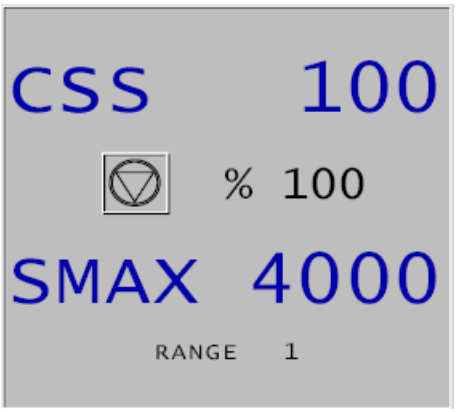
Rapid traverse
For rapid traverse enter F99999.

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





Input box for spindle speed in RPM



Input box for constant cutting speed (CSS)

Spindle speed S

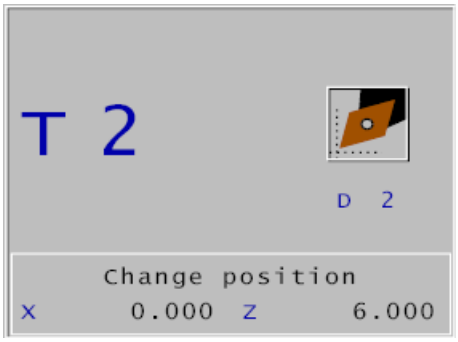
The WinNC offers the possibility to work with rotations per minute (RPM) and spindle orientation or with constant cutting speed (CSS).
In the constant cutting speed mode you define the tangential speed that should remain constant between tool tip and workpiece.
Therefore, the spindle speed depends on the position of the tool tip towards the rotation axis. When the tool tip moves away from the rotation axis, the spindle speed is reduced - when it it approaches the rotation axis it is increased.

Input

- Press  to switch between (RPM) and (CSS).
- To select another spindle speed press the  key. The highlight jumps on the current value.
- Enter the new value and press the  key.
- Spindle condition: clockwise rotation: press the key  for a short time, counterclockwise rotation: press the key  for at least 1 sec.
- Percentage (%) of the nominal spindle speed / constant cutting that is applied. If you wish to change it, press  ,  or  .
- If you wish to change the maximum spindle speed, press the key  twice. The highlight jumps on the current value.
- Enter the new value and press  .

Adjustment during program run

During program run you can adjust the spindle speed by means of the override turning knob S for the spindle speed.



Tool control

The standard screen displays the following information regarding the tool:

- In large figures the number "T" of the selected tool and a graphic display of its tip.
- The offset number "D" that is associated with the tool.
- The coordinate values of the tool changing point.

Selecting a new tool

- Press the T key. The highlight jumps on the current tool number.
- Enter the number of the required tool.
- After you have pressed the key, the WinNC will execute the tool change.
- When the new tool has been selected, the WinNC updates the graphic illustration of the new tool`s shape.

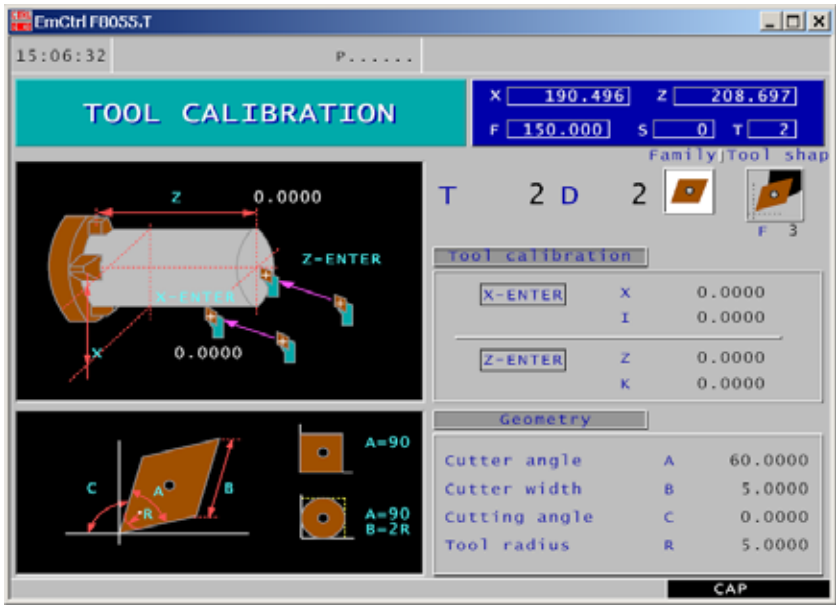
Tool changing point

There is only one tool changing point for the whole workpiece program.
The tool changing point that was programmed last, remains active until a new point will be defined.

- Press the T key. The highlight jumps on the current tool number.
- Press the keys X and Z.
  Now you have the possibility to:
  a.) enter the required value manually and confirm with ENTER.
  b.) move the axes to the required point by means of the JOG keys and enter the current position of the machine. Press the RECALL key. The WinNC assigns the coordinate value to the selected field. Confirm with ENTER.

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

Tool calibration



Press the key to access the tool calibration mode.

Clamping the workpiece

Clamp a workpiece with known dimensions in the workpiece holder.
Define the workpiece dimensions:

X (value) ENTER

Z (value) ENTER

Defining tool data

Define the tool number "T":

(tool number) and press .

Define the number of the tool offset "D":

(number of the tool offset) and press .

Select the tool type or the tool family with the key:

The following types are possible:

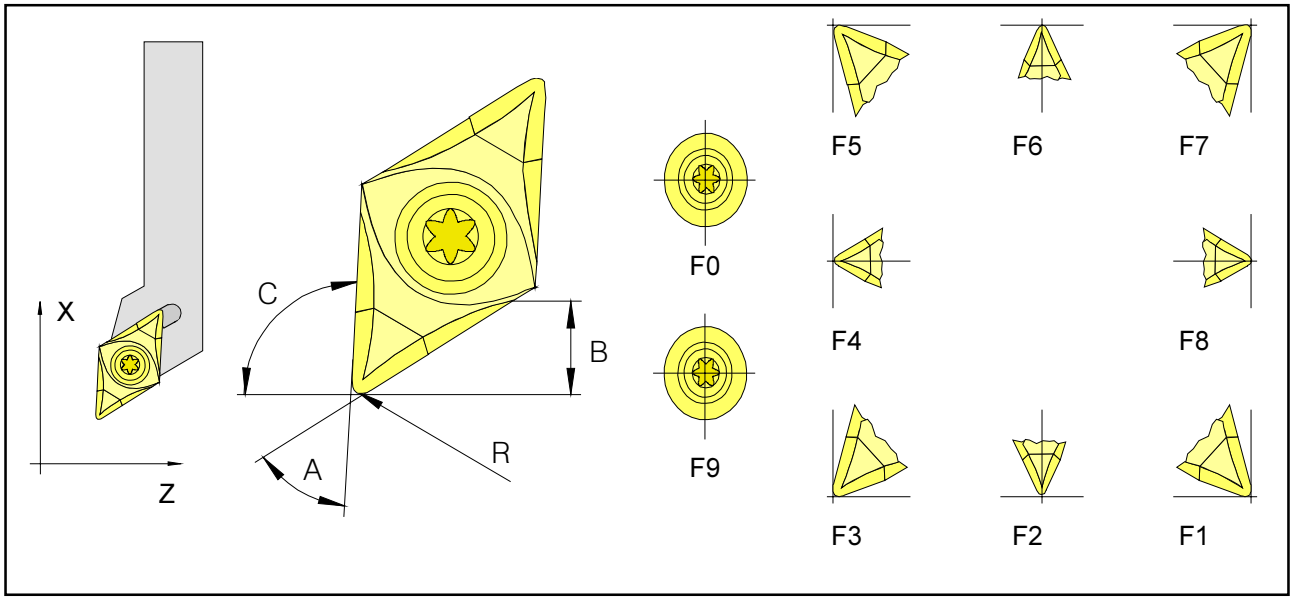
- (rhombic carbide tip)
- (carbide tip for threading)
- (carbide tip for grooving)
- (circular carbide tip)
- (drill or driven tool)

Define the tool shape with the key.

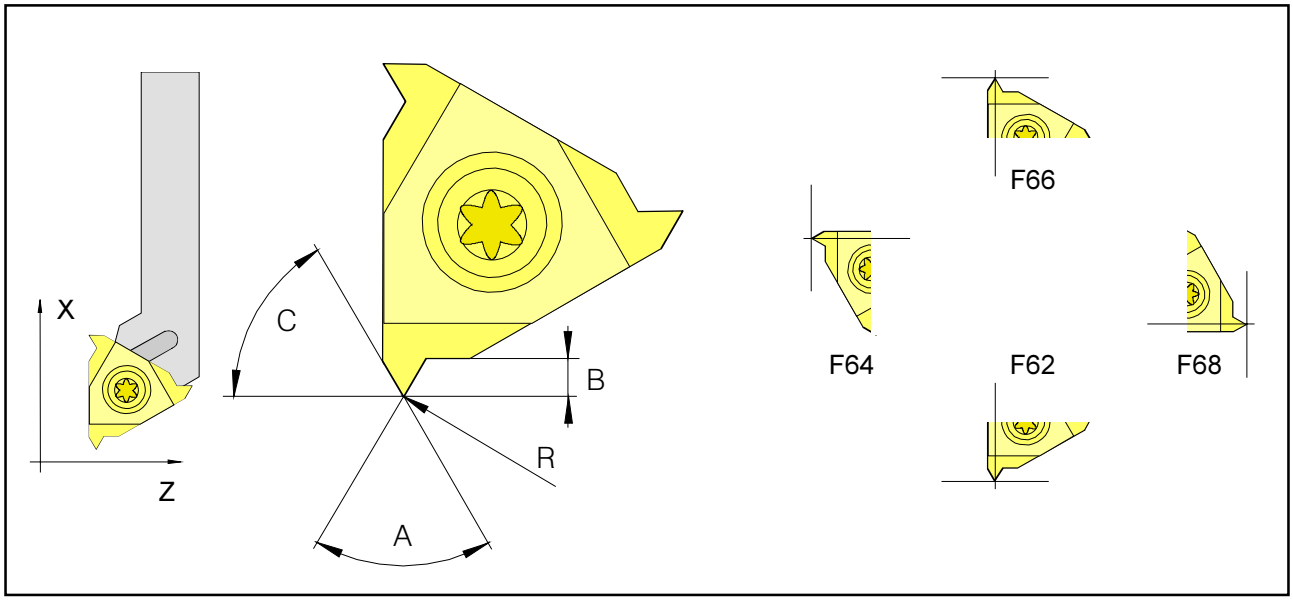
Note:

All inputs have to be confirmed with .

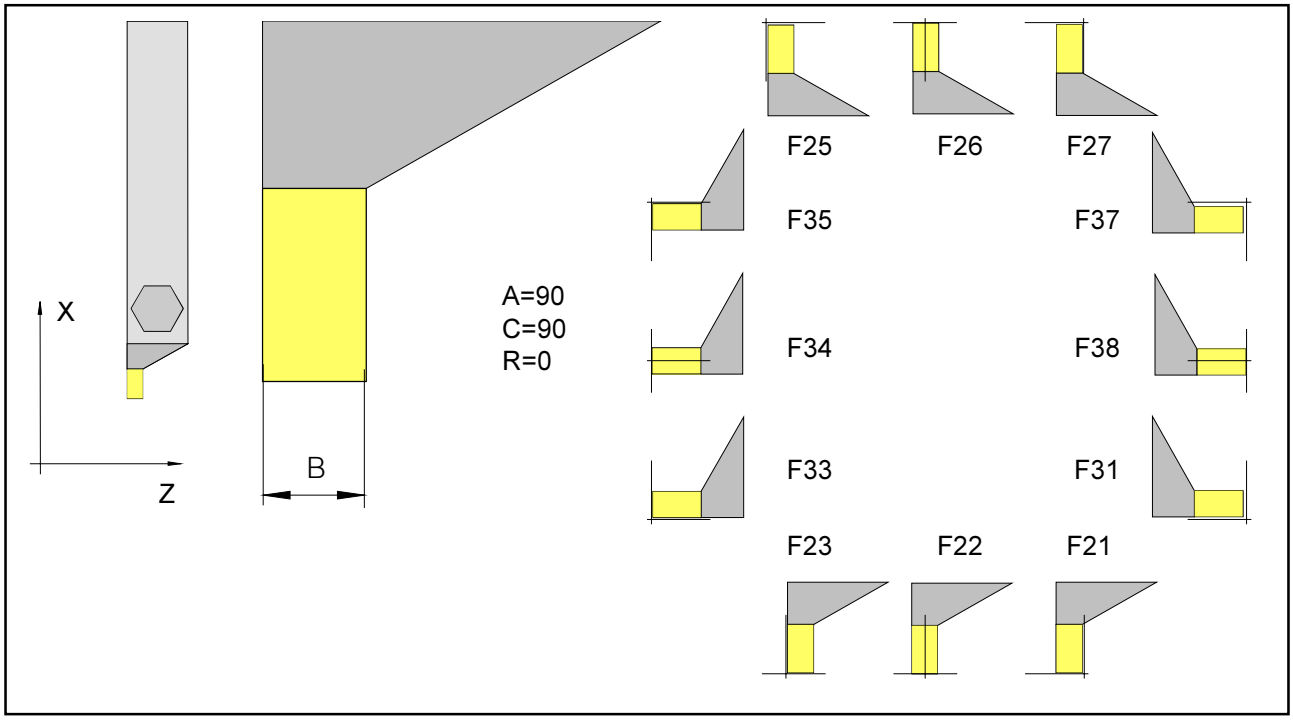
Available tool tip positions for the type:



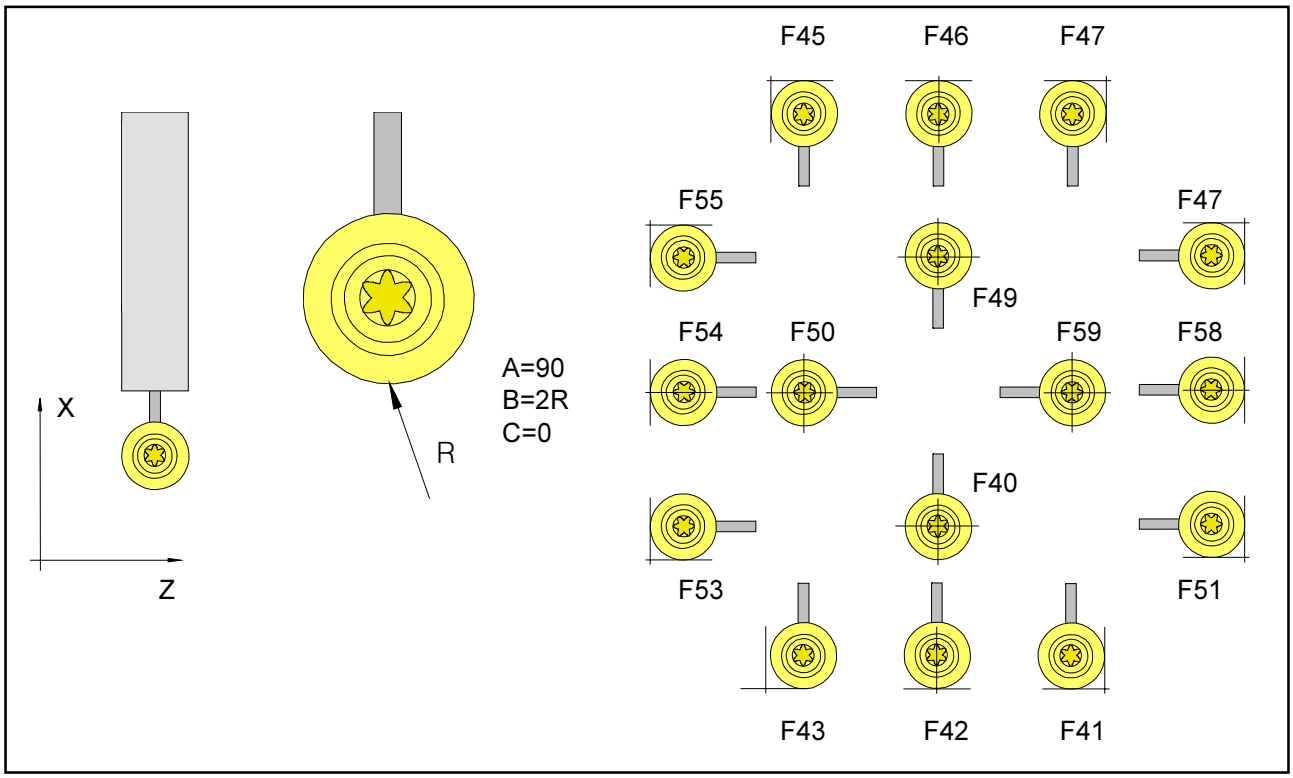
Available tool tip positions for the type:



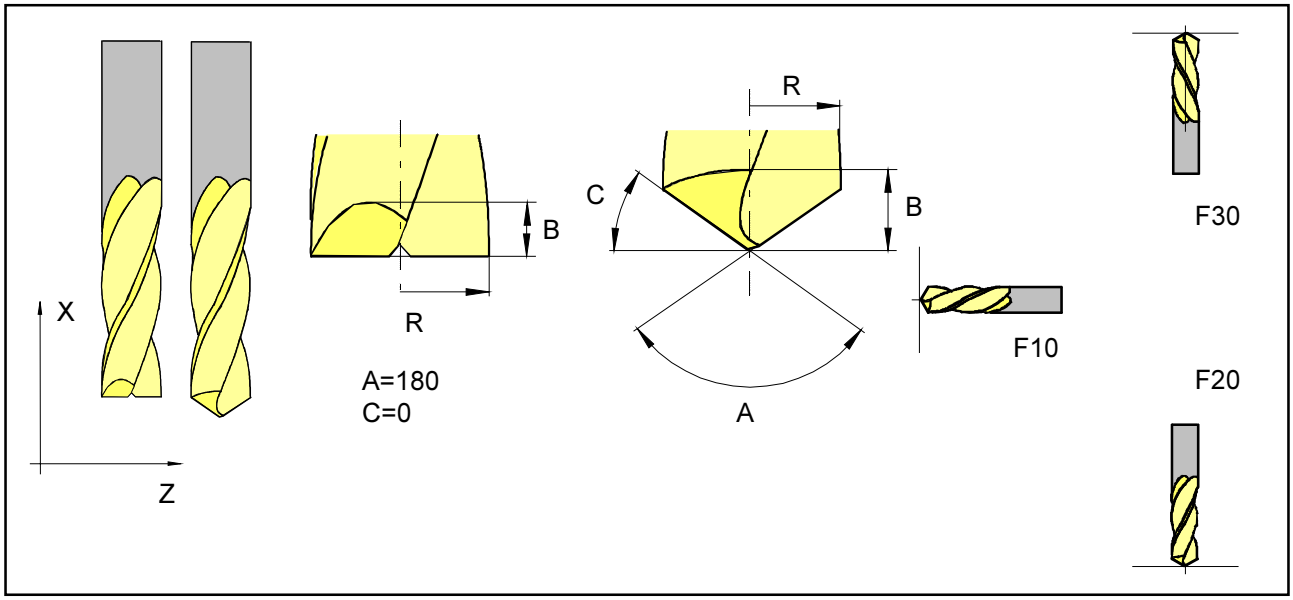
Available tool tip positions for the type:



Available tool tip positions for the type:



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 x ø

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.

Note:
The data X and Z indicate the dimensions of the tool in X and Z. The data I and K indicate the tool offset, which the WinNC must consider in order to compensate the tool wear.

The WinNC adds together the value of the tool offset (I, K) and the length (X, Z) to get the actual tool length (X+I) and (Z+K), that it must use.
The value "I" (tool offset of the longitudinal wear in X direction) has to be defined in radii.

Tool calibration with the touching method

- Enter the X value of the test part.
- Define the Z value of the test part with "0".
- The tool must touch the diameter of the workpiece, then press A R X ENTER.
- The tool must touch the face of the workpiece, then press E Y Z ENTER.
- Now the tool is calibrated. The WinNC updates the data X and Z and sets I and K to 0.
- Assign the geometry data of the tool to the respective tool types.

To leave the tool calibration mode, press ESC.

F: Program run

Requirements

Datum setting

The used datums must be measured and entered.

Tools

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

Reference point

The reference point must be traversed in all axes.

Machine

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

Alarms

There must not be any alarms activated.

Program start, Program stop

Select a program for machining.

Press the  key.

Stop program with  , continue with  .

Abort program with  .

H: Alarms and Messages

Input Device Alarms 3000 - 3999

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

Missing digitizer calibration
Cause: A digitizer tablet has been installed but not calibrated
Remedy: Calibrate digitizer tablet (set corner points), see External Input Devices

3001 General RS232 communication error
Remedy: Correct settings of serial interfacae.

3002 Control keyboard missing
Remedy: Connect control keyboard, switch on, ...

3003 Digitizer missing
Remedy: Connect digitizer, switch on, ...

3004 Check sum error in control keyboard
The keyboard tries an automatic re-initializing - when failed switch off / on keyboard.

3005 Error in control keyboard
The keyboard tries an automatic re-initializing - when failed switch off / on keyboard.

3006 Error with initializing control keyboard
The keyboard tries an automatic re-initializing - when failed switch off / on keyboard.

Machine Alarms 6000 - 7999

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

PC MILL 50 / 55, PC TURN 50 / 55

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

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: CYCLE TIME EXCEEDS LIMIT
Contact EMCO Service.

6002: NO PLC PROGRAM LOADED
Contact EMCO Service.

6003: DB NOT EXISTENT
Contact EMCO Service.

6004: RAM ERROR ON PLC BOARD
Contact EMCO Service.

6009: FAILURE SAFETY CIRCUIT
Defective door limit switch or main contactor. Operating the machine is not possible.
Contact EMCO Service.

6010: X-AXIS NOT READY
Step motor board defective, 24 V or 30 V fuse defective. Check fuses and switch box fan filter.
Contact EMCO Service.

6011: Y-AXIS NOT READY
see alarm 6010.

6012: Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, cable defective, main drive overload.
Check fuse, reduce load.
Contact EMCO service.

6014: NO SPEED FOR MAIN SPINDLE
This will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).

6019: VICE TIMEOUT
24 V fuse defective, hardware defective.
Contact EMCO service.

6020: VICE FAILURE
24 V fuse defective, hardware defective.
Contact EMCO service.

6024: DOOR NOT CLOSED
The door was opened while a machine movement. The program will be aborted.

6025: GEARBOX COVER NOT CLOSED
The gearbox cover was opened while a machine movement. A running CNC program will be aborted. Close the cover to continue.

6027: DOOR LIMIT SWITCH DEFECTIVE
The limit switch of the automatic door is displaced, defective, wrong cabled.
Contact EMCO service.

6028: DOOR TIMEOUT
The automatic door stucks, the pressured air supply is insufficient, the limit switch is displaced.
Check door, pressured air supply, limit switch or contact EMCO service.

6030: NO PART CLAMPED
No workpiece inserted, vice cheek displaced, control cam displaced, hardware defective.
Adjust or contact EMCO service.

6031: QUILL FAILURE

6037: CHUCK TIMEOUT

6039: CHUCK PRESSURE FAILURE

6041: TOOL CHANGE TIMEOUT
Tool turret stucks (collision?), 24 V fuse defective, hardware defective.
A running CNC program will be stopped.
Check for a collision or contact EMCO service.

6042: TOOL CHANGE TIMEOUT
see alarm 6041.

6043: TOOL CHANGE TIMEOUT
see alarm 6041.

6044: TOOL TURRET SYNC ERROR
Hardware defective.
Contact EMCO service.

6046: TOOL TURRET SYNC MISSING
Hardware defective.
Contact EMCO service.

6048: DIVIDING TIME EXCEEDED
Dividing head stucks, insufficient pressured air supply, hardware defective.
Check for collision, check pressured air supply or contact EMCO service.

6049: INTERLOCKING TIME EXCEEDED
see alarm 6048

6050: FAILURE DIVIDING DEVICE
Hardware defective.
Contact EMCO service.

7000: INVALID TOOL NUMBER
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7007: FEED HOLD
In the robotic mode a HIGH signal is at input E3.7.
Feed Stop is active until a low signal is at E3.7.

7017: GO FOR REFERENCE POINT
Approach the reference point.

7040: DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated.
Some accessories can be operated only with open machine door.
Close the machine to run a program.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7050: NO PART CLAMPED
After switching on or after an the vice is neither at the open position nor at the closed position.
NC-Start is locked.
Traverse the vice manually on a valid end position.

7051: DIVIDING DEVICE NOT INTERLOCKED
After switching on or after an the dividing head is not in a lock position. NC-Start is locked.

PC MILL 100/105/125/155

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

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed. Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
Contact EMCO Service.

6003: PLC - NO DATA UNIT
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
Contact EMCO Service.

6005: OVERHEAT BRAKEMODUL
Main drive was braked too often, large changes of speed within a short time. E4.2 active

6006: OVERLOAD BRAKE RESISTOR
see 6005

6007: SAFETY CIRCUIT FAULT
Axis and main drive contactor with machine switched off not disabled. Contactor got stuck or contact error. E4.7 was not active during switch-on.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse or cabling is defective.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6011: DRIVE Y-AXIS NOT READY
see alarm 6010.

6012: DRIVE Z-AXIS NOT READY
see alarm 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective, main drive too hot, fuse defective.
A running program will be stopped, the 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.

6024: MACHINE DOOR OPEN
The door was opened while a machine movement. The program will be aborted.

6040: TOOL TURRET INDEX FAILURE
After WZW procedure drum pressed down by Z-axis. Spindle position wrong or mechanical defect. E4.3=0 in lower state

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), main drive not ready, fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6043-6046: TOOL DISK POSITION FAULT
Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.
The Z axis could have been slipped out of the toothing while the machine was switched off.
A running CNC program will be stopped.
Contact EMCO service.

6047: TOOL DISK UNLOCKED
Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.
A running CNC program will be interrupted.
Contact EMCO service.
When the tool drum is turned out of locked position (no defect), act as following:
Turn the drum into locking position manually
Change into MANUAL (JOG) mode.
Turn the key switch. Traverse the Z slide upwards, until the alarm disappears.

6050: M25 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

6064: DOOR AUTOMATIC NOT READY
Cause: pressure failure automatic door
automatic door stucks mechanically
limit switch for open end position defective
security print circuits defect
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic door

6069: CLAMPING FOR TANI NOT OPEN
When opening the clamping pressure switch does not fall within 400ms. Pressure switch defective or mechanical problem. E22.3

6070: PRESSURE SWITCH FOR TANI MISSING
When closing the clamping pressure switch does not respond. No compressed air or mechanical problem. E22.3

6071: DIVIDING DEVICE NOT READY
Servo Ready Signal from frequency converter missing. Excess temperature drive TANI or frequency converter not ready for operation.

6072: VICE NOT READY
Attempt to start the spindle with an open vice or without clamped workpiece.
Vice stucks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.
Check the fuses or contact EMCO service.

6073: DIVIDING DEVICE NOT READY
Cause: locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device
lock the dividing device

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
locking switch defective
cabling defective
fuses defective
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: service automatic dividing device

6075: M27 AT RUNNING MAIN SPINDLE
Cause: Programming mistake in NC program.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Correct NC program

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 10.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives.

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is not possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: INITIALIZE TOOL TURRET
The tool turret operating was interrupted.
No traversing operation is possible.
Press tool turret key in JOG operation. Message occurs after alarm 6040.

7022: INITIALIZE TOOL TURRET !
see 7021

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated (except special operation mode)
Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7054: VICE OPEN
Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6073 (vice not ready) will be released.
Remedy: Clamp

7055: DIVIDING DEVICE NOT LOCKED
Cause: the dividing device is not locked
When switching on the main spindle with M3/M4 alarm 6073 (dividing device not ready) will be released.
Remedy: lock dividing device

7270: OFFSET COMPENSATION ACTIVE !
Only with PC-MILL 105
Offset compensation activated by the following operation sequence.
- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4

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

7271: COMPENSATION FINISHED,DATA SAVED !
see 7270

PC TURN 105/120/125/155

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

6000: EMERGENCY OFF
The EMERGENCY OFF key was pressed.
The reference position will be lost, the auxiliary drives will be switched off.
Remove the endangering situation and restart machine and software.

6001: PLC-CYCLE TIME EXCEEDING
The auxiliary drives will be switched off.
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
The auxiliary drives will be switched off.
Contact EMCO Service.

6003: PLC - NO DATA UNIT
The auxiliary drives will be switched off.
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
The auxiliary drives will be switched off.
Contact EMCO Service.

6008: MISSING CAN SUBSCRIBER
Check fuses or contact EMCO Service.

6009: SAFETY CIRCUIT FAULT
Defective step motor system.
A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.
Contact EMCO Service.

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6012: DRIVE Z-AXIS NOT READY
see 6010.

6013: MAIN DRIVE NOT READY
Main drive power supply defective or main drive too hot, fuse defective, over- or undervoltage from mains.
A running program will be stopped, the auxilliary drives will be switched off.
Check fuses or contact EMCO Service.

6014: NO MAIN SPINDLE SPEED
This alarm will be released, when the spindle speed is lower than 20 rpm because of overload.
Alter cutting data (feed, infeed, spindle speed).
The CNC program will be aborted, the auxiliary drives will be switched off.

6015: NO DRIVEN TOOL SPINDLE SPEED
see 6014.

6024: MACHINE DOOR OPEN
The door was opened while a machine movement.
The program will be aborted.

6040: TOOL TURRET INDEX FAILURE
The tool turret is in no locked position, tool turret sensor board defective, cabling defective, fuse defective.
A running CNC program will be stopped.
Swivel the tool turret with the tool turret key, check fuses or contact EMCO service.

6041: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6042: TOOL TURRET OVERHEAT
Tool turret motor too hot.
With the tool turret a max. of 14 swivel procedures a minute may be carried out.

6043: TOOL CHANGE TIMEOUT
Tool drum stucks (collision?), fuse defective, hardware defective.
A running CNC program will be stopped.
Check for collisions, check fuses or contact EMCO service.

6046: TOOL TURRET ENCODER FAULT
Fuse defective, hardware defective.
Check fuses or contact EMCO service.

6048: CHUCK NOT READY
Attempt to start the spindle with open chuck or without clamped workpiece.
Chuck stucks mechanically, insufficient pressured air supply, fuse defective, hardware defective.
Check fuses or contact EMCO service.

6049: COLLET NOT READY
see 6048

6050: M25 DURING SPINDLE ROTATION
With M25 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6055: NO PART CLAMPED
This alarm occurs when with rotating spindle the clamping device or the tailstock reach the end position.
The workpiece has been pushed out of the chuck or has been pushed into the chuck by the tailstock.
Check clamping device settings, clamping forces, alter cutting data.

6056: QUILL NOT READY
Attempt to start the spindle or to move an axis or to swivel the tool turret with undefined tailstock position.
Tailstock is locked mechanically (collision), insufficient pressured air supply, fuse defective, magnetic switch defective.
Check for collisions, check fuses or contact EMCO service.

6057: M20/M21 DURING SPINDLE ROTATION
With M20/M21 the main spindle must stand still (consider run-out time, evtl. program a dwell)

6058: M25/M26 DURING QUILL FORWARD
To actuate the clamping device in an NC program with M25 or M26 the tailstock must be in back end position.

6059: C-AXIS SWING IN TIMEOUT
C-axis does not swivel in within 4 seconds.
Reason: not sufficient air pressure, and/or mechanics stuck.

6060: C-AXIS INDEX FAILURE
When swivelling in the C-axis the limit switch does not respond.
Check pneumatics, mechanics and limit switch.

6064: AUTOMATIC DOOR NOT READY
Door stucks mechanically (collision), insufficient pressured air supply, limit switch defective, fuse defective.
Check for collisions, check fuses or contact EMCO service.

6065: LOADER MAGAZINE FAILURE
Loader not ready.
Check if the loader is switched on, correctly connected and ready for operation and/or disable loader (WinConfig).

6066: CLAMPING DEVICE FAILURE
No compressed air at the clamping device
Check pneumatics and position of the clamping device proximity detectors.

7000: INVALID TOOL NUMBER PROGRAMMED
The tool position was programmed larger than 8.
The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7016: SWITCH ON AUXILIARY DRIVES
The auxiliary drives are off. Press the AUX ON key for at least 0.5 sec. (to avoid accidentally switching on) to switch on the auxiliary drives (also a lubricating pulse will be released).

7017: REFERENCE MACHINE
Approach the reference point.
When the reference point is not active, manual movements are possible only with key switch at position "setting operation".

7018: TURN KEY SWITCH
With NC-Start the key switch was in position "setting operation".
NC-Start is locked.
Turn the key switch in the position "automatic" to run a program.

7019: PNEUMATIC LUBRICATION MONITORING!
Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE
Special operation mode: The machine door is opened, the auxiliary drives are switched on, the key switch is in position "setting operation" and the consent key is pressed.
Manual traversing the axes is possible with open door. Swivelling the tool turret is possible with open door. Running a CNC program is possible only with standing spindle (DRYRUN) and SINGLE block operation.
For safety: If the consent key is pressed for more than 40 sec. the function of this key is interrupted, the consent key must be released and pressed again.

7021: TOOL TURRET NOT LOCKED
The tool turret operating was interrupted.
NC start and spindle start are locked. Press the tool turret key in the RESET status of the control.

7038: LUBRICATION SYSTEM FAULT
The pressure switch is defective or gagged.
NC-Start is locked. This alarm can be reset only by switching off and on the machine.
Contact EMCO service.

7039: LUBRICATION SYSTEM FAULT
Not enough lubricant, the pressure switch is defective.
NC-Start is locked.
Check the lubricant and lubricate manually or contact EMCO service.

7040: MACHINE DOOR OPEN
The main drive can not be switched on and NC-Start can not be activated (except special operation mode)
Close the machine to run a program.

7042: INITIALIZE MACHINE DOOR
Every movement and NC-Start are locked.
Open and close the machine door to initialize the safety circuits.

7043: PIECE COUNT REACHED
A predetermined number of program runs was reached. NC-Start is locked. Reset the counter to continue.

7048: CHUCK OPEN
This message shows that the chuck is open. It will disappear if a workpiece will be clamped.

7049: CHUCK - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7050: COLLET OPEN
This message shows that the collet is open. It will disappear if a workpiece will be clamped.

7051: COLLET - NO PART CLAMPED
No part is clamped, the spindle can not be switched on.

7052: QUILL IN UNDEFINED POSITION
The tailstock is in no defined position.
All axis movements, the spindle and the tool turret are locked.
Drive the tailstock in back end position or clamp a workpiece with the tailstock.

7053: QUILL - NO PART CLAMPED
The tailstock reached the front end position. Traverse the tailstock back to the back end position to continue.

7054: NO PART CLAMPED
No part clamped, switch-on of the spindle is locked.

7055: CLAMPING DEVICE OPEN
This message indicates that the clamping device is not in clamping state. It disappears as soon as a part is clamped.

AC ALARMS

Axis Controller Alarms 8000 - 9999

8000 Fatal Error AC

8004 ORDxx Failure main-drive unit
8005 - 8009 ORDxx Internal error AC
Remedy: report to EMCO if repeatable

8010 ORDxx Syncr. error main drive
Cause: synchronisation mark missing for main drive
Remedy: report to service technician, if repeatable

8011 - 8013 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8014 ORDxx Decel.-time of axis too high
Remedy: report to service technician, if repeatable

8018 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8021 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8022 ORDxx Internal error AC
Remedy: report to EMCO, if repeatable

8023 ORDxx Invalid Z value for helix
Cause: The Z value of the helix must be smaller than the length of the arc to be traversed
Remedy: Program correction

8100 Fatal init error AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8101 Fatal init error AC

see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

8106 No PC-COM card found
Cause: PC-COM board can not be accessed (ev. not mounted).
Remedy: Mount board, adjust other address with jumper

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

8110 PC-COM init message missing
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

8117 Fatal init data error (pccom.hex)
see 8110.

8118 Fatal init error AC
see 8110, ev. insufficient RAM memory

8119 PC interrupt no. not valid
Cause: The PC interrupt number can not be used.
Remedy: Find out free interrupt number in the Windows95 system control (allowed: 5,7,10, 11, 12, 3, 4 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

8174 INC not allowed

8175 MSD file could not be opened

Cause: Internal error

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

8176 PLS file could not be opened

see 8175.

8177 PLS file could not be accessed

see 8175.

8178 PLS file could not be written

see 8175.

8179 ACS file could not be opened

see 8175.

8180 ACS file could not be accessed

see 8175.

8181 ACS file could not be written

see 8175.

8182 Gear change not allowed

8183 Gear too high

8184 Invalid interpolaton command

8185 Forbidden MSD data change

see 8175.

8186 MSD file could not be opened

see 8175.

8187 PLC program error

see 8175.

8188 Gear command invalid

see 8175.

8189 Invalid channel assignement

see 8175.

8190 Invalid channel within message

8191 Invalid jog feed unit

8192 Invalid axis in command

8193 Fatal PLC error

see 8175.

8194 Thread without length

8195 No thread slope in leading axis

Remedy: Program thread pitch

8196 Too 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.

8211 Feed too high (thread)
Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed
Remedy: Program correction, lower pitch or lower spindle speed for thread

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