

# **EMCO WinNC for Heidenhain TNC 640**

## **Software description, software release effective from 01.00**



### **Software description**

#### **Emco WinNC for Heidenhain TNC 640**

**Ref. no. EN 1844**  
**Issue B 2017-10**

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

### **Original operating instructions**

EMCO GmbH  
P.O. Box 131  
A-5400 Hallein-Taxach/Austria  
Phone ++43-(0)62 45-891-0  
Fax ++43-(0)62 45-869 65  
Internet: [www.emco-world.com](http://www.emco-world.com)  
E-Mail: [service@emco.at](mailto:service@emco.at)



Designed for your profit

**Note:**

Some of the functionality of the EMCO WinNC for Heidenhain TNC 640 control software is beyond the scope of these instructions. Emphasis is placed instead on representing the most important functions simply and clearly, in order to achieve the greatest possible learning success. Depending on the machine you are operating with EMCO WinNC for Heidenhain TNC 640, not all functions may be available.

**Note:**

Use of text and picture material (picture material in the form of screen shots) with the designation \*), °) and +) with kind permission of Dr. Johannes Heidenhain GmbH.

**Sources:**

User's Manual Heidenhain Conversational Programming TNC 640 \*)

User's Manual Cycle Programming TNC 640 °)

NC Software 340590-05, 340591-05, 340595-05

User's Manual Conversational Programming TNC 426 +)

NC-Software 280474-xx, 280475-xx

# Preface

The EMCO WinNC for Heidenhain TNC 640 software is a component of the EMCO training concept.

EMCO WinNC for Heidenhain TNC 640 makes it easy to operate CNC lathes/milling machines. Prior knowledge of ISO programming are not needed.

Interactive contour programming means that you can define workpiece contours with linear and circular contour elements.

A cycle is programmed interactively, with graphic support. A large number of processing cycles and programming commands which can be combined freely to form a single program is available to the user.

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

Some of the functionality of the EMCO WinNC for Heidenhain TNC 640 control software is beyond the scope of these instructions. Emphasis is placed instead on representing the most important functions simply and clearly, in order to achieve the greatest possible learning success.

If you have queries about or suggestions for improving these operating instructions, please contact directly

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



Designed for your profit

## EC compliance



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

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

# Table of contents

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

## A: Principles

EMCO milling machine reference points .....	A1
N (T) = Tool zero point .....	A1
M = Machine zero point .....	A1
W = Workpiece zero point .....	A1
R = Reference point.....	A1
Reference system on milling machines .....	A2
Polar coordinates.....	A3
Determining the pole and the angle reference axis .....	A3
Absolute and incremental workpiece positions.....	A4
Zero point offset.....	A5
Setting the zero point / reference point.....	A5
Milling operations.....	A6
Down-cut Milling .....	A6
Up-cut milling .....	A6
Down-cut/up-cut milling .....	A6
Tool data .....	A7

## B: Key description

EMCO WinNC for Heidenhain TNC 640 control keyboard .....	B1
EMCO WinNC for Heidenhain TNC 640 control keyboard variant with Easy2control and MOC-Touch .....	B3
Key functions .....	B5
Screen layout manual operation.....	B9
PC keyboard.....	B10
PC keyboard machinefunctions.....	B11
Overview key assignment control keyboard .....	B14
Language for context sensitive help .....	B14
Machine control panel .....	B15
Key description .....	B15
Skip (block mask) .....	B15
Dry run (test run feed).....	B15
Individual piece mode .....	B16
Optional stop.....	B16
Edit.....	B16
Handwheel mode (optional).....	B16
Reset key (Reset).....	B16
Feed Stop .....	B16
Feed Start.....	B16
Single block .....	B17
Cycle stop .....	B17
Cycle start.....	B17
Arrow keys .....	B17
Rapid traverse .....	B17
Reference point .....	B17
Chip conveyor (Option).....	B17
Swing tool drum .....	B18
Manual tool change .....	B18
Clamping device .....	B18
Coolant .....	B18
Modes of operation .....	B19
Auxiliary OFF .....	B20
Auxiliary ON.....	B20
Override switch (feed rate override) .....	B21
EMERGENCY SHUTDOWN .....	B21
Key Switch Special Operations Mode.....	B21
Multifunction control.....	B22

Key switch.....	B25
Additional clamping device key.....	B25
USB connection (USB 2.0) .....	B25
Enable key .....	B25

## C: Operation

Feed F [mm/min] .....	C1
Spindle speed S [rev/min].....	C2
Modes of operation.....	C3
Machine modes of operation .....	C3
Programming modes of operation .....	C4
Activating modes of operation .....	C5
Approaching the reference point .....	C7
Move carriages manually.....	C8
Move carriages in step-movement .....	C8
Administration of reference points in the preset table .....	C10
Switch off .....	C13
File management fundamentals .....	C14
Files .....	C14
File management .....	C15
Creating a new directory .....	C16
Creating a new file .....	C16
Selecting a file .....	C17
Delete file .....	C17
Deleting a directory .....	C17
Copying a file .....	C18
Selecting one of the last 10 selected files .....	C19
Renaming files .....	C19
Tagging files.....	C20
Additional functions .....	C21
Sorting .....	C21
Special functions .....	C22
Defining a raw part: BLK FORM .....	C22
Assigning string parameters .....	C22
Insert comment .....	C23
Opening and entering programs .....	C24
Opening a new machining program .....	C25
Programming tool movements in plain language dialogue .....	C26
Editing program .....	C28
MOD function.....	C30
Graphical simulation .....	C31
Screen layout graphical simulation .....	C32
Softkey functions .....	C33

## D: Programming

Overview.....	D1
M-commands .....	D1
Calculator .....	D2
Error messages .....	D3
Tool movements .....	D4
Fundamentals of Path Functions .....	D5
Approach and leave contour .....	D8
Approaching on a straight line with tangential connection: APPR LT .....	D10
Approaching on a straight line perpendicular to the first contour point: APPR LN .....	D11
Approaching on a circular path with tangential connection: APPR CT .....	D12
Approaching on a circular path with tangential connection from .....	

a straight line to the contour: APPR LCT .....	D13
Departing in a straight line with tangential connection: DEP LT .....	D14
Departing on a circular path with tangential connection: DEP CT .....	D15
Departing in a straight line perpendicular to the last contour point: DEP LN .....	D16
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT .....	D17
Path movements - rectangular coordinates .....	D18
Straight line L .....	D19
Insert a chamfer CHF between two straight lines .....	D20
Corner roundings RND .....	D21
Circle centre CC .....	D22
Circular path C around circle centre CC .....	D23
Circular path with a defined radius .....	D23
Path contours - Polar coordinates .....	D25
Polar coordinates origin: Pole CC .....	D25
Straight line LP .....	D26
Circular path CP around pole CC .....	D26
Helix .....	D27
Cycle overview .....	D29
Drilling / thread .....	D35
DRILLING (cycle 200) .....	D36
GRINDING (cycle 201) .....	D38
TURNING OUT (cycle 202) .....	D40
UNIVERSAL DRILLING (cycle 203) .....	D42
UNIVERSAL DEEP HOLE DRILLING (cycle 205) .....	D44
DRILL MILLING (cycle 208) .....	D48
THREAD TAPPING NEW (cycle 206) .....	D50
RIGID TAPPING (cycle 207) .....	D52
THREAD TAPPING CHIPS CRUSHING (cycle 209) .....	D54
CENTRING (cycle 240) .....	D56
THREAD MILLING (cycle 262) .....	D60
THREAD MILLING/COUNTERSINKING (cycle 263) .....	D62
THREAD DRILLING/MILLING (cycle 264) .....	D66
HELICAL THREAD DRILLING/MILLING (cycle 265) .....	D70
EXTERNAL THREAD MILLING (cycle 267) .....	D72
Converting the coordinates .....	D77
ZERO OFFSET (cycle 7) .....	D78
MIRRORING (cycle 8) .....	D79
ROTATION (cycle 10) .....	D80
DATUM SETTING (cycle 247) .....	D81
SL Cycles .....	D83
CONTOUR (cycle 14) .....	D85
CONTOUR DATA (cycle 20) .....	D88
CLEARING (cycle 22) .....	D90
FINISHING DEPTH (cycle 23) .....	D94
FINISHING SIDE (cycle 24) .....	D96
CONTOUR PULL (cycle 25) .....	D98
Dot patterns .....	D101
PATTERN ON CIRCLE (cycle 220) .....	D102
PATTERN ON LINES (cycle 221) .....	D104
Special cycles .....	D107
DWELL TIME (cycle 9) .....	D108
PGM CALL (cycle 12) .....	D109
Old Cycles .....	D111
DEEP DRILLING (cycle 1) .....	D112
THREAD TAPPING (cycle 2) .....	D114
RIGID TAPPING (cycle 17) .....	D116
SLOT MILLING (cycle 3) .....	D118
POCKET MILLING (cycle 4) .....	D120
CIRCULAR POCKET (cycle 5) .....	D122
POCKET FINISHING (cycle 212) .....	D124
STUD FINISHING (cycle 213) .....	D126
CIRCULAR POCKET FINISHING (cycle 214) .....	D128
CIRCULAR STUD FINISHING (cycle 215) .....	D130

SLOT RECIPROCAL (cycle 210) .....	D132
ROUND SLOT (cycle 211) .....	D134
MULTIPASS MILLING (cycle 230) .....	D136
RULED SURFACE (cycle 231) .....	D138
Subprograms .....	D142

## E: Tool programming

Tool related input data .....	E1
Feed rate F .....	E1
Spindle speed S .....	E2
Tool data .....	E3
Entering tool data in the table .....	E5
Opening any other tool table .....	E6
3D tool-name .....	E7
3D tool-color .....	E7
Pocket table for tool changer .....	E8
Call tool data .....	E10
Tool compensation .....	E11
Introduction .....	E11
Tool length compensation .....	E11
Tool radius compensation .....	E12

## F: Program sequence

Preconditions .....	F1
Program start, program stop .....	F2
Modes of operation program run .....	F2
Block scan .....	F3
F MAX .....	F4
Optional block skip .....	F4
Optional program-run interruption .....	F4

## G: Flexible NC programming

Q parameters .....	G1
Calling up Q parameters .....	G2
Basic mathematical functions .....	G2
Trigonometric functions .....	G3
If/then decisions with Q parameters .....	G4

## H: Alarms and Messages

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

**I: Heidenhain TNC 640 controller alarms**

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

**W: Accessory Functions**

Activating accessory functions.....	W1
Robotic Interface.....	W1
Automatic doors.....	W1
Win3D View .....	W1
Tool modelling with the 3D-Tool Generator.....	W2
DNC interface .....	W6

**X: EMConfig**

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

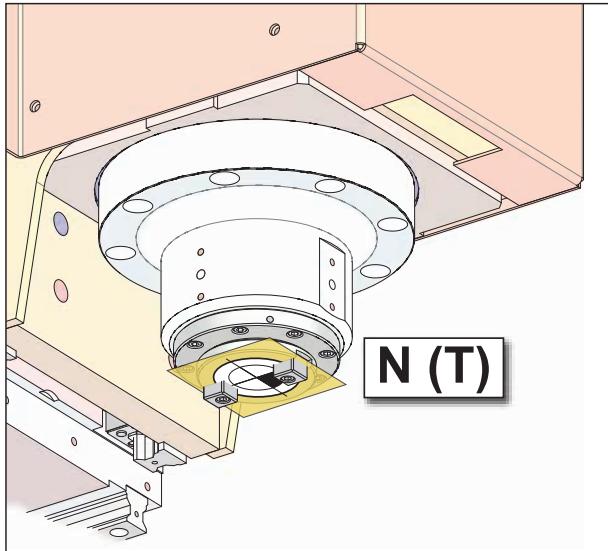
**Y: External Input Devices**

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

**Z: Software Installation Windows**

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

# A: Principles



Points on the machine

## EMCO milling machine reference points

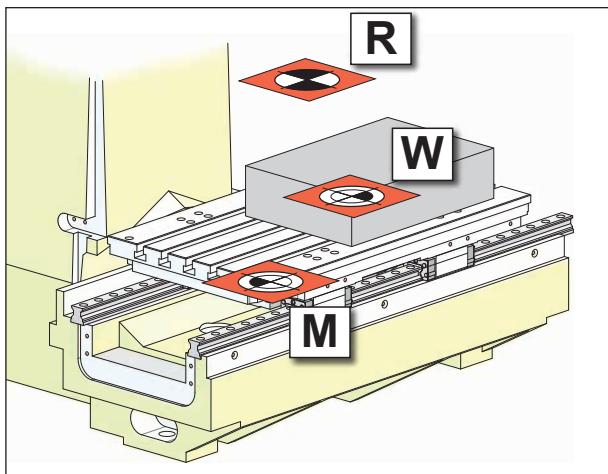
### **N (T) = Tool zero point**

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

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

**Note:**

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



Machine reference points



### **M = Machine zero point**

The machine zero point M is an unchangeable reference point defined by the manufacturer. The entire machine is measured from this point. The machine zero point M is the origin of the coordinate system.



### **W = Workpiece zero point**

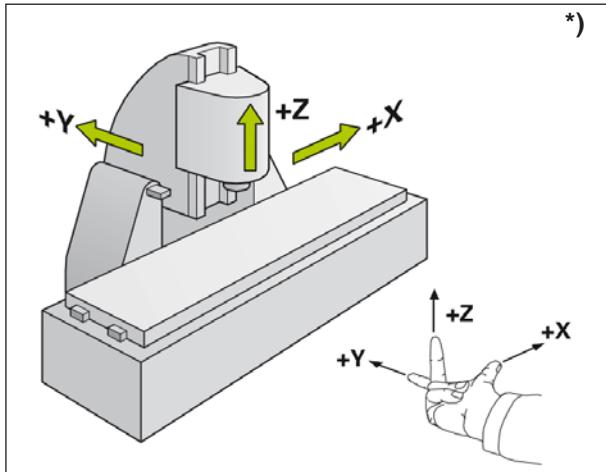
The workpiece zero point W can be freely programmed by the user. By programming a workpiece zero point, the origin of the coordinate system will be shifted from the machine zero point M to the workpiece zero point W.

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

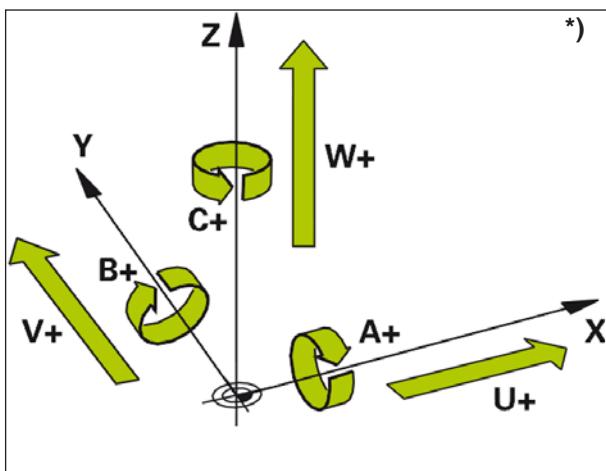


### **R = Reference point**

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



Coordinate system



Assignment of the rotating axes to the main axes

## Reference system on milling machines \*)

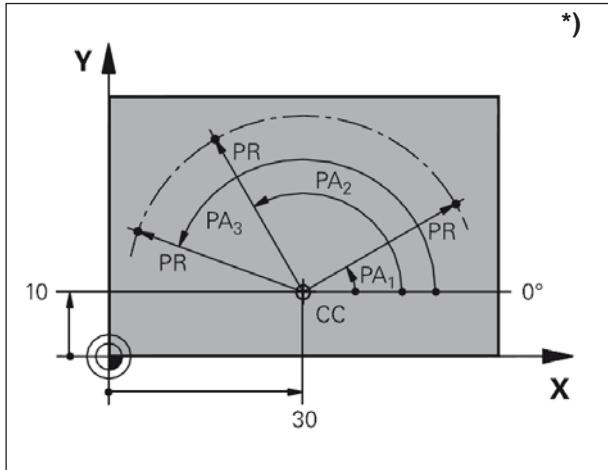
With a reference system, you define positions in a plane or in space unambiguously. Positional information always relates to a given point and is described by means of coordinates.

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

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

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

WinNC can control a maximum of 5 axes in total. Rotation axes are designated as A, B and C. The picture in the left bottom corner shows the assignment of the additional axes and/or rotation axes to the main axes.



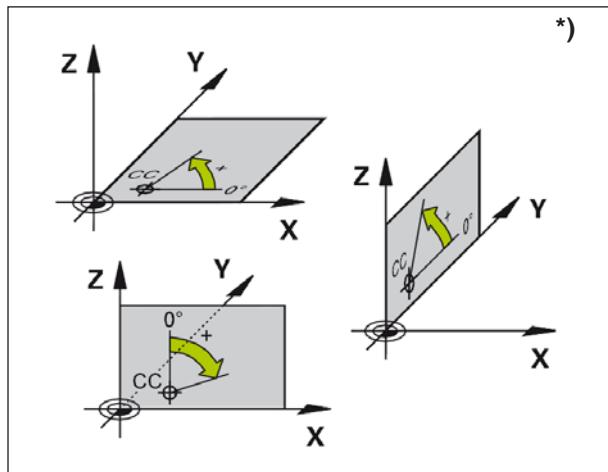
## Polar coordinates \*)

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

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

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

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

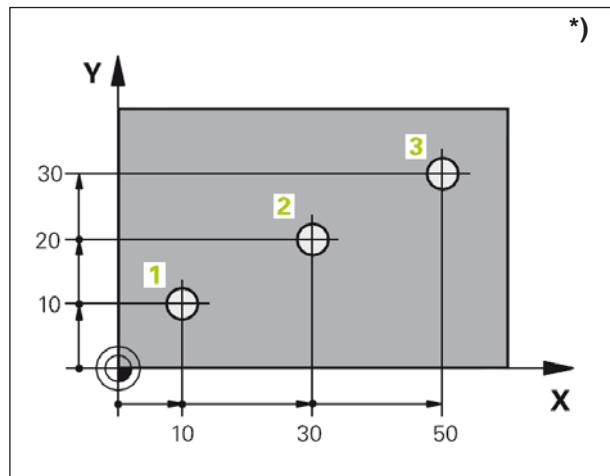


## Determining the pole and the angle reference axis

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

Polar coordinates (plane)	Angle reference axis
X/Y	+X

## Absolute and incremental workpiece positions \*)



### Absolute workpiece positions

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

Example 1: Drillings with absolute coordinates

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

### Incremental workpiece positions

Incremental coordinates refer to the last programmed position of the tool, which serves as the relative (conceptual) zero point. Incremental coordinates describe the actual paths of the tool. Consequently, it is also described as chain measurement.

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

Example 2: Drillings with incremental coordinates

Absolute coordinates of the bore 4

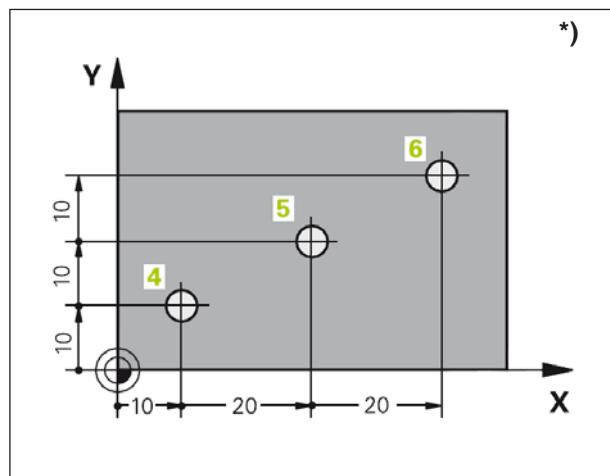
IX = 10 mm  
IY = 10 mm

Bore 5, related to 4

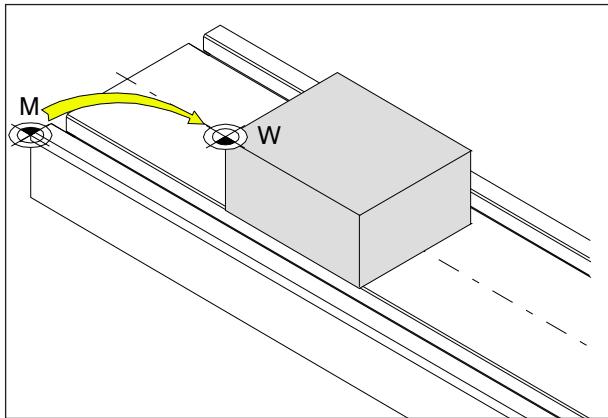
IX = 20 mm  
IY = 10 mm

Bore 6, related to 5

IX = 20 mm  
IY = 10 mm



## Zero point offset

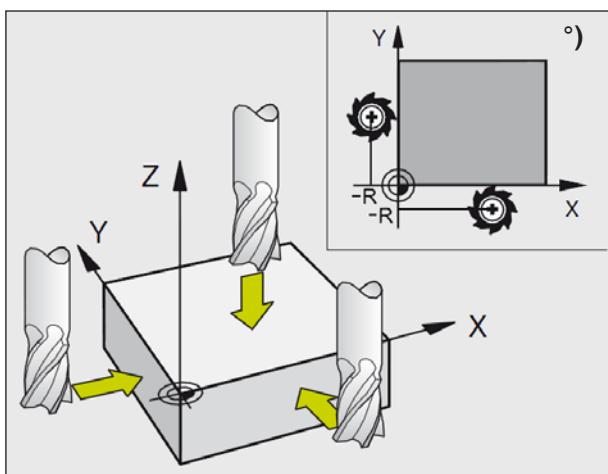
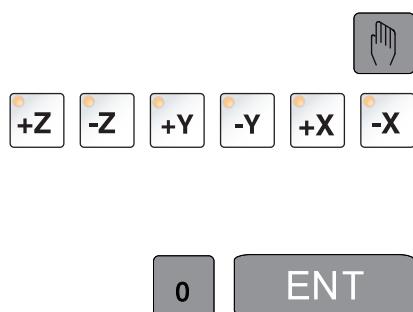


*Zero point offset of machine zero point M to tool zero point W*

On EMCO milling machines, the machine zero point "M" is on the left front edge of the machine table. This position is unsuitable as a start point for programming.

WinNC offers 2 methods, which can also be combined, to set a zero point:

- 1.) Set reference point (see below)
- 2.) Cycle 7 - zero point offset. Here, absolute or incremental coordinates can be used. (see Section D, cycles for coordinate conversion)



### Setting the zero point / reference point

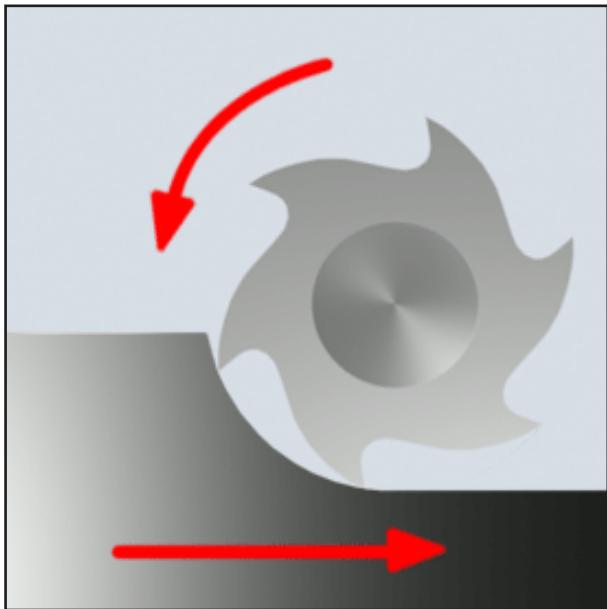
- Select **Manual mode of operation**
- Move the tool cautiously, until it touches the work piece (scratches it).
- Select axis (all axes can be selected via the ASCII keyboard as well)
- Zero tool, spindle axis: Set display to known work piece position (e. g. 0). In the machining plane: Take tool radius into account.
- Set the reference points for the remaining axes in the same way.

If you use a predefined tool in the infeed axis, set the display of the infeed axis to the length L of the tool.

Zero points / reference points are stored in the 1st line of the preset table, see also chapter "C".

## Milling operations

### Down-cut Milling

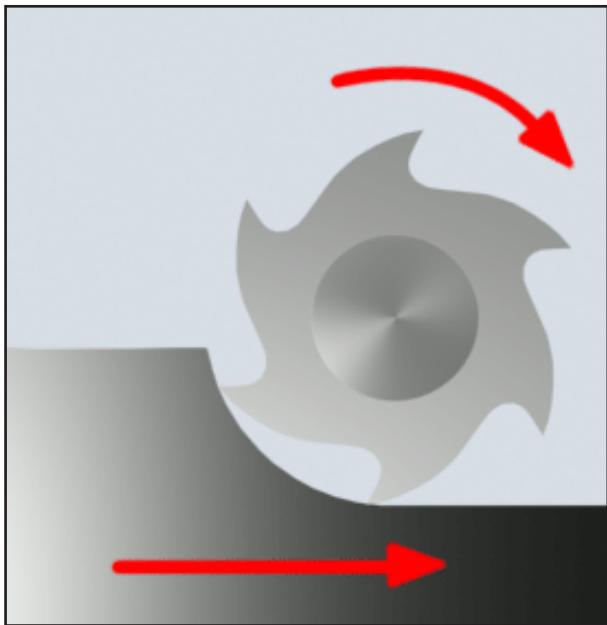


Down-cut Milling

In down-cut milling, feed direction and cutting direction of the mill are identical. The blade penetrates the surface of the raw part into the material first.. It has the advantage that a larger lead angle enables the immediate penetration of the blade into the material. Unlike up-cut milling, no certain sliding cutting path is traversed under pressure and friction.

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

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



Up-cut milling

### Up-cut milling

In up-cut milling, feed direction and cutting direction of the mill are opposite. The tool's blades hit the material in a very sharp angle ( $\varphi = 0$ ).

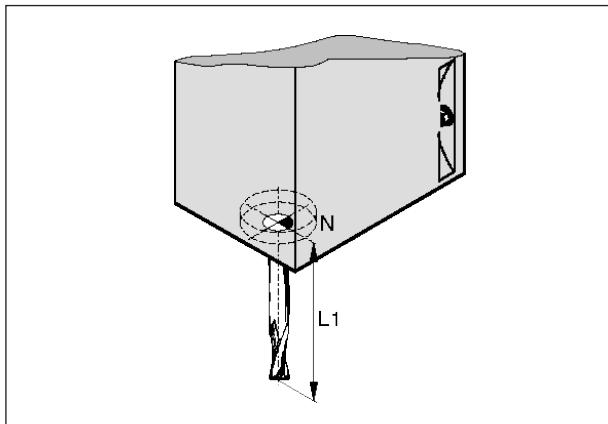
Before the blades penetrate into the material, they slide on the surface over a short distance with increasing contact force. After penetration, the chip cross-section slowly increases and quickly decreases towards the end.

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

### Down-cut/up-cut milling

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

## Tool data



Tool length

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

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

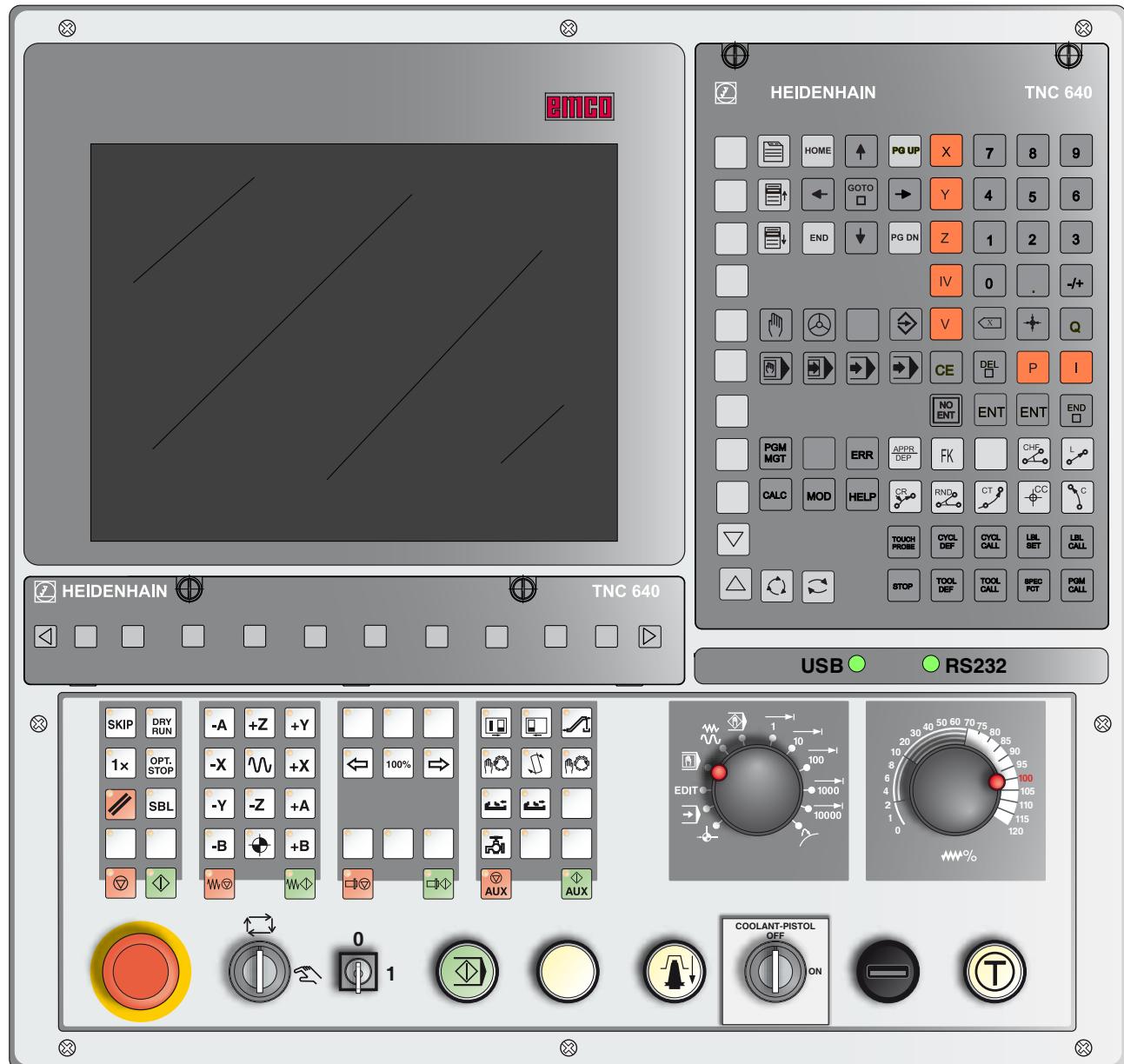
The measured lengths and mill radius can be stored in the tool list.

The mill radius information is **only** necessary if a **mill radius compensation** or a milling cycle is selected for the relevant tool!  
(see Chapter F Tool Programming)



# B: Key description

## EMCO WinNC for Heidenhain TNC 640 control keyboard

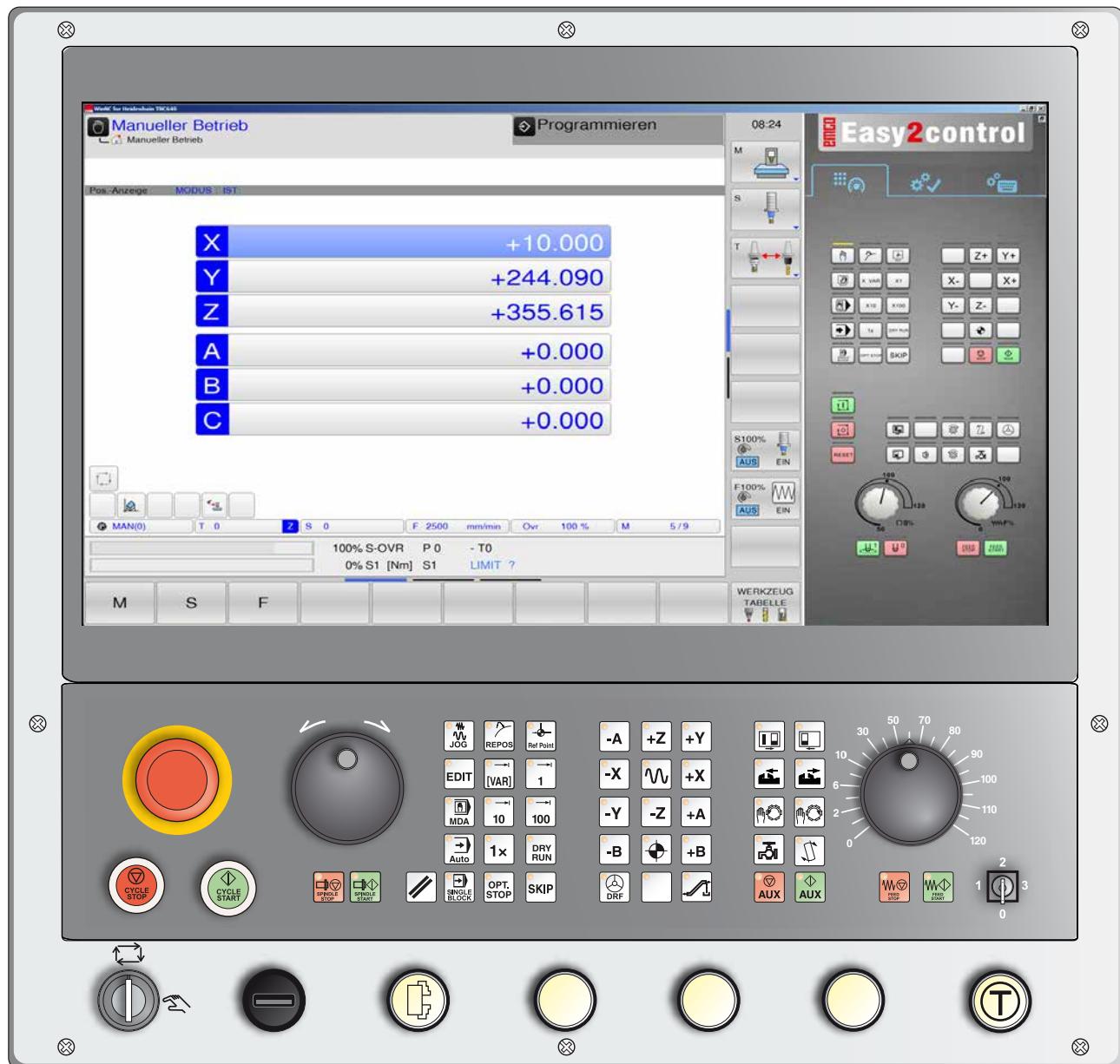


**Note:**

Depending on the machine you are operating with Emco WinNC for Heidenhain TNC 640, not all functions and machine keys may be available.



## EMCO WinNC for Heidenhain TNC 640 control keyboard variant with Easy2control and MOC-Touch

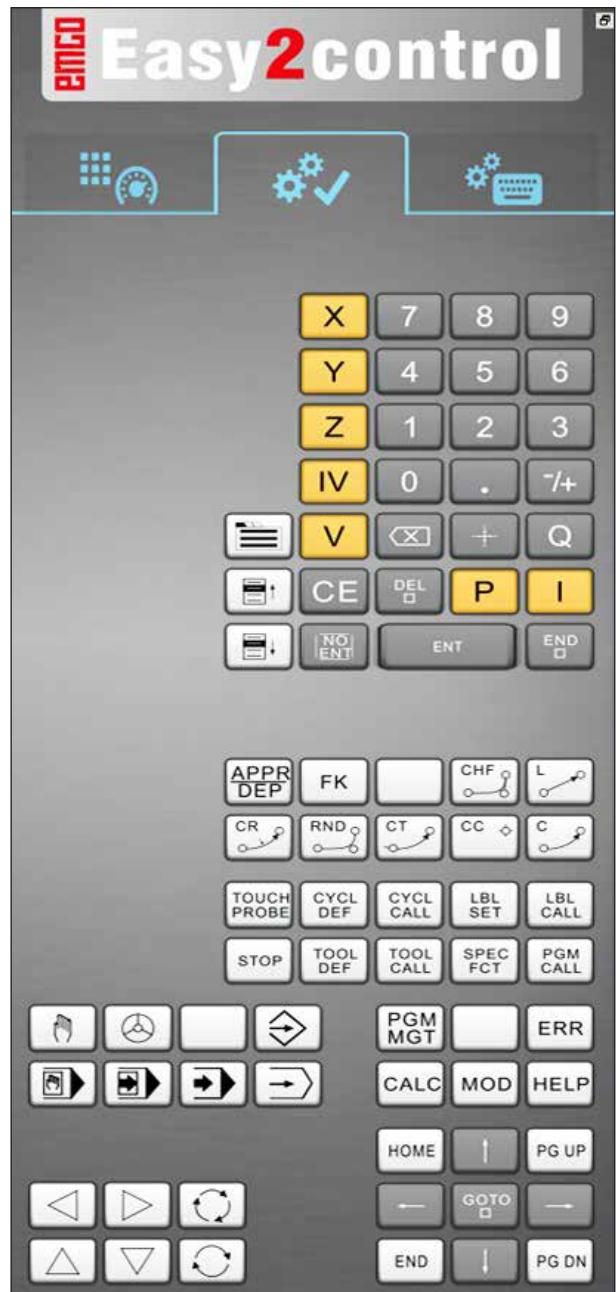


**Note:**

Depending on the machine you are operating with Emco WinNC for Heidenhain TNC 640, not all functions and machine keys may be available.



Address and number pad



Easy2Control

# Key functions

## Screen layout



Call up the softkey bar for the screen layout



Switch screen between machine and programming mode of operation



Select softkey in the horizontal and/or vertical softkey bar



Switch softkey bar

## Modes of operation



Manual operation



Electronic hand wheel



Positioning with manual input



Single block program run



Block sequence program run

## Programming modes of operation



Saving and/or editing program



Program test

## Manage programs and files, TNC functions



Select and delete programs/files  
External data transfer



Enter program call in a program



Select MOD function



Display context sensitive help



Display calculator



Displays all pending error messages

**Navigation keys**

Shift illuminated field



Directly select blocks, cycles and parameter functions

**Cycles, subprograms and program section repeats**

Define and call up cycles



Enter and call up subprograms and program section repeats



Enter program stop in a program

**Tools information**

Define tool data in the program



Call up tool data

**Program path movements**

Approach / leave contour



Free contour programming FK



Straight line



Circle centre / pole for polar coordinates



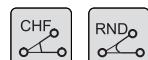
Circular path around circle centre



Circular path with radius



Circular path with tangential contact



Chamfer / Corner rounding

**Special functions**

Display special functions



Select next tab in form



Dialogue or button next / back

**Insert / edit coordinate axes and numerical values**

Select and/or enter coordinate axes in program



Numerals



Reverse decimal point and/or algebraic sign



Enter polar coordinates / incremental values



Q parameter programming / Q parameter status



Adopt current position, values from calculator



Skip dialogue questions and delete words



Conclude input and continue dialogue



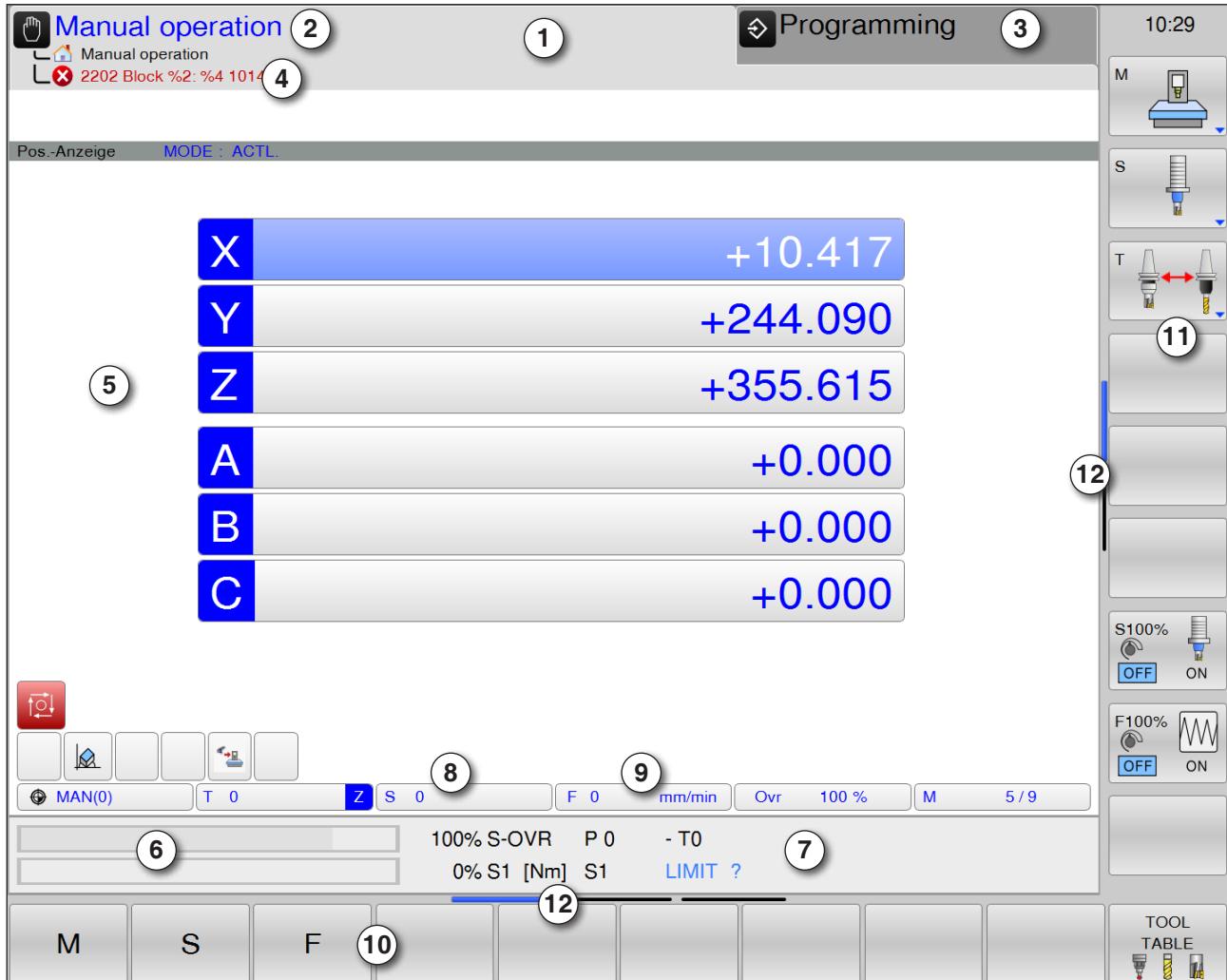
Conclude block, finish entry

Reset numerical value entry or delete TNC error message.  
Reset simulation

Cancel dialogue, delete program section



## Screen layout manual operation

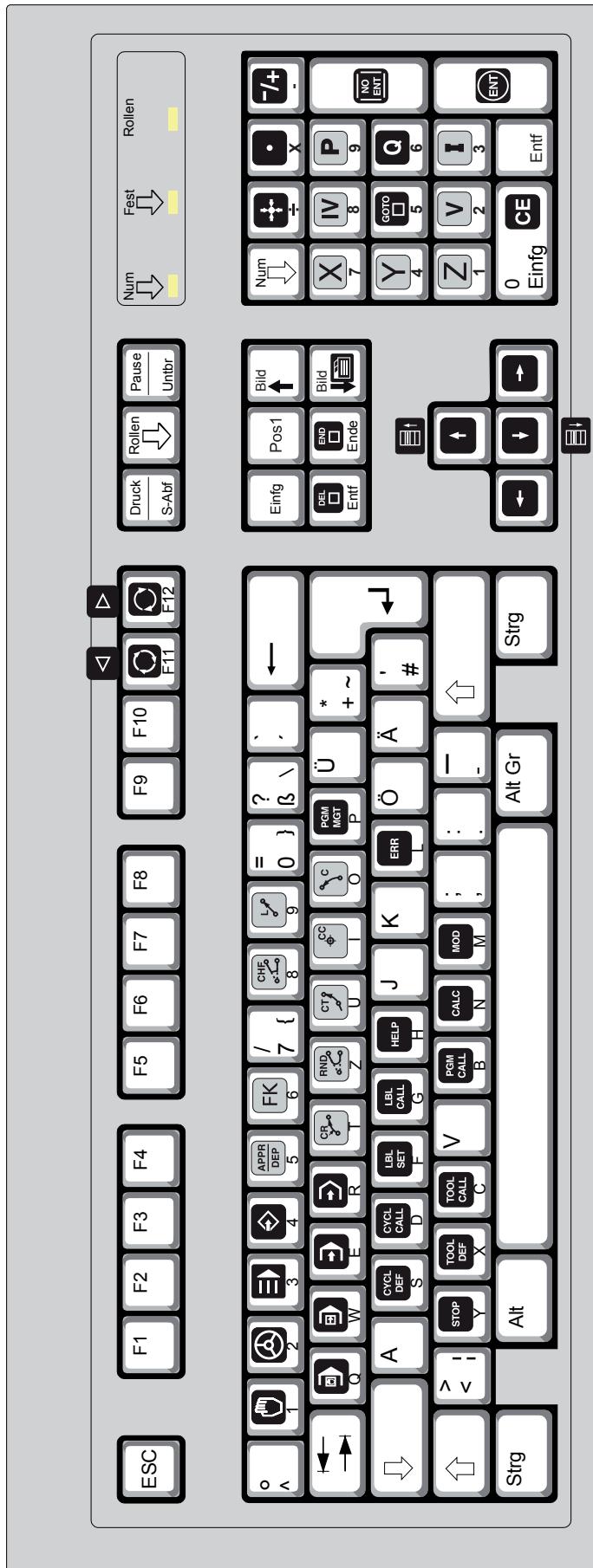


- |  |  |
|--|--|
| 1 Header: Display of selected modes of operation:  | 9 Programmed feed  |
| 2 Left tabs: Machine modes of operation  | 10 horizontal softkey bar  |
| 3 Right tabs: Programming modes of operation   | 11 vertical softkey bar  |
| 4 Alarm and notification lines   | 12 narrow bars above the softkey bar show the number of softkey bars.<br>The current bar is displayed in blue colour |
| 5 Working window, NC displays  |  |
| 6 Power display  |  |
| 7 General status display informs about the current state of the machine:<br>S-OVR, S1, T0, LIMIT | For detailed descriptions see chapter "C Operation"  |
| 8 Current spindle speed  |  |

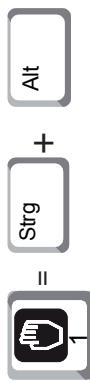
**Note:**

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

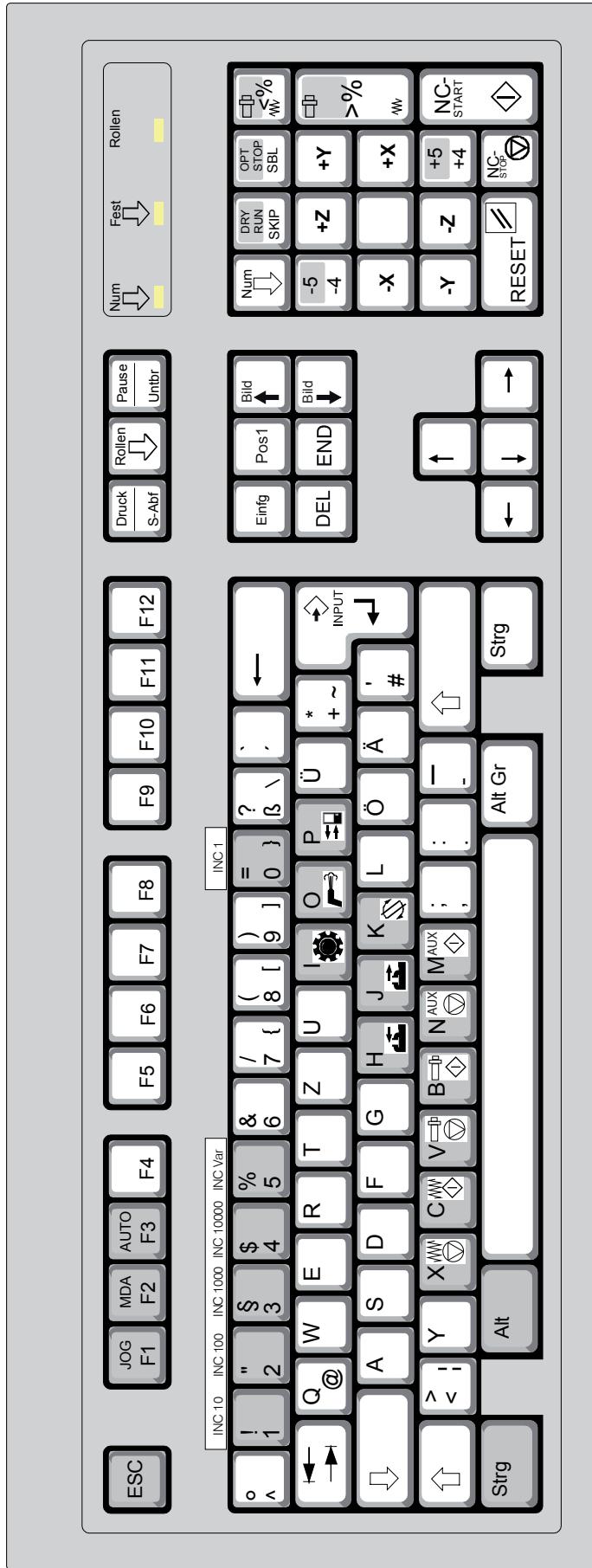
## PC keyboard



To program WinNC functions, the CTRL and ALT keys must be pressed at the same time.



## PC keyboard machinefunctions



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



**Note:**

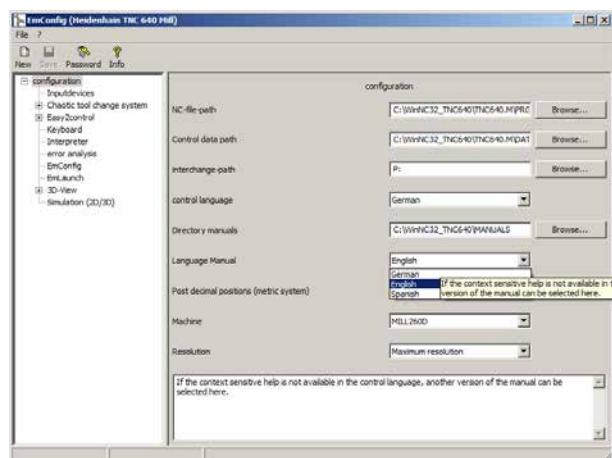
The machine functions in the numerical keypad are only active if NUM Lock is not active.

PC key	Control key	Function
		Softkey keys
		Display context sensitive help
		Select screen layout
		Switch screen between machine and programming mode of operation
		Switch softkey bar
		Switch softkey bar
		Conclude input and continue dialogue
		Move mark
		Manual operation
		Electronic hand wheel
		Saving / editing program
		Approach / leave contour
		Free contour programming
		Chamfer
		Straight line
		Positioning with manual input
		Single block program run
		Block sequence program run
		Program test
		Circular path with radius

PC key	Control key	Function		
				Corner rounding
				Circular path with tangential contact
				Circle centre / pole for polar coordinates
				Circular path around circle centre
				Select and delete programs/files, external data transfer
				Define cycles
				Call up cycles
				Enter subprograms and program section repeats
				Call up subprograms and program section repeats
				Display context sensitive help
				Display all pending error messages
				Enter program stop in a program
				Define tool data in the program
				Call up tool data
				Define program call up, select zero point and point tables
				Display calculator
				Select MOD function
				Cancel dialogue, delete program section
				Conclude block, finish entry
				Select next tab in form
				Next dialogue or key

## Overview key assignment control keyboard

PC key	Control key	Function
		Back to previous dialogue or key
		Adopt current position, values from calculator
		Reverse decimal point
		Reverse algebraic sign
		Directly select blocks, cycles and parameter functions
		Q parameter programming / Q parameter status
		Enter incremental values
		Enter polar coordinates
		Cancel dialogue
		Skip dialogue questions and delete words
		Conclude entry, delete program section



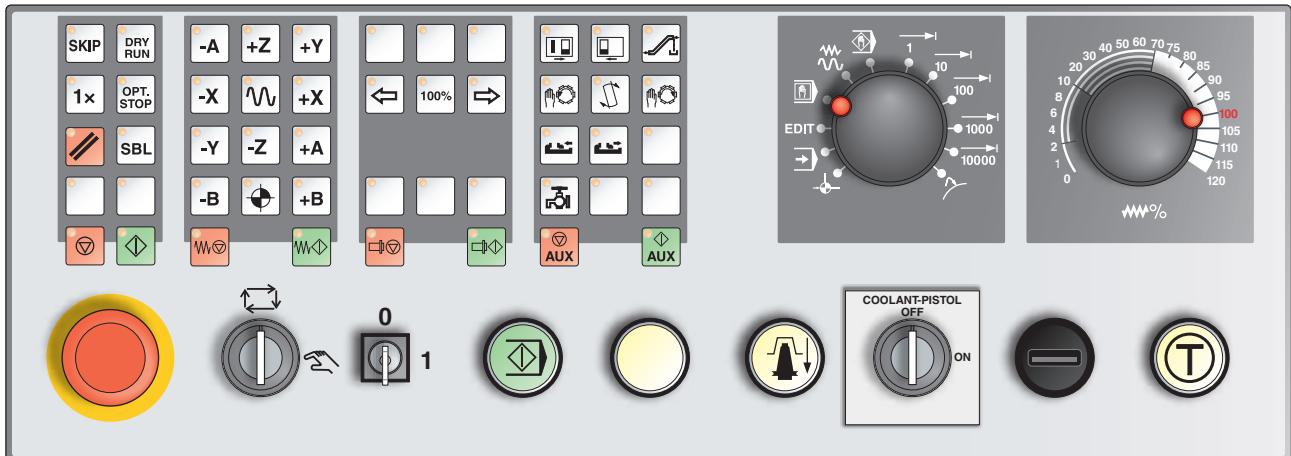
Language manual

## Language for context sensitive help

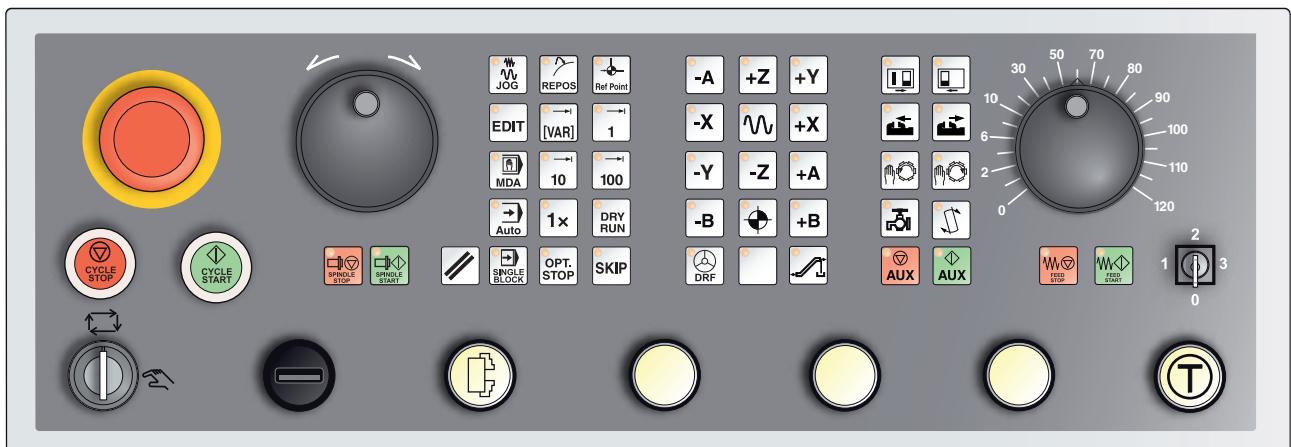
Setting in EMConfig software:

If the context sensitive help is not available in the control language, another version of the manual can be selected here.

## Machine control panel



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



Machine control panel variant with Easy2Control and MOC-Touch

## Key description

### Skip (block mask)



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

### Dry run (test run feed)



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

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

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

Only perform the test run without a workpiece to avoid the danger of collisions.

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

## Individual piece mode



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

## Optional stop



When this function is active (key held down), program processing will stop at blocks in which the M01 special function has been programmed. You start machining again with the NC Start key. If the function is not active, the M01 special function (from the parts program) will be ignored.

## Edit



Toggle into Edit mode.

## Handwheel mode (optional)



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

## Reset key (Reset)



Pressing the Reset key:

interrupts the machining of the current parts program.

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

## Feed Stop



This key interrupts a programmed carriage movement.

## Feed Start



This key continues a programmed but interrupted carriage movement.

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

## Single block

This function allows you to run a parts program block by block. You can enable the single block feature in Automatic mode.

### When single block processing is active:



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

## Cycle stop

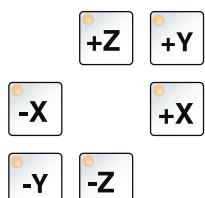


After pressing the Cycle stop key after the controller has taken over the function, machining of the current parts program is suspended. Machining can then be continued by pressing the Cycle start key.



## Cycle start

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



## Arrow keys

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

Depending on the machine configuration, various arrow keys are available

## Rapid traverse



If this key is pressed in addition to the arrow keys, the axis concerned moves in rapid traverse.

## Reference point



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

## Chip conveyor (Option)



Switch on chip conveyor:

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

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

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

This value is set in the factory.

## Swing tool drum

Pressing this key causes the tool drum to swivel by one position:



Cycle in the clockwise direction (one position further)



Cycle in the counterclockwise direction (one position back)

### Preconditions:

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

## Manual tool change



Pressing this key starts a manual tool change.

The tool clamped in the milling spindle will be removed and replaced with the tool from the currently swivelled-in tool drum.

### Preconditions:

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



### Information:

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

## Clamping device



These functions activate the clamping device.

## Coolant



This function switches the coolant equipment on or off.

## Modes of operation

### JOG



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

### MDA - Manual Data Automatic



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

### Automatic



Control the machine by automatically executing programs.

### REF - Reference mode



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

### Inc 1 - Incremental feed



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

Metric measurement system: Inc 1 corresponds to 1µm

Imperial (inch-based) measurement system: Inc 1 corresponds to 0.1 pinch

### Inc 10 - Incremental feed



Move step by step a predefined distance of 10 increments.

Metric measurement system: Inc 10 corresponds to 10µm

Imperial (inch-based) measurement system: Inc 10 corresponds to 1 pinch

### Inc 100 - Incremental feed



Move step by step a predefined distance of 100 increments.

Metric measurement system: Inc 100 corresponds to 100µm

Imperial (inch-based) measurement system: Inc 100 corresponds to 10 pinch

### Inc [VAR]



Move in steps with a continuously variable increment.

### REPOS - Repositioning



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

**Information:**

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

**Feed:**

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

**Constant cutting speed:**

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

## Auxiliary OFF



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

## Auxiliary ON



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

The key must be pressed for around 1 second.

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

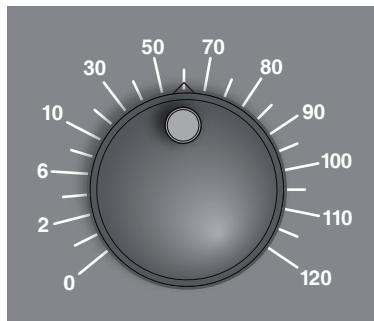
### Clearing before referencing

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

### Free swivelling of the tool turret

If the tool turret has to be swivelled free after an alarm had been pending, press the keys and then .

## Override switch (feed rate override)



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

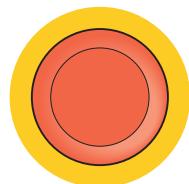
The set feed value F in % will be shown on the screen.

Adjustment range:

0% to 120% of the programmed feed.

In rapid traverse 100% will not be exceeded.

No effect with thread commands G33, G63



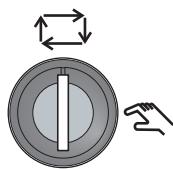
## EMERGENCY SHUTDOWN

Press the red button only in an emergency.

Effects:

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

To continue working, press the following keys:  
RESET, AUX ON, doors OPEN and CLOSED.



## Key Switch Special Operations Mode

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

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



### Danger:

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

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

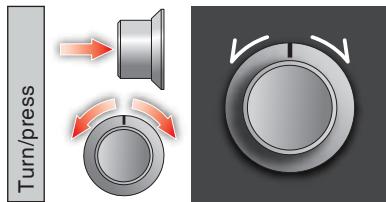
Keep the chip guard door closed even in Set-up mode.

Keys should only be used by authorized persons.

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

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

## Multifunction control

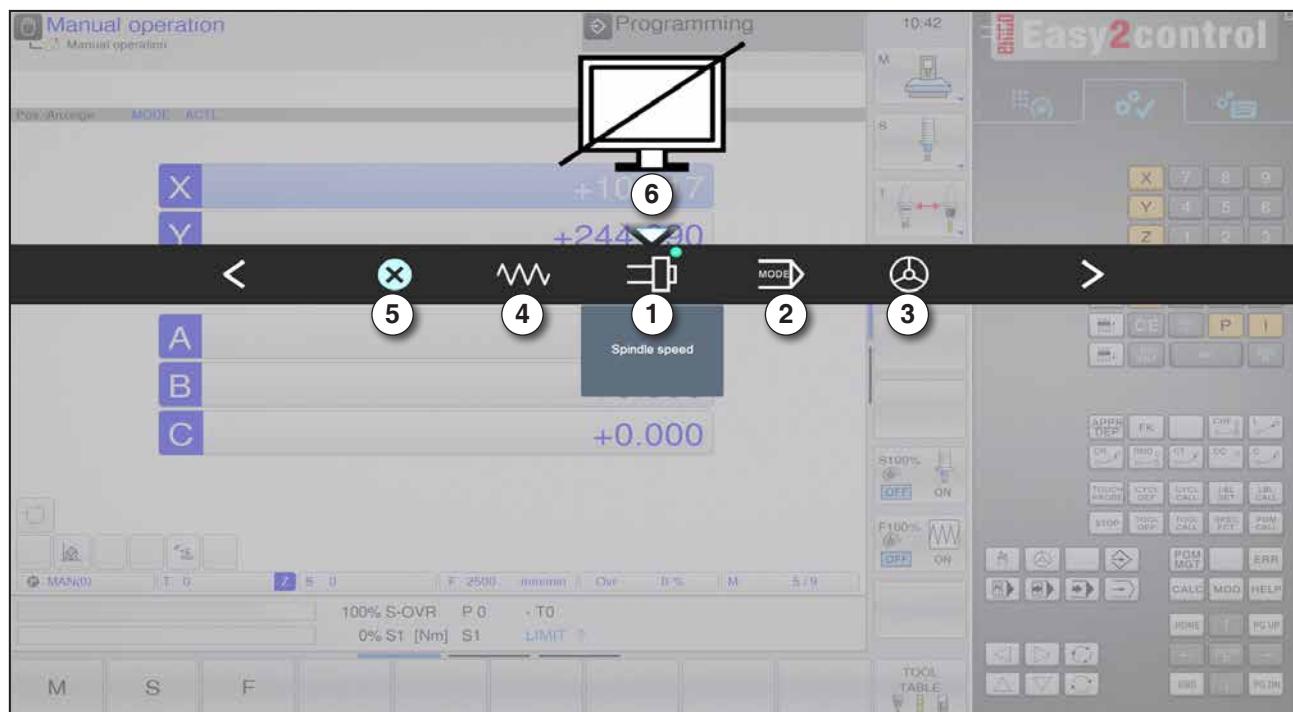


The multifunction control is designed as a rotary switch with a press feature.

### Method of operation

- Pressing the multifunction control once opens the user interface. The active function is marked by a green check mark.
- Switching between the functions is performed by rotating the switch. In doing so, the black bar with the symbols moves to the left and/or to the right.
- Activating a function or changing in a sub-menu is performed by pressing the knob.

The interface offers the following functions:



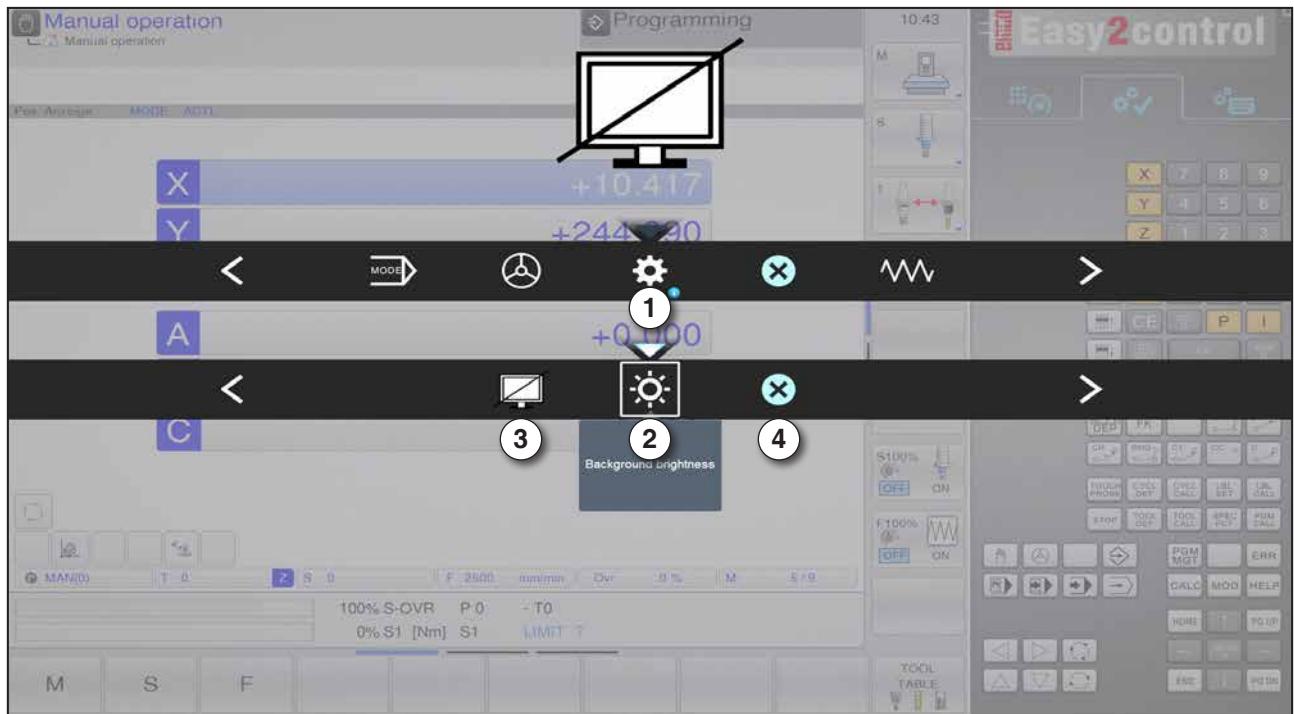
### Function overview

- 1 Feed override: controls the feed like a conventional feed controller
- 2 Spindle override: controls the spindle speed like a conventional speed controller
- 3 Modes of operation: Enables the selection of modes of operation via the multifunction control

- 4 Close: Closes the user interface. The menu disappears, return to the control interface
- 5 Settings: opens a further level with settings options
- 6 Cursor: indicates the current position in the menu

#### Note:

The number of available functions of the multifunction control can vary depending on the software version.



*Background brightness settings*

1 Settings

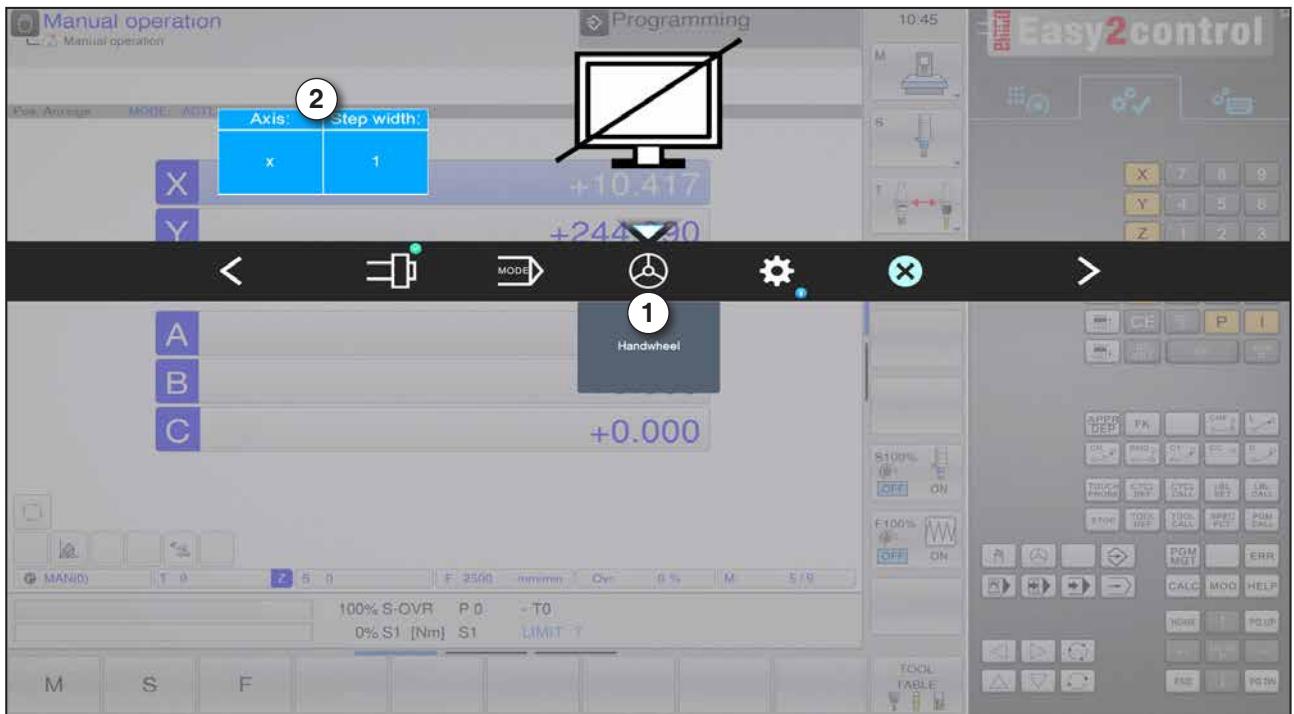
- 2 Background brightness: adjusts the transparency of the background
- 3 Lock screen: Pressing the knob again unlocks it.
- 4 Close: The sub-menu is closed. Return to the superordinate menu item.

#### Setting the background brightness



- Pressing once will cause a white frame to appear around the symbol. The menu item is activated.
- By turning the rotary switch, you can now change the transparency of the background:  
Turning left: brighter  
Turning right: darker
- By pressing the knob again you leave the menu item and the white frame disappears.





*Hand wheel function*

The hand wheel (1) activates the hand wheel mode. The parameters axis and increment (2) can be entered via the keys for axis and mode of operation on the machine keyboard.

### Operation

- The electronic hand wheel serves the purpose of moving the carriages at a predetermined increment distance.
- The increment distance is determined by the set Inc-mode of operation: Inc 1, Inc 10, Inc 100.
- An Inc-operating mode must be selected and an axis must be defined via an arrow key.
- See also "Description of the modes of operation" and "Description of the arrow keys" in section B.

#### Note:

The hand wheel cannot be used in the mode of operation "Inc 1000". "Inc 1000" moves at "Inc 100".



## Key switch

The key switch function is machine-specific.



## Additional clamping device key

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



## USB connection (USB 2.0)

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



## Enable key

When the door is open, axis movements via the arrow keys and tool turret movements are authorized by pressing the enable key (precondition key switch in SET-UP position).

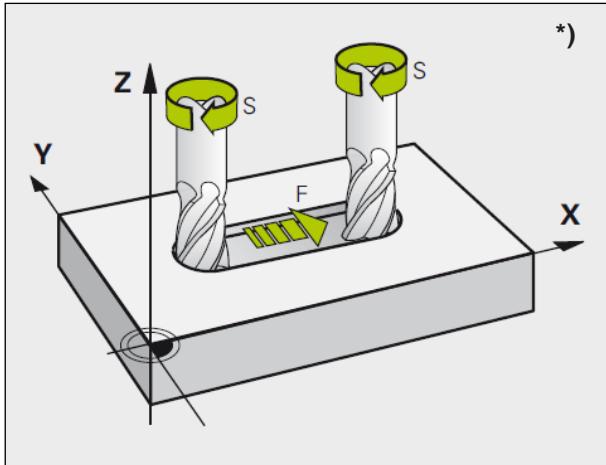
In machines with automated doors (option) pressing the enable switch opens the machine doors.



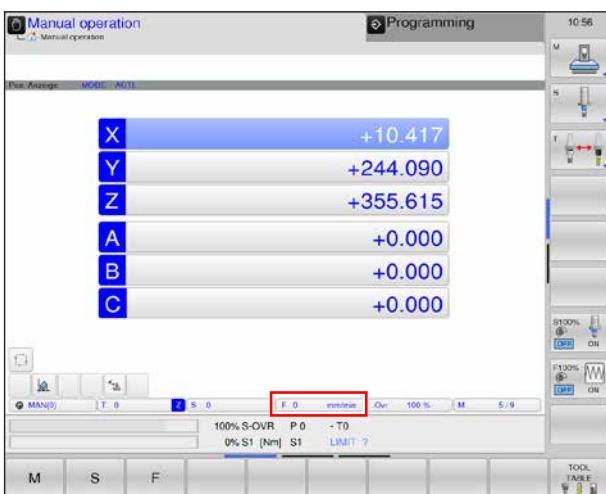
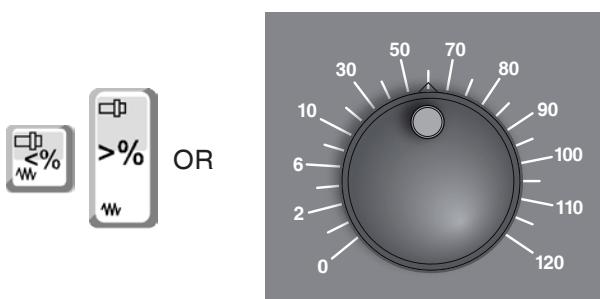
# C: Operation

## Feed F [mm/min]

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



Feed and spindle speed



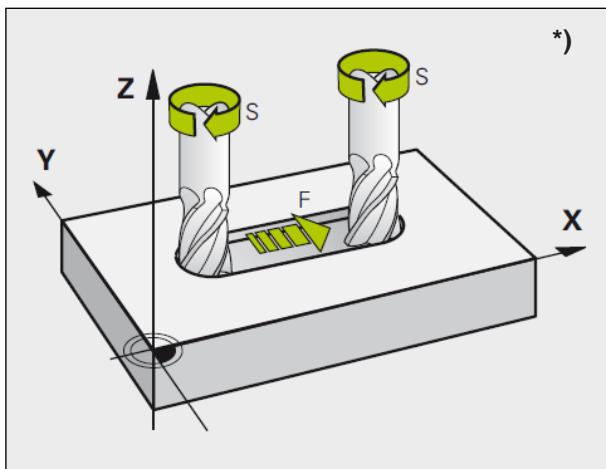
Feed

### Feed influence

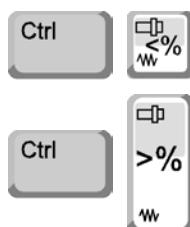
The feed value F programmed by you corresponds to 100%. With this button or with the feed override, the set feed value F can be changed in %.

Adjustment range:

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

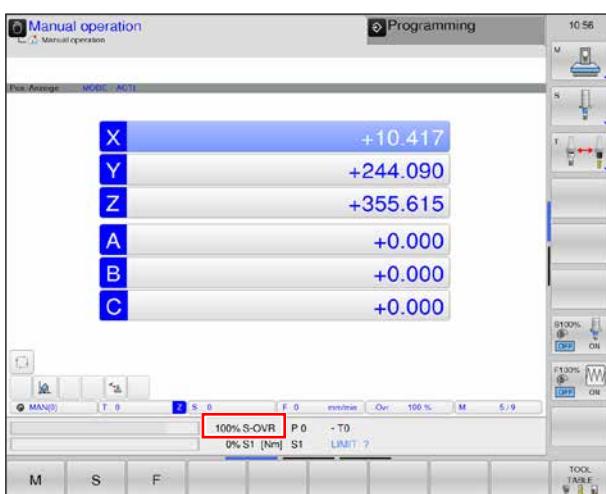


Feed and spindle speed



## Spindle speed S [rev/min]

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



Spindle speed

### Spindle speed correction

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

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

Adjustment range:

0% to 120% of the programmed spindle speed.  
Only the changed percentage and not the resulting effective value will be shown.

## Modes of operation

The operating areas of Emco WinNC for Heidenhain TNC 640 milling are divided into five machine modes of operation and two programming modes of operation.

The machine modes of operation are displayed in the header on the left side and the programming modes of operation on the right side. In the bigger field of the header, the mode of operation is displayed. Dialogue queries and message texts also appear here. See also Chapter "B" under screen layout

### Machine modes of operation



#### Manual operation

Setting of the machine: In this mode of operation, the machine axes are positioned manually or step by step and the reference points are set.



#### Electronic hand wheel

In the operating mode El. hand wheel, the machine axes can be traversed manually via an electronic hand wheel.



#### Positioning with manual data input

In this mode of operation, simple positioning movements are programmed, e. g. face milling or pre-positioning.

The program is always executed in single block mode.

A short program can also be entered and executed immediately. The cycles of the controller can also be called.

The program is stored in the file \$MDI.

#### Limitations

The following functions are not available in positioning with manual input:

- FK free contour programming
- LBL SET
- Program call PGM CALL

**Single block program run**

In the operating mode single block program run, every block is started individually via the external START key.

**Block sequence program run**

In block sequence program run, the control unit executes a program up to the end of program or up to a manual and/or programmed interruption. After an interruption, the program run can be resumed again.

**Program run single bloc/block sequence**

Control the machine by automatically executing programs.

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

Preconditions for executing part programs:

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

Options in Automatic mode:

- Block search run
- Program influence

(see Chapter F program sequence)

**Programming modes of operation****Saving / editing program**

In this mode of operation, the machining programs are created. Free contour programming, the different cycles and the Q parameter functions provide additional support for programming. On demand, the programming graphics will display the individual steps or you can use another window to create your program structure.

**Program test**

Programs and program sections are simulated by WinNC in the program test mode of operation. In this manner, geometrical incompatibilities, missing or wrong information in the program and violations of the working area can be detected. The simulation is graphically supported in different views.

**Note:**

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

## Activating modes of operation

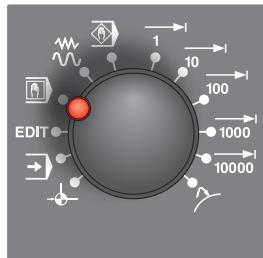
Depending on the machine configuration, the modes of operation can be activated as follows:



via the address and number pad



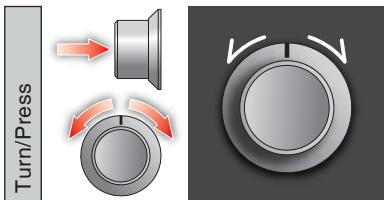
via the PC keyboard



via the mode selector switch of the machine control panel



via the machine control panel variant Easy2Control with MOC-Touch

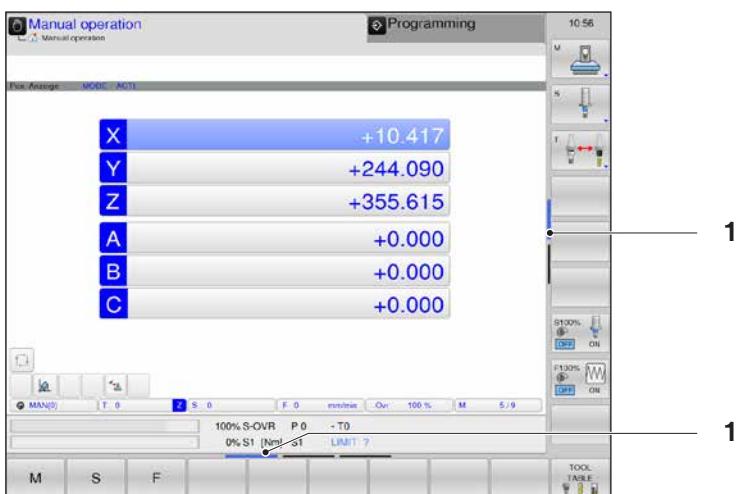


via multifunction wheel of the Easy2Operate

## Navigation in the menu window



In the footer, further functions of the softkey bar are displayed. For orientation purposes, narrow bars directly above the softkey bar show the number of softkey bars, which can be selected via the black arrow keys arranged on the outside or via the F11 and/or F12-key. The active bar is displayed as a blue bar (1).



### Inc 1 - Incremental feed

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



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

### Inc 10 - Incremental feed



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

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

### Inc 100 - Incremental feed



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

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

### Inc 1000 - Incremental feed



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

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

#### Note:

Metric is assigned in the imperial measuring system as follows:



#### Feed:

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

#### Constant cutting speed:

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

## Approaching the reference point

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



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

- Switch to the reference mode REF.

### Option A: Referencing the axes individually



Press the +Z and +X buttons.

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



#### Note:

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

### Option B: Reference automatically



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

## Move carriages manually

The machine axes are moved manually using the arrow keys.



- Switch to manual mode.



- The arrow keys move the axes in the corresponding direction as long as the key is pressed.
- The feed speed is set with the override switch.
- If the key is pressed simultaneously, the carriages move in rapid traverse.



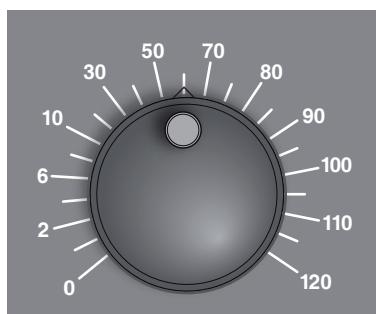
## Move carriages in step-movement

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

- Switch to INC mode.



- The arrow keys move the axes in the corresponding direction by the set incremental distance each time the button is pressed.
- Set the softkey INCREMENT to ON, in order to select incremental positioning.



- The feed speed is set with the override switch.
- If the key is pressed simultaneously, the carriages move in rapid traverse.



Input the traverse for linear axes in mm (1) and rotary-axes in degree ° (2).

- Values between 0.0001 and 10 mm or degrees can be entered.

The values entered correspond to "INC var" .

 CONFIRM VALUE

Confirm the entered value.

 OK

Complete the entry with OK.

 SWITCH OFF

Switch off increment positioning.

## Administration of reference points in the preset table

- The preset table is stored under the name PRESET.PR in the directory TNC: \ table \.



- Press softkey to open the preset table.



OR

- The preset table can only be used in manual mode and El. Handwheel.
- Press softkey "CHANGE PRESET" to change the preset table.
- In programming mode the preset table can be opened, but not be edited.

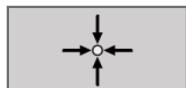


### Note:

- Copying the preset table to another directory for data backup is allowed. Write-protected lines are also read-only in copied tables and can not be changed.
- Do not change the number of lines in copied tables! This can cause problems if you want to re-activate the table.
- In order to activate the preset table that was copied to another directory, you must copy it back to the directory TNC: \ table \.



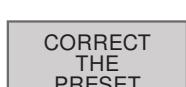
## Editing features



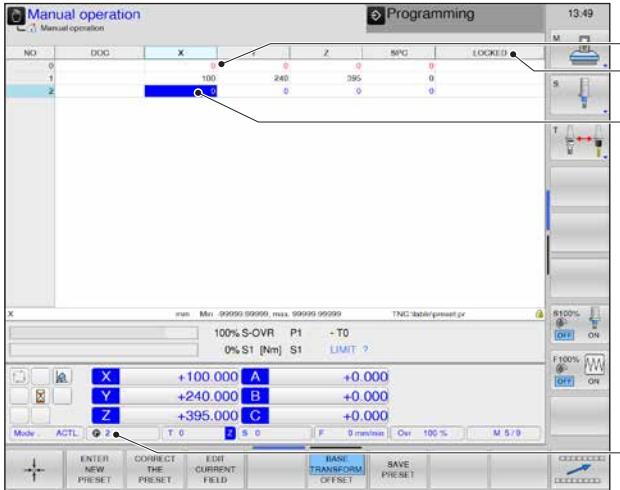
- Take over the actual tool position directly as a new reference point: The function stores the reference point only in the axis which is currently active.



- Assign an arbitrary value to the actual position of the tool: The function stores the reference point only in the axis which is currently active. Enter the desired value in the pop-up window.



- Moves a stored reference point incrementally: The function stores the reference point only in the axis which is currently active. Enter the desired value in the pop-up window.

*Structure of the preset table*

### Structure of the preset table

- 1
  - 2
  - 3
  - 4
- The first line (1) is marked red, is locked and can not be changed. A manually defined zero point and / or reference point is stored in here.

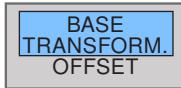
A blue colored field (2) marks the entry which is currently being edited.

A line marked in blue indicates that the preset defined in here is selected. If there is no blue line, line 0 is selected. If a line is locked via the LOCKED column (3), it is highlighted in red.

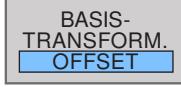
Display for the selected line of the preset table (4), the number represents the line. MAN stands for line 0.



- Enter the new reference point directly (axis-specific) without the kinematics being set. Use this function only if the machine is equipped with a round table and if you want to set the reference point to the round table center by entering 0 directly. The function stores the value only in the axis in which the field is currently active. Enter the desired value in the pop-up window.



- Select BASE TRANSFORMATION/ OFFSET:  
In standard view base transformation, the columns X, Y and Z are displayed. SPA, SPB and SPC columns are displayed depending on the machine. The control software stores the basic rotation (in the case of tool axis Z, the control uses column SPC).



The Offset view displays the offset values for the preset.



- Writes the currently active reference point to a selectable line: The function saves the reference point in all axes and then activates the respective table line automatically.

### Edit functions for the table



- Append N lines at the end of the table.



- Copy actual selected field.



- Paste field.



- Reset actual selected line: All fields of the line are reset to 0.



- Append a single line at the end of the table.



- Delete single line at the end of the table.

**Note:**

Preset tables can only be opened but not be edited in Program Manager.



## Switch off

In order to avoid data loss when switching off the machine, you must specifically shut down the operating system of the WinNC.



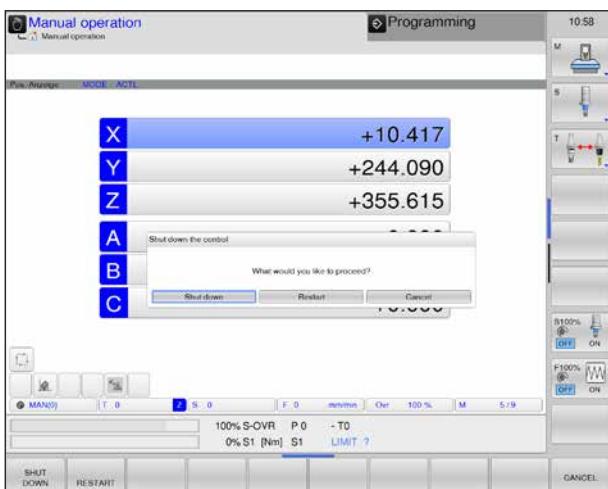
**1** Select manual mode of operation.

**2** Switch the softkey bar until the function for shut down appears.

**3** Select the shut down function

**4** SHUT DOWN or

**5** RESTART.



*Shut down*

As soon as the control unit has been shut down, the supply voltage of the control unit can be switched off.

Switching off the WinNC randomly can lead to data loss!

### Note:

In our reproduction, the "electronic hand wheel" mode of operation behaves in the same manner as the "manual operation" mode. In order to be able to move the axes with the hand wheel, you have to change into one of the INC modes (1 - 100) on the machine control panel and then select the appropriate axis (see machine operating instructions).

## File management fundamentals

### Files

Files in the control unit	Type
<b>Programs</b> in HEIDENHAIN format	.H
<b>Tables for</b> Tools Preset-table	.T .PR

For quick finding and management of files, WinNC is provided with a specific file management window. You can call up, copy, rename and delete the different files.

With WinNC you can manage any number of files, the total size of the files is only limited by the hard disc storage capacity.

#### File names

For programs, tables and texts an extension needs to be added, which extension is separated from the file name by a dot. This extension marks the file type.

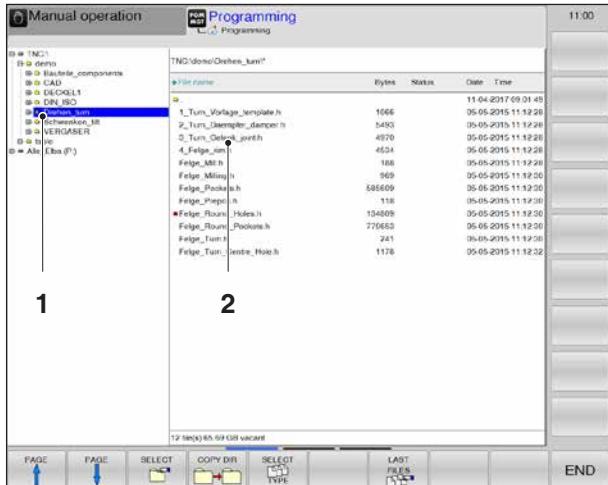
PROG20	.H
File name	File type

## File management

### Opening program manager

Press PGM MGT key.  
WinNC opens the file management window.

PGM  
MGT



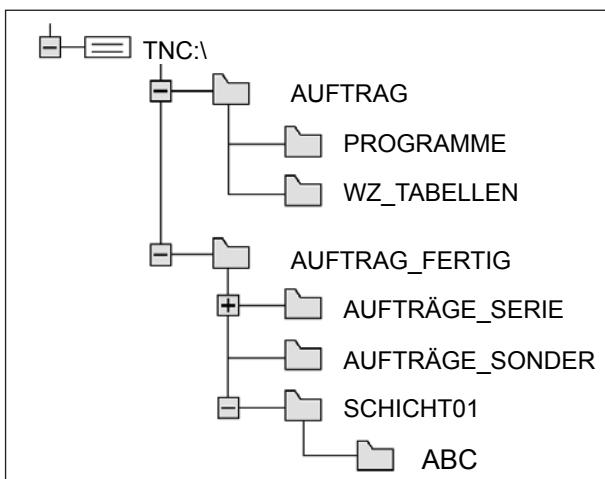
File management

The narrow window on the left-hand side shows the available drives and directories (1). Drives are those devices in which data are stored or transmitted. A drive is the hard disc drive of the WinNC, further drives are interfaces like network interfaces and USB port.

In **EMConfig** you can specify, which drives will be displayed (see startup information Chapter X "Processing Ini-data of WinNC").

The broad window on the right-hand side (2) displays all files that are saved in the selected directory. Information associated with every file is shown, the following table shows the information:

Display	Meaning
File name	Name with maximum 16 characters and file type
Byte	File size in bytes
Status	Characteristic of the file Program is in operating mode
E	Program is selected in the programming operating mode
S	Program is selected in the program test operating mode
M	Program is selected in the program run operating mode
	File is protected against deletion and change



Paths

## Paths

A path specifies the drive and all directories and subdirectories. The files are saved here. The individual pieces of information are separated by "\".

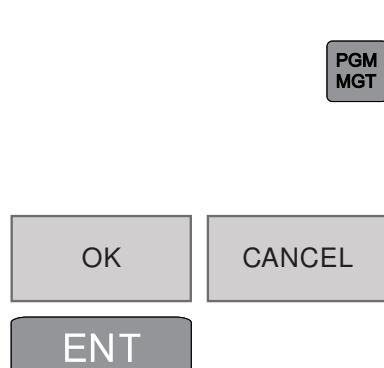
Example:

On the drive **TNC:\** the directory **ORDER** was created.

After that, the subdirectory **PROGRAMS** was created in the directory **ORDER**, and the machining program **PROG1.H** was saved there. Thus, the machining program has the path:

**TNC:\ORDER\PROGRAMS\PROG1.H**

The graph on the left-hand side shows an example of a directory display with different paths.

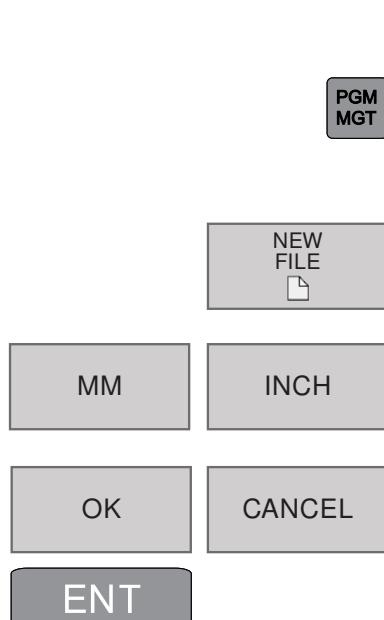


## Creating a new directory

**1 Call up file management**

Tag the directory in the left window, in which you want to create a subdirectory.

**2 Enter the new directory name and complete the entry with OK or ENT and/or cancel the entry with CANCEL.**



## Creating a new file

**1 Call up file management**

Select the directory, in which you want to create the new file

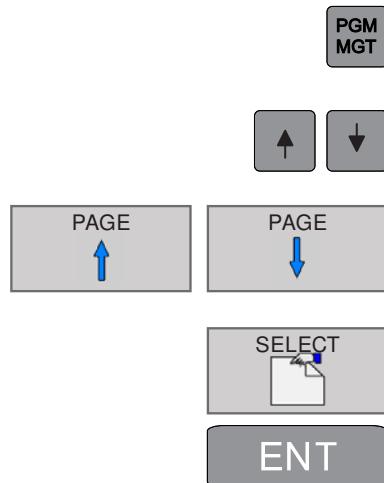
**2 Open the dialogue for creating a new file**

**3 Enter the new file name with the extension,**

**4 Enter measurement system**

**5 Complete the file entry with OK or ENT or cancel it with CANCEL.**

## Selecting a file



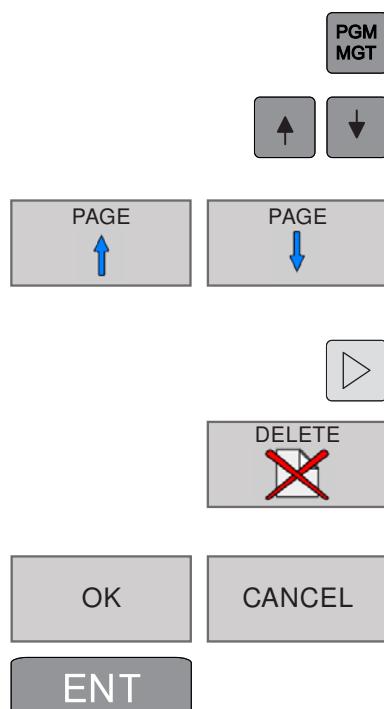
**1** Call up file management

**2** Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to select.

**3** Moves the illuminated field per page up and down in the window.

**4** Select file: Press softkey SELECT or ENT.

## Delete file



**1** Call up file management.

**2** Using the arrow keys or the arrow softkeys, move the illuminated field to the file that you want to delete.

**3** Moves the illuminated field per page up and down in the window.

**4** Switch the softkey bar until the function for DELETE appears.

**5** Delete file: Press softkey DELETE.

**5** Confirm with OK or ENT or cancel with CANCEL.

## Deleting a directory

### Note:

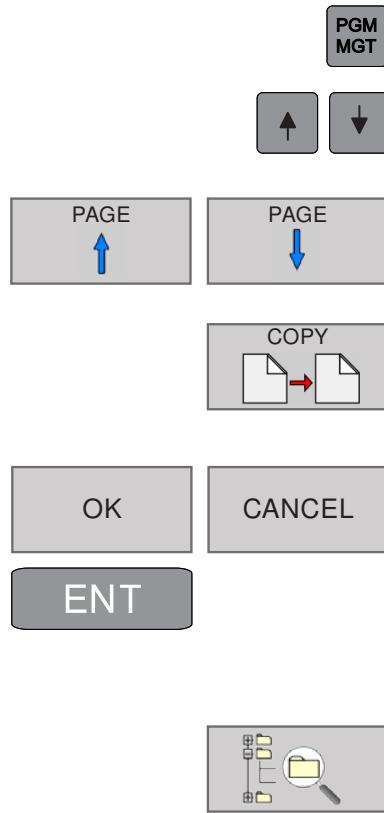
When deleting files and directories, data loss is possible!  
The deletion process cannot be reversed!



Move the illuminated field using the arrow keys or the arrow softkeys to the directory that you want to delete.

For the further deletion process, see deleting file above.

## Copying a file



**1** Call up file management

**2** Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to copy.

**3** Moves the illuminated field per page up and down in the window.

**4** Copy the file: Press softkey COPY.

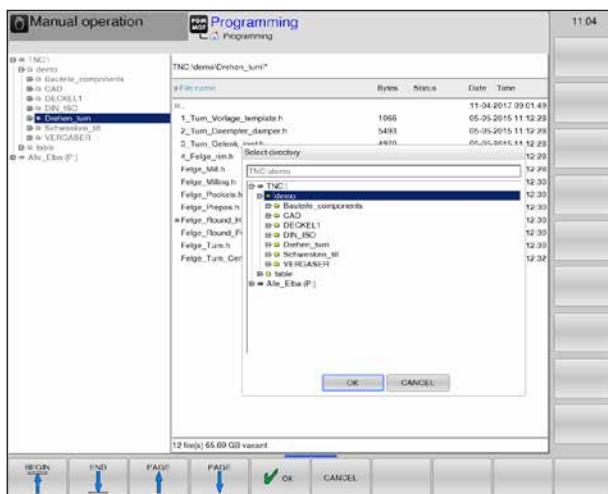
**5** Enter new file name.

**6** Confirm with OK or ENT or cancel with CANCEL.

The control unit copies the file into the current directory, and/or into the selected target directory. The initial file remains unchanged.

**7** You can also select a target directory.

**8** Press the softkey and select the desired directory.

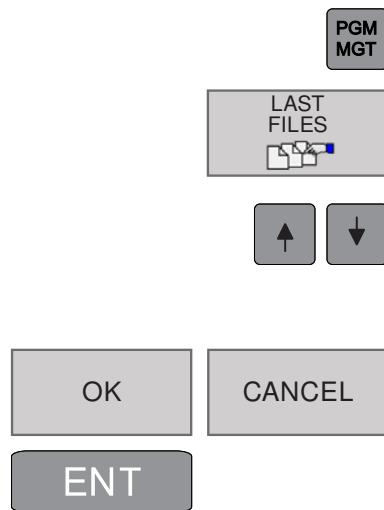


**9** Complete the entry with OK or ENT.

### Note:

The control unit displays a status window with information about the progress of the copying process. As long as data are being copied, work cannot be resumed.

## Selecting one of the last 10 selected files



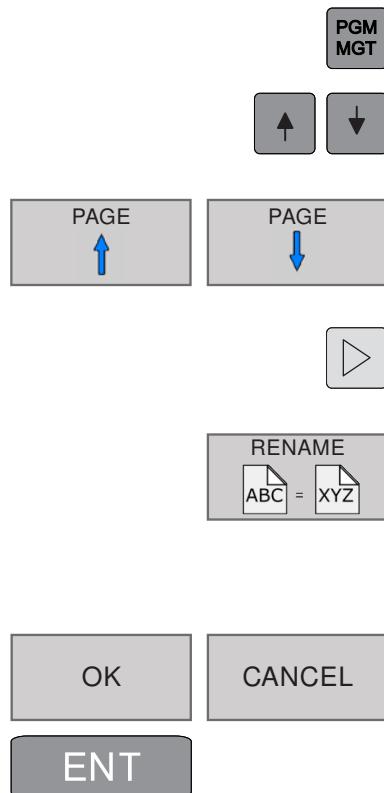
**1** Call up file management

**2** Display the last 10 selected files: Press softkey LAST FILES

**2** Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to select.

**4** Select file with OK or ENT or cancel with CANCEL.

## Renaming files



**1** Call up file management

**2** Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to rename.

**3** Moves the illuminated field per page up and down in the window.

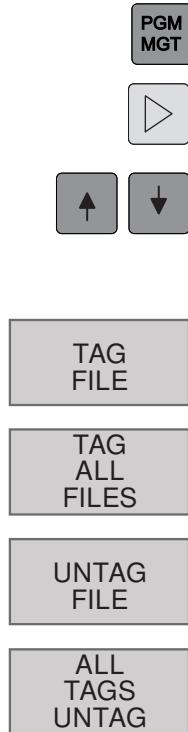
**4** Switch the softkey bar until the function for RENAME appears.

**5** Rename file: Press softkey RENAME.

**6** Enter new file name. The file type cannot be changed.

**7** Select file with OK or ENT or cancel with CANCEL.

## Tagging files



**1** Call up file management

**2** Switch the softkey bar until the function for TAG appears.

**3** Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to rename.

The following functions are available:

**4** Tag individual file

**5** Tag all files in the directory

**6** Remove tag for individual file

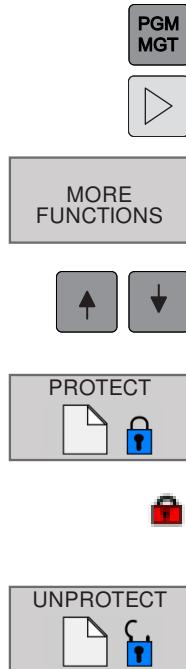
**7** Remove tag for all files

**Note:**

Functions like copying or deleting files can be applied to individual files as well as to several files simultaneously.

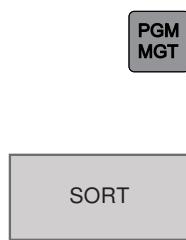


## Additional functions



- 1** Call up file management
- 2** Switch the softkey bar until the function for MORE FUNCTIONS appears.
- 3** Press softkey
- 4** Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to select.
- 5** Press softkey PROTECT to activate file protection.
- 6** The file receives the status P and is now protected against changes and deletion.
- 7** Cancelling file protection: Press softkey UNPROTECT.  
The protect status is cancelled.

## Sorting



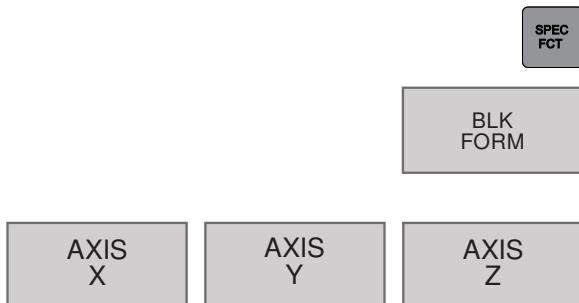
- 1** Call up file management
  - 2** Select the folder, in which you want to sort files
  - 3** Press SORT softkey.
- The files can be sorted according to the following criteria:
- Name
  - Size
  - Date
  - Type
  - Status

## Special functions

### Defining a raw part: BLK FORM

After opening a new program, a rectangular-shaped unmachined work piece is defined.

For subsequent definition of the raw part proceed as follows:



1 Call up Special Functions

2 Press softkey BLK FORM. This definition is required by WinNC for graphical simulation.

The raw part is determined by two of its corner points.

3 MIN-Point: smallest X-,Y- and Z-coordinate of the cuboid; enter absolute values.

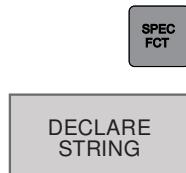
4 MAX-Point: biggest X-,Y- and Z-coordinate of the cuboid.

Enter absolute or incremental values.

**Note:**

The raw part definition is required if you want to test the program graphically.

## Assigning string parameters



1 Call up Special Functions

2 Press softkey DECLARE STRING.

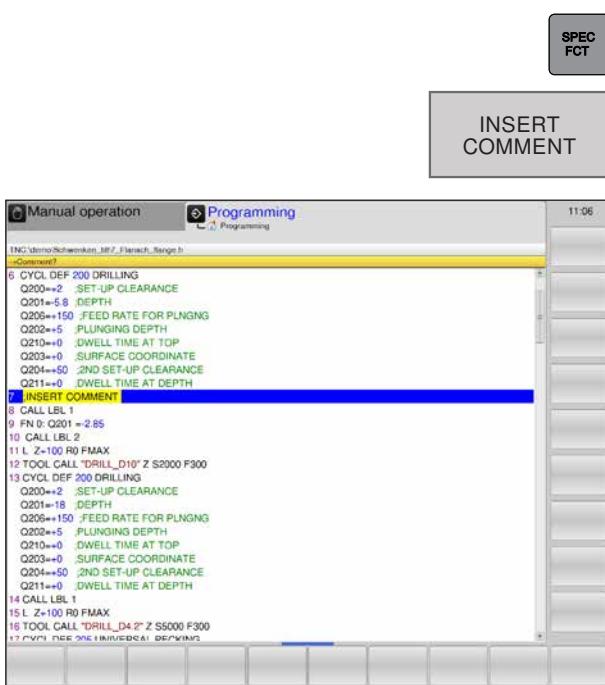
Before string variables are used, they must be assigned with the DECLARE STRING command:

Example: DECLARE STRING QS10 ="WORK PIECE"

## Insert comment

You can insert comments in the machining program, in order to give information about the programming or to explain program steps.

- 1 Call up Special Functions
- 2 Press softkey INSERT COMMENT.
- 3 WinNC inserts a new line that begins with a semicolon (;).
- 4 Enter the comment and complete entry with END.

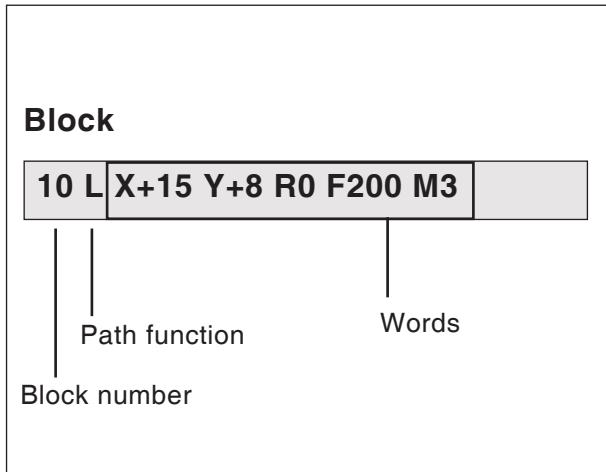


*Insert comment*



End Special Functions.

## Opening and entering programs



*Elements of the program block*

### Structure of an NC program in HEIDENHAIN plain text format

NC machining programs consist of a number of program blocks.

The figure on the left shows the elements of a block.

The blocks of a machining program are numbered by WinNC in ascending order.

The first block of a program consists of:

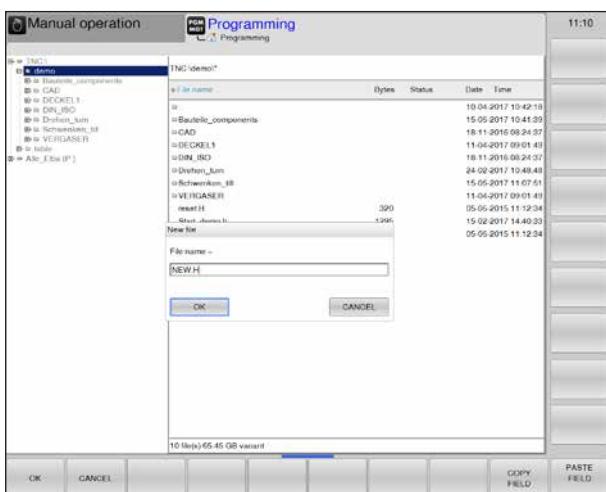
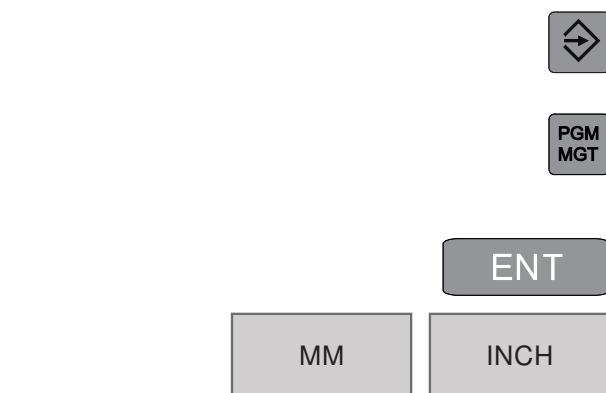
- BEGIN PGM
- the program name
- the valid measurement unit.

The subsequent blocks contain information about:

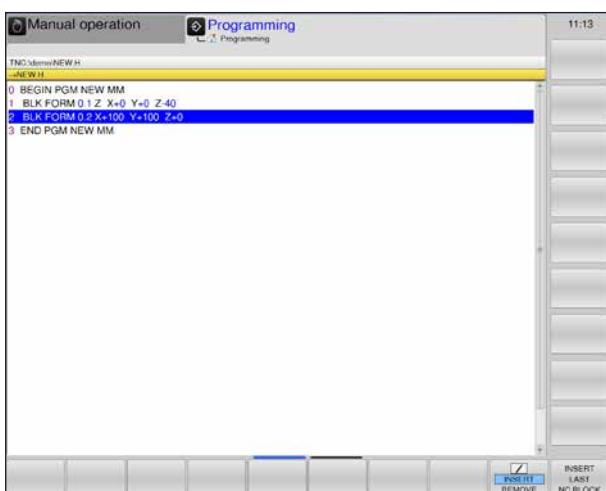
- the raw part
- tool definitions and calls
- feed rates and speeds
- Path movements, cycles and further functions

The last block of a program consists of:

- END PGM
- the program name
- the valid measurement unit.



Entering a new program



Displaying BLK Form in the program

## Opening a new machining program

**1** Select operating mode saving/editing program.

**2** Call up file management.

Select the directory, in which you want to save the new program.

**3** Enter the new program name and confirm by pressing the ENT key.

**4** Select measurement unit: Press softkey MM or INCH.

WinNC changes into the program window and opens the dialogue for definition of the **BLK-FORM** (raw part).

**Machining plane in graph: XY  
Spindle axis parallel X/Y/Z?**  
Enter spindle axis

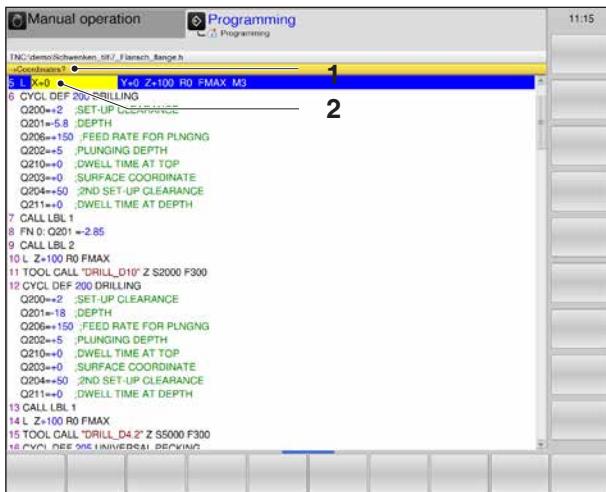
**Raw part definition minimum:**

Enter the X-, Y- and Z-coordinates of the MIN-point one after the other and confirm with the ENT key.

**Raw part definition maximum:**

Enter the X-, Y- and Z-coordinates of the MAX-point one after the other and confirm with the ENT key.

WinNC creates the block numbers, the BEGIN and END block automatically.

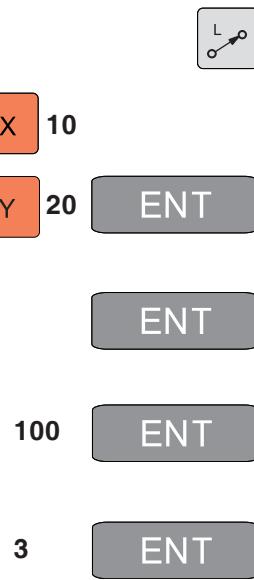


Programming a program block

## Programming tool movements in plain language dialogue

Programming of a block starts with a dialogue key. In the header of the screen, a query (1) appears in conjunction with the data entry. The current position for the data entry is additionally highlighted in yellow colour (2).

### Example of a dialogue



1 Open dialogue

2 Coordinates?

Enter the target coordinate for the X-axis

3 Enter target coordinate for the Y-axis  
with ENT key to the next question

4 Radius compensation: RL/RR/No compensation?  
Enter "No radius compensation", with ENT key to  
the next question

5 Feed F=? / F MAX = ENT

Feed for this path movement 100 mm/min, with  
ENT key to the next question

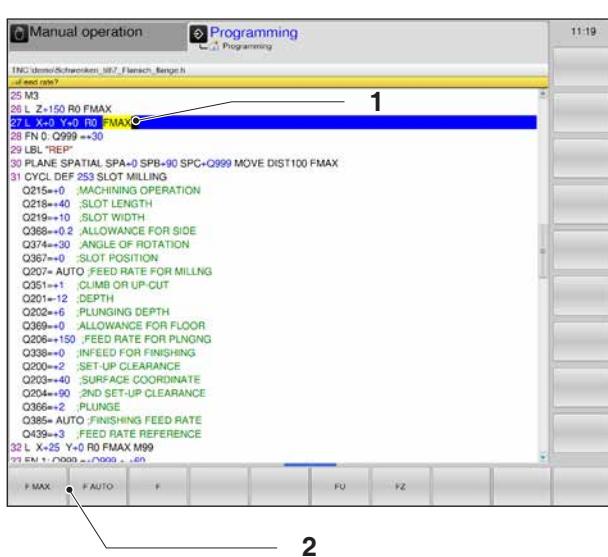
6 Additional function M?

Additional function **M3** "Spindle clockwise",  
pressing the ENT key ends the dialogue.

The program window shows the following line:  
**3 L X+10 Y+20 R0 F100 M3**

### Functions for dialogue guidance

Function	Key
Skip dialogue question	
Prematurely skip dialogue question	
Cancel and delete dialogue	



### Additional softkeys for data entry

In the context of the query (1), several entry possibilities via softkeys (2) are available for certain data, such as feed, radius compensation etc.

F MAX

F AUTO

F

FU

FZ

### Functions for determining the feed

**1** Move in rapid traverse, effective per block.

**2** Automatically calculated feed.

**3** Programmed feed: Unit mm/min.

For rotation axes, the control unit interprets the feed in degrees/min, independently of the program being written in mm or inch.

**4** Define revolution feed: Unit mm/rev.

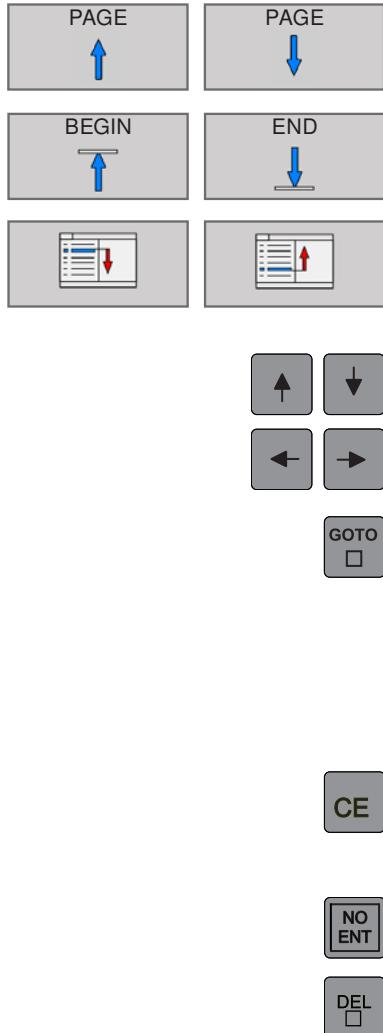
**5** Define tooth feed:

Unit mm/tooth.

The number of teeth must be defined in the tool table in the CUT. column.

## Editing program

When creating or changing a machining program, you can select every line in the program as well as individual words of a block via arrow keys (or softkeys):



**1** Scroll page up / down

**2** Jump to program start / end

**3** Change the position of the current block on the screen. Gives you the possibility to display further program blocks that are programmed before and/or after the current block.

**4** Jump from block to block

**5** Select single words in the block

**6** Select a specific block:

Press the GOTO key, enter the desired block number and confirm with the ENT key.

Or:

Enter the block number step and skip the number of entered lines by pressing the softkey N LINES up or down.

**7** Set the value of a selected word to zero

Delete wrong value

Delete error message (not blinking)

**8** Delete selected word

**9** Delete selected block

Delete cycles and program sections

### Inserting blocks at any desired position

Select the block after which you want to insert a new block and open the dialogue.

### Changing and inserting words

Select a word in a block and overwrite it with the new word. While you have selected the word, the plain language dialogue is available.

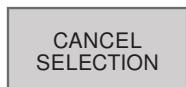
Complete the change: Press END key.

If you want to insert a word, use the arrow keys (to the left or to the right) until the desired dialogue appears and enter the desired value.

## Marking, copying, cutting and inserting program sections

The WinNC provides the following functions for copying program sections within an NC program or into another NC program:

### Coying program sections



- 1 Switch the softkey bar until the function marking appears.

Select the first (last) block of the section you want to copy.

- 2 Mark the first (last) block: Press the SELECT BLOCK soft key.

- 3 The first digit of the block number is marked with a bright field and activates the CANCEL SELECTION softkey.

- 4 Move the highlight to the last (or first) block of the program section you wish to copy or cut. The WinNC shows the marked blocks in a different color.

Press the softkey CANCEL SELECTION to end the marking function.



### Copying marked program part

- 1 Press softkey COPY BLOCK.

- 2 Press softkey CUT OUT BLOCK to cut out the selected program section.

The WinNC stores the marked block.

Using the arrow keys, select the block after which you wish to insert the copied (cut) program section.

- 3 Press softkey INSERT BLOCK to insert the stored program section, or

- 4 Press softkey CANCEL SELECTION to end the marking function.

## MOD function

Via MOD functions you can select additional display and entry possibilities. It depends on the selected mode of operation, which MOD functions are available.

### Selecting MOD functions

Press the MOD key and open the MOD screen.

- The available options and functions depend on the WinNC software version.



MOD functions



### Exiting MOD functions

Press the MOD key again or press the softkey CANCEL.

## Graphical simulation

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

### Raw part definition

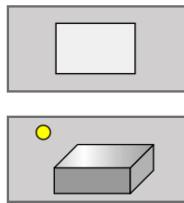
The raw part dimensions entered in the program editor are used for the workpiece.

The raw part will be clamped relative to the coordinate system valid at the time of the raw part definition.

The graphic simulation is available in all operating modes. .

The control unit offers the following views (see further view options).

- Top view
- 3D display

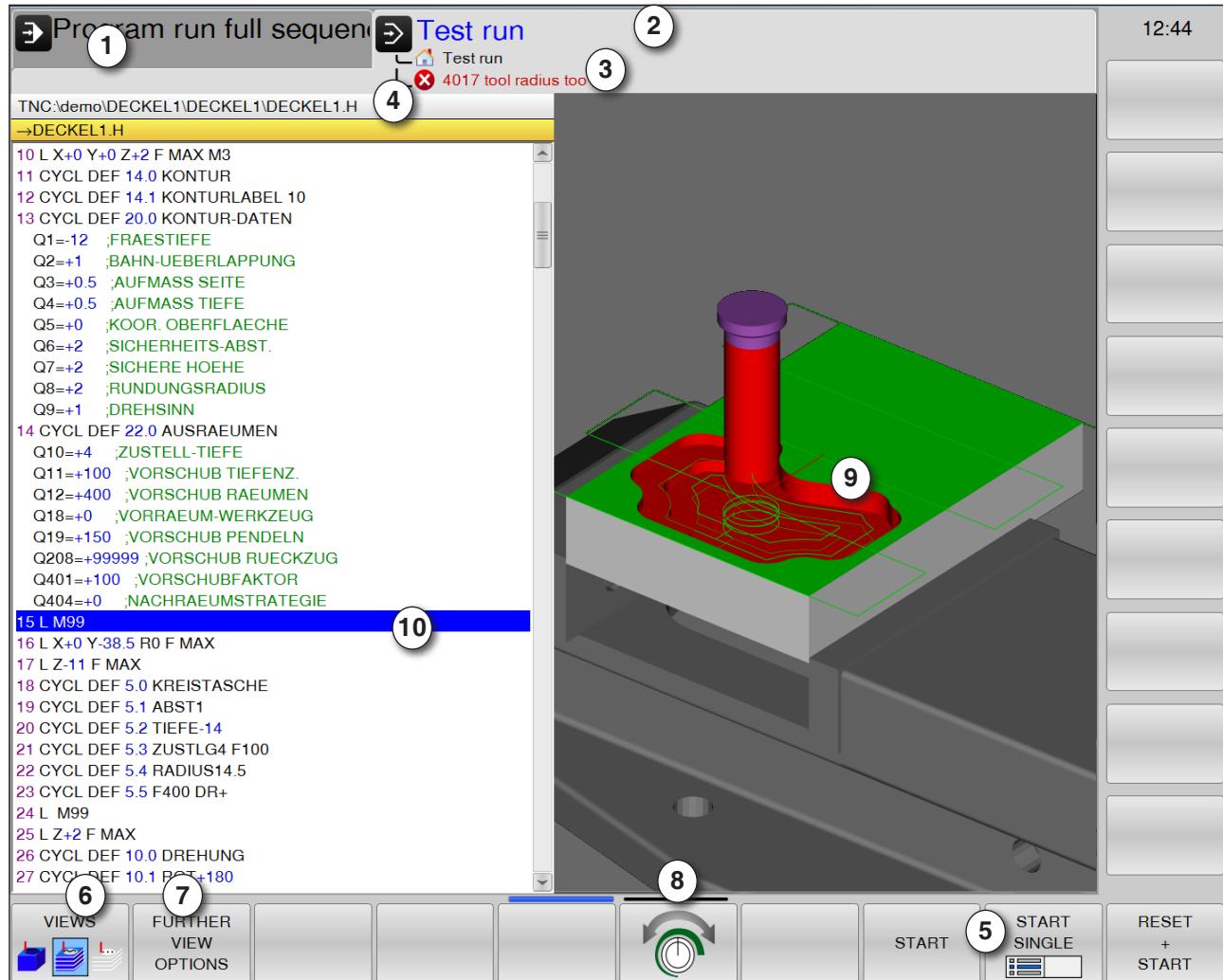


### Testing the program



Change to programming operating mode program test in order to test the current program.

## Screen layout graphical simulation



- 1 Mode
- 2 Active operating area / program test
- 3 Simulation alarm and notification lines
- 4 Program name
- 5 Softkeys "Start", "Start single block" and "Reset + Start" of the simulation
- 6 Select view
- 7 retrieve further view options
- 8 Set the simulation speed
- 9 Coloured positioning movements:
  - Red positioning movement = tool moves in rapid traverse.
  - Green positioning movement = tool moves in machining feed.
- 10 Current block in the program

## Softkey functions

### Simulation speed

Set the speed of the simulation



Incrementally raise and/or lower the simulation speed.

Incrementally lower the simulation speed



Test the program at maximum possible speed (predefined setting)



### Simulation start / reset

Start simulation



Reset and start simulation



Start simulation single block



Simulation reset



### Selecting views

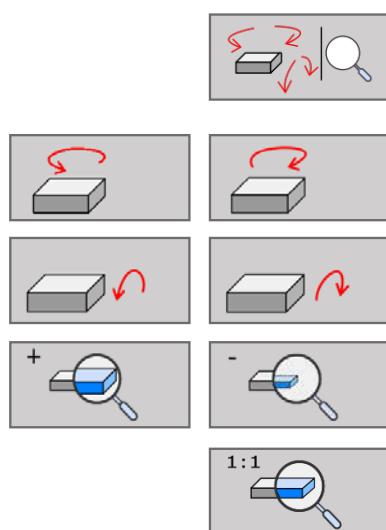
Volume view



Volume view and tool paths



Tool paths

**3D display**

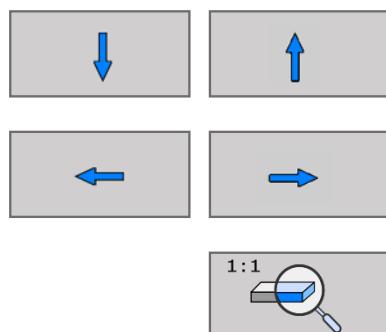
Functions for rotating, enlarging and reducing

Rotate picture in 5° steps

Tilt picture horizontally in 5° steps

Incrementally enlarge / reduce picture

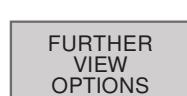
Reset picture to original size and angle.

**Shifting the display**

Shift the picture up and down

Shift the picture to the right and to the left

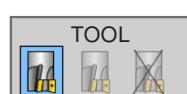
Reset picture to original position and angle.

**Further view options**

Press softkey to show further options

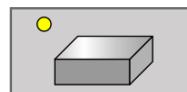


Display the work piece as transparent



Display the tool during simulation:

- Display tool
- Display tool as transparent
- Fade out tool
- Display work piece in top view
- Display work piece in 3D viewD



### Skiping blocks



Do not test and/or execute program blocks with the "/" sign.

### Shifting the graphic display with the mouse

Keep the right mouse button pressed and move the mouse to shift the model.

With the shift key pressed simultaneously, you can shift the model only in horizontal or vertical direction.

With the CTRL key pressed simultaneously, you can enlarge and/or reduce the picture by dragging the mouse.

Instead of the right mouse key, you can also use the mouse wheel.

By pressing the left mouse key, you can rotate the picture horizontally and vertically.



# D: Programming

**Note:**

This programming manual describes all functions that can be executed with EMCO WinNC for Heidenhain TNC 640.

Depending on the machine you are operating with WinNC, not all functions may be available.

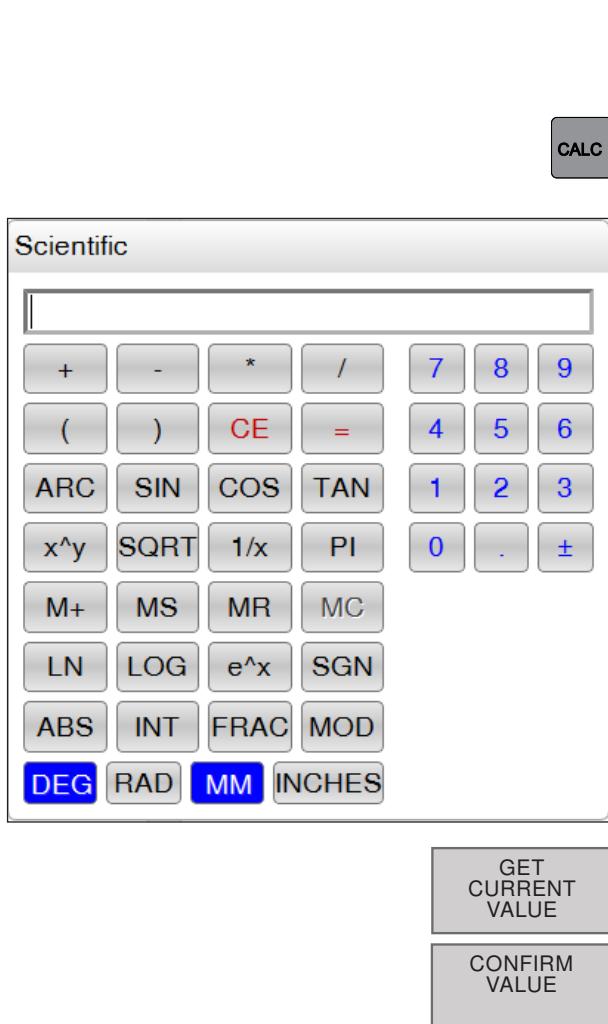
**Example:**

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

## Overview

### M-commands

M00	Programmed stop	M08	Coolant ON
M01	Optional stop (program stop only at OPT. STOP)	M09	Coolant OFF
M02	End of program	M13	M3 + M8
M03	Spindle ON clockwise	M14	M4 + M8
M04	Spindle ON counterclockwise	M15	M5 + M9
M05	Spindle stop	M30	End of main program
M06	Perform tool change	M91	Move in machine coordinates
		M89	Cycle call modal
		M99	Block wise cycle call



## Calculator

The control unit has a calculator with the main mathematical functions.

Press the CALC button to open and/or close it.

When entering a program in dialogue mode, you can copy the display of the calculator directly into the marked field by pressing the "Adopt current positions" button.

- Transfer current value from the program to the calculator.
- Transfer calculated value into the program.

## Arithmetic operators

Function	Meaning
+,-,*,/	Fundamental arithmetic operations
()	Bracket calculation
ARC	Arc cosine
SIN	Sine function
COS	Cosine function
TAN	Tangent function
X <sup>Y</sup>	Exponentiate
SQR	Square root computation
1/x	Inverse function
PI	Circle division number PI 3.14159265359
M+	Add value to buffer memory
MS	Store value temporarily
MR	Retrieve buffer memory
MC	Delete buffer memory

Function	Meaning
LN	Logarithm Naturalis
LOG	Logarithm
e <sup>x</sup>	Exponential function
SGN	Check sign
ABS	Create absolute value
INT	Cut off decimals
FRAC	Cut off pre-decimal positions
MOD	Modulus value
View	View: Formula editor or scientific
CE	Delete value
MM or INCH	Measurement unit
DEG or RAD	Display of angle values
DEC or HEX	Display type of numerical values

## Error messages

Errors are displayed in the event of:

- wrong entries
- logical errors in the program
- non-executable contour elements
- improper touch probe system operations
- retrieving error window with ERR.



If an error occurs, it is displayed in red letters in the header; if an error message is long and/or running over several lines, it is abbreviated. If errors occur in the background operating mode, they are displayed by the word "Error" in red letters.

If an "Error in data processing" exceptionally occurs, the error window opens automatically. Such errors cannot be rectified by the operator. In such a case, shut down the system and restart the control unit.

Error messages are displayed in the header until they are either deleted or replaced by an error of a higher priority.

Error messages that contain the number of a program set have been caused by this set or by a preceding set.



- Delete the error message with the CE button.

## Tool movements \*)

### Path functions

The contour of a work piece is composed of multiple contour elements such as straight lines and arcs. The tool movements for straight lines and arcs are programmed by means of the path functions.

### Additional functions M

With the additional functions, the WinC controls the following:

- the program run, e. g. an interruption of the program run
- the machine functions, e. g. switching the spindle rotation and the coolant on and off
- the path behaviour of the tool

### Subprograms and program section repeats

Repeated machining steps are entered only once, namely as a subprogram or a repetitions of a program part.

If a part of a program is to be executed only under specific conditions, these program steps are also defined in a subprogram.

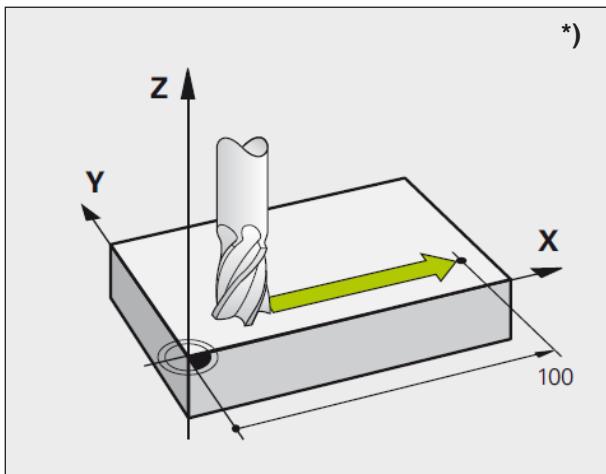
A machining program can activate and execute another machining program.

### Programming with Q parameters

In the program, Q parameters represent numerical values: A Q parameter is assigned a numerical value elsewhere. Q parameters are used to program mathematical functions, which control the program run or describe a contour.

## Fundamentals of Path Functions

\*)



*Movements parallel to the machine axes*

### Programming tool movements for a machining operation

In the machining program, you consecutively program the path functions for the individual elements of the work piece contour. To this end, enter the coordinates for the end points of the contour elements from the dimension drawing. WinNC determines the actual travel path of the tool from these coordinate values, the tool data and the radius compensation.

WinNC simultaneously moves all machine axes you have programmed in the program block of a path function.

### Movements parallel to the machine axes

The program block contains the coordinate value: WinNC moves the tool parallel to the programmed machine axis.

During machining, the machine table moves with the clamped work piece. A programmed path movement presupposes that the tool is moving.

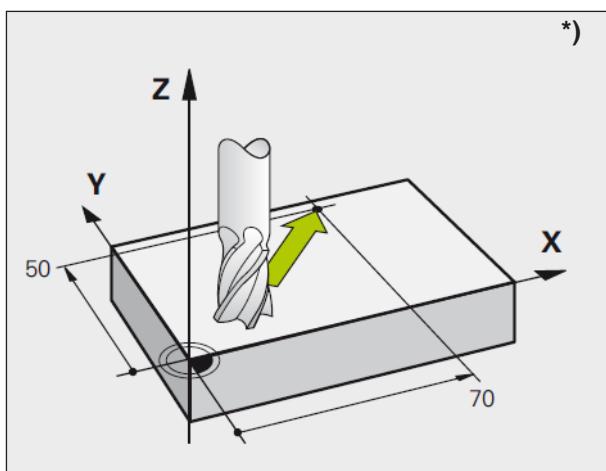
Example:

**L X+100**

L ..... Path function "straight line"

X+100 .. Coordinates of the end point

The tool maintains the Y- and Z-coordinates and moves to position X=100.



*Movements in the main planes*

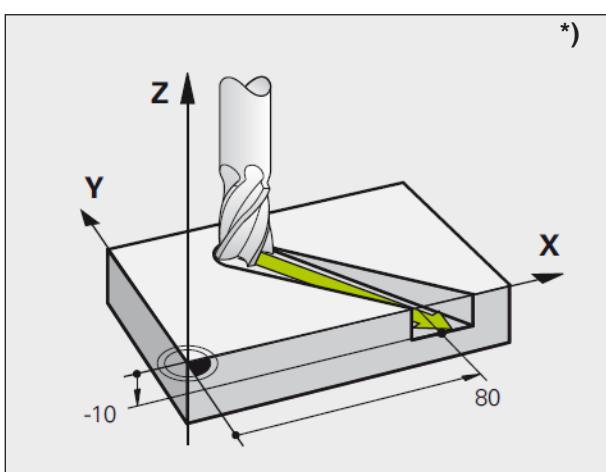
### Movements in the main planes

The program block contains two coordinate values: WinNC moves the tool in the programmed plane.

Example:

**L X+70 Y+50**

The tool maintains the Z-coordinate and moves in the XY plane to the position X=70, Y=50.



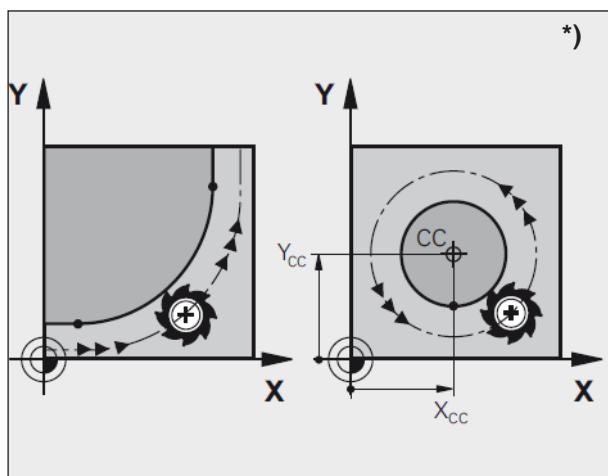
*Three-dimensional movement*

### Three-dimensional movement

The program block contains three coordinate values: WinNC moves the tool to the programmed position spatially.

Example:

**L X+80 Y+0 Z-10**

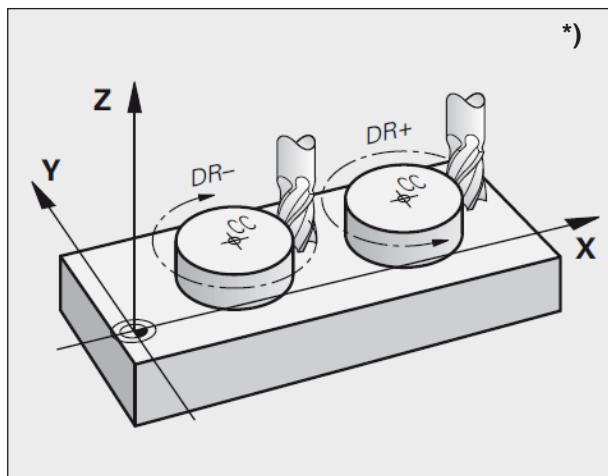
*Circular movements*

### Circles and arcs

For circular movements, two machine axes are moved simultaneously: In doing so, the tool is moving on a circular path relative to the work piece. For circular movements, a circle centre CC is entered.

For path functions for arcs, you program circles in the main planes: In doing so, the main plane is defined during "TOOL CALL" when the spindle axis is defined:

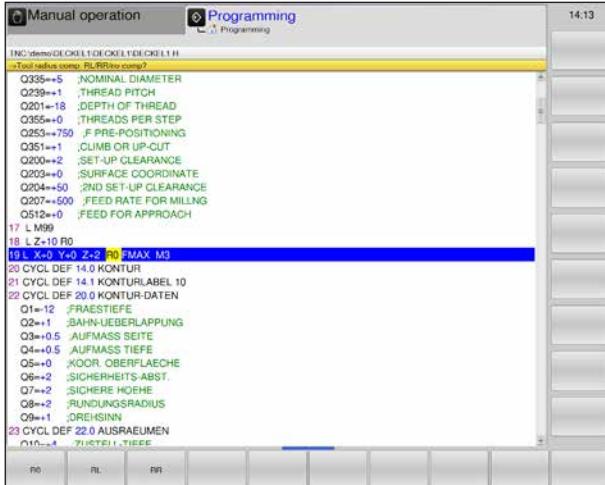
Spindle axis	Main plane
Z	XY, also UV, XV, UY

*Direction of rotation of the circular movements*

### Direction of rotation DR of the circular movements

The direction of rotation DR of the circular movements is defined as follows:

Rotation clockwise: DR-  
Rotation counterclockwise: DR+

*Radius compensation*

## Radius compensation

Program the radius compensation in the block that you use to move to the first contour element. You cannot begin programming the radius compensation in a block for a circular path, but program it before that in a straight line block (see "Path movements - rectangular coordinates") or in the approach block (APPR block, see "Approaching and leaving a contour").

### Preliminary positioning

Prior to starting the program, the tool must be preliminary positioned in such a way as to exclude any damage of the tool and the work piece.

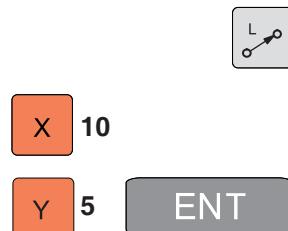
### Creating program blocks via path function keys

The grey path function keys open the plain language dialogue.

WinNC consecutively requests all information and inserts the program block into the machining program.

**Example:** Programming an straight line.

- Open programming dialogue: e. g. straight line
- Enter coordinates of the end point of the straight line



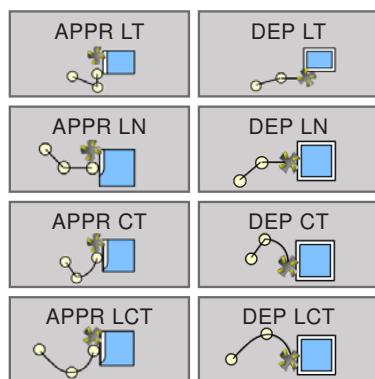
- Select radius compensation: e. g. press softkey `RL`, the tool moves away from the contour on the left side
- Enter feed and confirm with `ENT` button: e. g. 100 mm/min. When programming in INCH mode: Entry of 100 corresponds to a feed of 10 inch/min.
- Move in rapid traverse: Press softkey `F MAX`.
- Enter additional function, e. g. `M3`, and end the dialogue by pressing the key `END`.

Line in the machining program  
**L X+10 Y+5 RL F100 M3**

## Approach and leave contour

### Overview: Path form for approaching and leaving the contour

APPR  
DEP



- The functions APPR (approach) and DEP (departure) are activated via the APPR/DEP key. You can select the following path forms via softkeys:
- Straight line with tangential contact **approach / leave**
- Straight line perpendicular to contour point **approach / leave**
- Circular path with tangential contact **approach / leave**
- **Approach / leave** on a circular path with tangential contact to contour and straight line section

### Polar coordinates

The contour points for the following approach/leave functions can also be programmed via polar coordinates:

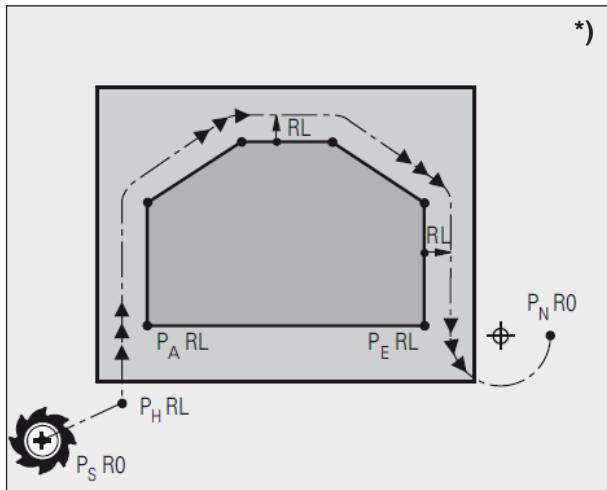
APPR LT changes to APPR PLT  
 APPR LN changes to APPR PLN  
 APPR CT changes to APPR PCT  
 APPR LCT changes to APPR PLCT  
 DEP LCT changes to DEP PLCT

P

Press the P key, after you have selected an approach and/or a departure function via a softkey.

### Approach and leave a helix

When approaching and leaving a helix, the tool moves in an extension of the helix and contacts the contour on a tangential circular path in this way. Use the function APPR CT and/or DEP CT for this.



## Important positions for approach and departure \*)

- Starting point  $P_s$   
You program this position immediately prior to the APPR block.  
 $P_s$  lies outside the contour and is approached without radius compensation ( $R0$ ).
- Auxiliary point  $P_h$   
For some path forms, approaching and departing is performed via an auxiliary point  $P_h$ , which WinNC computes from values in the APPR and DEP block.
- First contour point  $P_a$  and last contour point  $P_e$   
Program the first contour point  $P_a$  in the APPR block; the last contour point  $P_e$  can be programmed with any path function. If the APPR block contains the Z-coordinate, WinNC first moves the tool in the machining plane to point  $P_{h1}$ , and there, in the tool axis, to the specified depth.
- End point  $P_n$   
The position  $P_n$  lies outside the contour and results from your values in the DEP block. If the DEP block contains the Z-coordinate, WinNC first moves the tool in the machining plane to point  $P_{h2}$ , and there, in the tool axis, to the specified height.

When positioning from the actual position to the auxiliary point  $P_h$ , WinNC does not check whether the programmed contour is going to be damaged. You have to check it using the test graphic!

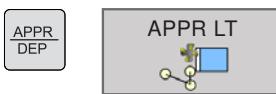
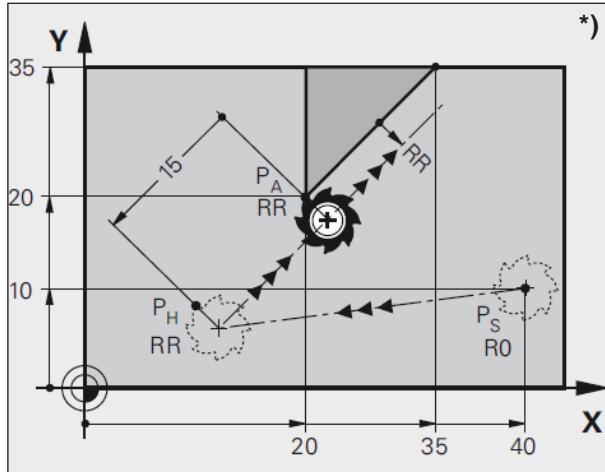
With the functions APPR LT, APPR LN and APPR CT, WinNC moves from the actual position to the auxiliary point  $P_h$  with the last programmed feed/rapid traverse.

With the function APPR LCT, WinNC approaches the auxiliary point  $P_h$  with the feed programmed in the APPR block.

The coordinates can be entered in an absolute or incremental manner as rectangular or polar coordinates.

### Radius compensation

You program the radius compensation along with the first contour point  $P_a$  in the APPR block. The DEP blocks automatically preserve the radius compensation!

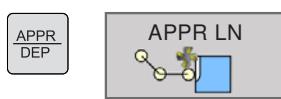
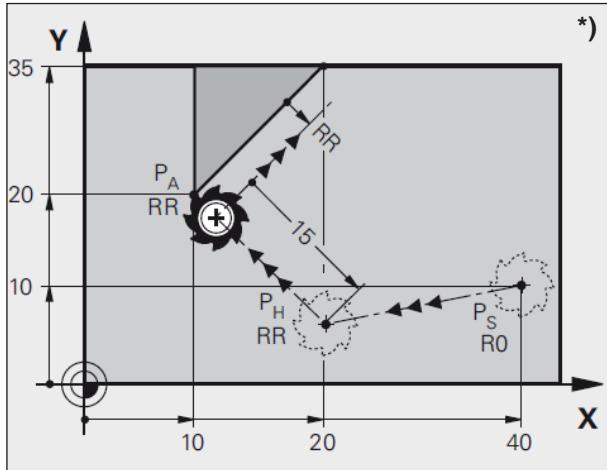


## Approaching on a straight line with tangential connection: APPR LT \*)

WinNC moves the tool on a straight line from starting point  $P_s$  to an auxiliary point  $P_h$ . From there, it approaches the first contour point  $P_a$  on a straight line in a tangential manner. The auxiliary point  $P_h$  is at a distance  $LEN$  to the first contour point  $P_a$ .

- Any path function: Approach starting point  $P_s$
- Open dialogue using the key APPR/DEP and softkey APPR LT:
- Coordinates of the first contour point  $P_a$ .
- $LEN$ : Distance of auxiliary point  $P_h$  to the first contour point  $P_a$ .
- Radius compensation RR/RL for machining.

Example of NC program	Comment
7 L X+40 Y+10 R0 FMAX M3	Approach $P_s$ without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN+15 RR F100	$P_a$ w. radius offs. RR, distance $P_h$ to $P_a$ : $LEN=15$
9 L Y+35 Y+35	End point of first contour element
10 L ...	Next contour element

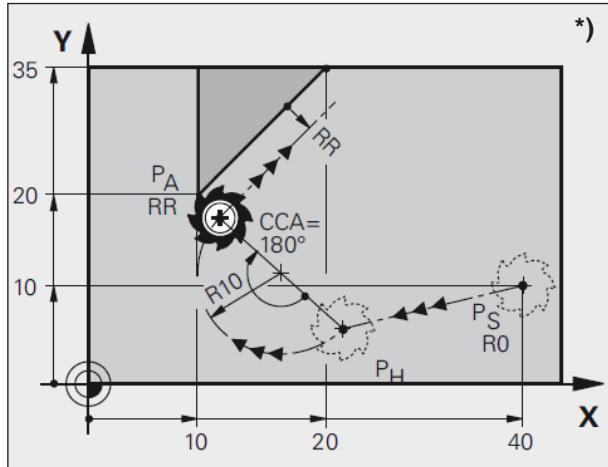


## Approaching on a straight line perpendicular to the first contour point: APPR LN \*)

WinNC moves the tool on a straight line from starting point  $P_s$  to an auxiliary point  $P_h$ . From there, it approaches the first contour point  $P_a$  on a straight line in a perpendicular manner. The auxiliary point  $P_h$  is at a distance  $LEN + \text{tool radius}$  to the first contour point  $P_a$ .

- Any path function: Approach starting point  $P_s$ .
- Open dialogue using the key APPR/DEP and softkey APPR LN:
- Coordinates of the first contour point  $P_a$
- Length: Distance of auxiliary point  $P_h$ .  $LEN$  always enter positive.
- Radius compensation RR/RL for machining.

Example of NC program	Comment
7 L X+40 Y+10 R0 FMAX M3	Approach $P_s$ without radius compensation
8 APPR LN X+10 Y+20 Z-10 LEN+15 RR F100	$P_a$ w. radius offs. RR, distance $P_h$ to $P_a$ : $LEN=15$
9 L X+20 Y+35	End point of first contour element
10 L ...	Next contour element



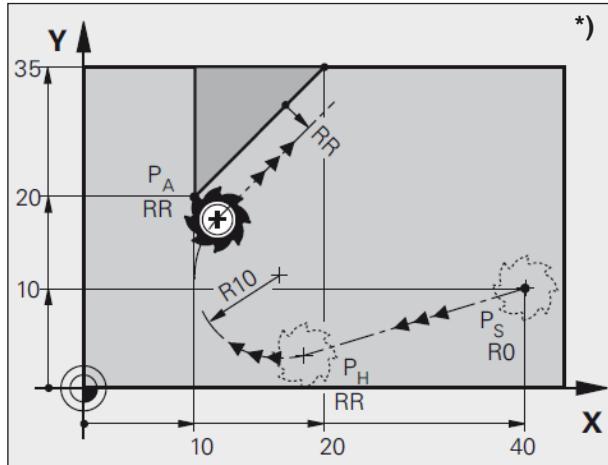
## Approaching on a circular path with tangential connection: APPR CT \*)

WinNC moves the tool on a straight line from starting point  $P_s$  to an auxiliary point  $P_h$ . From there, it approaches the first contour point  $P_a$  on a circular path that tangentially transitions into the first contour element.

The circular path from  $P_h$  to  $P_a$  is defined by the radius  $R$  and the centre angle  $CCA$ . The direction of rotation of the circular path is determined by the course of the first contour element.

- Any path function: Approach starting point  $P_s$ .
- Open dialogue using the key APPR/DEP and softkey APPR CT:
- Coordinates of the first contour point  $P_a$ .
- Radius  $R$  of the circular path
- Approaching the tool from the side, which is defined by the radius compensation: Enter  $R$  positive.
- Approaching from the tool side: Enter  $R$  negative.
- Centre angle  $CCA$  of the circular path
- Enter  $CCA$  only positive
- Maximum input value  $360^\circ$
- Radius compensation RR/RL for machining

Example of NC program	Comment
7 L X+40 Y+10 R0 FMAX M3	Approach $P_s$ without radius compensation
8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100	$P_a$ w. radius offs. RR, radius $R=10$
9 L X+20 Y+35	End point of first contour element
10 L ...	Next contour element



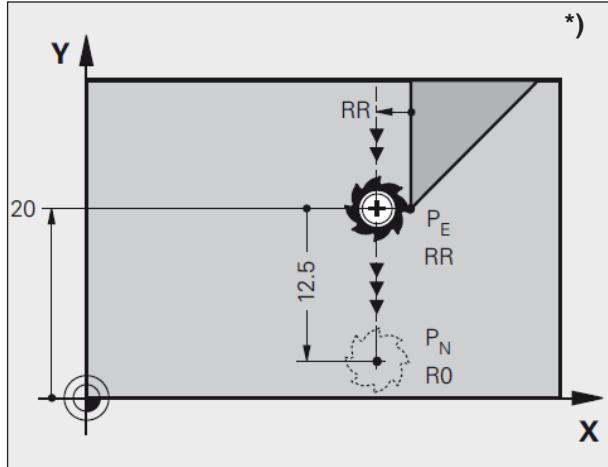
## Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT \*)

WinNC moves the tool on a straight line from starting point  $P_s$  to an auxiliary point  $P_h$ . From there, it approaches the first contour point  $P_a$  on a circular path. The feed programmed in the APPR block is active.

The circular path connects to both, the straight line  $P_s - P_h$  and also to the first contour element, tangentially. Consequently, it is unambiguously determined by the radius  $R$ .

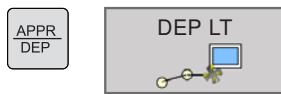
- Any path function: Approach starting point  $P_s$
- Open dialogue using the key APPR/DEP and softkey APPR LCT:
- Coordinates of the first contour point  $P_a$
- Radius  $R$  of the circular path. Specify  $R$  positive
- Radius compensation RR/RL for machining

Example of NC program	Comment
7 L X+40 Y+10 R0 FMAX M3	Approach $P_s$ without radius compensation
8 APPR LCT X+10 Y+20 Z-10 R+10 RR F100	$P_a$ w. radius offs. RR, radius $R=10$
9 L X+20 Y+35	End point of first contour element
10 L ...	Next contour element



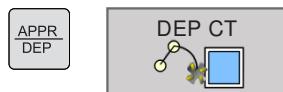
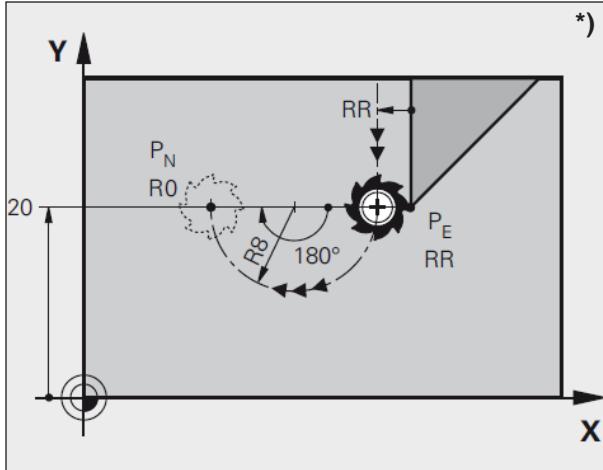
## Departing in a straight line with tangential connection: DEP LT \*)

WinNC moves the tool on a straight line from the last contour point  $P_s$  to the end point  $P_h$ . The straight line lies in the extension of the last contour element..  $P_n$  is positioned in a distance LEN from  $P_e$ .



- Programming the last contour element with the end point  $P_e$  and radius compensation
- Open dialogue using the key APPR/DEP and softkey DEP LT:
- LEN: Enter distance of end point  $P_n$  from last contour element  $P_e$

Example of NC program	Comment
23 L Y+20 RR F100	Last contour element: $P_e$ with radius compensation
24 DEP LT LEN12.5 F100	Move away by LEN=12.5 mm
25 L Z+100 FMAX M2	Clear Z, jump back, end of program

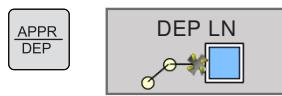
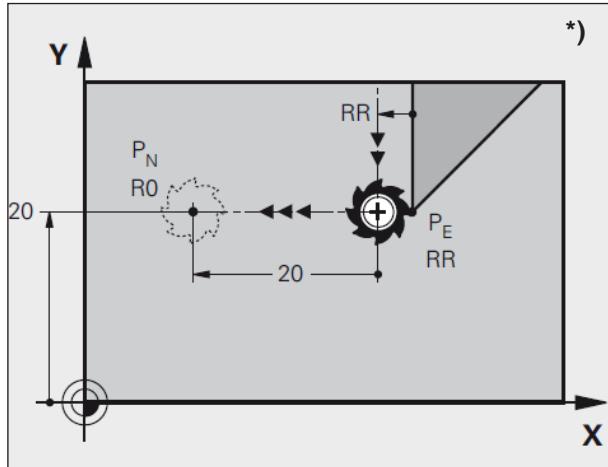


## Departing on a circular path with tangential connection: DEP CT \*)

WinNC moves the tool on a circular path from the last contour point  $P_E$  to the end point  $P_N$ . The circular path tangentially connects to the last contour element..

- Programming the last contour element with the end point  $P_E$  and radius compensation
- Open dialogue using the key APPR/DEP and softkey DEP CT:
- Centre angle CCA of the circular path
- Radius R of the circular path
- The tool is to depart from the work piece to the side, which is determined by the radius compensation: Enter R positive
- The tool is to depart from the work piece on **the opposite side**, which is determined by the radius compensation:  
Enter R negative

Example of NC program	Comment
23 L Y+20 RR F100	Last contour element: $P_E$ with radius compensation
24 DEP CT CCA 180 R+8 F100	Centre angle=180°, circular path radius=8 mm
25 L Z+100 FMAX M2	Clear Z, jump back, end of program

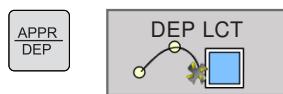
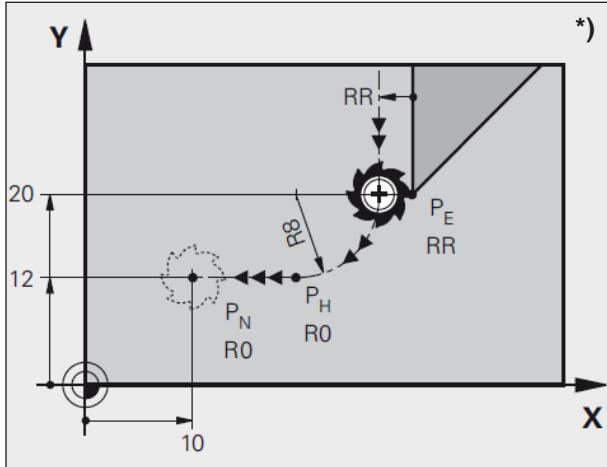


## Departing in a straight line perpendicular to the last contour point: DEP LN \*)

WinNC moves the tool on a straight line from the last contour point  $P_E$  to the end point  $P_N$ . The straight line leads vertically away from the last contour point  $P_E$ .  $P_N$  is positioned in a distance  $LEN + \text{tool radius}$  from  $P_E$ .

- Programming the last contour element with the end point  $P_E$  and radius compensation
- Open dialogue using the key APPR/DEP and softkey DEP LN:
- $LEN$ : Enter distance of the end point  $P_N$   
Important: Enter  $LEN$  positive

Example of NC program	Comment
23 L Y+20 RR F100	Last contour element: $P_E$ with radius compensation
24 DEP LN LEN+20 F100	Move away by $LEN=20$ mm
25 L Z+100 FMAX M2	Clear Z, jump back, end of program



## Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT \*)

WinNC moves the tool on a circular path from the last contour point  $P_E$  to an auxiliary point  $P_H$ . From there, it moves on a straight line to the end point  $P_N$ . The last contour element and the straight line from  $P_H - P_N$  have tangential transitions with the circular path. Consequently, the circular path is unambiguously determined by the radius  $R$ .

- Programming the last contour element with the end point  $P_E$  and radius compensation
- Open dialogue using the key APPR/DEP and softkey DEP LCT:
- Enter the coordinates of the end point  $P_N$
- Radius  $R$  of the circular path. Enter  $R$  positive

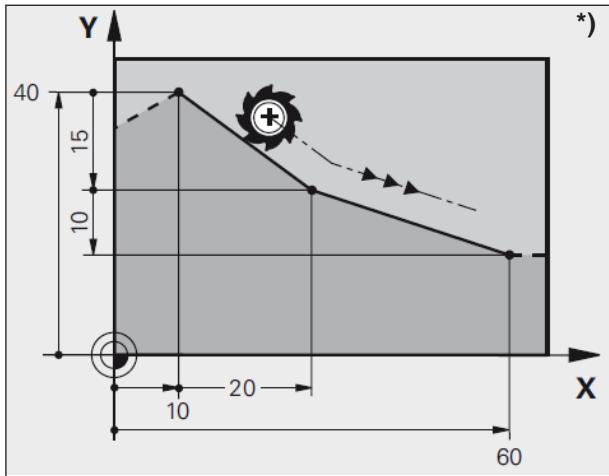
Example of NC program	Comment
23 L Y+20 RR F100	Last contour element: $P_E$ with radius compensation
24 DEP LCT X+10 Y+12 R+8 F10	Coordinates $P_N$ , circular path radius=8 mm
25 L Z+100 FMAX M2	Clear Z, jump back, end of program

# Path movements - rectangular coordinates

## Overview of path functions

Function	Path functions key	Tool movement	Required inputs
Straight line <b>L</b> Line		Straight line	Coordinates of the straight line end point
Chamfer <b>CHF</b> <b>CHamFer</b>		Chamfer between two straight lines	Chamfer length
Circle centre <b>CC</b> <b>Circle Centre</b>		No	coordinates of the circle centre and/or pole
Arc <b>C</b> <b>Circle</b>		Circular path around the circle centre CC to the arc end point	Circle path end point coordinates, direction of rotation
Arc <b>CR</b> <b>Circle by Radius</b>		Circular path with a specific radius	Circle path end point coordinates, circle radius, direction of rotation
Corner rounding <b>RND</b> <b>RouNDing of Corner</b>		Circular path with tangential contact with the preceding and the following element	Corner radius R

## Straight line L



WinNC moves the tool on a straight line from its current position to the end point of the straight line.

Starting point is the end point of the preceding block.

- **Coordinates** of the straight line end point  
If required:
- **Radius compensation RL/RR/R0**
- **Feed F**
- **Additional function M**

### Example

7 L X+10 Y+40 RL F200 M3

8 L IX+20 IY-15

9 L X+60 IY-10

### Adopt current position

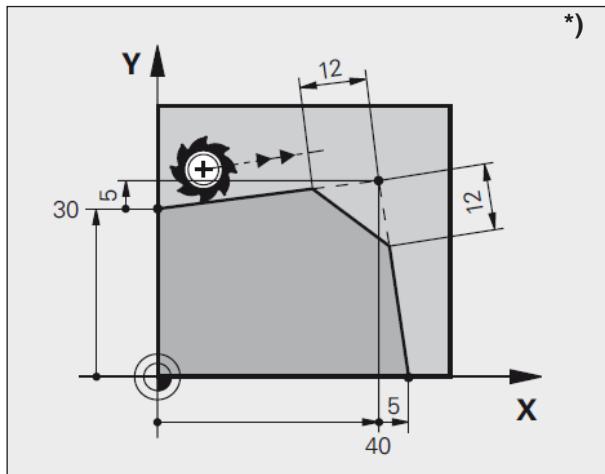
You can generate a straight line block (L-block) via the key "ADOPT CURRENT POSITION":



- Set the operating mode "Manual mode"
- Move the tool to the position to be adopted.
- Change the screen display to "Save/edit program".
- Select the program block, after which the L-block is to be inserted.
- Press the "ADOPT CURRENT POSITION" key: WinNC generates an L-block with the coordinates of the current position.



## Insert a chamfer CHF between two straight lines



The intersection between two straight lines creates contour edges, which edges can be provided with a chamfer.

- In each straight line block prior to and after the CHF-block, program both coordinates of the plane, in which the chamfer is to be executed.
- The radius compensation prior to and after the CHF-block must be the same.
- The chamfer must be executable with the current tool.
- Chamfer section: Length of the chamfer, if required:
- Feed F (only effective in CHF-block).

### Example

7 L X+0 Y+30 RL F300 M3

8 L X+40 IY+5

9 CHF 12 F250

10 L IX+5 Y+0

### Information:

A contour cannot start with a CHF-block.

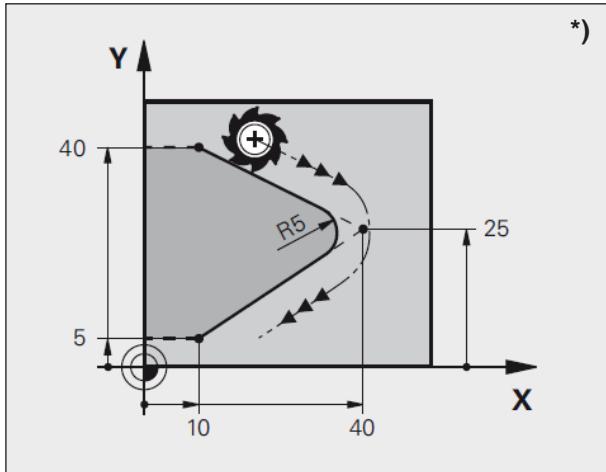
Execute the chamfer only in the machining plane.

Do not approach the point that was cut off by the chamfer.

The feed programmed in the CHF-block is only effective in this CHF-block. After that, the feed programmed prior to the CHF-block is effective again.



## Corner roundings RND



Contour corners are rounded with the RND function.

In doing so, the tool moves along a circular path, which is tangentially connected to both, the preceding as well as the following contour element.

The called tool must be able to execute the rounding circle.

- **Rounding radius:** Radius of the arc.  
If required:
- **Feed F** (only effective in RND-block).

### Example

```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5
```

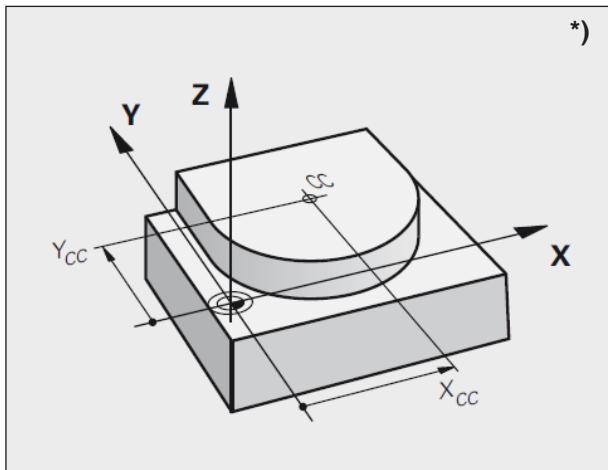
### Note:

The preceding and following contour element must contain both coordinates of the plane, in which the rounding of the corner is executed. If you are machining the contour without tool radius compensation, you have to program both coordinates of the machining plane. The corner point will not be approached.

The feed programmed in the RND-block is only effective in this RND-block. After that, the feed programmed prior to the RND-block is effective again.

An RND-block can also be utilised for soft approach to the contour, if the APPR functions are not to be used.

## Circle centre CC \*)



You define the circle centre for the circle paths that you program using the C key (circular path C). To that end,

- enter the rectangular coordinates of the circle centre, or
- adopt the last programmed position, or
- adopt the coordinates with the key "ADOPT CURRENT POSITIONS"
- **coordinates CC:**  
Enter the coordinates for the circle centre, or  
In order to adopt the last programmed position:  
Do not enter any coordinates.

### Validity

The circle centre remains defined until you program a new circle center.

### Entering the circle centre CC incrementally

An incrementally entered coordinate for the circle centre always refers to the last programmed tool position.

#### Note:

With CC, you mark a position as a circle centre:  
The tool does not move to this position.  
The circle centre is simultaneously the pole for polar coordinates.



### Example

5 CC X+25 Y+25

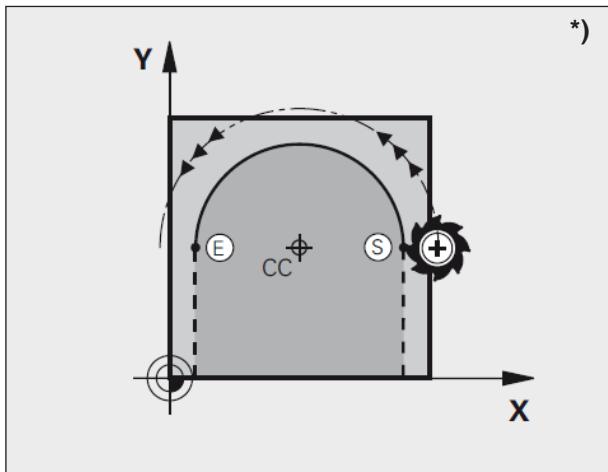
or

10 L X+25 Y+25

11 CC

The program lines 10 and 11 do not refer to the picture.

## Circular path C around circle centre CC \*)



Define the circle centre CC, before you program the circular path C.

The last programmed tool position prior to the C-block is the starting point of the circular path.

- Move the tool to the starting point of the circular path
- **Coordinates** of the circle centre
- **Coordinates** of the arc end point
- **Direction of rotation DR**

If required:

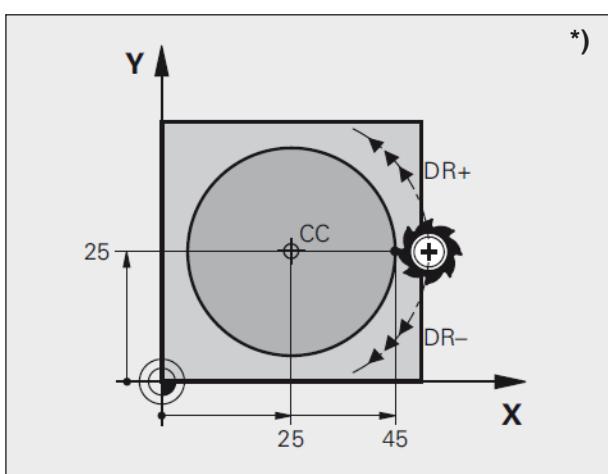
- **Feed F**
- **Additional function M**

### Full circle

Program the same coordinates for the end point as for the starting point.

### Example

```
5 CC X+25 Y+25
6 L X+45 Y+25 RR F200 M3
7 C X+45 Y+25 DR+
```



### Note:

Start and end point of the circular movement must be positioned on the circular path.

The input tolerance can be set in EMConfig.  
Standard tolerance: 3µm



## Circular path with a defined radius



The tool moves along a circular path with the radius R.

- **Coordinates** of the arc end point
- **Radius R**

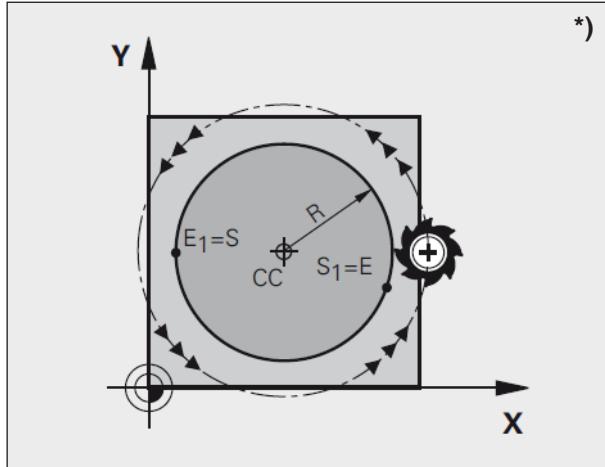
Important: The sign determines the dimension of the arc!

- **Direction of rotation DR**

Important: The sign determines the concave or convex curvature!

If required:

- **Additional function M**
- **Feed F**

**Full circle**

For a full circle, you have to program two CR-blocks one after the other:

The end point of the first semicircle is the starting point of the second. The end point of the second semicircle is the starting point of the first.

**Centre angle CCA and arc radius R**

Starting point and end point of the contour can be connected with each other via four different arcs with the same radius:

Smaller arc:  $CCA < 180^\circ$

Radius has a positive sign  $R > 0$

Larger arc:  $CCA > 180^\circ$

Radius has a negative sign  $R < 0$

Via the direction of rotation you determine, whether the arc is curved towards the outside (convex) or towards the inside (concave):

Convex: Direction of rotation DR- (with radius compensation RL)

Concave: Direction of rotation DR+ (with radius compensation RL)

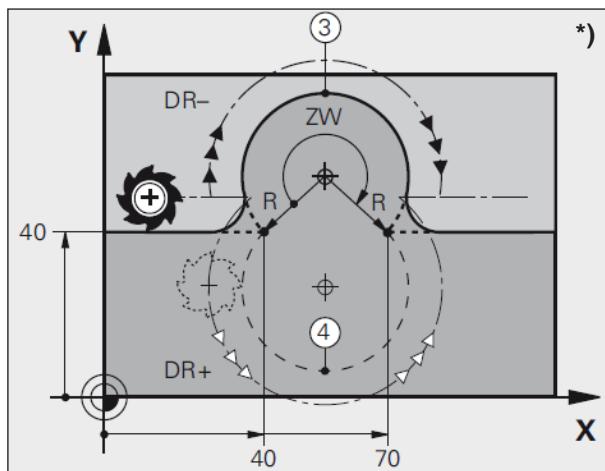
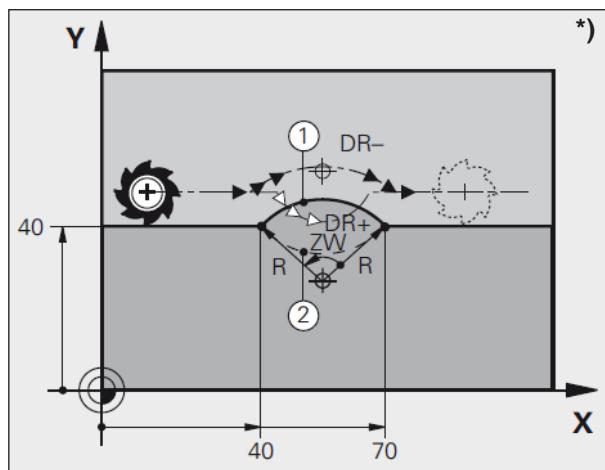
**Example**

10 L X+40 Y+40 RL F200 M3

11 CR X+70 Y+40 R+20 DR- (arc 1)

or

11 CR X+70 Y+40 R+20 DR+ (arc 2)

**Example**

11 CR X+70 Y+40 R-20 DR- (arc 3)

or

11 CR X+70 Y+40 R-20 DR+ (arc 4)

**Note:**

The distance between the starting point and the end point of the circle diameter must not exceed the circle diameter.

The maximum radius is 99.9999 m.

## Path contours - Polar coordinates

\*)

### Overview

With the polar coordinates, you define a position via an angle PA and a distance PR to a previously defined pole CC (see "Free contour programming FK").

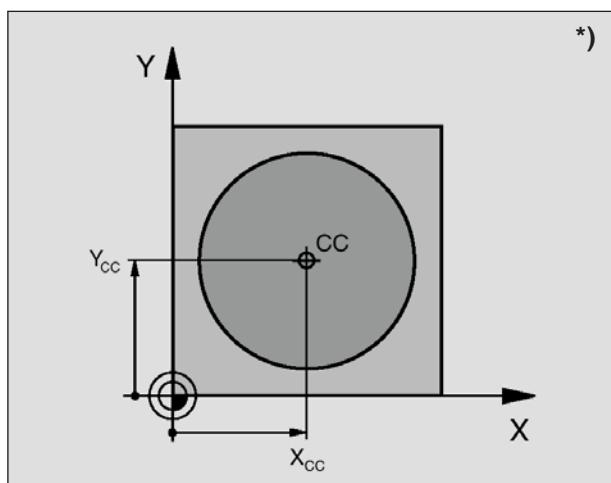
You can use polar coordinates advantageously with:

- Positions on arcs
- Work piece drawings with angle specifications, e. g. in case of hole circles

### Overview of the path function with polar coordinates

Function	Path function key	Tool movement	Required inputs
Straight line LP	 + 	Straight line	Polar radius, polar angle of the straight line end point
Arc CP	 + 	Circular path around the circle centre pole CC to the arc end point	Polar angle of the circle end point, direction of rotation
Helix	 + 	Overlap of a circular path with a straight line	Polar radius, polar angle of the circle end point in the tool axis

### Polar coordinates origin: Pole CC

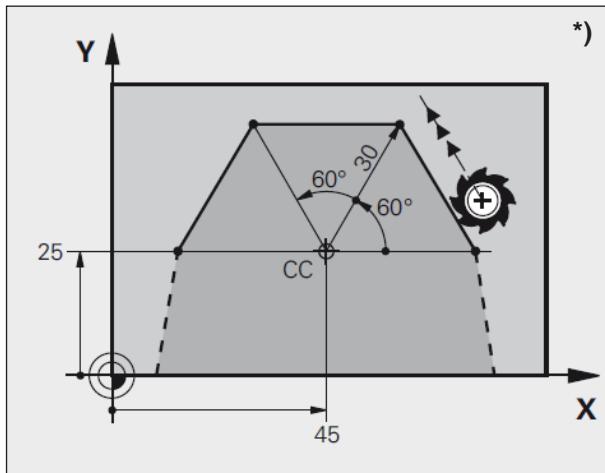


You can define the pole CC at random positions in the machining program, before you specify positions via polar coordinates. When defining the pole, proceed as you would when programming the circle centre CC.

- **Coordinates CC:** Enter rectangular coordinate for the pole. The pole CC is valid until you define a new pole CC.

**Example**  
12 CC X+45 Y+25

## Straight line LP



The tool moves on a straight line from its current position to the end point of the straight line. The starting point is the end point of the preceding block.

- Polar coordinates radius PR:** Enter the distance from the straight line end point to the pole CC
- Polar coordinates angle PA:** Angle position of the straight line end point between  $-360^\circ$  and  $+360^\circ$

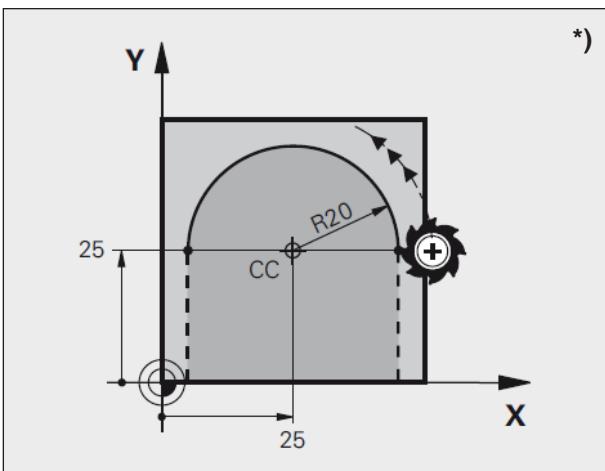
The sign of PA is determined by the angle reference axis:

- Angle of the angle reference axis to PR counterclockwise: PA>0
- Angle of the angle reference axis to PR clockwise: PA<0

### Example

```
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F300 M3
14 LP PA+60
15 LP IPA+60
16 LP PA+180
```

## Circular path CP around pole CC



The polar coordinates radius PR is simultaneously the radius of the arc. PR is determined by the distance from the starting point to the pole CC. The last programmed tool position prior to the CP-block is the starting point of the circular path.

- Polar coordinates angle PA:** Angle position of the circular path end point between  $-5400^\circ$  and  $+5400^\circ$
- Direction of rotation DR**

### Example

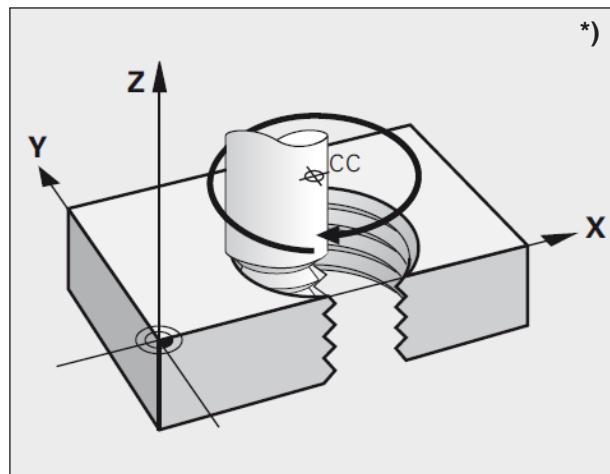
```
18 CC X+25 Y+25
19 LP PR+20 PA+0 RR F250 M3
20 CP PA+180 DR+
```

### Note:

Be sure to enter the same sign for DR and PA when using incremental coordinates.



## Helix



A helix results from an overlap of a circular movement and a straight line movement that runs perpendicular to it. You program the circular path in a main plane.

The path movements for the helix can only be programmed in polar coordinates.

### Application

- Internal and external threads with bigger diameters
- Lubrication grooves

### Calculating the helix

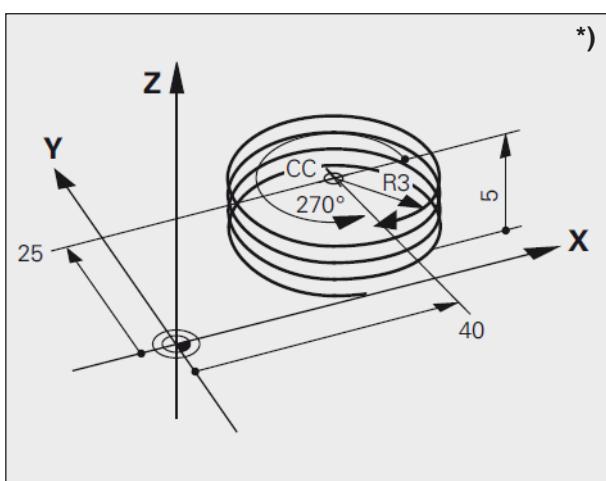
For programming, you need the incremental specification of the total angle, with which the tool travels on the helix, and the total height of the helix.

For calculation in milling direction from bottom to top the following applies:

### Form of the helix

The table shows the relation between working direction, direction of rotation and radius compensation for specific path forms.

Internal thread	Working direction	Direction of rotation	Radius compensation
right-handed	Z+	DR+	RL
left-handed	Z+	DR-	RR
right-handed	Z-	DR-	RR
left-handed	Z-	DR+	RL
External thread	Working direction	Direction of rotation	Radius compensation
right-handed	Z+	DR+	RR
left-handed	Z+	DR-	RL
right-handed	Z-	DR-	RL
left-handed	Z-	DR+	RR



### Programming a helix

- **Polar coordinates angle:** Incrementally enter the total angle with that the tool travels on the helix. After entering the angle, select the tool axis using the axis selection key.
- Incrementally enter the **coordinate** for the height of the helix
- **Direction of rotation DR**  
helix clockwise: DR-  
Helix counterclockwise: DR+
- **Radius compensation RL/RR/R0**  
Enter the radius compensation according to the table

#### Example

```

12 CC X+40 Y+25
13 L Z+0 F100 M3
14 LP PR+3 PA+270 RL F50
15 CP IPA-1800 IZ+5 DR-

```

#### Note:

Enter the direction of rotation DR and the incremental total angle IPA with the same sign, as the tool can possibly travel in a wrong path. For the total angle IPA, you can enter a value between  $-5400^\circ$  and  $+5400^\circ$ . If the thread has more than 15 convolutions, you should program the helix in a repetitions of a program part.

## Cycle overview

Here, the cycle groups with the defined cycles of EMCO WinNC for Heidenhain TNC 640 are listed.

### DRILLING / THREAD

#### Drilling / thread

- 200 Drilling
- 201 Grinding
- 202 Turning out
- 203 Universal drilling
- 205 Universal deep hole drilling
- 208 Drill milling
- 206 Thread tapping new
- 207 Rigid tapping
- 209 Thread tapping chips crushing
- 240 Centring
- 262 Thread milling
- 263 Thread milling/countersinking
- 264 Thread drilling/milling
- 265 Helix-thread tapping/milling
- 267 Outer thread tapping/milling

### POCKETS / STUDS / SLOTS

#### Pockets / studs / slots

- For a description of the cycles see Old Cycles

### COORD. TRANSF.

#### Converting the coordinates

- 7 Zero offset
- 8 Mirroring
- 10 Rotation

### SL CYCLES

#### SL Cycles

- 14 Contour
- 20 Contour data
- 22 Clearing
- 23 Finishing depth
- 24 Finishing side
- 25 Contour pull

### PATTERN

#### Dot patterns

- 220 Pattern on circle
- 221 Pattern on line



**Special cycles**

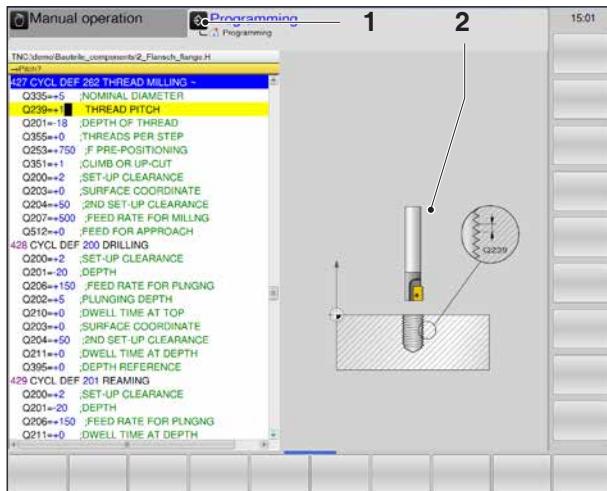
SPECIAL CYCLES

- 9 Dwell time
- 12 PGM CALL

**Old Cycles**

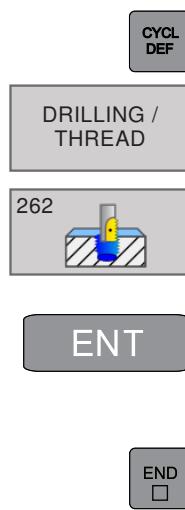
OLD CYCLS

- 1 Deep hole drilling
- 2 Thread tapping
- 17 Rigid tapping
- 3 Slot milling
- 4 Pocket milling
- 5 Circular pocket
- 212 Finishing pocket
- 213 Finishing stud
- 214 Finishing circular pocket
- 215 Finishing circular stud
- 210 Slot reciprocal
- 211 Round slot
- 230 Multipass milling
- 231 Ruled surface



## Working with cycles

Often used and continuously repeated machining operations, which comprise multiple machining steps, the control unit saves as cycles. Some special functions also are available as cycles. Machining cycles with numbers from 200 upwards use Q parameters as transfer parameters. Parameter with the same number, which WinNC uses in different cycles, always have the same number: Q200 is always the safety distance, Q202 is always the infeed depth etc.



## Defining a cycle via softkeys

- Retrieve the softkey bar for the different cycle groups.
- Select the cycle group: DRILLING/THREAD
- Select cycle: 262 THREAD MILLING. WinNC opens a dialogue with the query for all input values from (1). On the right side of the screen, the control unit displays a diagram (2), where the parameters that need to be entered are highlighted.
- Enter all parameters required by WinNC and confirm every entry by pressing the ENT key.
- After you have entered all required data, WinNC ends the dialogue.
- Press END, if you want to end the entry prematurely.

### Note:

For machining cycles with numbers from 200 upward, which use indirect parameter assignment (e.g. Q210 = Q1), any change of the assigned parameter (e.g. Q1) after the cycle definition will not be effective. In such cases, the cycle parameter (e.g. Q210) must be defined directly.

## Example

### 21 CYCL DEF 262 THREAD MILLING

```

Q335=5 ;NOMINAL-DIAMETER
Q239=1 ;THREAD PITCH
Q201=-18 ;DEPTH OF THREAD
Q235=+0 ;THREADS PER STEP
Q351=+1 ;CLIMB OR UP-CUT
Q200=2 ;SET-UP CLEARANCE
Q203=+0 ;SURFACE COORDINATE
Q204=+50 ;SAFETY DISTANCE
Q207=+500 FEED RATE FOR MILLING
Q512=AUTO ;FEED FOR APPROACH

```

## Call up cycle

**Note:**

Prior to a cycle call-up, program in every case:

- **BLK FORM** for graphic display (only required for test graphic)
- Tool call
- Direction of rotation of spindle (additional function M3/M4)
- Cycle definition (CYCL DEF).

Observe further requirements that are specified in the following cycle descriptions.

The following cycles are effective from the point of their definition in the machining program. You cannot and must not call up these cycles:

- the cycles 220 dot patterns on circle and 221 dot patterns on lines
- the SL cycle 14 CONTOUR
- the SL cycle 20 CONTOUR DATA
- cycles for converting coordinates
- the cycle 9 DWELL TIME

You can call up all other cycles as described below:

If you want WinNC to execute the cycle after the last programmed block once, you have to program the cycle call via the additional function M99 or via CYCL CALL:



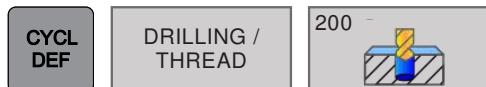
- Programming a cycle call: Press CYCL CALL key
- Entering a cycle call: Press softkey CYCL CALL M
- Enter additional function M, or end dialogue by pressing the END key



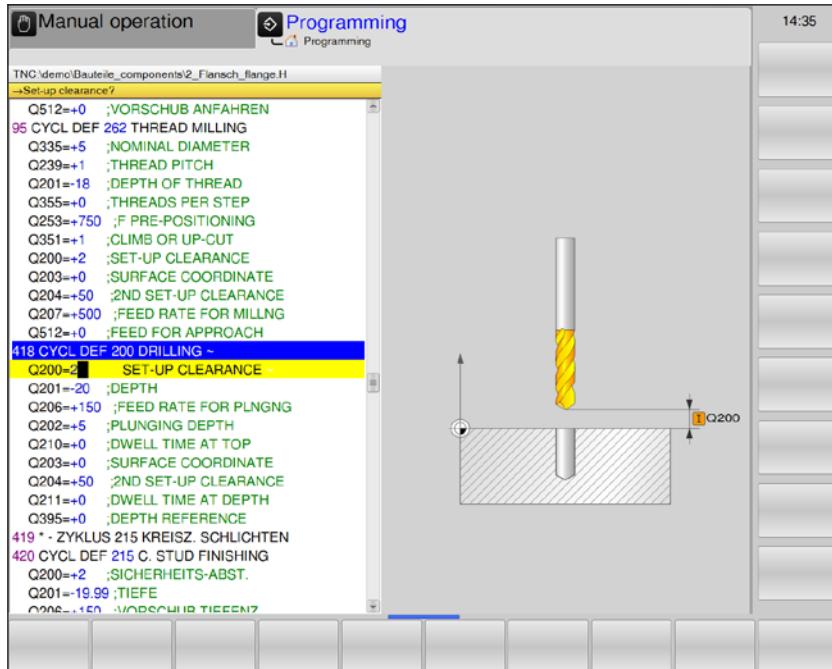
## Drilling / thread

### DRILLING / THREAD

- 200 Drilling
- 201 Grinding
- 202 Turning out
- 203 Universal drilling
- 205 Universal deep hole drilling
- 208 Drill milling
- 206 Thread tapping new
- 207 Rigid tapping
- 209 Thread tapping chips crushing
- 240 Centring
- 262 Thread milling
- 263 Thread milling/countersinking
- 264 Thread drilling/milling
- 265 Helix-thread tapping/milling
- 267 Outer thread tapping/milling



## DRILLING (cycle 200)



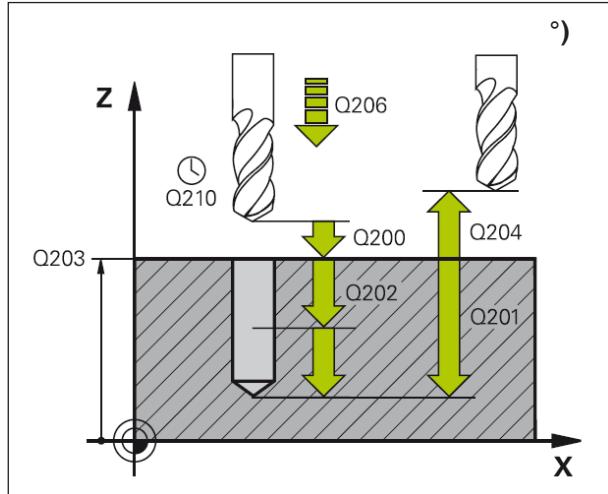
Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Depth (incremental): Distance work piece surface - hole bottom (drill taper tip)	mm
Q206	Feed for depth infeed: Tool traverse speed during drilling.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in. The depth does not have to be a multiple of the infeed depth. WinNC moves in one work step to the depth, if: <ul style="list-style-type: none"><li>• infeed depth and depth are identical</li><li>• infeed depth is bigger than depth</li></ul>	mm
Q210	Dwell time above: Dwell time in seconds, during which the tool remains at the safety distance, after WinNC has moved it out of the hole for chip clearing	s
Q203	Coordinates of the work piece surface: (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm

Parameter	Description	Unit
Q211	Dwell time below: Time in seconds, during which the tool remains at the hole bottom.	s
Q395	Depth relation: Selection, whether depth relates to the tool point or to the cylindrical part of the tool. <ul style="list-style-type: none"> <li>• 0 = entered depth related to the tool point</li> <li>• 1= entered depth related to the cylindrical part of the tool.</li> </ul> The point angle must be defined in the column T-ANGLE of the tool table TOOL.T.	

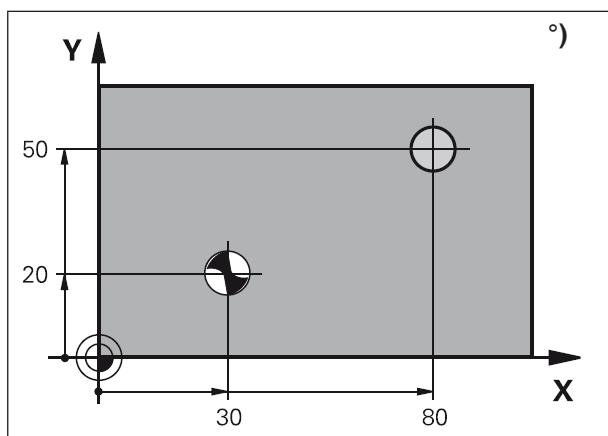
**Note:**

Currently, Q395 cannot be changed, you can only work with the default value.

If external programs are loaded, which work according to other values for Q395, WinNC will automatically switch to the default value.

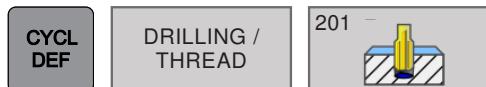
**Cycle description**

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool drills at the programmed feed F to the first infeed depth.
- 3 WinNC retracts the tool at FMAX back to the safety distance, dwells there, if entered, and subsequently moves at FMAX to the safety distance above the first infeed depth.
- 4 Subsequently, the tool drills at the programmed feed F by a further infeed depth.
- 5 WinNC repeats the sequence (2 to 4), until the entered drilling depth is reached.
- 6 From the hole bottom, the tool moves at FMAX to the safety distance, or, if entered, to the 2nd. safety distance.

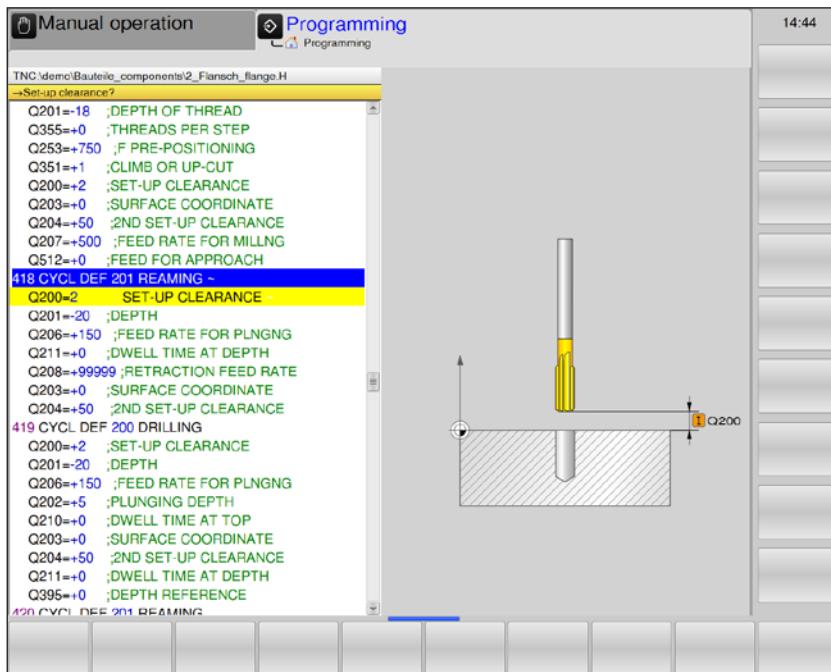
**Note:**

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.

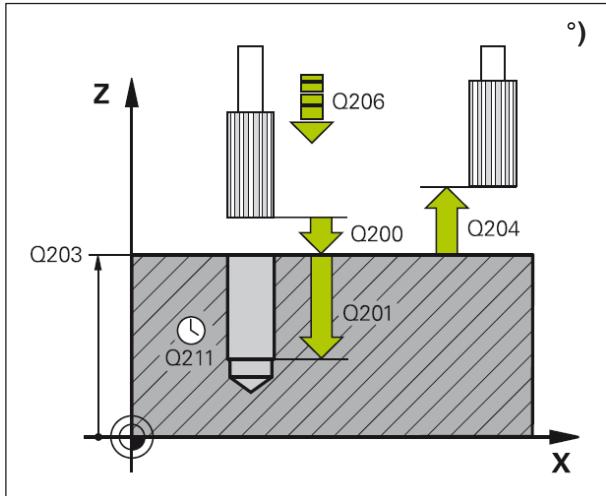




## GRINDING (cycle 201)

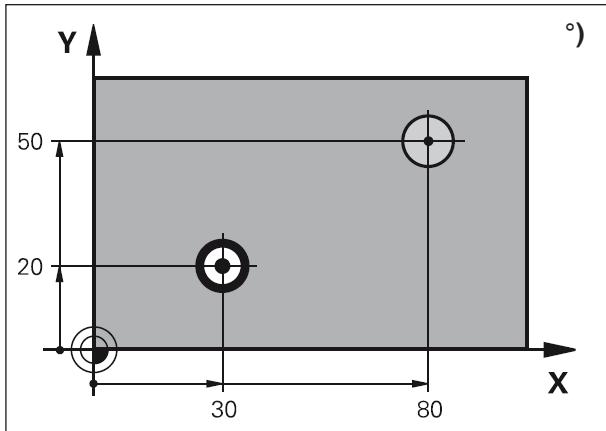


Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Depth (incremental): Distance work piece surface - hole bottom	mm
Q206	Feed for depth infeed: Tool traverse speed during grinding.	mm/min
Q211	Dwell time below: Time in seconds, during which the tool remains at the hole bottom.	s
Q208	Retraction feed: Traverse speed of the tool during retraction from the hole. If 0 is entered, the feed for grinding applies.	mm/min
Q203	Coordinates of the work piece surface: (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm



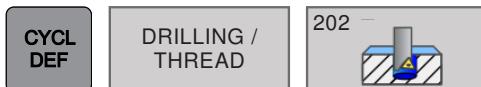
### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool grinds at the programmed feed F to the programmed depth.
- 3 At the hole bottom (Q201), the tool dwells, if entered.
- 4 Subsequently, WinNC moves the tool back at feed F to the safety distance and from there, if entered, with FMAX to the 2nd safety distance.

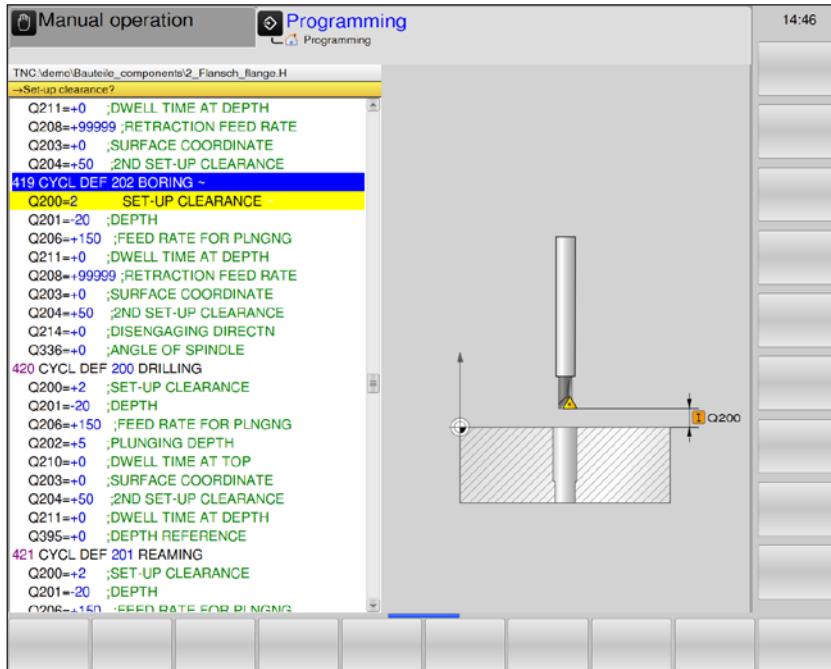


### Note:

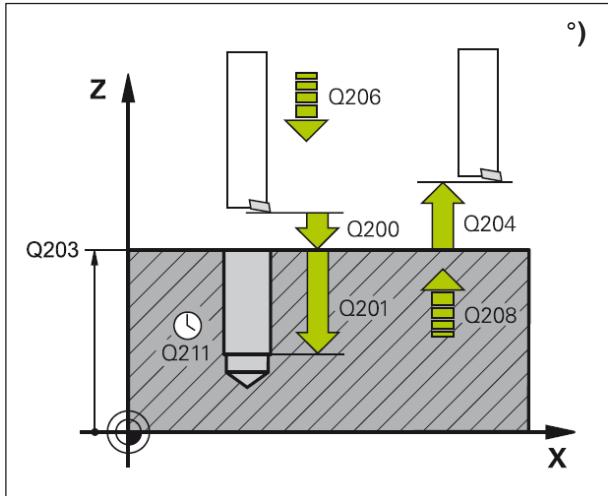
Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.



## TURNING OUT (cycle 202)

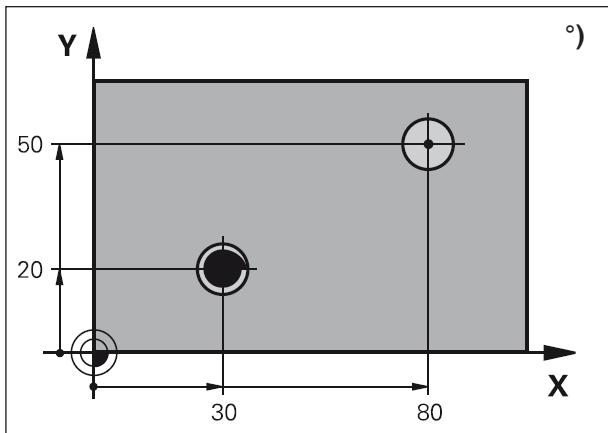


Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Depth (incremental): Distance work piece surface - hole bottom	mm
Q206	Feed for depth infeed: Tool traverse speed during turning out.	mm/min
Q211	Dwell time below: Time in seconds, during which the tool remains at the hole bottom.	s
Q208	Retraction feed: Traverse speed of the tool during retraction from the hole. If 0 is entered, the feed for grinding applies.	mm/min
Q203	Coordinates of the work piece surface: (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q214	Clearing direction: Determine the direction, in which WinNC will clear the tool at the hole bottom (according to the spindle orientation) <ul style="list-style-type: none"> <li>• 0 = Do not clear tool</li> <li>• 1 = Clear tool in minus direction of the main axis</li> <li>• 2 = Clear tool in minus direction of the minor axis</li> <li>• 3 = Clear tool in plus direction of the main axis</li> <li>• 4 = Clear tool in plus direction of the minor axis</li> </ul>	
Q336	Angle for the spindle orientation (absolute): Angle, at which WinNC will position the tool prior to clearing. Input: -360.000 to 360.000	degrees



### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool drills at the drilling feed F to the depth.
- 3 At the hole bottom (Q201), the tool dwells, if entered, with the spindle running to cut free.
- 4 Subsequently, WinNC performs a spindle orientation to the 0° position.
- 5 If clearing has been selected, WinNC clears the tool in the entered direction for 0.2 mm (fixed value).
- 6 Subsequently, WinNC moves the tool back at retraction feed to the safety distance and from there, if entered, with FMAX to the 2nd safety distance. If Q214=0, retraction occurs at the hole wall.



#### Danger of collision:

Select the clearing direction such that the tool moves away from the hole wall. Check, where the tool point is located, when programming a spindle orientation to the angle that you enter in Q336 (e. g. via manual input in the positioning operating mode). Select the angle such that the tool point is positioned parallel to a coordinate axis.



#### Note:

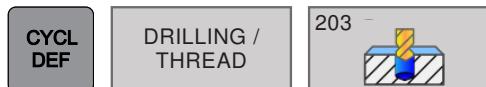
Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.

A negative sign means:

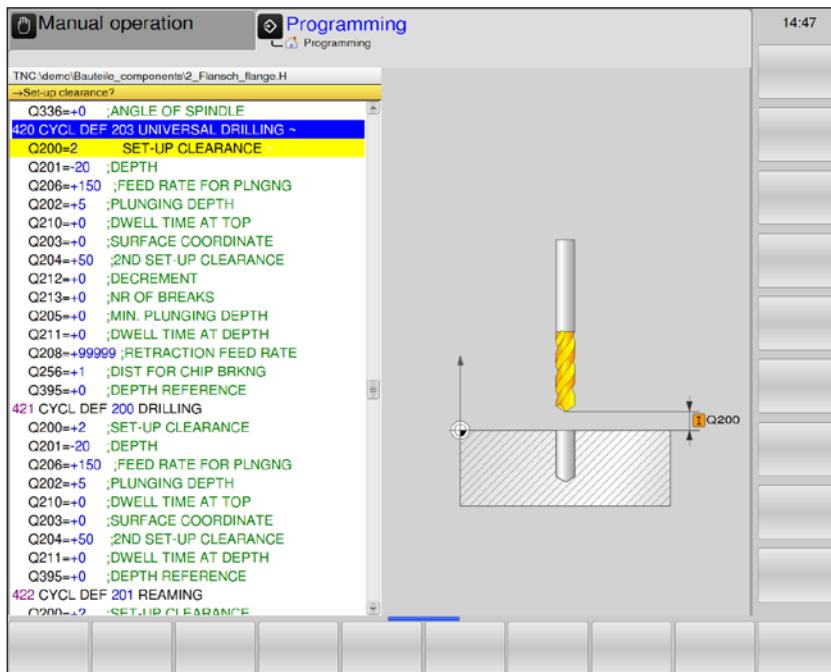
Machining in the direction of the negative spindle axis.

If depth = 0 is programmed, the cycle will not be executed.





## UNIVERSAL DRILLING (cycle 203)



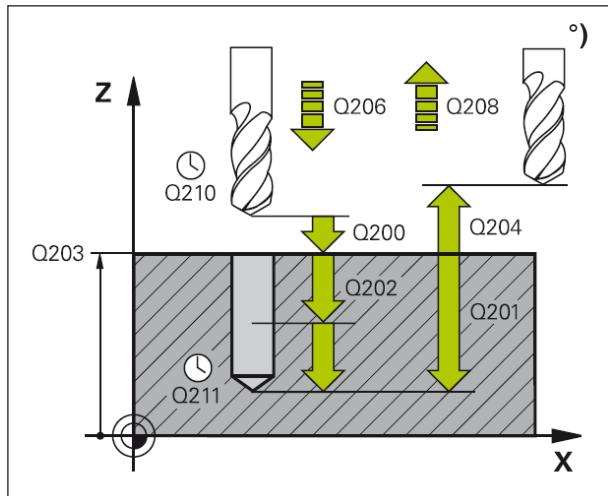
Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Depth (incremental): Distance work piece surface - hole bottom	mm
Q206	Feed for depth infeed: Tool traverse speed during drilling.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in. The depth does not have to be a multiple of the infeed depth. WinNC moves in one work step to the depth, if: <ul style="list-style-type: none"><li>• infeed depth and depth are identical</li><li>• infeed depth is bigger than depth</li></ul>	mm
Q210	Dwell time above: Dwell time in seconds, during which the tool remains at the safety distance, after WinNC has moved it out of the hole for chip clearing.	s
Q203	Coordinates of the work piece surface: (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q212	Reduction value (incremental): Value by which WinNC reduces the infeed depth Q202 after every infeed.	mm
Q213	Number of chips crushing operations before retraction: Number of chips crushing operations, before WinNC is to retract the tool out of the hole for chip clearing. For chips crushing, WinNC retracts the tool each time by the retraction value Q256.	
Q205	Minimum infeed depth (incremental): If a reduction value has been entered, WinNC limits the infeed to the value entered with Q205	mm

Parameter	Description	Unit
Q211	Dwell time below: Time in seconds, during which the tool remains at the hole bottom.	s
Q208	Retraction feed: Traverse speed of the tool during retraction from the hole in mm/min. If 0 is entered, WinNC will retract the tool at feed Q206	mm/min
Q256	Retraction at chips crushing (incremental): Value, by which WinNC retracts the tool during chips crushing.	mm
Q395	Depth relation: Selection, whether depth relates to the tool point or to the cylindrical part of the tool. • 0 = entered depth related to the tool point • 1= entered depth related to the cylindrical part of the tool. The point angle must be defined in the column T-ANGLE of the tool table TOOL.T.	

**Note:**

Currently, Q395 cannot be changed, you can only work with the default value.

If external programs are loaded, which work according to other values for Q395, WinNC will automatically switch to the default value.

**Cycle description**

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool drills at the drilling feed F to the first infeed depth.
- 3 If chips crushing has been entered, WinNC retracts the tool by the entered retraction value. If working without chips crushing, WinNC retracts the tool at retraction feed back to the safety distance, dwells there, if entered, and subsequently moves at FMAX to the safety distance above the first infeed depth.
- 4 Subsequently, the tool drills at the feed by a further infeed depth. With every infeed, the infeed depth is reduced by the reduction value, if entered,, however, at least by the minimum infeed depth.
- 5 WinNC repeats this sequence (2 to 4), until the drilling depth is reached.
- 6 At the hole bottom, the tool dwells, if entered, to cut free, and is subsequently retracted at retraction feed to the safety distance after the dwell time. If a 2nd safety distance has been entered, WinNC moves the tool at FMAX there.

**Note:**

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:

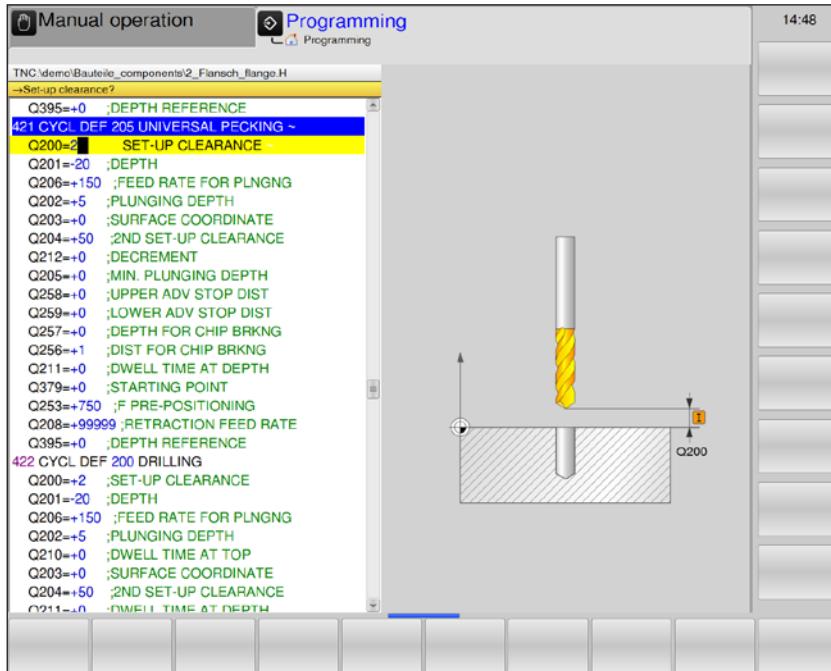
Machining in the direction of the negative spindle axis.

If depth = 0 is programmed, the cycle will not be executed.





## UNIVERSAL DEEP HOLE DRILLING (cycle 205)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Depth (incremental): Distance work piece surface - hole bottom	mm
Q206	Feed for depth infeed: Tool traverse speed during drilling.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in. The depth does not have to be a multiple of the infeed depth. WinNC moves in one work step to the depth, if: <ul style="list-style-type: none"><li>• infeed depth and depth are identical</li><li>• infeed depth is bigger than depth</li></ul>	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q212	Reduction value (incremental): Value by which WinNC reduces the infeed depth Q202 after every infeed.	mm
Q205	Minimum infeed depth (incremental): If a reduction value has been entered, WinNC limits the infeed to the value entered with Q205	mm
Q258	Advance stop distance (incremental): Safety distance for rapid traverse positioning, when WinNC moves the tool after a retraction from the hole back to the current infeed depth; value at first infeed.	mm

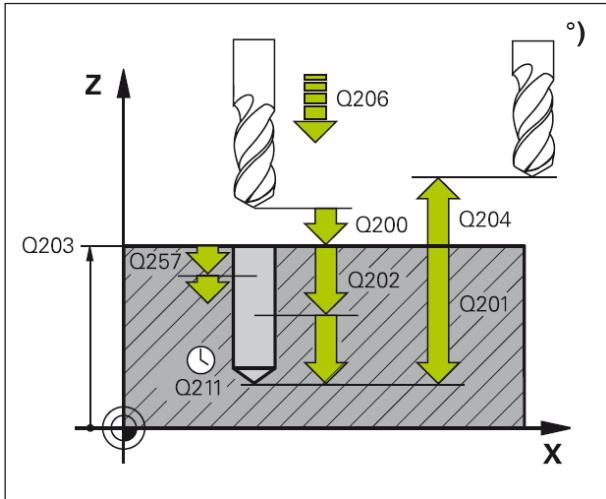
Parameter	Description	Unit
Q259	Advance stop distance below (incremental): Safety distance for rapid traverse positioning, when WinNC moves the tool after a retraction from the hole back to the current infeed depth; value at last infeed.	mm
Q257	Drilling depth until chips crushing (incremental): Infeed, after which WinNC executes a chips crushing operation. No chips crushing at input 0.	mm
Q256	Retraction at chips crushing (incremental): Value by which WinNC retracts the tool during chips crushing.	mm
Q211	Dwell time below: Time in seconds, during which the tool remains at the hole bottom.	s
Q379	Retraction feed (incremental, related to the work piece surface): Starting point of the actual drilling operation.	mm/min
Q253	Preliminary positioning feed: Defines the traverse speed of the tool during re-approaching the drilling depth after chips crushing retraction. This feed also applies if the tool is positioned on a deepened starting point.	mm/min
Q208	Retraction feed: Traverse speed of the tool during retraction after machining. If 0 is entered, the feed for grinding applies.	mm/min
Q395	Depth relation: Selection, whether depth relates to the tool point or to the cylindrical part of the tool. <ul style="list-style-type: none"> <li>• 0 = entered depth related to the tool point</li> <li>• 1= entered depth related to the cylindrical part of the tool.</li> </ul> The point angle must be defined in the column T-ANGLE of the tool table TOOL.T.	

**Note:**

Currently, Q379, Q253, Q208 and Q395 cannot be changed, you can only work with the default values. If external programs are loaded, which work according to other values for these parameters, WinNC will automatically switch to the default values.







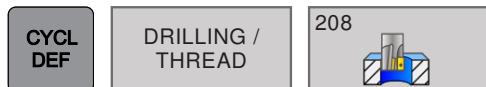
### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 If a deepened starting point has been entered, WinNC travels at the defined positioning feed to the safety distance above the deepened starting point.
- 3 The tool drills at the entered feed F to the first infeed depth.
- 4 If chips crushing has been entered, WinNC retracts the tool by the entered retraction value. If you work without chips crushing, WinNC retracts the tool in rapid traverse to the safety distance, and subsequently again at FMAX to the entered advance stop distance above the first infeed depth.
- 5 Subsequently, the tool drills at the feed by a further infeed depth. With every infeed, the infeed depth is reduced by the reduction value - if entered.
- 6 WinNC repeats this sequence (2 to 4), until the drilling depth is reached.
- 7 At the hole bottom, the tool dwells, if entered, to cut free, and is subsequently retracted at retraction feed to the safety distance after the dwell time. If you entered a 2nd safety distance, WinNC moves the tool at FMAX there.

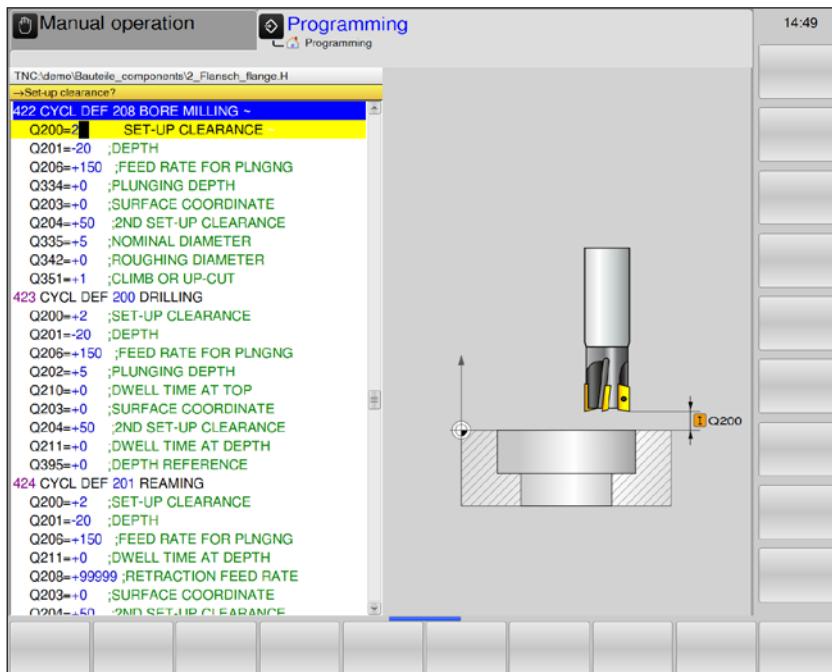
#### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.

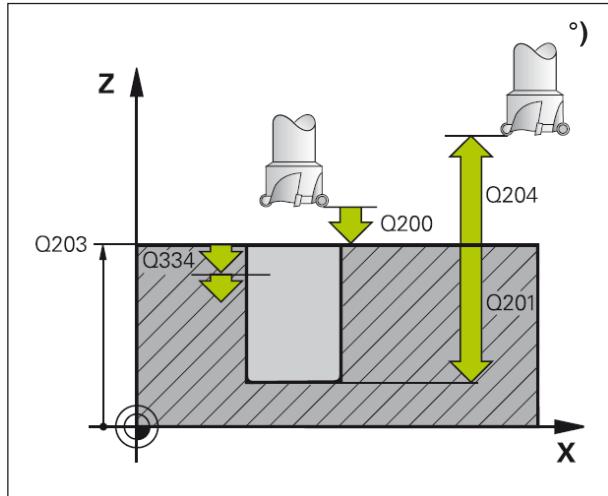
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.



## DRILL MILLING (cycle 208)

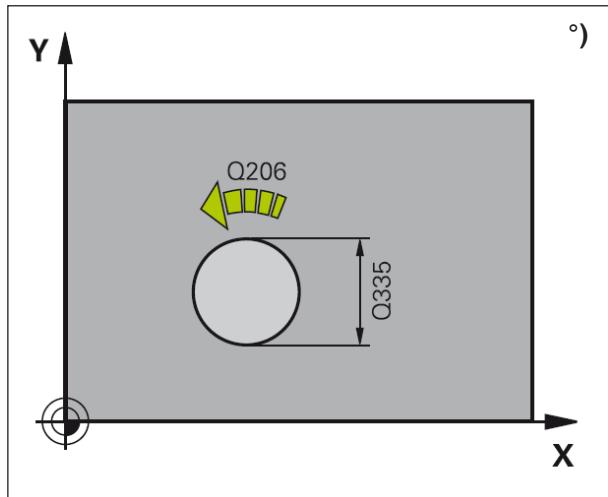


Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Depth (incremental): Distance work piece surface - hole bottom	mm
Q206	Feed for depth infeed: Traverse speed of the tool during drilling on the helix.	mm/min
Q334	Infeed per helix (incremental): Dimension by which the tool is fed in on a helix (=360°).	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q335	Nominal diameter (absolute): Hole diameter. If the nominal diameter is entered to be identical with the tool diameter, WinNC drills without any helix interpolation directly to the entered depth.	mm
Q342	Pre-drilled diameter (absolute): If a value > 0 is entered, WinNC does not conduct any check with regard to the relation between the nominal diameter and the tool diameter. Thus, holes can be milled that have a diameter more than double the size of the tool diameter.	mm
Q351	Type of milling: Type of milling machining at M3. • + = climb milling • - = up-cut milling	



### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool mills at the programmed feed F in a helix to the entered depth.
- 3 After reaching the full drilling depth, WinNC moves again in a full circle, in order to remove the material that was left at plunging.
- 4 After that, WinNC positions the tool back in the hole centre.
- 5 Finally, WinNC returns at FMAX to the safety distance. If a 2nd safety distance has been entered, WinNC moves the tool at FMAX there.



### Note:

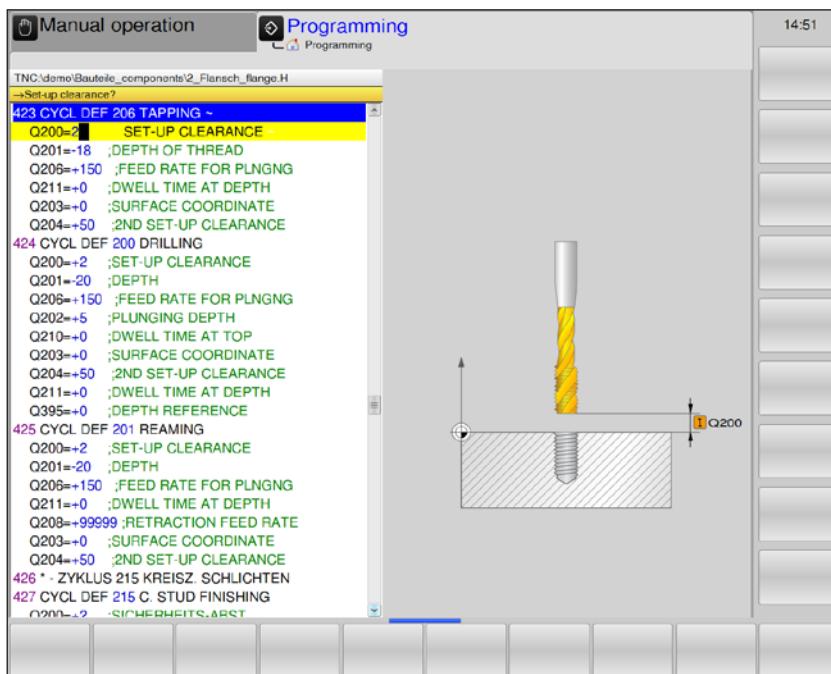
Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.

A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.

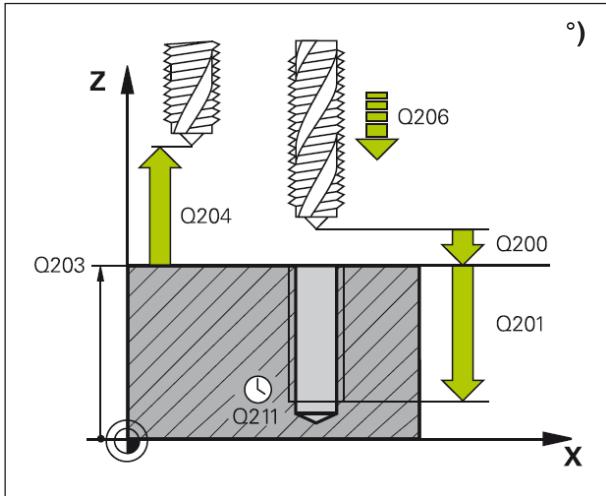




## THREAD TAPPING NEW (cycle 206)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q206	Feed: Tool traverse speed during thread tapping.	mm/min
Q211	Dwell time below: Enter a value between 0 and 0.5 seconds in order to prevent wedging of the tool during retraction.	s
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm



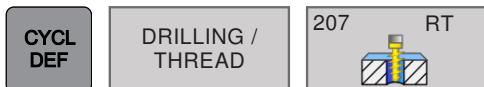
### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool moves to the drilling depth in one work step.
- 3 After that, the spindle rotation direction is reversed and, after the dwell time, the tool is retracted to the safety distance. If a 2nd safety distance has been entered, WinNC moves the tool at FMAX there.
- 4 At the safety distance, the spindle rotation direction is again reversed.

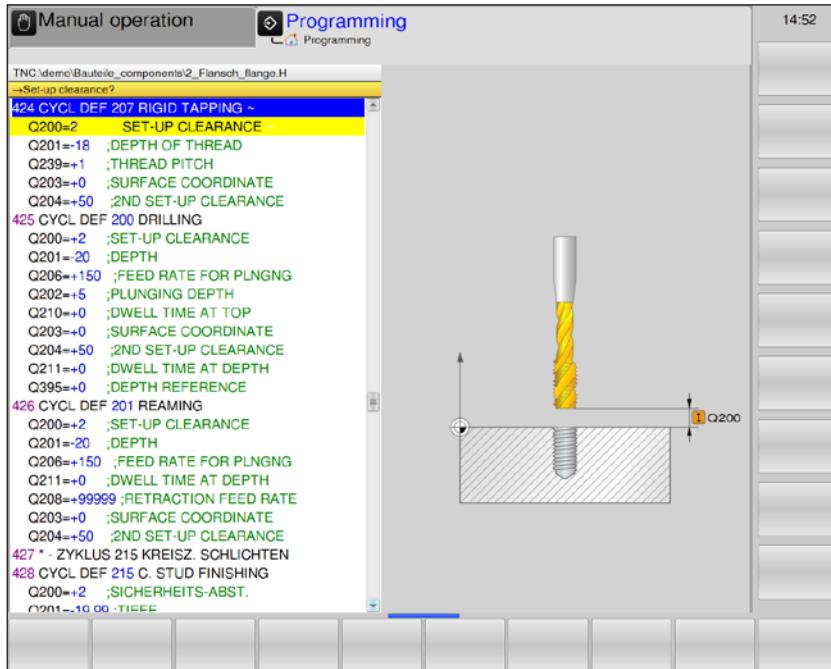
#### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.

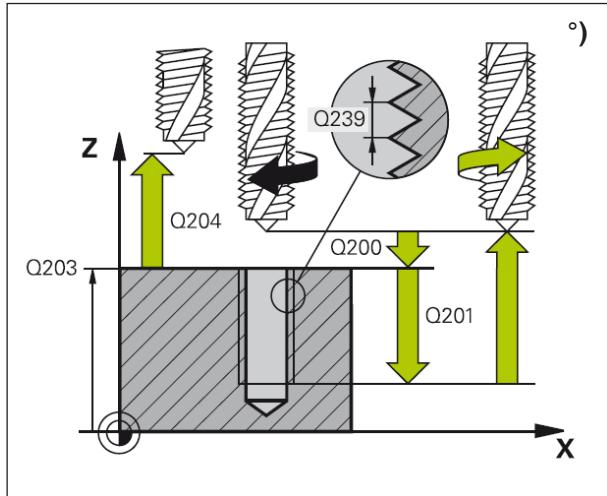




## RIGID TAPPING (cycle 207)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q239	Thread pitch: Sign determines right-hand or left-hand thread • +: • Right-hand thread • -: • Left-hand thread	mm/min
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm



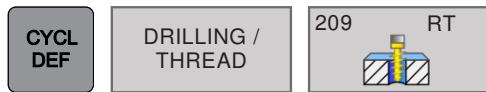
### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool moves to the drilling depth in one work step.
- 3 After that, the spindle rotation direction is reversed and, after the dwell time, the tool is retracted to the safety distance. If a 2nd safety distance has been entered, WinNC moves the tool at FMAX there.
- 4 At the safety distance, WinNC stops the spindle.

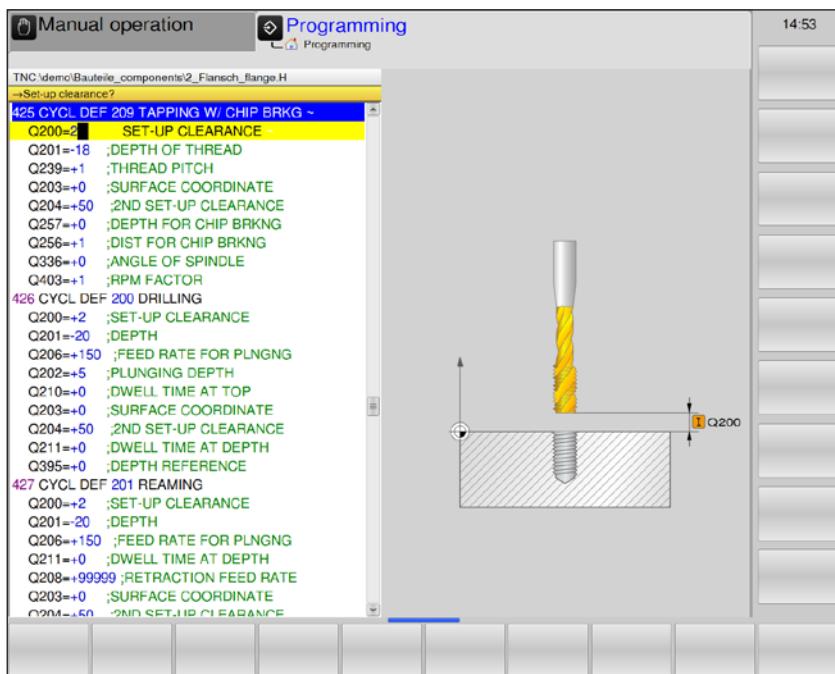
#### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.

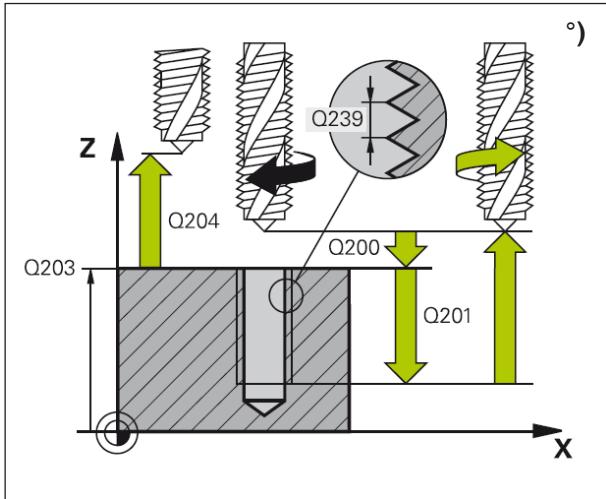




## THREAD TAPPING CHIPS CRUSH- ING (cycle 209)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q239	Thread pitch: Sign determines right-hand or left-hand thread • +: • Right-hand thread • -: • Left-hand thread	mm/min
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q257	Drilling depth until chips crushing (incremental): Infeed after which WinNC executes the chips crushing operation. Input 0: no chips crushing	mm
Q256	Retraction at chips crushing: WinNC multiplies the pitch by the entered value and retracts the tool during chips crushing by this calculated value. Input 0: WinNC completely retracts out of the hole for chip removal.	mm
Q336	Angle for the spindle orientation (absolute): the angle at which WinNC positions the tool prior to the thread tapping operation. Thus the thread can be re-cut, if required.	degrees
Q403	Speed change factor at retraction: Factor by which WinNC increases the spindle speed and thus the retracting feed when cutting out of the hole. Can be maximally increased to the maximum speed of the active gear range.	



### Cycle description

WinNC cuts the thread in multiple infeeds to the specified depth. You can determine via a parameter whether during chips crushing the tool should be retracted fully out of the hole or not.

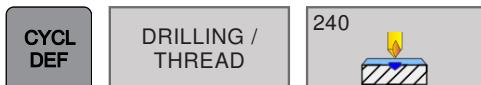
- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool travels to the specified infeed depth, reverses the spindle rotation direction and - depending on the definition - retracts by a certain value or retracts fully out of the hole for chip clearing.
- 3 After that, the spindle rotation direction is reversed again and the tool travels to the next infeed depth.
- 4 WinNC repeats this sequence (2 to 3) until the entered thread depth is reached.
- 5 After that, the tool is retracted to the safety distance. If a 2nd safety distance has been entered, WinNC moves the tool at FMAX there.
- 6 At the safety distance, WinNC stops the spindle.

#### Note:

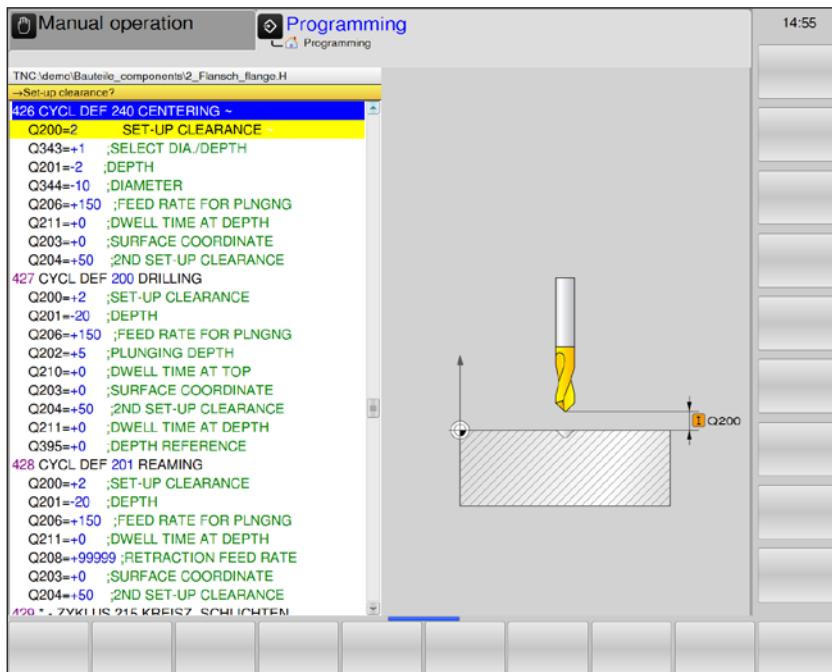
Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.

#### Note:

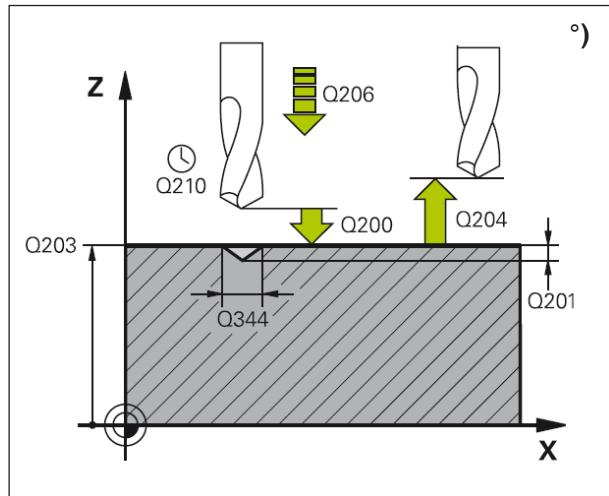
Currently, Q403 cannot be changed, you can only work with the default value.  
If external programs are loaded, which work according to other values for Q403, WinNC will automatically switch to the default value.



## CENTRING (cycle 240)

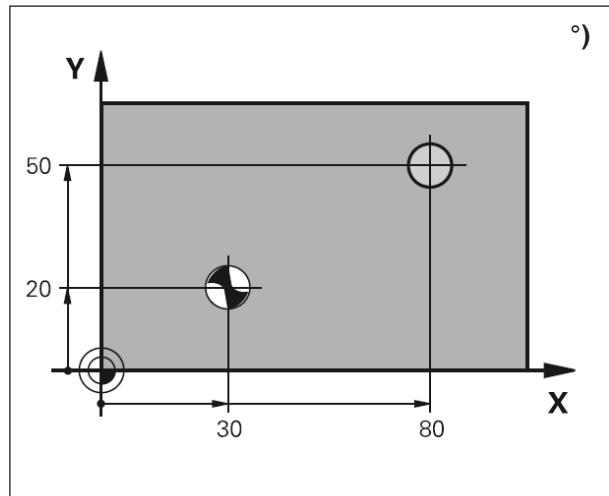


Parameter	Description	Unit
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q343	Selection depth / diameter: Selection, whether centring should be performed relative to the diameter or relative to the depth. <ul style="list-style-type: none"><li>• 0 = centring relative to the entered depth</li><li>• 1 = centring relative to the entered diameter.</li></ul> When diameter is selected: The point angle must be defined in the column T-ANGLE of the tool table TOOL.T.	
Q201	Thread depth (incremental): Distance work piece surface - centering bottom (centring taper tip). Only effective, if at Q343 input = 0.	mm
Q344	Diameter (sign): Centring diameter, only effective if Q343=1	mm
Q206	Feed for depth infeed: Tool traverse speed during centring.	mm/min
Q211	Dwell time below. Time in seconds, during which the tool remains at the hole bottom.	s
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm



### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool centres at the programmed feed F up to the entered centring diameter and/or the entered centring depth.
- 3 If entered, the tool dwells at the centering bottom.
- 4 After that, the tool is retracted to the safety distance. If a 2nd safety distance has been entered, WinNC moves the tool at FMAX there.



### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.





## Fundamentals of Thread Milling °)

### Preconditions

- As during thread milling distortions of the thread profile occur, tool-specific corrections are usually required, which you can find in the tool catalogue or request from your tool manufacturer. The correction is executed during TOOL CALL via the delta radius DR.
- The cycles 262, 263, 264 and 267 can only be used with clockwise-rotating tools. For cycle 265, you can use both, clockwise-turning and counterclockwise-turning tools.
- The working direction results from the following input parameters:  
Sign of the thread pitch Q239 (+ = right-hand thread /- = left-hand thread) and type of milling Q351 (+1 = climb milling /-1 = up-cut milling). From the adjacent table you can see the interrelation of the input parameters of clockwise-rotating tools.

Internal thread	Pitch	Type of milling	Working direction
right-handed	+	+1 (RL)	Z+
left-handed	-	-1 (RR)	Z+
right-handed	+	-1 (RR)	Z-
left-handed	-	+1 (RL)	Z-
External thread	Pitch	Type of milling	Working direction
right-handed	+	+1 (RL)	Z-
left-handed	-	-1 (RR)	Z-
right-handed	+	-1 (RR)	Z+
left-handed	-	+1 (RL)	Z+

### Note:

For thread milling, WinNC relates the programmed feed to the tool blade. However, as WinNC displays the feed relative to the centre point path, the displayed value does not match the programmed value.  
The turning direction of the thread changes if you machine one thread milling cycle in conjunction with cycle 8 MIRRORING in only one axis.



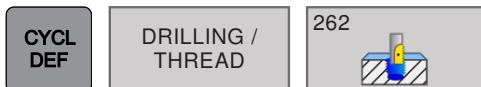
### Danger of collision:



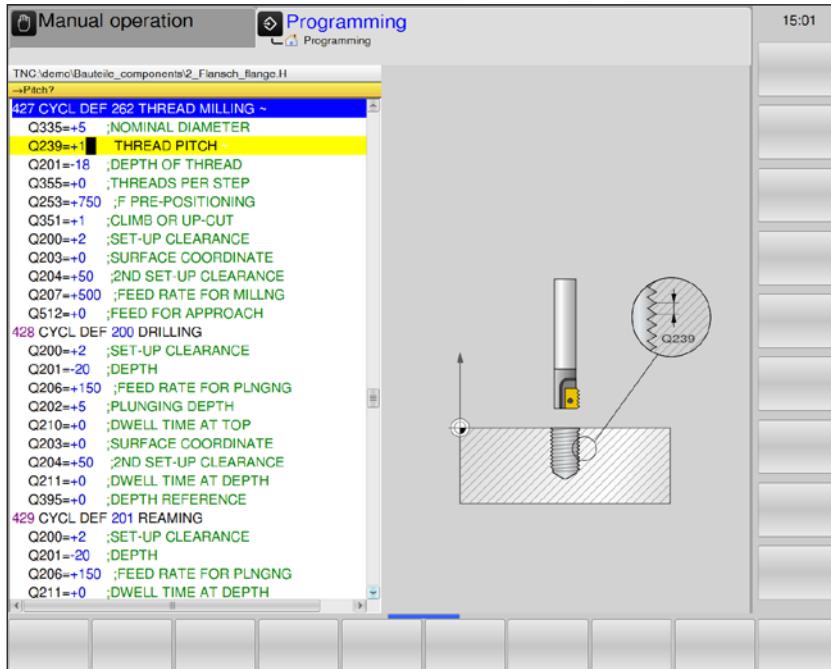
For the depth infeeds, always program the same signs, as the cycles contain multiple sequences, which are independent of each other. The order of priority, which governs the selection of the working direction, is described in the context of the respective cycles. If you want to repeat e. g. a cycle with only the countersinking procedure, just enter 0 for the thread depth, the working direction will then be determined via the countersinking depth.

### What to do when tools break:

If a tool breakage occurs during thread cutting, stop the program run, change into the positioning operating mode with manual input and move the tool then in a linear movement to the hole centre. Subsequently, you can clear the tool in the infeed axis and change it.

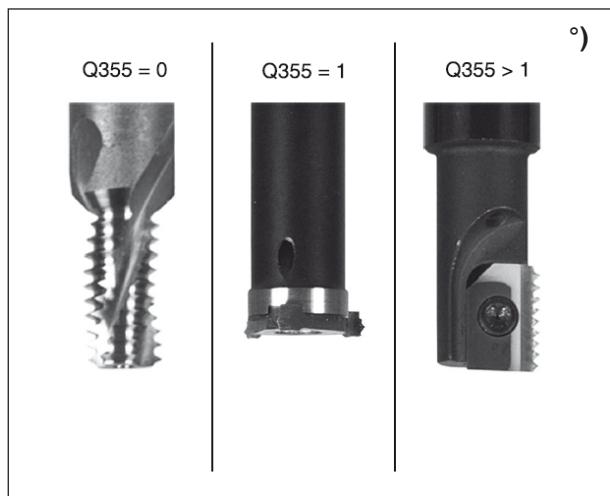
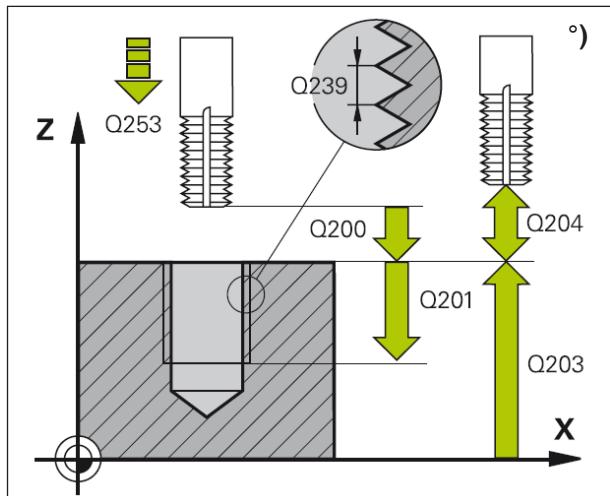


## THREAD MILLING (cycle 262)



Parameter	Description	Unit
Q335	Nominal diameter: Thread diameter	mm
Q239	Thread pitch: the sign determines the direction of rotation: • + = Right-hand thread • - = Left-hand thread	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q355	Threads per step: Number of threads by which the tool is offset: • 0 = one helix line to the thread depth • 1 = continuous helix line on the total thread length. • >1 = multiple helix paths with approaching and departing. Between them, WinNC offsets the tool by Q355 multiplied by the pitch.	
Q253	Preliminary positioning feed. Traverse speed of the tool during plunging and/or during retracting from the work piece.	mm/min
Q351	Type of milling: Type of milling machining at M3. • +1 = down-cut milling • -1 = up-cut milling with input 0 = down-cut milling	
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm

Parameter	Description	Unit
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q512	Approach feed: traverse speed of the tool during approaching. In case of small thread diameters, the risk of tool breakage can be reduced by reducing the approach feed.	mm/min



### Cycle description

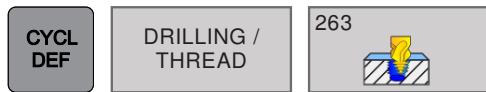
- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.
- 2 The tool travels at the programmed preliminary positioning feed to the starting plane, which results from the algebraic sign of the thread pitch, from the type of milling and from the number of threads specified as the offset number.
- 3 Subsequently, the tool tangentially approaches the nominal diameter of the thread in a helical movement. However, prior to the helix approach movement, a compensation movement in the tool axis is executed, in order to start with the thread path on the programmed start plane.
- 4 Depending on the threads per step parameter, the tool mills the thread in one helical line, in several offset helical lines or in one continuous helical movement.
- 5 After that, the tool tangentially retracts from the contour to the starting point in the machining plane.
- 6 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance.

#### Note:

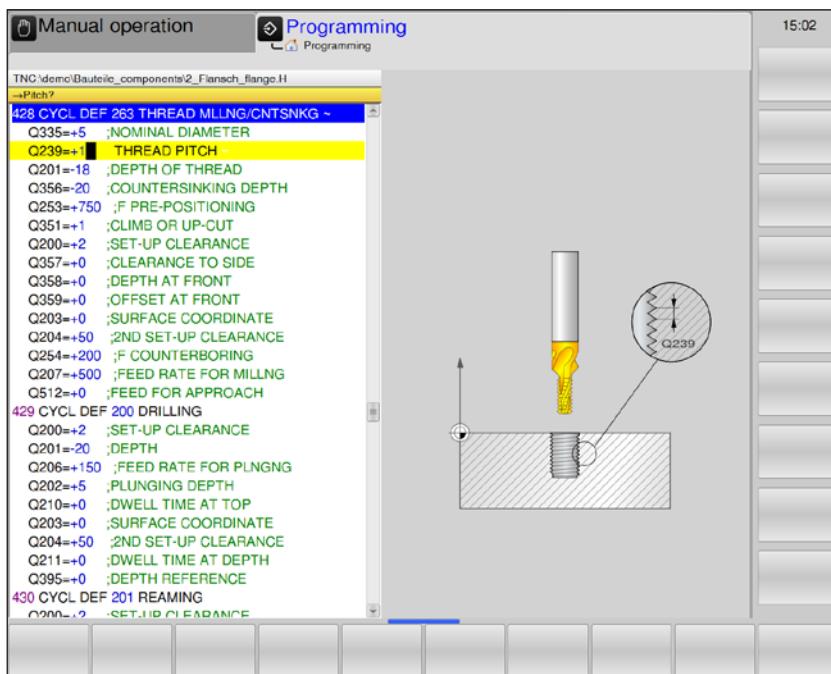
Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.

A negative sign means:  
Machining in the direction of the negative spindle axis.

If depth = 0 is programmed, the cycle will not be executed.



## THREAD MILLING/COUNTER-SINKING (cycle 263)



Parameter	Description	Unit
Q335	Nominal diameter: Thread diameter	mm
Q239	Thread pitch: the sign determines the direction of rotation: • + = Right-hand thread • - = Left-hand thread	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q356	Countersinking depth (incremental): Distance between work piece surface and tool point:	mm
Q253	Preliminary positioning feed. Traverse speed of the tool during plunging and/or during retracting from the work piece.	mm/min
Q351	Type of milling: Type of milling machining at M3. • +1 = down-cut milling • -1 = up-cut milling with input 0 = down-cut milling	
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q357	Side safety distance (incremental): Distance between tool blade and hole wall.	mm
Q358	Front side depth (incremental): Distance between work piece surface and tool point during countersinking procedure from the front side.	mm
Q359	Offset countersinking from the front side (incremental): Distance by which WinNC offsets the tool centre from the centre.	mm
Q203	Work piece surface coordinates (absolute)	mm

Parameter	Description	Unit
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q254	Countersinking feed: Tool traverse speed during countersinking	mm/min
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q512	Approach feed: traverse speed of the tool during approaching. In case of small thread diameters, the risk of tool breakage can be reduced by reducing the approach feed.	mm/min

**Note:**

Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The algebraic signs of the cycle parameters thread depth, countersinking depth and/or front side depth determine the working direction.

A negative sign means:

Machining in the direction of the negative spindle axis.

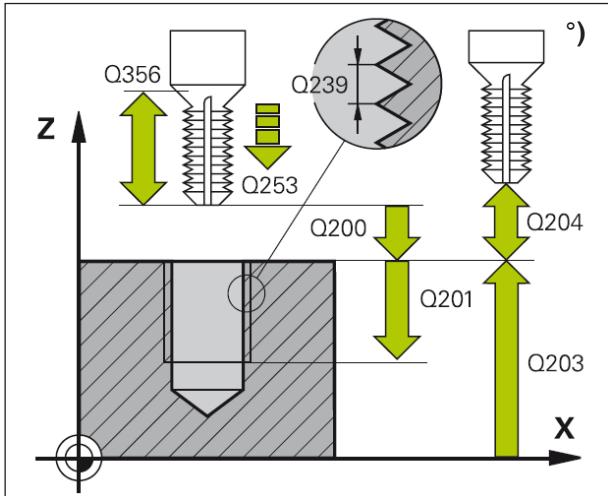
The working direction is determined according to the following order:

1. Thread depth
- 2nd Drilling depth
3. Front side depth

If depth = 0 is programmed, the cycle will not be executed. Program the thread depth at least one third multiplied by the thread pitch smaller than the drilling depth.







### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.

### Countersinking

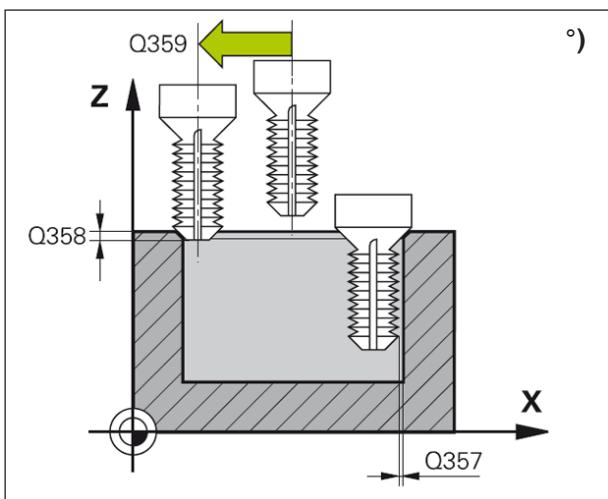
- 2 The tool travels at preliminary positioning feed to the countersinking depth minus safety distance, and subsequently at countersinking feed to the countersinking depth.
- 3 If a side safety distance has been specified, WinNC positions the tool directly at the countersinking depth at preliminary positioning feed.
- 4 Subsequently, depending on the space conditions, WinNC departs out of the centre, or it approaches the core diameter smoothly with side preliminary positioning and executes a circular movement.

### Front side countersinking

- 5 The tool moves at preliminary positioning feed to the front side countersinking depth.
- 6 WinNC positions the tool uncorrected from the centre via a semicircle on the front side offset and executes a circular movement at countersinking feed.
- 7 After that, WinNC moves the tool on a semi-circle back to the hole centre.

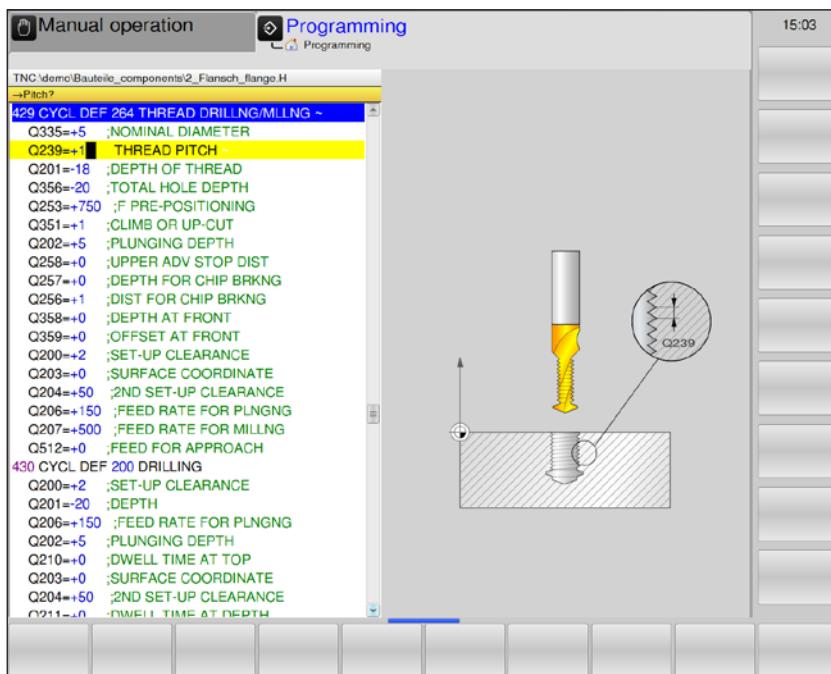
### Thread milling

- 8 WinNC moves the tool at the programmed preliminary positioning feed to that starting plane for the thread, which results from the algebraic sign of the thread pitch and the type of milling.
- 9 Subsequently, the tool tangentially approaches the nominal thread diameter in a helical movement and mills the thread with a 360° helical movement.
- 10 After that, the tool tangentially retracts from the contour to the starting point on the machining plane.
- 11 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance.





## THREAD DRILLING/MILLING (cycle 264)



Parameter	Description	Unit
Q335	Nominal diameter: Thread diameter	mm
Q239	Thread pitch: the sign determines the direction of rotation: • + = Right-hand thread • - = Left-hand thread	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q356	Drilling depth (incremental): Distance between work piece surface and hole bottom.	mm
Q253	Preliminary positioning feed. Traverse speed of the tool during plunging and/or during retracting from the work piece.	mm/min
Q351	Type of milling: Type of milling machining at M3. • +1 = down-cut milling • -1 = up-cut milling with input 0 = down-cut milling	
Q202	Infeed depth: (incremental): dimension by which the tool is fed in respectively. The depth does not have to be a multiple of the infeed depth. WinNC moves in one work step to the depth, if • infeed depth and depth are identical • infeed depth is bigger than depth	mm
Q258	Advance stop distance (incremental): Safety distance for rapid traverse positioning, when WinNC moves the tool after a retraction from the hole back to the current infeed depth.	mm
Q257	Drilling depth until chips crushing (incremental): Value by which WinNC retracts the tool during chips crushing.	mm

Parameter	Description	Unit
Q256	Retraction at chips crushing (incremental): Value by which the tool retracts during chips crushing.	mm
Q358	Front side depth (incremental): Distance between work piece surface and tool point during countersinking procedure from the front side.	mm
Q359	Offset countersinking from the front side (incremental): Distance by which WinNC offsets the tool centre from the centre.	mm
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q206	Feed for depth infeed: Tool traverse speed during plunging	mm/min
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q512	Approach feed: traverse speed of the tool during approaching. In case of small thread diameters, the risk of tool breakage can be reduced by reducing the approach feed.	mm/min

**Note:**

Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The algebraic signs of the cycle parameters thread depth, countersinking depth and/or front side depth determine the working direction.

A negative sign means:

Machining in the direction of the negative spindle axis.

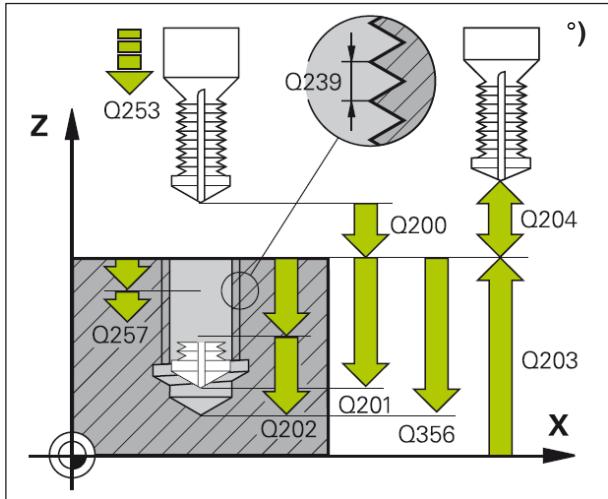
The working direction is determined according to the following order:

1. Thread depth
- 2nd Drilling depth
3. Front side depth

If depth = 0 is programmed, the cycle will not be executed. Program the thread depth at least one third multiplied by the thread pitch smaller than the drilling depth.







### Cycle description

1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.

### Drilling

- 2 The tool drills at the entered feed for depth infeed to the first infeed depth.
- 3 If chips crushing has been entered, WinNC retracts the tool by the entered retraction value. If you work without chips crushing, WinNC retracts the tool in rapid traverse to the safety distance, and subsequently again at FMAX to the entered advance stop distance above the first infeed depth.
- 4 Subsequently, the tool drills at the feed by a further infeed depth.
- 5 WinNC repeats this sequence (2 to 4), until the drilling depth is reached.

### Front side countersinking

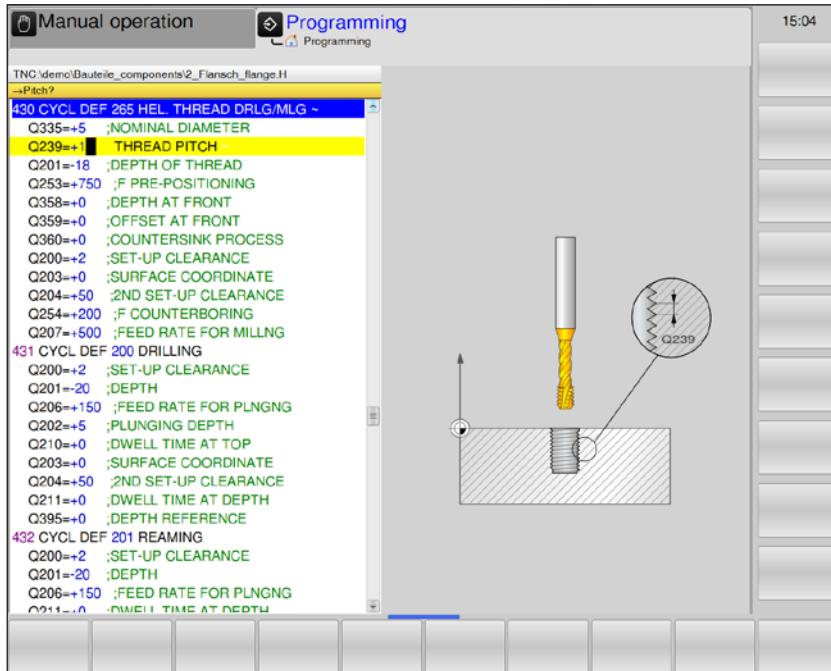
- 6 The tool moves at preliminary positioning feed to the front side countersinking depth.
- 7 WinNC positions the tool uncorrected from the centre via a semicircle on the front side offset and executes a circular movement at countersinking feed.
- 8 After that, WinNC moves the tool on a semi-circle back to the hole centre.

### Thread milling

- 9 WinNC moves the tool at the programmed preliminary positioning feed to that starting plane for the thread, which results from the algebraic sign of the thread pitch and the type of milling.
- 10 Subsequently, the tool tangentially approaches the nominal thread diameter in a helical movement and mills the thread with a 360° helical movement.
- 11 After that, the tool tangentially retracts from the contour to the starting point on the machining plane.
- 12 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance.

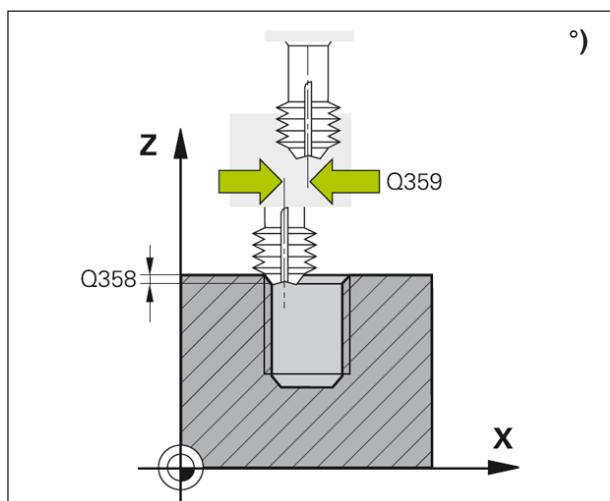
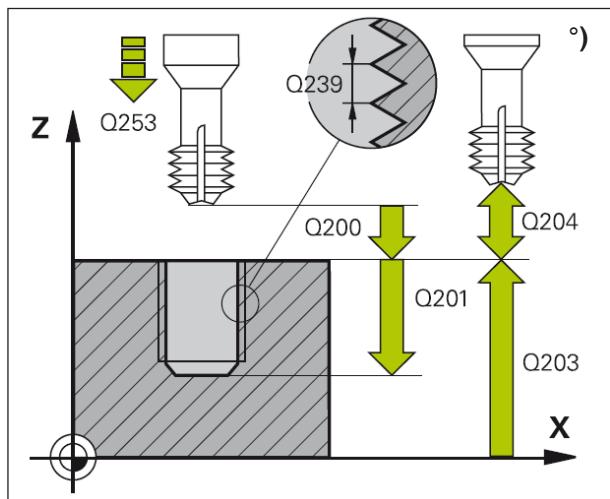


## HELICAL THREAD DRILLING/MILLING (cycle 265)



Parameter	Description	Unit
Q335	Nominal diameter: Thread diameter	mm
Q239	Thread pitch: the sign determines the direction of rotation: <ul style="list-style-type: none"> <li>+ = Right-hand thread</li> <li>- = Left-hand thread</li> </ul>	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q253	Preliminary positioning feed: Traverse speed of the tool during plunging and/or during retracting from the work piece.	mm/min
Q358	Front side depth (incremental): Distance between work piece surface and tool point during countersinking procedure from the front side.	mm
Q359	Offset countersinking from the front side (incremental): Distance by which WinNC offsets the tool centre from the centre.	mm
Q360	Countersinking procedure: Execution of the chamfer <ul style="list-style-type: none"> <li>0 = prior to the thread machining</li> <li>1 = after the thread machining</li> </ul>	
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q254	Countersinking feed: Tool traverse speed during countersinking.	mm/min

Parameter	Description	Unit
Q207	Milling feed: Tool traverse speed during milling	mm/min

**Note:**

Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The algebraic signs of the cycle parameters thread depth and/or front side depth determine the working direction.

The working direction is determined according to the following order:

1. Thread depth

2nd Front side depth

If depth = 0 is programmed, the cycle will not be executed.

The type of milling (up-cut/down-cut) is determined by the thread (right-hand/left-hand thread) and the direction of rotation of the tool, as only the working direction from the work piece surface into the work piece is possible.

**Cycle description**

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.

**Front side countersinking**

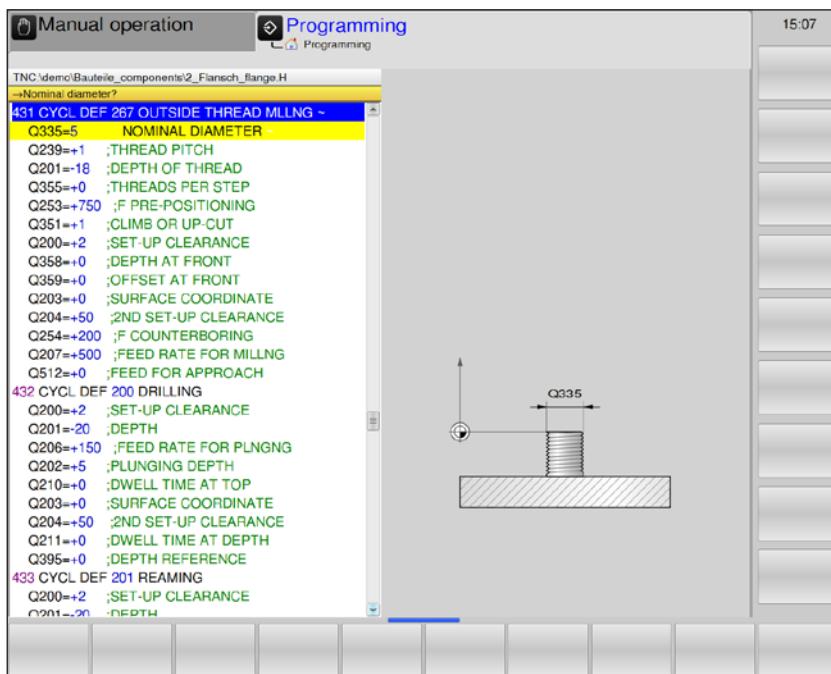
- 2 When countersinking prior to thread machining, the tool moves at countersinking feed to the front side countersinking depth. When the countersinking procedure is to occur after the thread machining, WinNC moves the tool to the countersinking depth at preliminary positioning feed.
- 3 WinNC positions the tool uncorrected from the centre via a semicircle on the front side offset and executes a circular movement at countersinking feed.
- 4 After that, WinNC moves the tool on a semi-circle back to the hole centre.

**Thread milling**

- 5 WinNC moves the tool at the programmed preliminary positioning feed to the start plane for the thread.
- 6 Subsequently, the tool tangentially approaches the nominal diameter of the thread in a helical movement.
- 7 WinNC moves the tool down along a continuous helical line, until the thread depth is reached.
- 8 After that, the tool tangentially retracts from the contour to the starting point on the machining plane.
- 9 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance.



## EXTERNAL THREAD MILLING (cycle 267)



Parameter	Description	Unit
Q335	Nominal diameter: Thread diameter	mm
Q239	Thread pitch: the sign determines the direction of rotation: • + = Right-hand thread • - = Left-hand thread	mm
Q201	Thread depth (incremental): Distance work piece surface - thread base	mm
Q355	Threads per step: Number of threads by which the tool is offset: • 0 = one helix line to the thread depth • 1 = continuous helix line on the total thread length • >1 = multiple helix paths with approaching and departing by Q355 multiplied by the pitch.	
Q253	Preliminary positioning feed: Traverse speed of the tool during plunging and/or during retracting from the work piece.	mm/min
Q351	Type of milling: Type of milling machining at M3 • +1 = down-cut milling • -1 = up-cut milling With input 0: down-cut machining	
Q200	Safety distance (incremental): Distance tool point - work piece surface; enter positive value	mm
Q358	Front side depth (incremental): Distance between work piece surface and tool point during countersinking procedure from the front side.	mm
Q359	Offset countersinking from the front side (incremental): Distance by which WinNC offsets the tool centre from the centre.	mm

Parameter	Description	Unit
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece (clamping device) can occur.	mm
Q254	Countersinking feed: Tool traverse speed during countersinking.	mm/min
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q512	Approach feed: Tool traverse speed during approaching. In case of small thread diameters, the risk of tool breakage can be reduced by reducing the approach feed.	mm/min

**Note:**

Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The required offset for front side countersinking should be determined beforehand. You have to specify the value from the stud centre to the tool centre (uncorrected value). The algebraic signs of the cycle parameters thread depth, countersinking depth and/or front side depth determine the working direction.

A negative sign means:

Machining in the direction of the negative spindle axis.

The working direction is determined according to the following order:

1. Thread depth

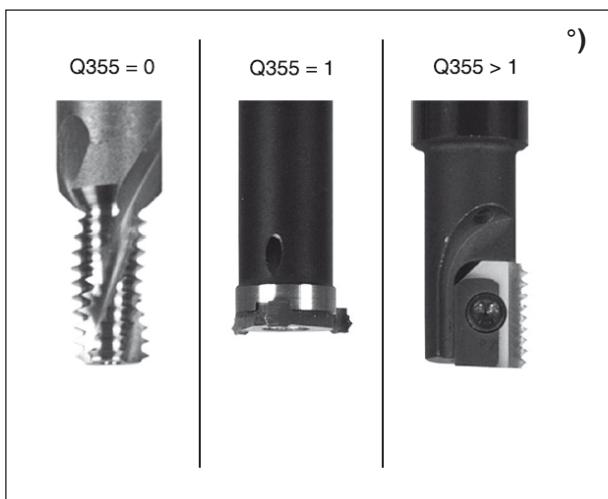
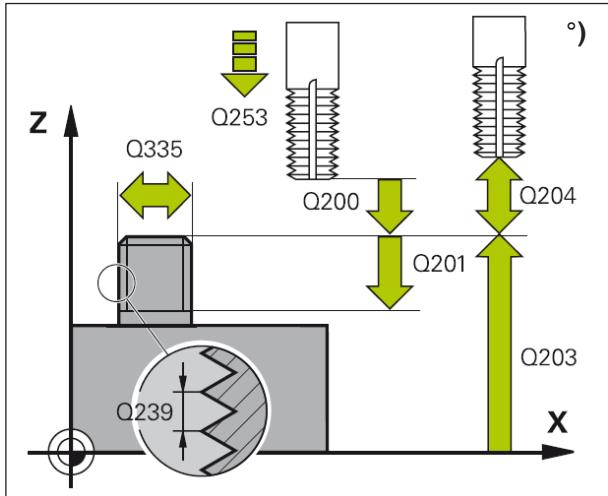
2nd Front side depth

If depth = 0 is programmed, the cycle will not be executed.

The sign of the cycle parameter thread depth determines the working direction.







### Cycle description

- 1 WinNC positions the tool at rapid traverse FMAX in the spindle axis at the safety distance above the work piece surface.

### Front side countersinking

- 2 WinNC approaches the starting point for front side countersinking originating from the stud centre on the main axis of the machining plane. The starting point position results from the thread radius, tool radius and pitch.
- 3 The tool moves at preliminary positioning feed to the front side countersinking depth.
- 4 WinNC positions the tool uncorrected from the centre via a semicircle on the front side offset and executes a circular movement at countersinking feed.
- 5 After that, WinNC moves the tool on a semi-circle back to the starting point.

### Thread milling

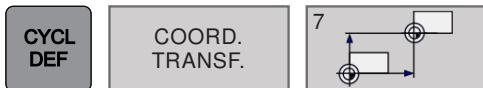
- 6 WinNC positions the tool on the starting point if no front side countersinking has been executed prior to the operation. Starting point thread milling = starting point front side countersinking.
- 7 The tool travels at the programmed preliminary positioning feed to the starting plane which results from the algebraic sign of the thread pitch, from the type of milling and from the number of threads specified as the offset number.
- 8 Subsequently, the tool tangentially approaches the nominal diameter of the thread in a helical movement.
- 9 Depending on the threads per step parameter, the tool mills the thread in one helical line, in several offset helical lines or in one continuous helical movement.
- 10 After that, the tool tangentially retracts from the contour to the starting point on the machining plane.
- 11 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance.



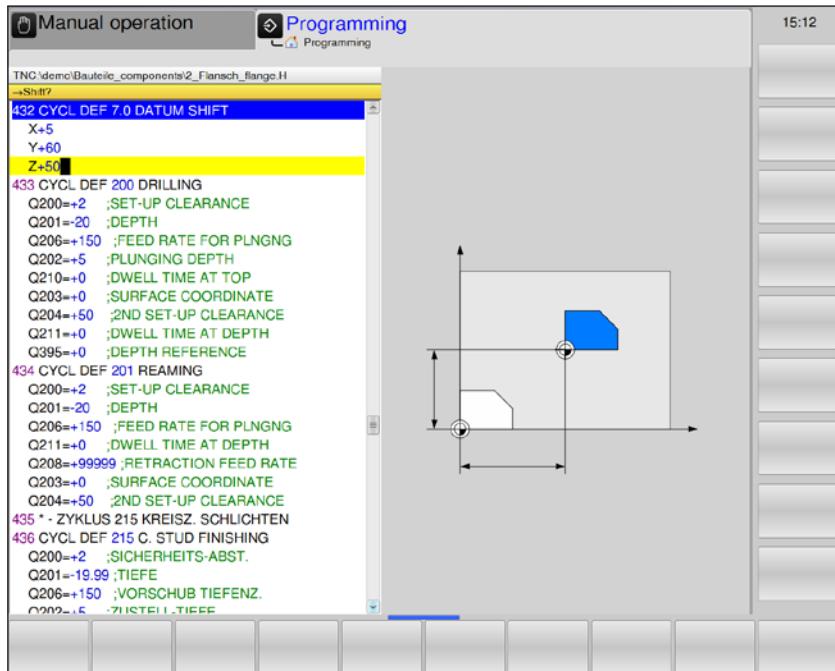
## Converting the coordinates

COORD.  
TRANSF.

- 7 Zero offset
- 8 Mirroring
- 10 Rotation



## ZERO OFFSET (cycle 7)



### Cycle description

#### Effect

Zero offset allows you to repeat machining processes at any position on the work piece by shifting the coordinate system to a suitable point in the working area of the machine.

The zero point of the work piece can be shifted within a parts program any number of times.

After a cycle definition zero offset, all coordinate specifications relate to the new zero point. WinNC displays the shift in every axis in the additional status display. Entry of rotation axes is also possible.

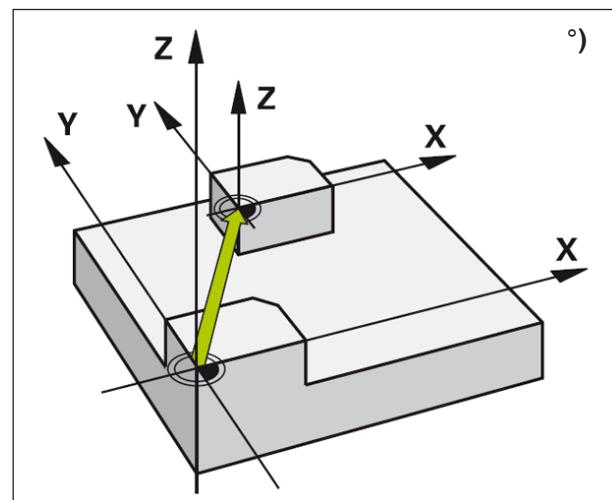
- **Offsetting:** Enter the coordinates of the new zero point.

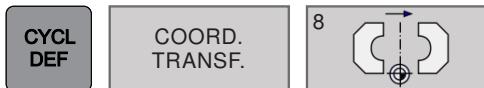
The absolute values relate to the work piece zero point that is determined by way of reference point setting.

The incremental values always refer to the last valid zero point - the latter can be shifted already.

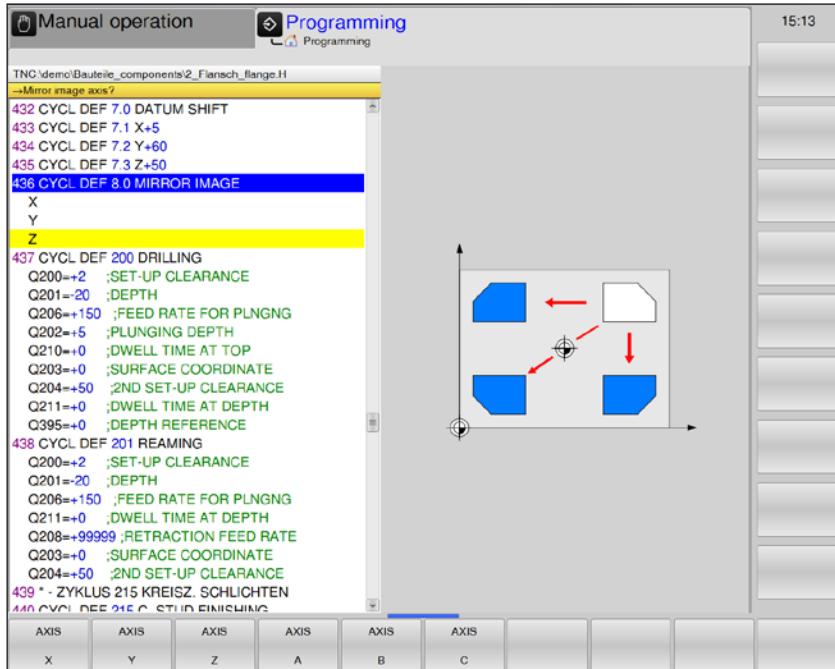
#### Resetting

A zero offset using the coordinates X=0, Y=0 and Z=0 cancels the previous zero offset.





## MIRRORING (cycle 8)



### Cycle description

WinNC can execute a machining process in the machining plane in a mirrored manner.

### Effect

As soon as mirroring has been defined in the program, it is also effective in the positioning operating mode with manual input. WinNC displays active mirroring axes in the additional status display.

- If only one axis is mirrored, the turning direction of the tool changes, this does not apply in machining cycles.
- If two axes are mirrored, the turning direction remains the same.

The result of the mirroring is dependent on the zero point position:

- The zero point is on the contour that is to be mirrored: The element is mirrored directly at the zero point.
- The zero point is outside of the contour that is to be mirrored: The element is additionally shifted.

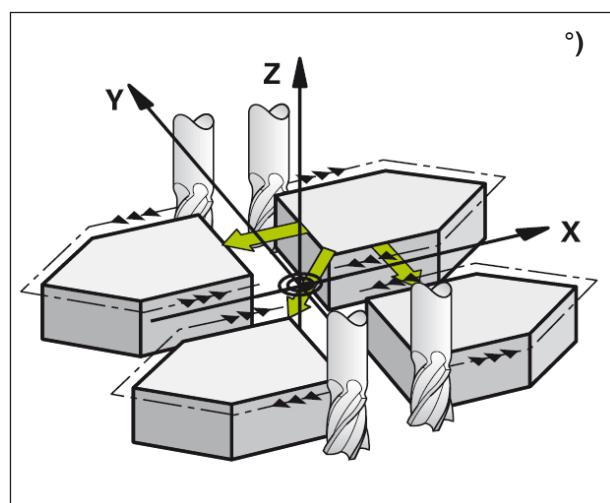
- **Mirrored axis?:**

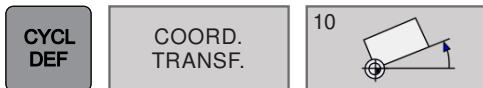
Enter the axes that need to be mirrored. All axes can be mirrored - including the rotation axes - except the spindle axis and the associated minor axis.

Entry of maximum three axes is allowed.

### Resetting

Program the mirroring cycle anew with the entry NO ENT.





## ROTATION (cycle 10)

```

TNC\demo\Bauteile_components\2_Flansch_flange.H
→Rotation angle?
432 CYCL DEF 7.0 DATUM SHIFT
433 CYCL DEF 7.1 X+5
434 CYCL DEF 7.2 Y-60
435 CYCL DEF 7.3 Z+50
436 CYCL DEF 8.0 MIRROR IMAGE
437 CYCL DEF 8.1 X,Y,Z
438 CYCL DEF 10.0 ROTATION
    ROT+10
440 CYCL DEF 200 DRILLING
    Q200=+2 ;SET-UP CLEARANCE
    Q201=-20 ;DEPTH
    Q206=+150 ;FEED RATE FOR PLNGNG
    Q202=+5 ;PLUNGING DEPTH
    Q210=+0 ;DWELL TIME AT TOP
    Q203=+0 ;SURFACE COORDINATE
    Q204=+50 ;2ND SET-UP CLEARANCE
    Q211=+0 ;DWELL TIME AT DEPTH
    Q395=+0 ;DEPTH REFERENCE
441 CYCL DEF 201 REAMING
    Q200=+2 ;SET-UP CLEARANCE
    Q201=+20 ;DEPTH
    Q206=+150 ;FEED RATE FOR PLNGNG
    Q211=+0 ;DWELL TIME AT DEPTH
    Q208=+99999 ;RETRACTION FEED RATE
    Q203=+0 ;SURFACE COORDINATE
    Q204=+50 ;2ND SET-UP CLEARANCE
442 * - ZYKLUS 215 KREISZ. SCHLICHTEN
443 CYCL DEF 215 C STUD FINISHING

```

### Cycle description

In a program, WinNC can rotate the coordinate system in the machining plane about the active zero point.

### Effect

As soon as rotation has been defined in the program, it is also effective in the positioning operating mode with manual input. WinNC displays the active rotation angle in the additional status display.

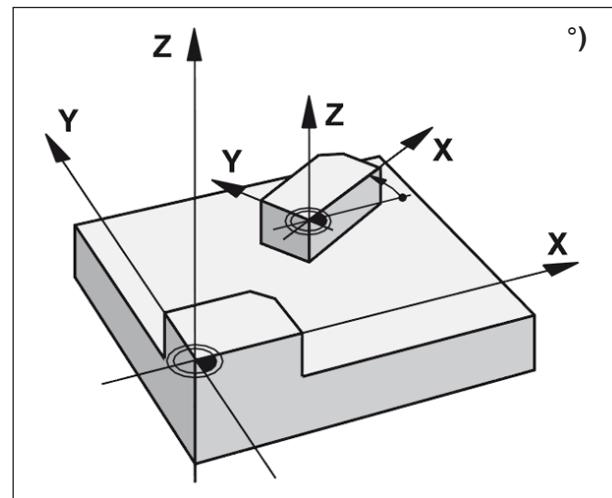
Reference axis for the angle of rotation:

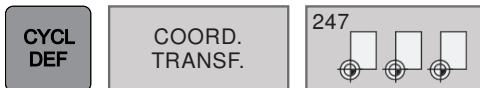
- X/Y-plane X-axis

**Rotation:** Enter the angle of rotation in degrees (°). Input range: -360° to +360° (absolute or incremental values)

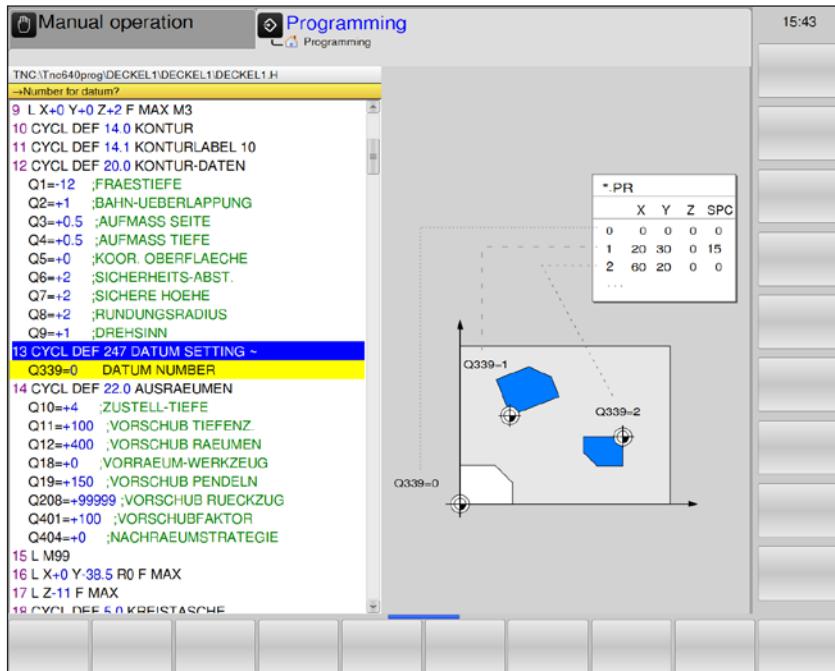
### Resetting

Program the rotation cycle anew with angle of rotation 0°.





## DATUM SETTING (cycle 247)



### Cycle description

With the DATUM SETTING cycle you can activate as the new datum a preset defined in a preset table.

After a DATUM SETTING cycle definition, all of the coordinate inputs and datum shifts (absolute and incremental) are referenced to the new preset.

### Status display

In the status display the WinNC shows the active preset number behind the datum symbol.

### Number for datum?:

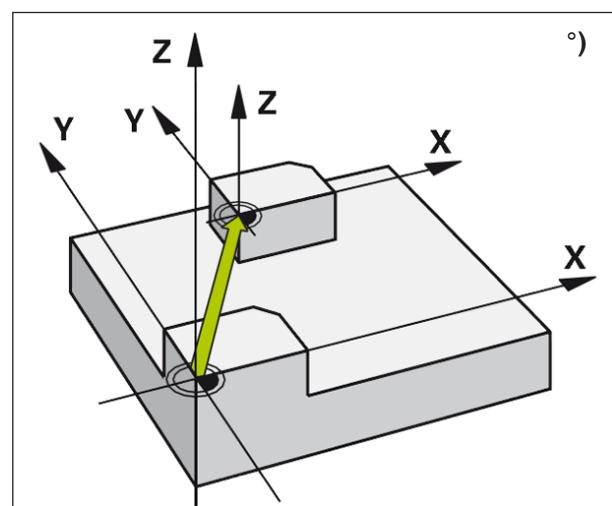
Enter the number of the line from the preset table in which the desired reference point is defined. Input range: 0 to 65535.

**Note:**

When activating a datum from the preset table, the WinNC resets the datum shift, mirroring, rotation, scaling factor and axis-specific scaling factor.

If you activate preset number 0 (line 0), then you activate the datum that you last set in the Manual Operation or El. Handwheel operating mode.

Cycle 247 is not functional in Test Run mode.





## SL Cycles



- 14 Contour
- 20 Contour data
- 22 Clearing
- 23 Finishing depth
- 24 Finishing side
- 25 Contour pull

## SL cycle fundamentals

Using SL cycles, complex contours consisting of up to 12 subcontours (pockets or islands) can be composed. The individual subcontours are entered as subprograms. From the list of subcontours (subprogram numbers), which are specified in the cycle 14 CONTOUR, WinNC computes the overall contour.

### Characteristics of the subprograms

- Coordinate conversions are permissible. If they are programmed within a subcontour, they are also effective in the subprograms that follow, however, they do not have to be reset after the cycle call.
- WinNC ignores feeds F and additional functions M.
- WinNC recognises a pocket if the tool path lies inside the contour, e. g. description of the contour clockwise with radius compensation RR.
- WinNC recognises an island if the tool path lies outside the contour, e. g. description of the contour clockwise with radius compensation RL.
- The subprograms must not contain any coordinates in the spindle axis.
- You define the machining plane in the first coordinate block of the subprogram.

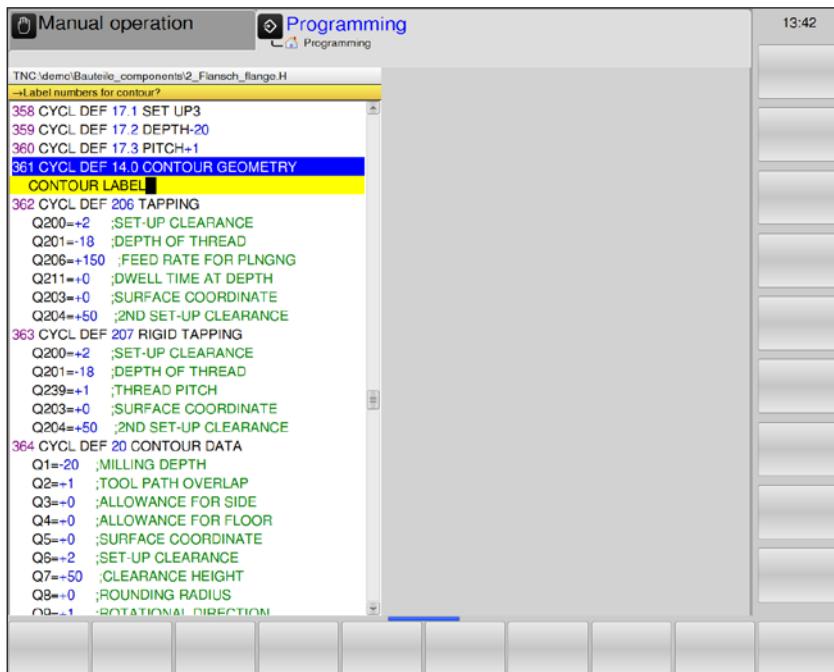
### Characteristics of the machining cycles

- Prior to every cycle, WinNC positions the tool automatically at the safety distance.
- Every depth level is milled without tool lifting, the tool moves around islands.
- The radius of "inside corners" can be programmed - the tool does not stop, clearance cutting marks are prevented (applies to the outermost path when reaming and side finishing).
- When side finishing, WinNC approaches the contour on a tangential circular path.
- When floor finishing, WinNC also moves the tool toward the work piece on a tangential circular path (e. g.: Spindle axis Z: circular path in plane Z/X).
- WinNC machines the contour continuously in down-cut and/or up-cut milling.

The dimension specifications for the machining such as cutting depth, allowances and safety distance are entered in cycle 20 as CONTOUR DATA.

CYCL DEF      SL CYCLES      14 LBL 1..N

## CONTOUR (cycle 14)



14  
LBL 1..N

In cycle 14 CONTOUR, you list all subprograms that are to be superimposed to form an overall contour.

ENT

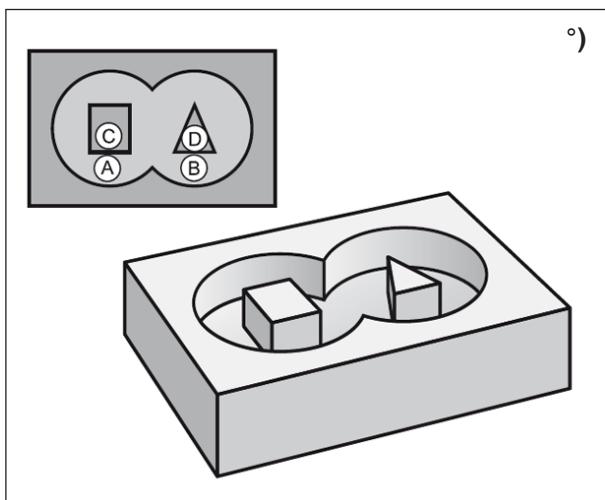
END

**Label numbers for the contour:** Enter all label numbers of the individual subprograms that are to be superimposed to form one contour.

Confirm every number with the ENT key and conclude the entry by pressing the END key.

### Example:

12 CYCL DEF 14.0 CONTOUR  
13 CYCL DEF 14.1 CONTOUR LABEL 1 /2 /3 /4



### Note:

Prior to programming, please take note:  
Cycle 14 is DEF-active, which means that it is effective in the program as soon as it is defined.

In cycle 14, a maximum of 12 subprograms (subcontours) can be listed.



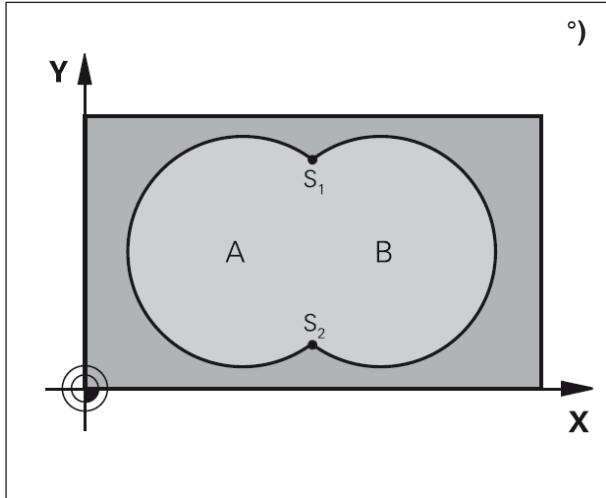
**Note:**

The following program examples are contour subprograms, which are called in a main program by cycle 14 CONTOUR:



## Superimposed contours

Pockets and islands can be superimposed to form a new contour. In this way, the area of a pocket can be enlarged by a superimposed pocket or an island can be made smaller.



### Subprograms Superimposed pockets

The pockets A and B are superimposed on each other.

WinNC computes the points of intersection S1 and S2; these do not have to be programmed.

The pockets are programmed as full circles.

#### Subprogram 1: Pocket A

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

#### Subprogram 2: Pocket B

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-60
60 LBL 0
```

### "Sum" surface

Both partial surfaces A and B including the mutually covered surface have to be machined:

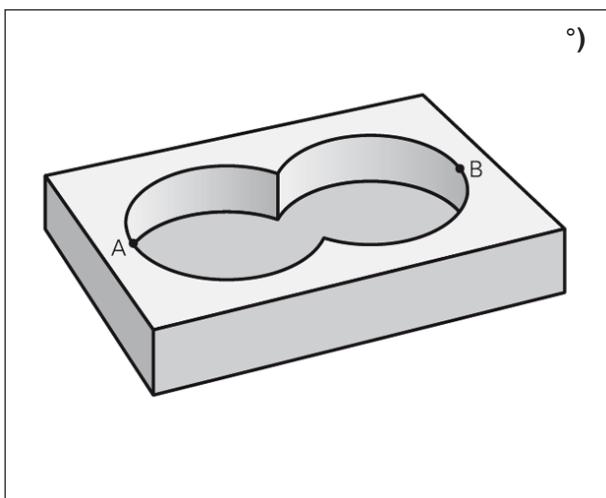
- Surfaces A and B have to be pockets.
- The first pocket (in cycle 14) has to start outside the second pocket.

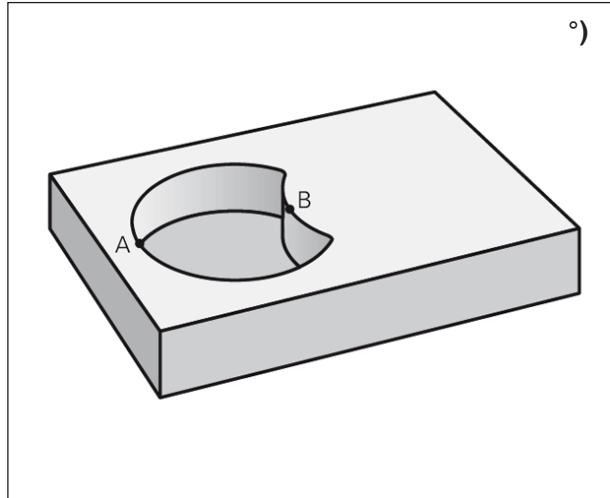
#### Surface A:

```
51 LBL 1
52 L X+10 Y+50 RR
53 CC X+35 Y+50
54 C X+10 Y+50 DR-
55 LBL 0
```

#### Surface B:

```
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```



**"Difference" surface**

Surface A has to be machined without the part that is covered by surface B:

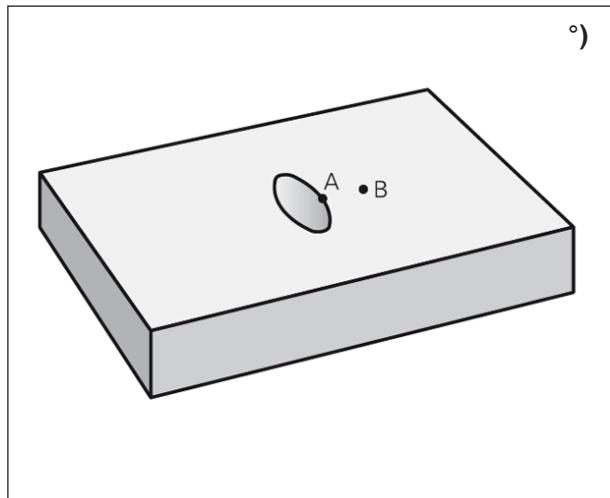
- Surface A has to be a pocket and B has to be an island.
- A has to start outside of B.

**Surface A:**

```
51 LBL 1  
52 L X+10 Y+50 RR  
53 CC X+35 Y+50  
54 C X+10 Y+50 DR-  
55 LBL 0
```

**Surface B:**

```
56 LBL 2  
57 L X+90 Y+50 RL  
58 CC X+65 Y+50  
59 C X+90 Y+50 DR-  
60 LBL 0
```

**"Overlap" surface**

The surface that is covered by A and B has to be machined. (Single overlapping surfaces have to be machined.)

- A and B have to be pockets.
- A has to start inside of B.

**Surface A:**

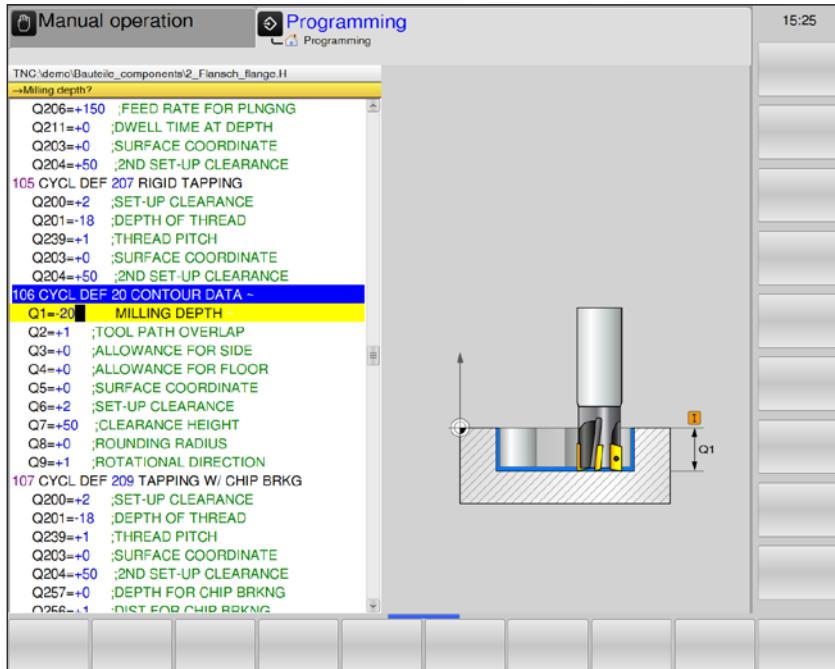
```
51 LBL 1  
52 L X+60 Y+50 RR  
53 CC X+35 Y+50  
54 C X+60 Y+50 DR-  
55 LBL 0
```

**Surface B:**

```
56 LBL 2  
57 L X+90 Y+50 RR  
58 CC X+65 Y+50  
59 C X+90 Y+50 DR-  
60 LBL 0
```

CYCL DEF      SL CYCLES      20 CONTOUR DATA

## CONTOUR DATA (cycle 20)

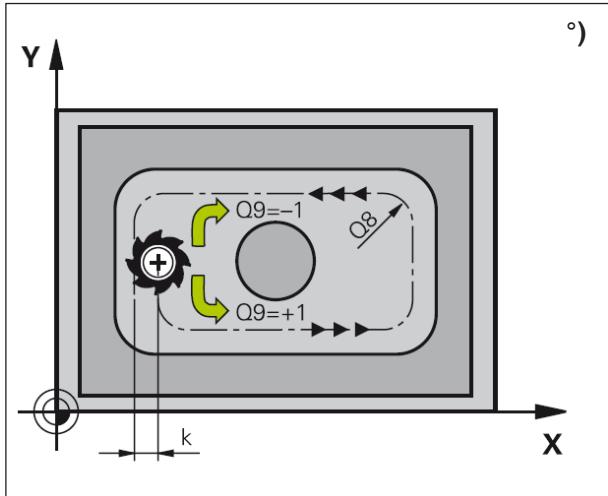


Parameter	Description	Unit
Q1	Cutting depth (incremental): Distance work piece surface - pocket bottom	mm
Q2	Path overlap factor Q2: Q2 x tool radius equals the side infeed k	
Q3	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q4	Bottom finishing allowance (incremental): Finishing allowance for the bottom.	mm
Q5	Work piece surface coordinate (absolute)	mm
Q6	Safety distance (incremental): Distance between work piece front side and work piece surface.	mm
Q7	Safe height (absolute): Absolute height, where no collision with the work piece is possible (for intermediate positioning and retraction at the end of the cycle).	mm
Q8	Inner rounding radius: Rounding radius at inside "corners": The specified value relates to the tool centre point path and is used in order to achieve smoother traverse movements between contour elements. <b>Note:</b> Q8 is not a radius which WinNC inserts between programmed elements as a separate contour element.	mm
Q9	Direction of rotation: Machining direction for pockets <ul style="list-style-type: none"><li>• -1 = up-cut for pocket and island</li><li>• +1 = down-cut for pocket and island</li></ul>	

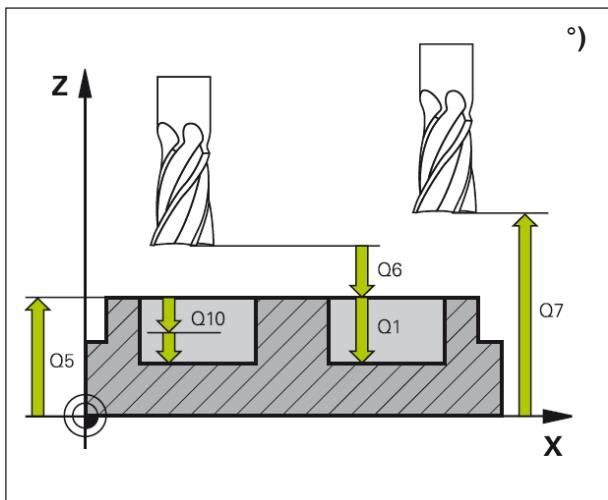
**Note:**

The machining parameters can be checked during a program interruption and overwritten, if needed.



**Cycle description**

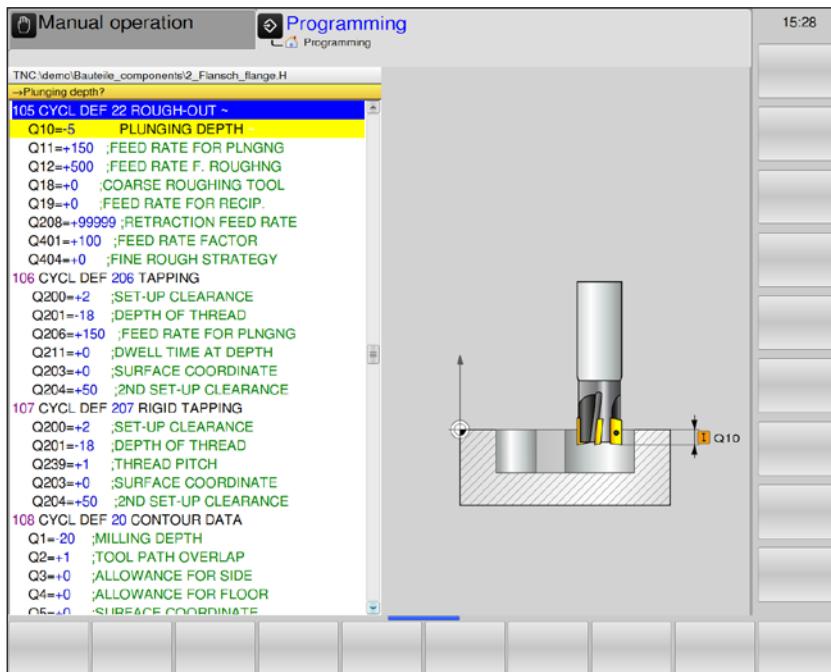
In cycle 20, machining information for the subprograms with the subcontours is specified.

**Note:**

Prior to programming, please take note:  
Cycle 20 is DEF-active, which means that it is effective in the machining program as soon as it is defined. The sign of the cycle parameter depth determines the working direction.  
A negative sign means:  
Machining in the direction of the negative spindle axis.  
If depth = 0 is programmed, the cycle will not be executed.  
Information specified in cycle 20 is valid for cycles 21 to 24. If you are using SL cycles in Q parameter programs, the parameters Q1 to Q19 must not be used as program parameters.



## CLEARING (cycle 22)

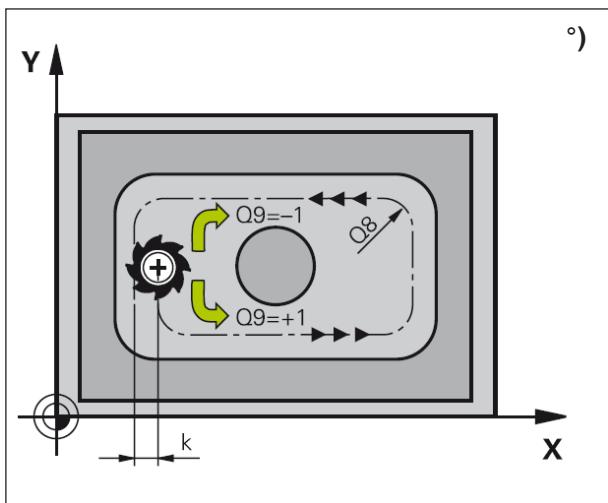


Parameter	Description	Unit
Q10	Infeed depth (incremental): dimension, by which the tool is fed in respectively.	mm
Q11	Feed for depth infeed: Feed during movements in the spindle axis	mm/min
Q12	Milling feed: Feed during movements in the machining plane	mm/min
Q18	Coarse roughing tool: Number or name of tool that was already used by WinNC for coarse roughing. If no coarse roughing took place, enter "0". If a number is entered here, WinNC will only ream the part that could not be machined by the coarse roughing tool. (Only available for contours without islands.)	
Q19	Reciprocation feed	mm/min
Q208	Retraction feed: Traverse speed of the tool during retraction after machining. With input =0: The tool retracts with feed Q12	mm/min
Q401	Feed factor: Percentage factor to which WinNC reduces the machining feed Q12, as soon as the tool starts moving with the full extent in the material during clearing. If this feed reduction is used, the clearing feed can be defined such that during the path overlap Q2 defined in cycle 20 optimal cutting conditions prevail. WinNC reduces the feed in transition areas or bottlenecks according to the way it has been defined so that the overall machining time should be shorter,	%

Parameter	Description	Unit
Q404	<p>Fine roughing strategy: defines how WinNC should move the tool during fine roughing when the radius of the fine-roughing tool is larger than half the coarse roughing tool.</p> <ul style="list-style-type: none"> <li>• Q404 = 0: WinNC moves the tool between the fine roughing areas at the current depth along the contour.</li> <li>• Q404 = 1: WinNC moves the tool between the fine roughing areas back to the safety distance and subsequently moves to the starting point of the next fine roughing area.</li> </ul>	mm

**Note:**

Currently, Q18, Q208, Q401 and Q404 cannot be changed, you can only work with the default values. If external programs are loaded, which work according to other values for Q18, Q208, Q401 and Q404, WinNC will automatically switch to the default values.

**Cycle description**

- 1 WinNC positions the tool above the cutter infeed point in consideration of the side finishing allowance.
- 2 In the first infeed depth, the tool cuts the contour from the inside to the outside at milling feed Q12.
- 3 In doing so, the island contours (here: C/D) with an approach to the pocket contour (here: A/B) are cleared out.
- 4 Subsequently, WinNC moves the tool to the next infeed depth and repeats the roughing operation until the programmed depth is reached.
- 5 Finally, the tool moves in the tool axis back to the safe height or to that position which was programmed last before the cycle.

**Note:**

Before cycle 22 can be called, further cycles need to be programmed:

- cycle 14 CONTOUR
- cycle 20 CONTOUR DATA
- possibly cycle 21 PRE-DRILLING





**Danger of collision:**

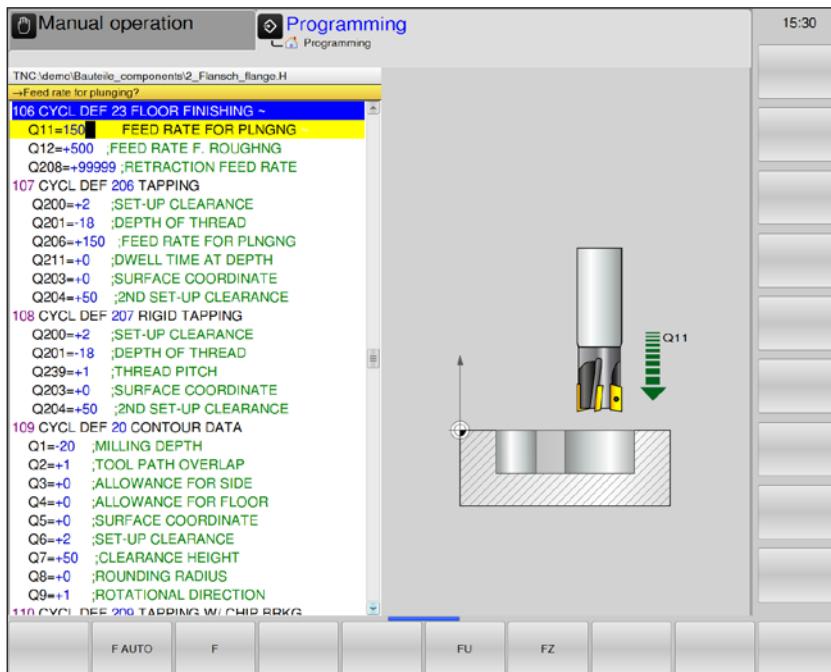
After conducting an SL cycle, the first traverse movement in the machining plane must be programmed with both coordinate values.  
Example: L X+80 Y+0 R0 FMAX

**Instructions fro programming**

- 1 Use a centre-cut end mill (DIN 844) or pre-drilling with cycle 21.
- 2 The plunging behaviour for cycle 22 is determined by the parameter Q19 and by the specifications under ANGLE and LCUTS in the tool table.
  - If Q19=0: WinNC plunges the tool perpendicularly, even if a plunging angle (ANGLE) is defined for the active tool.
  - If ANGLE =90°: WinNC plunges the tool perpendicularly. The reciprocation feed Q19 is used as plunging feed.
  - If the reciprocation feed Q19 is defined in cycle 22 and ANGLE is defined between 0.1 and 98.999 in the tool table, the tool plunges at the defined ANGLE in a helical line.
  - If the reciprocation feed Q19 is defined in the cycle 22 and no ANGLE is specified in the tool table, the control unit issues an error message.
- 3 In case of pocket contours with sharp inner corners, when using an overlap factor >1, residual material can remain during clearing. In particular, check the innermost path by means of a test graphic and slightly change the overlap factor, if needed. In this way, a different cut division can be achieved, in order to arrive at the desired result.



## FINISHING DEPTH (cycle 23)

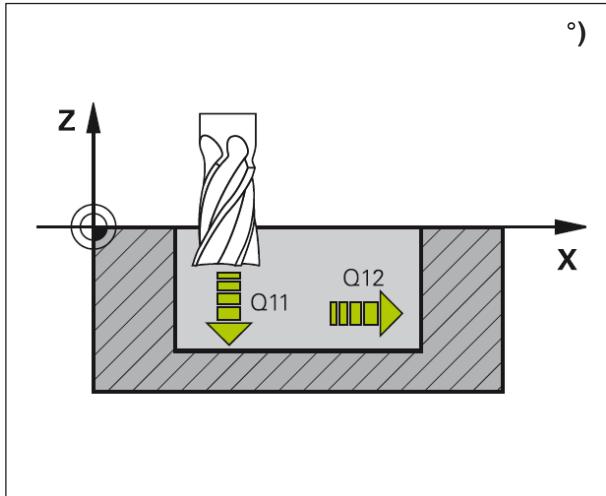


Parameter	Description	Unit
Q11	Feed for depth infeed: Tool traverse speed during plunging.	mm/min
Q12	Milling feed: Feed during movements in the machining plane	mm/min
Q208	Retraction feed: Traverse speed of the tool during retraction after machining. With input =0: The tool retracts with feed Q12	mm/min

**Note:**

Currently, Q208 cannot be changed, you can only work with the default value.

If external programs are loaded, which work according to other values for Q208, WinNC will automatically switch to the default value.



### Cycle description

- 1 WinNC positions the tool in rapid traverse at the safe height.
- 2 A movement in the tool axis at feed Q11 follows.
- 3 The control unit moves the tool smoothly (vertical tangential circle) to the surface that is to be machined if there is enough space. In narrow conditions, WinNC moves the tool vertically to the depth.
- 4 The finishing allowance that was left during clearing is removed.
- 5 Finally, the tool moves in the tool axis back to the safe height or to that position which was programmed last before the cycle.

#### Note:

Before cycle 23 can be called, further cycles need to be programmed:

- cycle 14 CONTOUR
- cycle 20 CONTOUR DATA
- possibly cycle 21 PRE-DRILLING
- possibly cycle 22 CLEARING

#### Danger of collision:

After conducting an SL cycle, the first traverse movement in the machining plane must be programmed with both coordinate values.  
Example: L X+80 Y+0 R0 FMAX

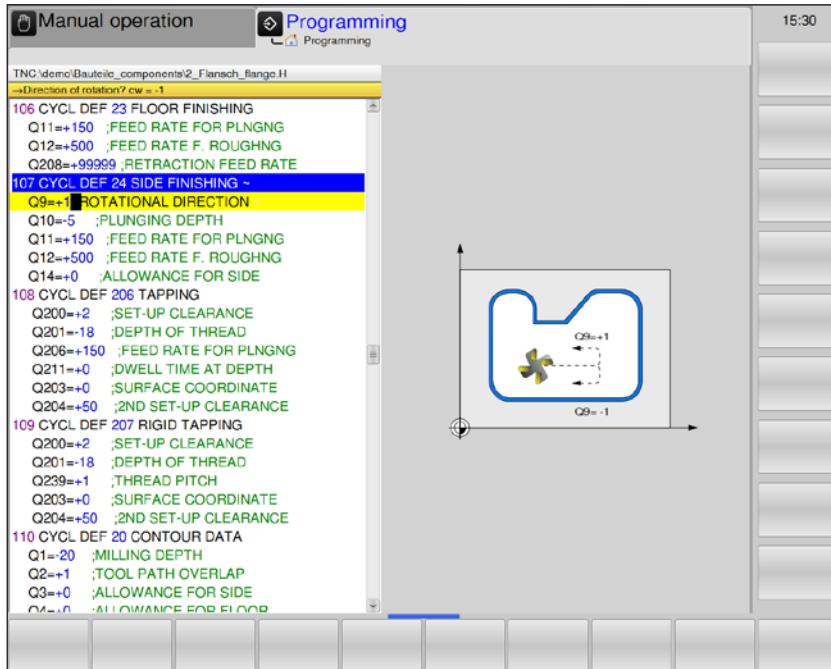


### Instructions for programming

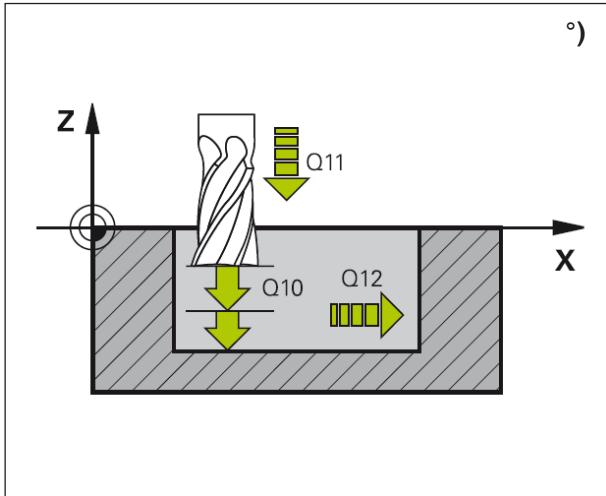
- 1 The control unit independently determines the starting point for finishing depth. The starting point depends on the spatial conditions in the pocket.
- 2 The approaching radius for pre-positioning to the final depth is internally permanently defined and dependent on the plunging angle of the tool.



## FINISHING SIDE (cycle 24)



Parameter	Description	Unit
Q9	Direction of rotation: Machining direction <ul style="list-style-type: none"> <li>+1: Rotation counterclockwise.</li> <li>-1: Rotation clockwise</li> </ul>	
Q10	Infeed depth (incremental): Dimension by which the tool is fed in respectively.	mm
Q11	Feed for depth infeed: Tool traverse speed during plunging.	mm/min
Q12	Milling feed: Feed during movements in the machining plane	mm/min
Q14	Side finishing allowance (incremental): The side allowance Q14 remains standing after finishing. This allowance must be smaller than the allowance in cycle 20.	mm



### Cycle description

- 1 WinNC positions the tool above the work piece on the starting point of the approach position. This position results from a tangential circular path on which WinNC guides the tool to the contour.
- 2 Subsequently, the control unit moves the tool to the first infeed depth at the feed of the depth infeed.
- 3 WinNC approaches the contour smoothly, until the whole contour is finished. Every subcontour is finished separately.
- 4 Finally, the tool moves in the tool axis back to the safe height or to that position which was programmed last before the cycle.

#### Danger of collision:

After conducting an SL cycle, the first traverse movement in the machining plane must be programmed with both coordinate values.  
Example: L X+80 Y+0 R0 FMAX



#### Note:

Before cycle 23 can be called, further cycles need to be programmed:

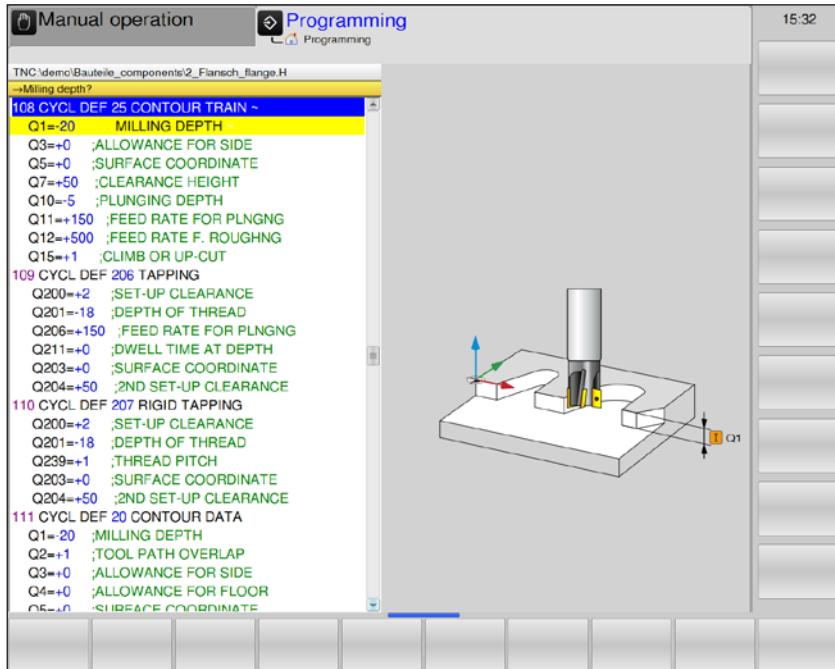
- cycle 14 CONTOUR
- cycle 20 CONTOUR DATA
- possibly cycle 21 PRE-DRILLING
- possibly cycle 22 CLEARING

### Instructions fro programming

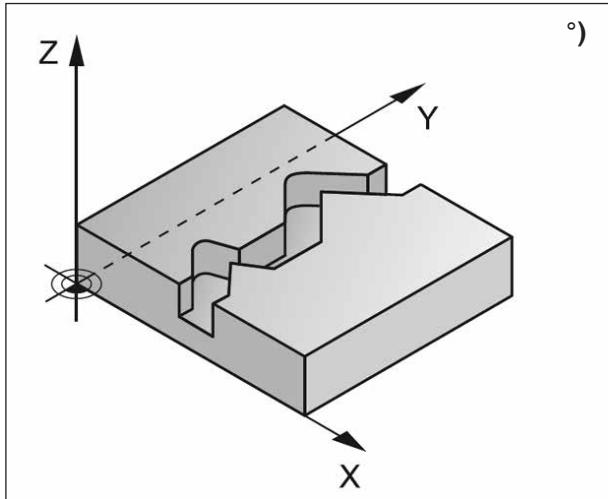
- 1 The sum total of finishing allowance (Q14) and finishing tool radius must be smaller than the sum total of side finishing allowance (Q3 cycle 20) and reaming tool radius.
- 2 If no allowance is programmed in cycle 20, the control unit issues the error message "Tool radius too big".
- 3 The side allowance Q14 remains standing after finishing; it must be smaller than the allowance in cycle 20.
- 4 If cycle 24 is executed without prior reaming with cycle 22, the above calculation applies. The reaming tool radius has then the value "0".
- 5 Cycle 24 can be used also for contour milling. Then
  - the contour to be milled must be defined as an individual island (without pocket limitation) and
  - the finishing allowance (Q3) in cycle 20 must be specified bigger than the sum total of finishing allowance Q14 + radius of the used tool.
- 6 The control unit independently determines the starting point for finishing. The starting point depends on the spacial conditions in the pocket and the allowance programmed in cycle 20.
- 7 The control unit calculates the starting point also in dependence from the order of processing. If the finishing cycle is selected via the GOTO key and the program is started, the starting point can be at a different position than if the program was to be processed according to the defined order.



## CONTOUR PULL (cycle 25)



Parameter	Description	Unit
Q1	Cutting depth (incremental): Distance between work piece surface and contour bottom:	mm
Q3	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q5	Work piece surface coordinate (absolute)	mm
Q7	Safe height (absolute): Absolute height, where no collision with the work piece is possible; for intermediate positioning and retraction at the end of the cycle.	mm
Q10	Infeed depth (incremental): Dimension by which the tool is fed in.	mm
Q11	Feed for depth infeed: Feed during movements in the spindle axis.	mm/min
Q12	Milling feed: Feed during movements in the machining plane.	mm/min
Q15	Type of milling: Machining direction • +1: Down-cut Milling • -1: Up-cut milling Input 0: Alternating up-cut and down-cut milling with multiple infeeds.	



### Cycle description

This cycle, along with cycle 14 CONTOUR can be used for machining of open and closed contours: Beginning and end of the contour are not identical.

Cycle 25 CONTOUR PULL provides significant advantages compared to machining of a contour with positioning blocks:

- WinNC monitors the machining with regard to undercuts and contour damage. The contour must be checked with the test graphic.
- If the tool radius is too big, possibly the contour has to be refinished at the inner corners.
- Machining can be executed continuously in up-cut or down-cut machining. The type of milling is maintained, even if the contours are mirrored.
- In case of multiple infeeds, WinNC can move the tool back and forth: This reduces the machining time.
- You can specify allowances, in order to execute roughing and finishing in multiple work steps.

#### Danger of collision:



- After cycle 25, no chain measurements must be programmed, as these refer to the tool position at the end of the cycle.
- Move to a defined (absolute) position in all main axes, as the tool position at the end of the cycle does not match the tool position at the beginning of the cycle.

#### Note:

Prior to programming, please take note:  
The sign of the cycle parameter depth determines the working direction.

A negative sign means:

Machining in the direction of the negative spindle axis.

If depth = 0 is programmed, the cycle will not be executed.

WinNC takes only the first label from the cycle 14 CONTOUR into account. Cycle 20 CONTOUR DATA is not needed.



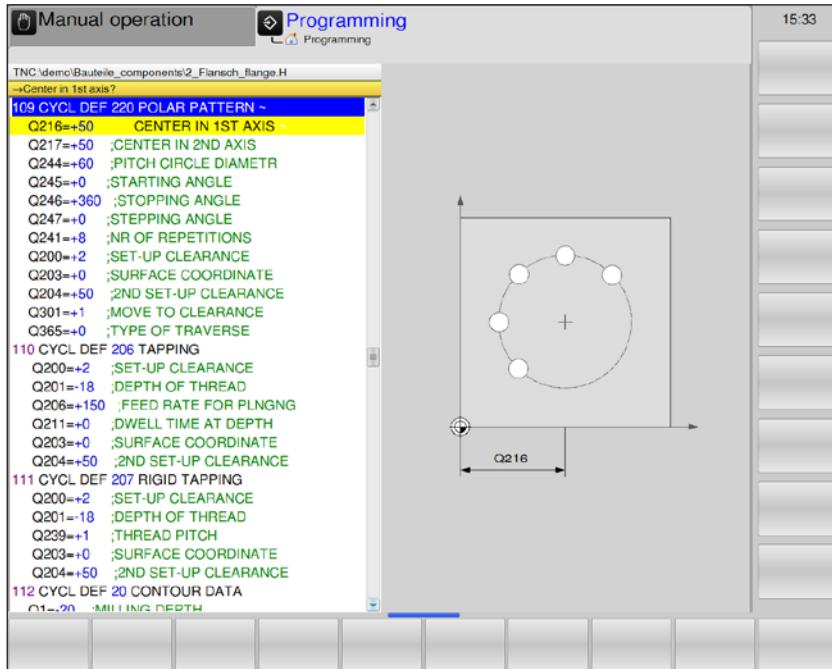
## Dot patterns

PATTERN

- 220 Pattern on circle
- 221 Pattern on line



## PATTERN ON CIRCLE (cycle 220)



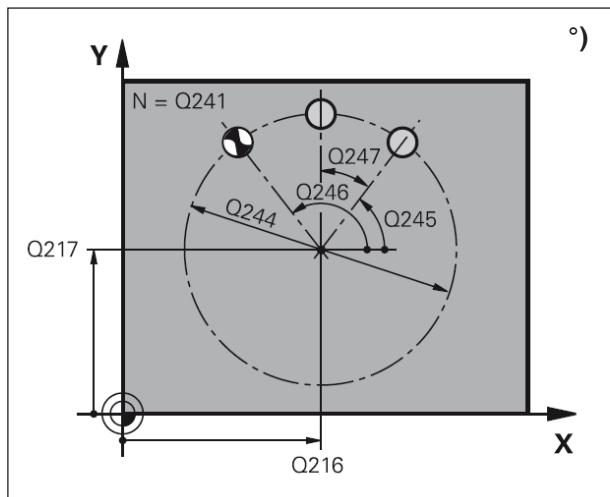
Parameter	Description	Unit
Q216	Centre 1st axis (absolute): Pitch circle centre in the main axis of the machining plane	mm
Q217	Centre 2nd axis (absolute): Pitch circle centre in the minor axis of the machining plane	mm
Q244	Pitch circle diameter	mm
Q245	Start angle (absolute): Angle between the main axis of the machining plane and the starting point of the first machining operation on the pitch circle.	degrees
Q246	End angle (absolute): Angle between the main axis of the machining plane and the starting point of the last machining operation on the pitch circle (does not apply to full circles). The end angle must be different from the start angle. If the end angle is bigger than the start angle, machining will be done counterclockwise. Otherwise, machining will be done clockwise.	degrees
Q247	Angle step (incremental): Angle between two machining operations on the pitch circle. If angle step =0, it is calculated by WinNC from the start angle, the end angle and the number of machining operations.	degrees
Q241	Number of machining operations on the pitch circle	
Q200	Safety distance (incremental): Distance between tool point and work piece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece can occur.	mm

Parameter	Description	Unit
Q301	Move on safe height: Determine, how the tool should move between the machining operations: <ul style="list-style-type: none"><li>• 0: Move to the safety distance between the machining operations</li><li>• 1: Move between the machining operations to 2nd safety distance</li></ul>	
Q365	Type of movement? Straight line or circle: Determine, according to which path function the tool should move between the machining operations: <ul style="list-style-type: none"><li>• 0: Move on a straight line between the machining operations.</li><li>• 1: Move circularly on the pitch circle diameter between the machining operations.</li></ul>	

**Note:**

Currently, Q365 cannot be changed, you can only work with the default value.

If external programs are loaded, which work according to other values for Q365, WinNC will automatically switch to the default value.

**Cycle description**

1 WinNC positions the tool in rapid traverse from the current position to the starting point of the first machining operation.

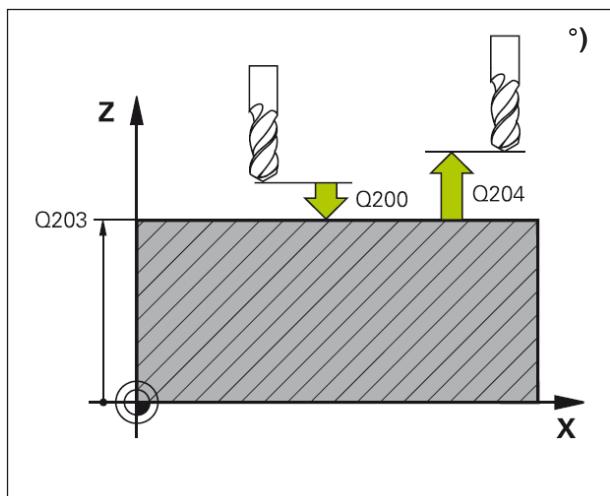
Order:

- 2. Approach safety distance (spindle axis).
- Approach starting point in the machining plane.
- Move to the safety distance above work piece surface (spindle axis).

2 From this position, WinNC executes the last defined machining cycle.

3 Subsequently, WinNC positions the tool with a straight line movement on the starting point of the next machining operation. The tool is positioned at the safety distance (or 2nd safety distance).

4 The procedure 1 to 3 is repeated, until all machining operations are executed.

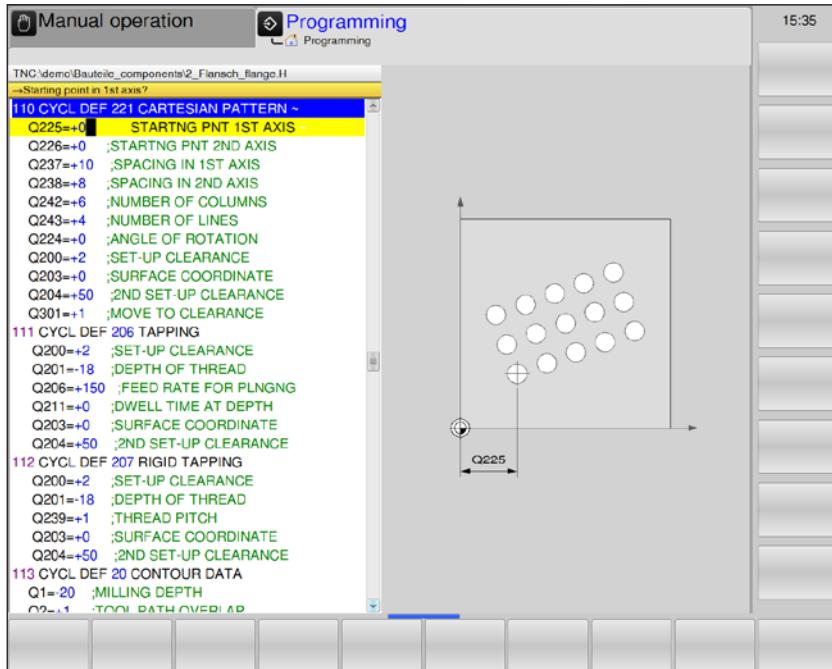
**Note:**

Prior to programming, please take note:  
Cycle 220 is DEF-active, which means that cycle 220 automatically calls the last defined machining cycle. If you combine one of the machining cycles 200 to 209 and 251 to 267 with cycle 220, the safety distance, the work piece surface and the 2nd safety distance from cycle 220 are effective.

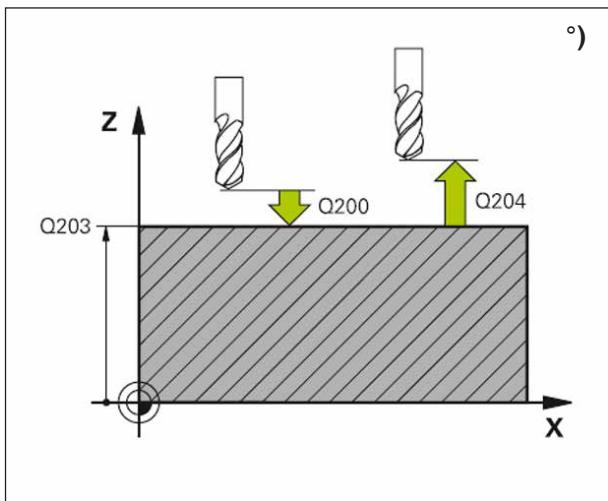
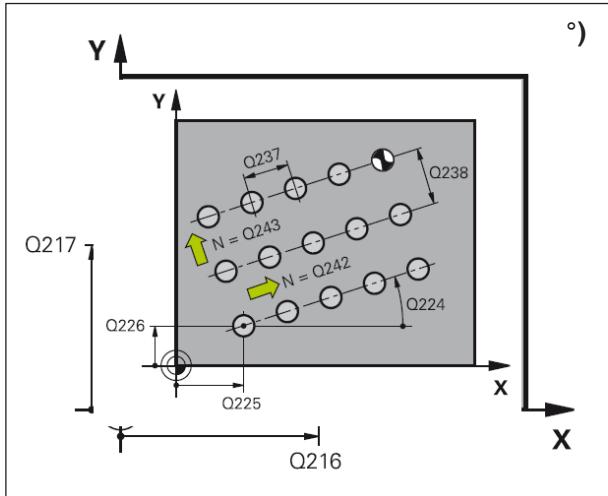




## PATTERN ON LINES (cycle 221)



Parameter	Description	Unit
Q225	Starting point 1st axis (absolute): Coordinate of the starting point in the main axis of the machining plane.	mm
Q226	Starting point 2nd axis (absolute): Coordinate of the starting point in the minor axis of the machining plane.	mm
Q237	Distance 1st axis (incremental): Distance of the individual points on the line.	mm
Q238	Distance 2nd axis (incremental): Distance of the individual lines from each other.	mm
Q242	Number of columns: Number of machining operations on the line	
Q243	Number of lines	
Q224	Turning position (absolute): Angle, about which the whole arrangement picture is turned. The centre of rotation lies in the starting point.	degrees
Q200	Safety distance (incremental): Distance between tool point and work piece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece can occur.	mm
Q301	Move on safe height: Determine, how the tool should move between the machining operations: <ul style="list-style-type: none"> <li>• 0: Move to the safety distance between the machining operations</li> <li>• 1: Move between the machining operations to 2nd safety distance</li> </ul>	



### Cycle description

1 WinNC positions the tool in rapid traverse from the current position to the starting point of the first machining operation.

Order:

- 2. Approach safety distance (spindle axis).
  - Approach starting point in the machining plane.
  - Move to the safety distance above work piece surface (spindle axis).
- 2 From this position, WinNC executes the last defined machining cycle.
- 3 Subsequently, WinNC positions the tool in positive direction of the main axis on the starting point of the next machining operation. The tool is positioned at the safety distance (or 2nd safety distance).
- 4 The procedure 1 to 3 is repeated, until all machining operations are executed.
- 5 After that, WinNC moves the tool to the last point of the second line and executes the machining operation there.
- 6 From there, WinNC positions the tool in negative direction of the main axis on the starting point of the next machining operation.
- 7 The procedure 6 is repeated, until all machining operations of the second line are executed.
- 8 Subsequently, WinNC moves the tool to the starting point of the next line.
- 9 All further lines are machined in a reciprocal movement.

#### Note:

Prior to programming, please take note:  
Cycle 221 is DEF-active, which means that cycle 221 automatically calls the last defined machining cycle. If you combine one of the machining cycles 200 to 209 and 251 to 267 with cycle 221, the safety distance, the work piece surface, the 2nd safety distance and the turning position from cycle 221 are effective.



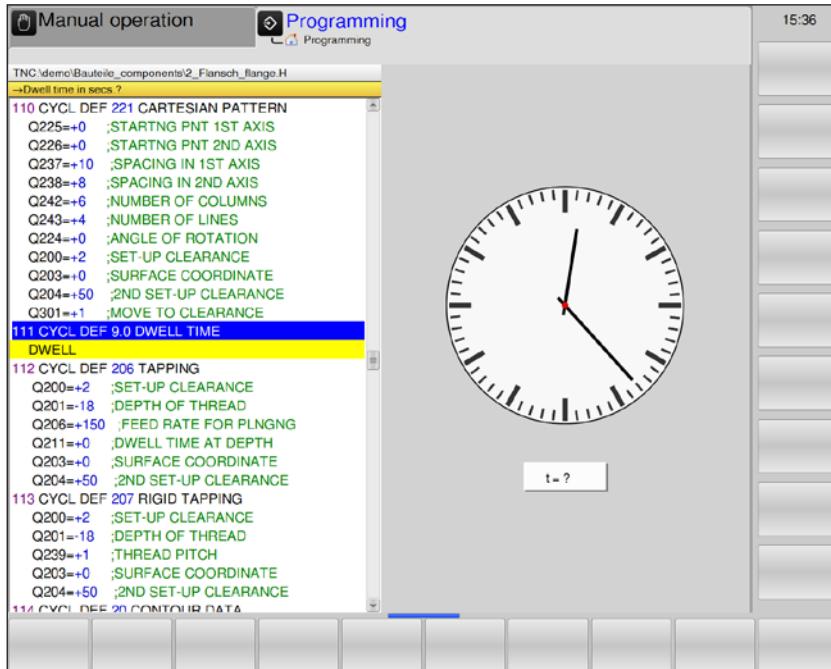
## Special cycles

SPECIAL  
CYCLES

- 9 Dwell time
- 12 PGM CALL



## DWELL TIME (cycle 9)



Parameter	Description	Unit
	Enter dwell time in seconds	s

### Cycle description

The program run is stopped for the duration of the DWELL TIME. For example, the dwell time can be used for chips crushing.

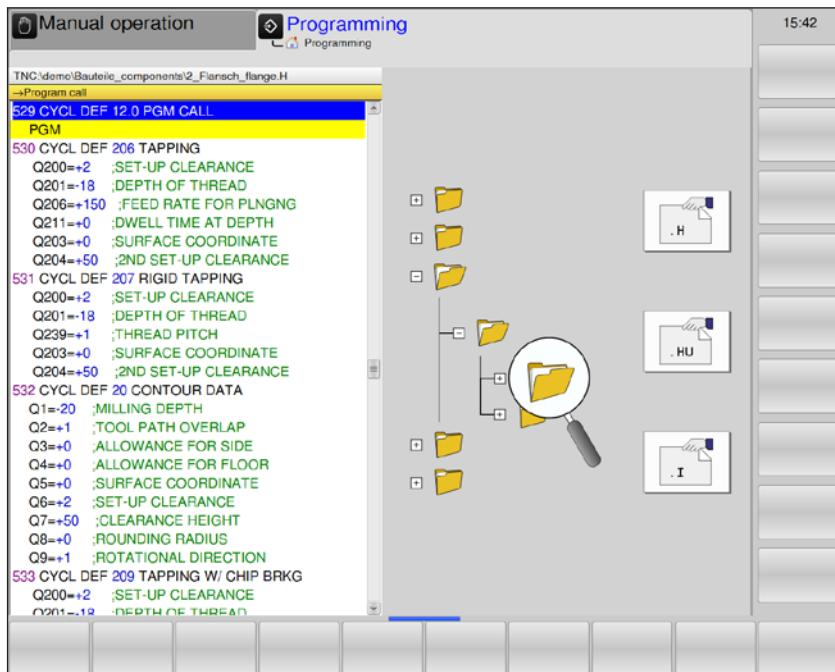
The cycle is effective from the moment of its definition in the program. Modal acting (permanent) conditions are not influenced by it, like, for example, the spindle rotation.

- **Dwell time in seconds:** Enter dwell time in seconds

Input range 0 to 3 600 s (1 hour) in 0.001 second steps

CYCL DEF      SPECIAL CYCLES      12 PGM CALL

## PGM CALL (cycle 12)



Parameter	Description	Unit
	Enter program name and extension	

### Cycle description

If the program to be called is in the same directory as the currently processed program, it is enough to enter the program name and the extension H.

Otherwise, the program name and the whole path must be specified:

TNC:\... \ ...



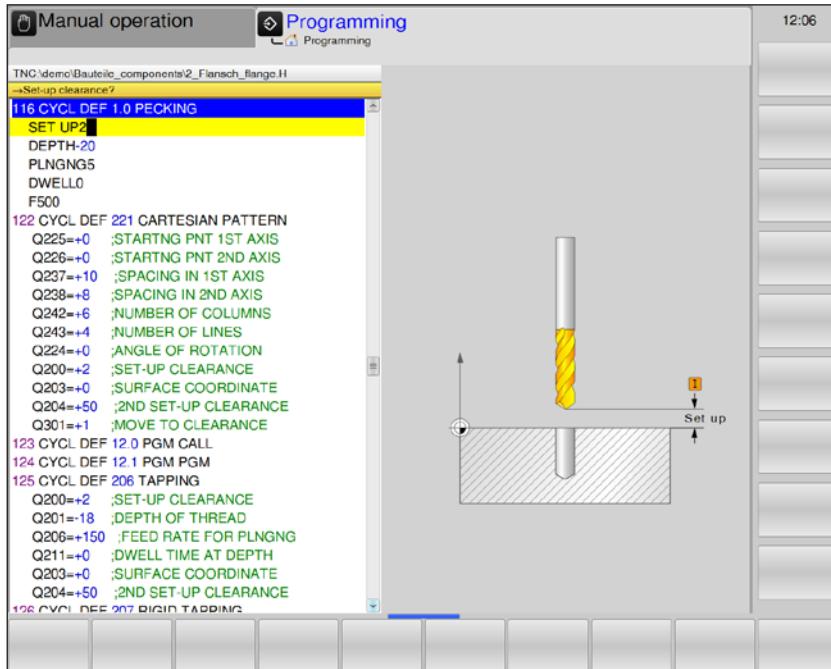
## Old Cycles

OLD  
CYCLS

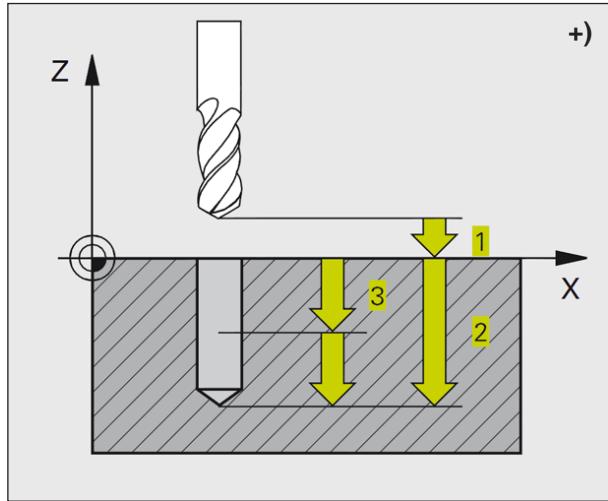
- 1 Deep hole drilling
- 2 Thread tapping
- 17 Rigid tapping
- 3 Slot milling
- 4 Pocket milling
- 5 Circular pocket
- 212 Finishing pocket
- 213 Finishing stud
- 214 Finishing circular pocket
- 215 Finishing circular stud
- 210 Slot reciprocal
- 211 Round slot
- 230 Multipass milling
- 231 Ruled surface



## DEEP DRILLING (cycle 1)



Parameter	Description	Unit
DIST	Safety distance <b>1</b> (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Drilling depth <b>2</b> (incremental): Distance between work piece surface - hole bottom (= drill taper tip).	mm
INF.DT	Infeed depth <b>3</b> (incremental): Dimension by which the tool is fed in respectively. The control unit moves in one work step to the drilling depth, if: <ul style="list-style-type: none"> <li>• infeed depth and depth are identical</li> <li>• the infeed depth is bigger than the drilling depth</li> </ul>	mm
D.TIME	Dwell time in seconds: Time duration, during which the tool dwells at the hole bottom to cut clear.	s
F	Feed F: Tool traverse speed during drilling.	mm/min



### Cycle description

- 1 The tool drills at the entered feed F from the current position to the first infeed depth.
- 2 After that, WinNC moves the tool back at rapid traverse FMAX to the first infeed depth reduced by the advance stop distance t.
- 3 The control unit determines the advance stop distance independently:
  - Drilling depth up to 30 mm:  $t = 0.6 \text{ mm}$
  - Drilling depth over 30 mm:  $t = \text{drilling depth}/50$
  - Maximum advance stop distance: 7 mm
- 4 Subsequently, the tool drills at the programmed feed F by a further infeed depth.
- 5 WinNC repeats the sequence 1 to 4, until the entered drilling depth is reached.
- 6 At the hole bottom, WinNC retracts the tool, after dwell time to cut clear, at FMAX to the starting position.

### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. Program the positioning block on the starting point in the spindle axis (safety distance above work piece surface).

The sign of the cycle parameter depth determines the working direction.

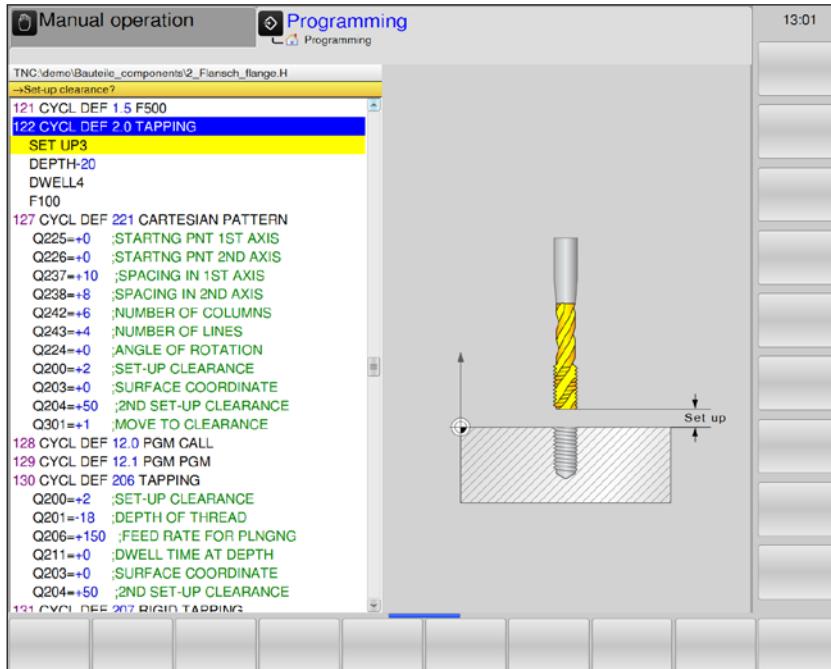
A negative sign means:

Machining in the direction of the negative spindle axis.

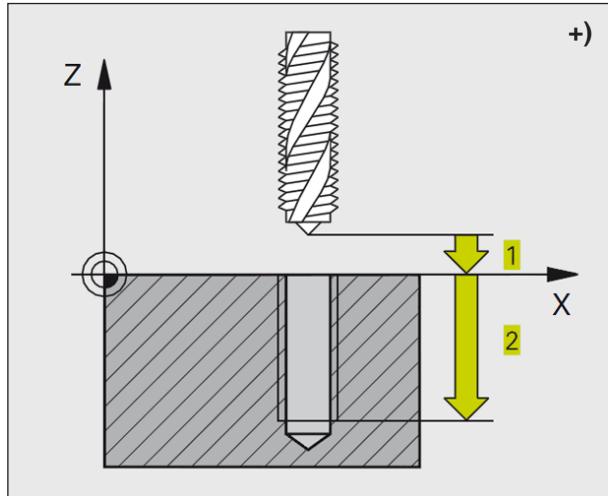
If depth = 0 is programmed, the cycle will not be executed.



## THREAD TAPPING (cycle 2)



Parameter	Description	Unit
DIST	Safety distance <b>1</b> (incremental): Distance between tool point (= starting position) and work piece surface. Guideline value: 4x thread pitch	mm
DEPTH	Drilling depth <b>2</b> (thread length, incremental): Distance work piece surface - thread end.	mm
D.TIME	Dwell time in seconds: Enter a value between 0 and 0.5 seconds, in order to prevent wedging of the tool during retraction.	s
F	Feed F: Tool traverse speed during thread tapping.  Feed calculation: $F = S \times p$ F Feed (mm/min) S: Spindle speed (rpm) p: Thread pitch (mm)	mm/min



### Cycle description

- 1 The tool moves to the drilling depth in one work step.
- 2 After that, the spindle rotation direction is reversed and, after the dwell time, the tool is retracted to the starting position.
- 3 At the starting position, the spindle rotation direction is again reversed.

#### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. Program the positioning block on the starting point in the spindle axis (safety distance above work piece surface).

The sign of the cycle parameter depth determines the working direction.

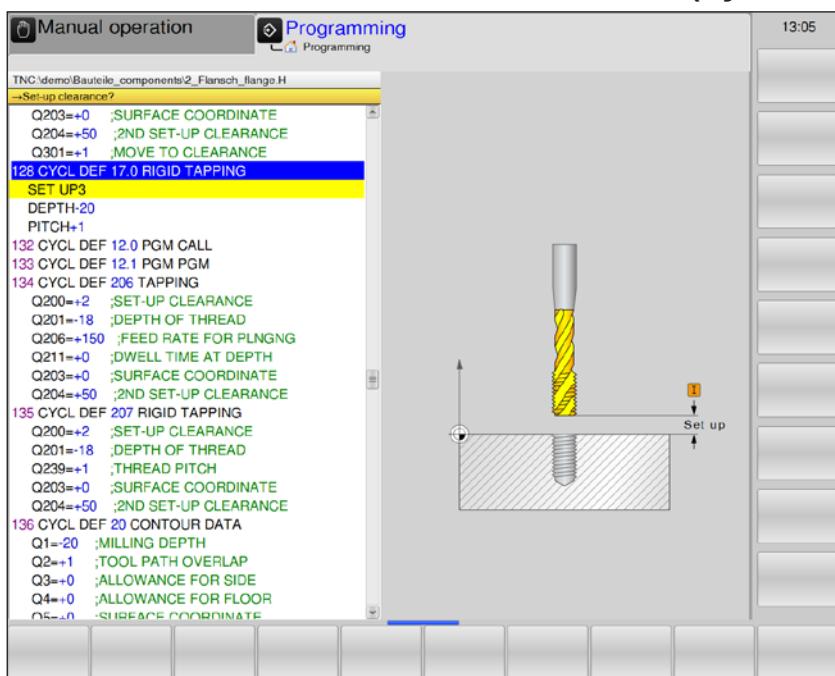
A negative sign means:

Machining in the direction of the negative spindle axis.

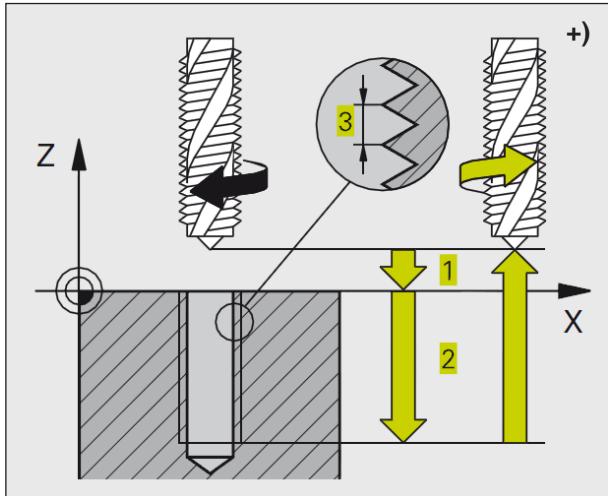
If depth = 0 is programmed, the cycle will not be executed.

The tool must be clamped in a length compensation chuck. The length compensation chuck compensates tolerances of feed and speed during machining. While the cycle is executed, the knob for speed override is deactivated. For right-hand thread, activate the spindle via M3, for left-hand thread, via M4.





Parameter	Description	Unit
DIST	Safety distance <b>1</b> (incremental): Distance between tool point (= starting position) and work piece surface. Guideline value: 4x thread pitch	mm
DEPTH	Thread depth <b>2</b> (incremental): Distance work piece surface (thread start) - thread end.	mm
PITCH	Thread pitch <b>3</b> : Sign determines right-hand or left-hand thread: • + = Right-hand thread • - = Left-hand thread	mm



### Cycle description

WinNC cuts the thread either in one or in multiple work steps without a length compensation chuck.

Advantages compared to the cycle thread tapping with compensating chuck:

- Higher machining speed
- The same thread can be repeated, as the spindle aligns to the 0° position at cycle call.
- Larger traverse range of the spindle axis, as the compensating chuck is dropped.

### Note:

Prior to programming, please take note:  
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. Program the positioning block on the starting point in the spindle axis (safety distance above work piece surface).

The sign of the cycle parameter depth determines the working direction.

A negative sign means:

Machining in the direction of the negative spindle axis.

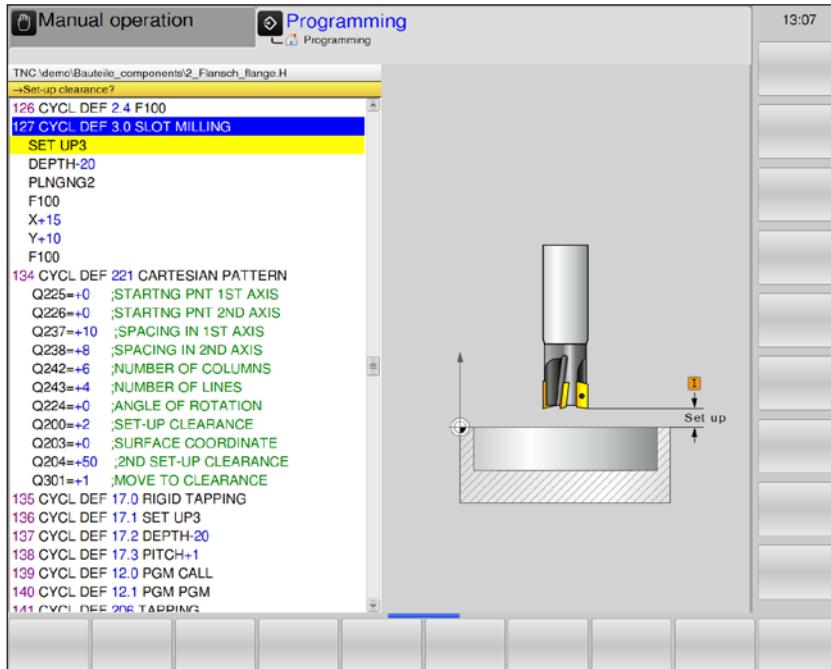
If depth = 0 is programmed, the cycle will not be executed.

WinNC calculates the feed as a function of the speed. If you actuate the knob for speed override during thread tapping, WinNC will automatically adjust the feed. The knob for the feed override is not active. At the end of the cycle, the spindle stops. Switch on the spindle via M3 (and/or M4), prior to the next machining operation.

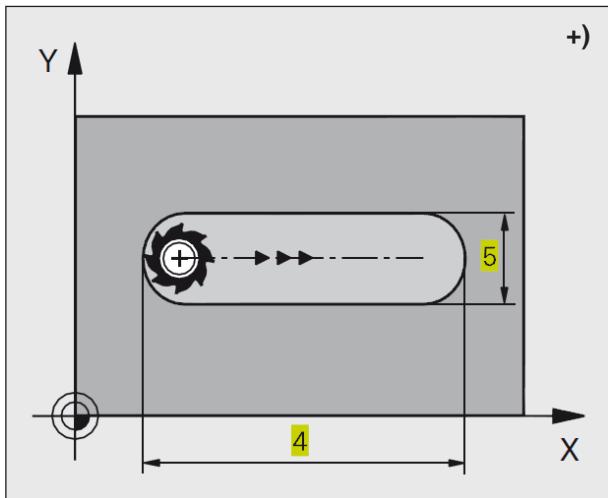
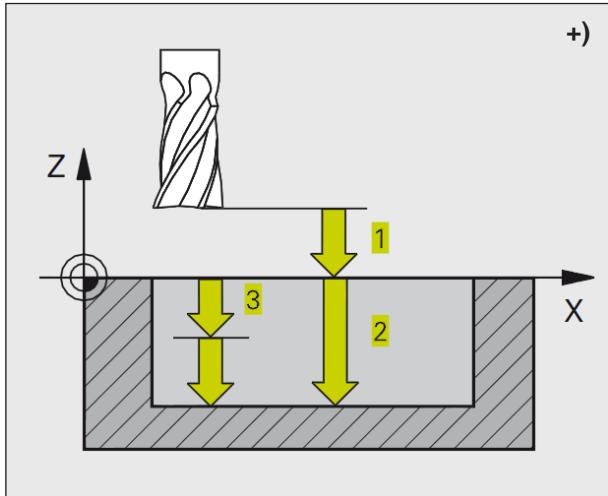




## SLOT MILLING (cycle 3)



Parameter	Description	Unit
DIST	Safety distance <b>1</b> (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Cutting depth <b>2</b> (incremental): Distance work piece surface - pocket bottom.	mm
INF.DT	Infeed depth <b>3</b> : Dimension by which the tool is fed in respectively. The control unit moves in one work step to the depth, if: <ul style="list-style-type: none"><li>• Infeed depth = depth</li><li>• Infeed depth &gt; depth</li></ul>	mm
F	Feed for depth infeed: Tool traverse speed during plunging	mm/min
X	1. Side length <b>4</b> : Length of the slot. Determine the cutting direction via the algebraic sign.	mm
Y	2nd Side length <b>5</b> : Width of the slot.	mm
F	Feed: Traverse speed of the tool in the machining plane.	mm/min



### Cycle description

#### Roughing

- 1 WinNC shifts the tool by the finishing allowance (half the difference between slot width and tool diameter) toward the inside. From there, the tool cuts into the work piece and mills in longitudinal direction of the slot.
- 2 At the end of the slot, a depth infeed is executed, and the tool mills in the opposite direction. This operation is repeated until the programmed cutting depth is reached.

#### Finishing

- 3 At the milling bottom, WinNC moves the tool on a circular path tangentially toward the outer contour. After that, the contour is finished in down-cut milling (at M3).
- 4 Finally, the tool moves back to the safety distance in rapid traverse FMAX. In case of an uneven number of infeeds, the tool moves at the safety distance back to the starting position.

#### Note:

Prior to programming, please take note:  
Use a centre-cut end mill (DIN 844) or pre-drilling at the starting point.

Pre-positioning in the centre of the slot and offset by the tool radius into the slot with radius compensation R0. Select milling cutter diameter not larger than the slot width and not smaller than half the slot width.

Program the positioning block on the starting point in the spindle axis (safety distance above work piece surface).

The sign of the cycle parameter depth determines the working direction.

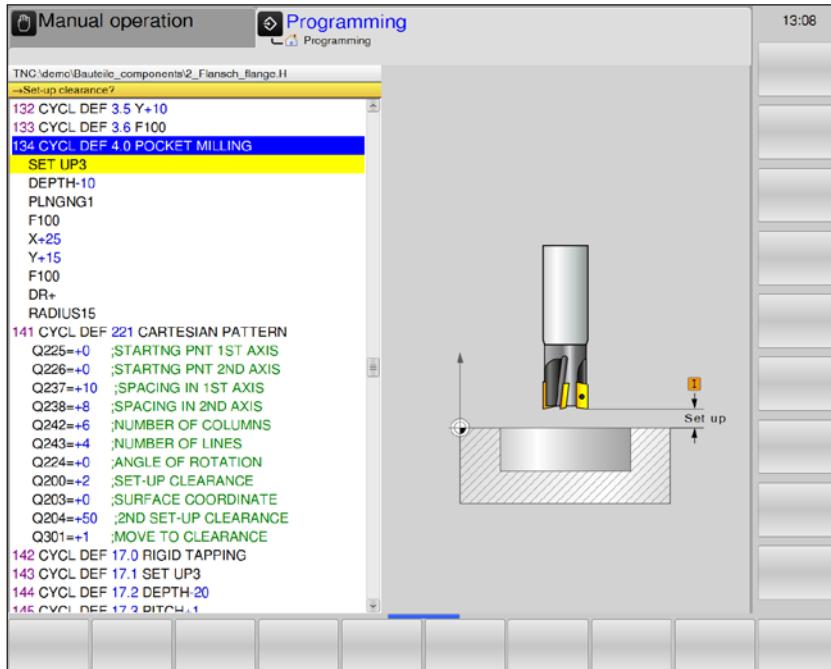
A negative sign means:

Machining in the direction of the negative spindle axis.

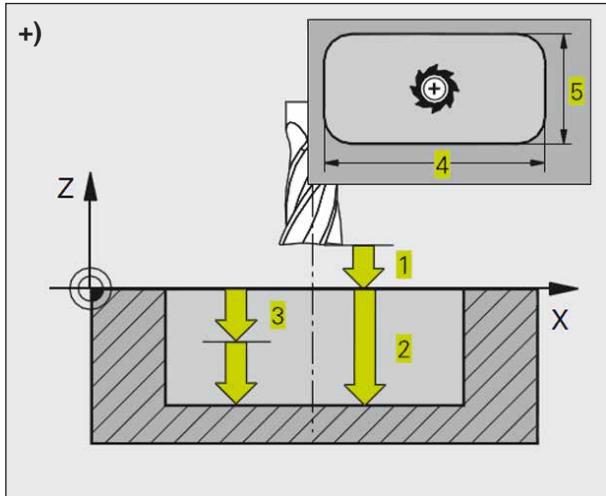
If depth = 0 is programmed, the cycle will not be executed.



## POCKET MILLING (cycle 4)



Parameter	Description	Unit
DIST	Safety distance <b>1</b> (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Cutting depth <b>2</b> (incremental): Distance work piece surface - pocket bottom.	mm
INF.DT	Infeed depth <b>3</b> : Dimension by which the tool is fed in respectively. The control unit moves in one work step to the depth, if: <ul style="list-style-type: none"> <li>• Infeed depth = depth</li> <li>• Infeed depth &gt; depth</li> </ul>	mm
F	Feed for depth infeed: Tool traverse speed during plunging:	mm/min
X	1. Side length <b>4</b> : Length of the pocket. Parallel to the main axis of the machining plane.	mm
Y	2nd Side length <b>5</b> : Width of the pocket.	mm
F	Feed: Traverse speed of the tool in the machining plane.	mm/min
DR	Rotation clockwise: <ul style="list-style-type: none"> <li>• DR + : Down-cut milling at M3</li> <li>• DR - : Up-cut milling at M3</li> </ul>	
RADIUS	Rounding radius: Radius for the pocket corners. Radius =0: Rounding radius equals the tool radius.	



### Cycle description

#### Roughing

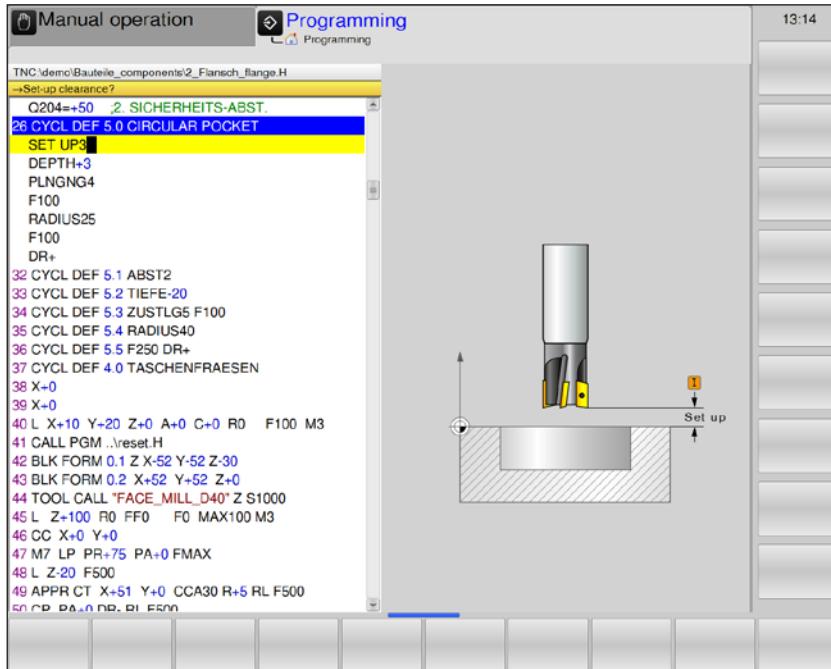
- 1 The tool cuts into the work piece at the starting position (pocket centre) and moves to the first infeed depth.
- 2 Subsequently, the tool initially moves in the positive direction of the longer side, in case of square pockets in positive Y-direction, and then clears the pocket from the inside to the outside.
- 3 This procedure 1 to 2 is repeated until the depth is reached.
- 4 At the end of the cycle, WinNC moves the tool back to the starting position.

#### Note:

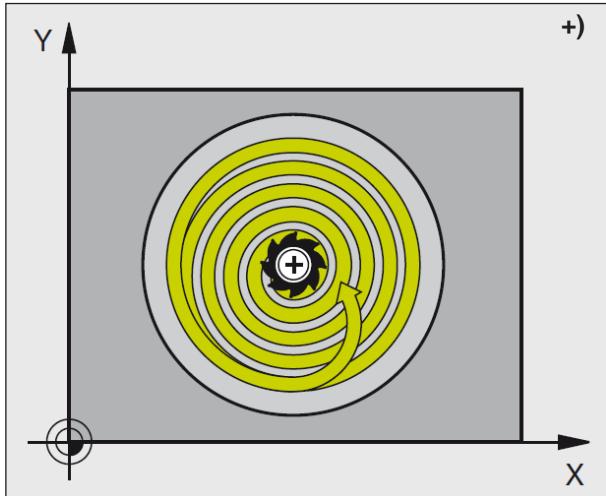
Prior to programming, please take note:  
Use a centre-cut end mill (DIN 844) or  
pre-drilling at the starting point.  
Program the positioning block on the start-  
ing point in the spindle axis (safety distance  
above work piece surface).  
The sign of the cycle parameter depth deter-  
mines the working direction.  
A negative sign means:  
Machining in the direction of the negative  
spindle axis.  
If depth = 0 is programmed, the cycle will not  
be executed.  
For the 2nd side length, the following applies:  
2nd side length larger than (2x rounding  
radius + side infeed k)



## CIRCULAR POCKET (cycle 5)



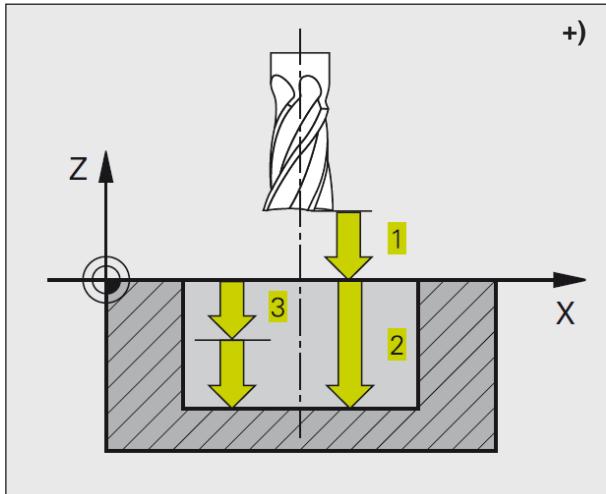
Parameter	Description	Unit
DIST	Safety distance <b>1</b> (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Cutting depth <b>2</b> (incremental): Distance work piece surface - pocket bottom.	mm
INF.DT	Infeed depth <b>3</b> : Dimension by which the tool is fed in respectively. The control unit moves in one work step to the depth, if: <ul style="list-style-type: none"><li>• Infeed depth = depth</li><li>• Infeed depth &gt; depth</li></ul>	mm
F	Feed for depth infeed: Tool traverse speed during plunging.	mm/min
RADIUS	Radius of the circular pocket	mm
F	Feed: Traverse speed of the tool in the machining plane.	mm/min
DR	Rotation clockwise: <ul style="list-style-type: none"><li>• DR + : Down-cut milling at M3</li><li>• DR - : Up-cut milling at M3</li></ul>	



### Cycle description

#### Roughing

- 1 The tool cuts into the work piece at the starting position (pocket centre) and moves to the first infeed depth.
- 2 Subsequently, the tool moves at the feed F along a circular path. Regarding side infeed k, see cycle 4 pocket milling.
- 3 This procedure 2 is repeated until the depth is reached.
- 4 At the end of the cycle, WinNC moves the tool back to the starting position.



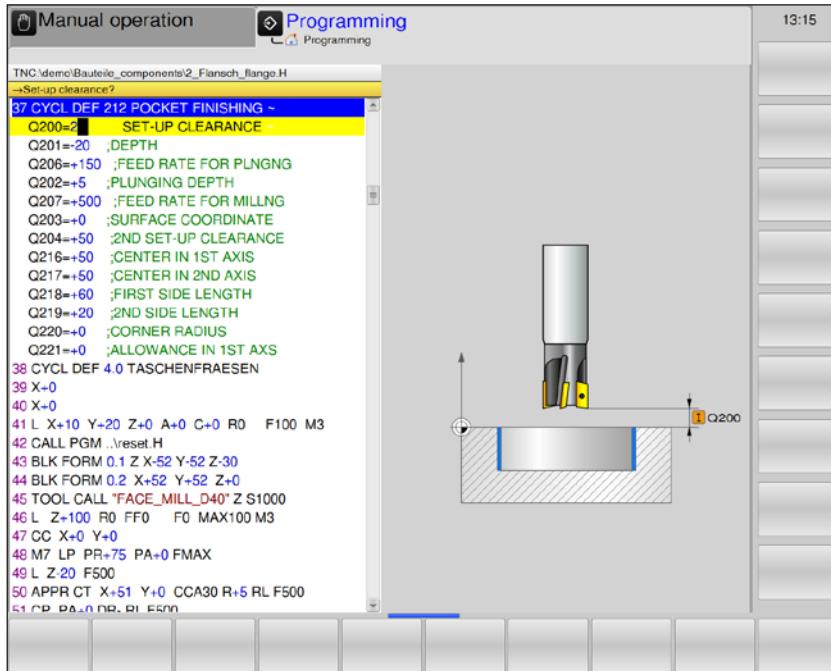
#### Note:

Prior to programming, please take note:  
 Use a centre-cut end mill (DIN 844) or  
 pre-drilling at the starting point.  
 Program the positioning block on the starting  
 point (= pocket centre) of the machining plane  
 with radius compensation R0.  
 Program the positioning block on the start-  
 ing point in the spindle axis (safety distance  
 above work piece surface).  
 The sign of the cycle parameter depth deter-  
 mines the working direction.

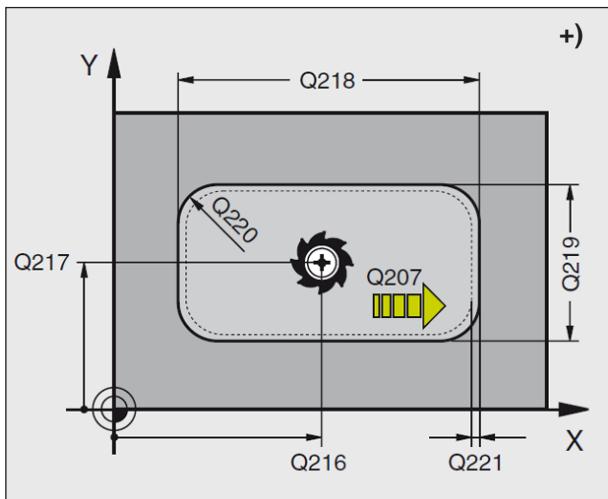
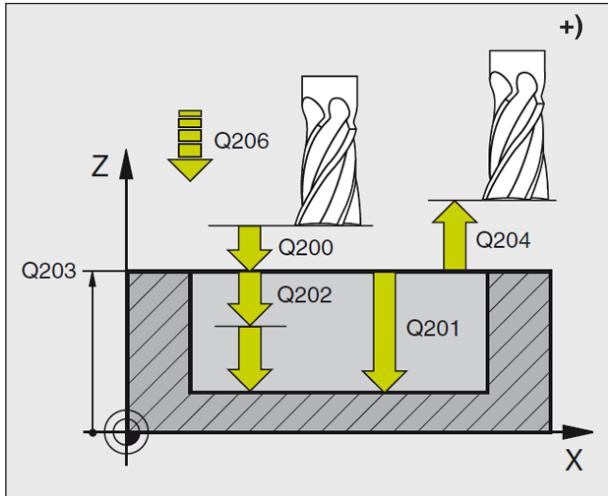


CYCL  
DEFSPECIAL  
CYCLESOLD  
CYCLES

## POCKET FINISHING (cycle 212)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance between tool point (= starting position) and work piece surface.	mm
Q201	Depth (incremental): Distance work piece surface - pocket bottom.	mm
Q206	Feed for depth infeed: Traverse speed of the tool during movement to the depth. When plunging into the material, a value smaller than the one defined in Q207 must be specified.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in respectively. Enter a positive value.	mm
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece can occur.	mm
Q216	Centre 1st axis (absolute): Centre of pocket in the main axis of the machining plane	mm
Q217	Centre 2nd axis (absolute): Centre of pocket in the minor axis of the machining plane	mm
Q218	1. Side length (incremental): Length of pocket, parallel to the main axis of the machining plane.	mm
Q219	2nd Side length (incremental): Length of pocket, parallel to the minor axis of the machining plane.	mm
Q220	Corner radius: Radius of the pocket corner. If no radius is specified, the control unit sets the corner radius to be equal to the tool radius.	mm
Q221	Allowance 1st axis (incremental): Allowance in the main axis of the machining plane, related to the length of the pocket.	mm



### Cycle description

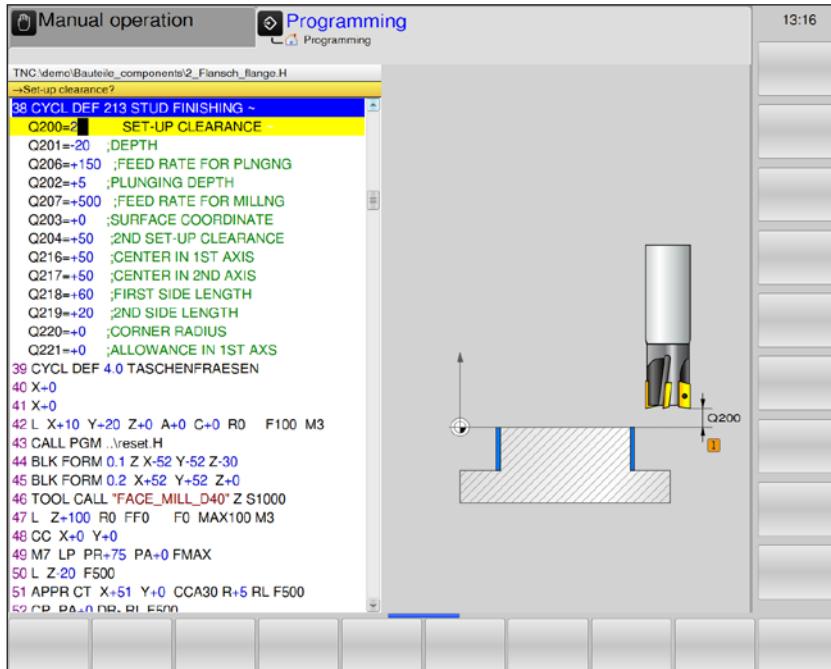
- 1 WinNC moves the tool automatically in the spindle axis to the safety distance, or, if specified, to the 2nd safety distance and subsequently to the pocket centre.
- 2 Starting from the pocket centre, the tool moves to the starting point of the machining operation in the machining plane. For the calculation of the starting point, WinNC takes the allowance and the tool radius into account. Possibly, WinNC plunges into the material in the pocket centre.
- 3 If the tool is at the 2nd safety distance, WinNC moves at rapid traverse FMAX to the safety distance, and from there, at the depth infeed feed to the first infeed depth.
- 4 Subsequently, the tool approaches the finished part contour tangentially and cuts one circumference in down-cut milling.
- 5 After that, the tool tangentially retracts from the contour back to the starting point in the machining plane.
- 6 This procedure 3 to 5 is repeated until the programmed depth is reached.
- 7 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance, and subsequently to the centre of the pocket (end position = starting position).

#### Note:

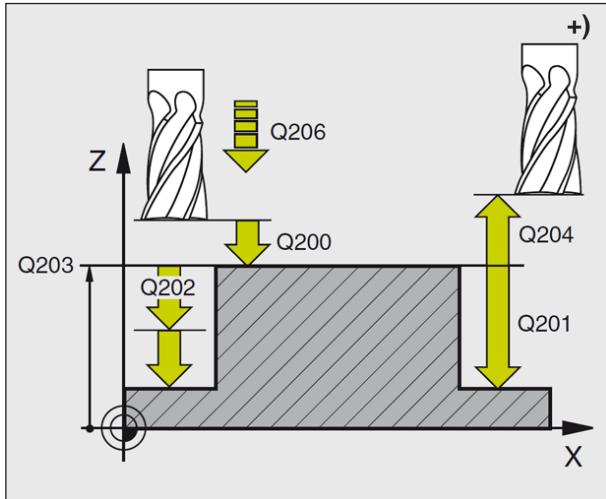
Prior to programming, please take note:  
 The sign of the cycle parameter depth determines the working direction.  
 If you want to clear and finish the pocket with the same tool, use a centre-cut end mill (DIN 844) and enter a low feed for depth infeed.  
 Minimum size of the pocket: threefold tool radius.



## STUD FINISHING (cycle 213)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance between tool point (= starting position) and work piece surface.	mm
Q201	Depth (incremental): Distance work piece surface - stud bottom.	mm
Q206	Feed for depth infeed: Traverse speed of the tool during movement to the depth. When plunging into the material, a value smaller than the one defined in Q207 must be specified.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in respectively. Enter a positive value.	mm
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece can occur.	mm
Q216	Centre 1st axis (absolute): Centre of stud in the main axis of the machining plane	mm
Q217	Centre 2nd axis (absolute): Centre of stud in the minor axis of the machining plane	mm
Q218	1. Side length (incremental): Length of stud, parallel to the main axis of the machining plane.	mm
Q219	2nd Side length (incremental): Length of stud, parallel to the minor axis of the machining plane.	mm
Q220	Corner radius: Radius of the stud corner.	mm
Q221	Allowance 1st axis (incremental): Allowance in the main axis of the machining plane, related to the length of the stud.	mm



### Cycle description

- 1 WinNC moves the tool automatically in the spindle axis to the safety distance, or, if specified, to the 2nd safety distance and subsequently to the pocket centre.
- 2 Starting from the stud centre, the tool moves to the starting point of the machining operation in the machining plane. The starting point is approx. 3.5 times the tool radius to the right from the stud.
- 3 If the tool is at the 2nd safety distance, WinNC moves at rapid traverse FMAX to the safety distance, and from there, at the depth infeed feed to the first infeed depth.
- 4 Subsequently, the tool approaches the finished part contour tangentially and cuts one circumference in down-cut milling.
- 5 After that, the tool tangentially retracts from the contour back to the starting point in the machining plane.
- 6 This procedure 3 to 5 is repeated until the programmed depth is reached.
- 7 At the end of the cycle, WinNC moves the tool in rapid traverse to the safety distance or, if entered, to the 2nd safety distance, and subsequently to the centre of the pocket (end position = starting position).

#### Note:

Prior to programming, please take note:  
The sign of the cycle parameter depth determines the working direction.  
If you want to clear and finish the stud with the same tool, use a centre-cut end mill (DIN 844) and enter a low feed for depth infeed.



## CIRCULAR POCKET FINISHING (cycle 214)

Manual operation      Programming

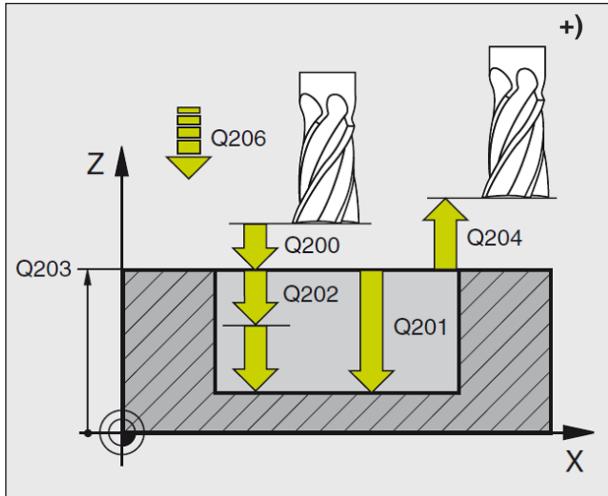
TNC:\demo\Bauteile\_components\2\_Flansch\_flange.H  
→Set-up clearance?

```

39 CYCL DEF 214 C. POCKET FINISHING ~
C200=2 SET-UP CLEARANCE ~
C201=20 ,DEPTH
C206=+150 ,FEED RATE FOR PLNGNG
C202=+5 ,PLUNGING DEPTH
C207=+500 ,FEED RATE FOR MILLNG
C203=+0 ;SURFACE COORDINATE
C204=+50 ;2ND SET-UP CLEARANCE
C216=+50 ;CENTER IN 1ST AXIS
C217=+50 ;CENTER IN 2ND AXIS
C222=+52 ;WORKPIECE BLANK DIA.
C223=+50 ;FINISHED PART DIA.
40 CYCL DEF 4.0 TASCHENFRAESEN
41 X+0
42 X+0
43 L X+10 Y+20 Z+0 A+0 C+0 R0 F100 M3
44 CALL PGM .\Reset.H
45 BLK FORM 0.1 Z X-52 Y-52 Z-30
46 BLK FORM 0.2 X-52 Y-52 Z+0
47 TOOL CALL "FACE_MILL_D40" Z S1000
48 L Z-100 R0 FF0 F0 MAX100 M3
49 CC X+0 Y+0
50 M7 LP PR+75 PA+0 FMAX
51 L Z-20 F500
52 APPR CT X+51 Y+0 CCA30 R+5 RL F500
53 CP PA+0 DR: RL F500
54 DEP PLCT PR+80 PA+0 R3
55 L Z+100 R0 FMAX

```

Parameter	Description	Unit
Q200	Safety distance (incremental): Distance between tool point (= starting position) and work piece surface.	mm
Q201	Depth (incremental): Distance work piece surface - pocket bottom.	mm
Q206	Feed for depth infeed: Traverse speed of the tool during movement to the depth. When plunging into the material, a value smaller than the one defined in Q207 must be specified.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in respectively. Enter a positive value.	mm
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece can occur.	mm
Q216	Centre 1st axis (absolute): Centre of pocket in the main axis of the machining plane.	mm
Q217	Centre 2nd axis (absolute): Centre of pocket in the minor axis of the machining plane.	mm
Q222	Raw part diameter: Diameter of the pre-machined pocket. Enter the raw part diameter smaller than the finished part diameter.	mm
Q223	Finished part diameter: Diameter of the finished pocket. The specified finished part diameter must be bigger than the raw part diameter and bigger than the tool diameter.	mm



### Cycle description

- 1 WinNC moves the tool automatically in the spindle axis to the safety distance, or, if specified, to the 2nd safety distance and subsequently to the pocket centre
- 2 Starting from the pocket centre, the tool moves to the starting point of the machining operation in the machining plane. For the calculation of the starting point, WinNC takes the raw part diameter and the tool radius into account. If you have specified the raw part diameter to be 0, WinNC starts cutting in the pocket centre.
- 3 If the tool is at the 2nd safety distance, WinNC moves the tool at rapid traverse FMAX to the safety distance, and from there, at the depth infeed feed to the first infeed depth.
- 4 Subsequently, the tool approaches the finished part contour tangentially and cuts one circumference in down-cut milling
- 5 After that, the tool tangentially retracts from the contour to the starting point in the machining plane
- 6 This procedure (3 to 5) is repeated until the programmed depth is reached.
- 7 At the end of the cycle, WinNC moves the tool at FMAX to the safety distance or, if entered, to the 2nd safety distance and subsequently to the centre of the pocket (end position = starting position)

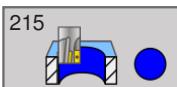
#### Note:

Prior to programming, please take note:  
The sign of the cycle parameter depth determines the working direction.  
If you want to clear and finish the pocket with the same tool, use a centre-cut end mill (DIN 844) and enter a low feed for depth infeed.

CYCL DEF

SPECIAL CYCLES

OLD CYCLES



## CIRCULAR STUD FINISHING (cycle 215)

Manual operation      Programming

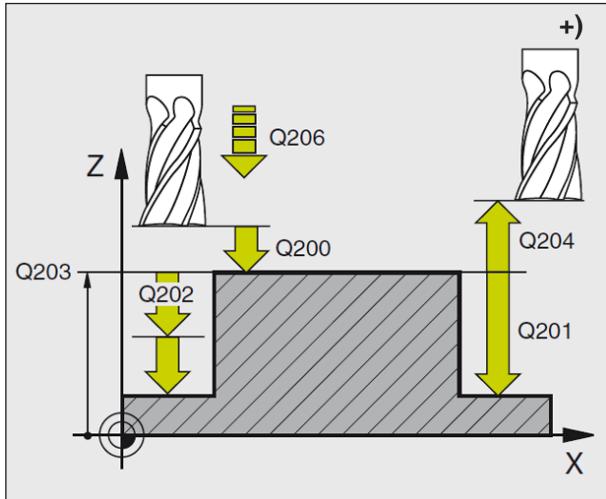
TNC \demo\Bauteile\_components\2\_Flansch\_flange.H  
→Set-up clearance?

```

40 CYCL DEF 215 C. STUD FINISHING ~
  C200=2 SET-UP CLEARANCE ~
  C201=20 .DEPTH
  Q206=+150 ;FEED RATE FOR PLNGNG
  Q202=+5 .PLUNGING DEPTH
  Q207=+500 ;FEED RATE FOR MILLNG
  Q203=+0 ;SURFACE COORDINATE
  Q204=+50 ;2ND SET-UP CLEARANCE
  Q216=+50 ;CENTER IN 1ST AXIS
  Q217=+50 ;CENTER IN 2ND AXIS
  Q222=+52 ;WORKPIECE BLANK DIA.
  Q223=+50 ;FINISHED PART DIA.
41 CYCL DEF 4.0 TASCHENFRAESEN
42 X=0
43 X=0
44 L X+10 Y+20 Z+0 A+0 C+0 R0 F100 M3
45 CALL PGM ..\reset.H
46 BLK FORM 0.1 Z X-52 Y-52 Z-30
47 BLK FORM 0.2 X-52 Y-52 Z+0
48 TOOL CALL "FACE_MILL_D40" Z S1000
49 L Z-100 R0 FF0 F0 MAX100 M3
50 CC X+0 Y+0
51 M7 LP PR+75 PA+0 FMAX
52 L Z-20 F500
53 APPR CT X+51 Y+0 CCA30 R+5 RL F500
54 CP PA+0 DR- RL F500
55 DEP PLCT PR+80 PA+0 R3
56 L Z+100 R0 FMAX

```

Parameter	Description	Unit
Q200	Safety distance (incremental): Distance between tool point (= starting position) and work piece surface.	mm
Q201	Depth (incremental): Distance work piece surface - stud bottom.	mm
Q206	Feed for depth inflection: Traverse speed of the tool during movement to the depth. If plunging takes place into the material, a lower value must be entered, and when plunging takes place outside, a higher value must be entered.	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is fed in respectively. Enter a positive value.	mm
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Coordinate of the spindle axis, in which no collision between tool and work piece can occur.	mm
Q216	Centre 1st axis (absolute): Centre of stud in the main axis of the machining plane.	mm
Q217	Centre 2nd axis (absolute): Centre of stud in the minor axis of the machining plane.	mm
Q222	Raw part diameter: Diameter of the pre-machined stud. Enter the raw part diameter bigger than the finished part diameter.	mm
Q223	Finished part diameter: Diameter of the finished stud. Enter the finished part diameter smaller than the raw part diameter.	mm

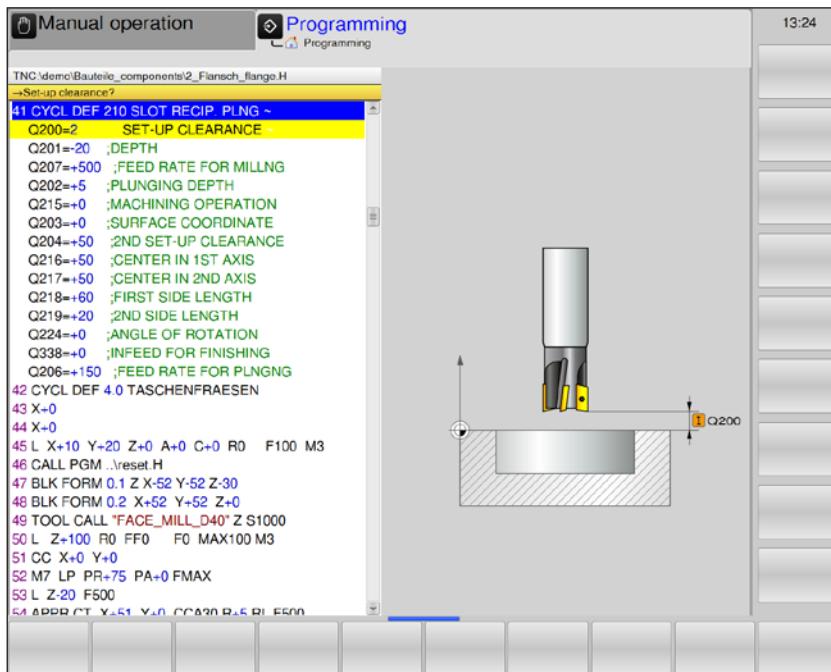


### Cycle description

- 1 WinNC moves the tool automatically in the spindle axis to the safety distance, or, if specified, to the 2nd safety distance and subsequently to the stud centre.
- 2 Starting from the stud centre, the tool moves to the starting point of the machining operation in the machining plane. The starting point is approx. 3.5 times the tool radius to the right from the stud.
- 3 If the tool is at the 2nd safety distance, WinNC moves the tool at rapid traverse FMAX to the safety distance, and from there, at the depth infeed feed to the first infeed depth.
- 4 Subsequently, the tool approaches the finished part contour tangentially and cuts one circumference in down-cut milling.
- 5 After that, the tool tangentially retracts from the contour back to the starting point in the machining plane.
- 6 This procedure 3 to 5 is repeated until the programmed depth is reached.
- 7 At the end of the cycle, WinNC moves the tool at FMAX to the safety distance or, if entered, to the 2nd safety distance and subsequently to the centre of the stud (end position = starting position).

#### Note:

Prior to programming, please take note:  
The sign of the cycle parameter depth determines the working direction.  
If you want to clear and finish the pocket with the same tool, use a centre-cut end mill (DIN 844) and enter a low feed for depth infeed.

CYCL  
DEFSPECIAL  
CYCLESOLD  
CYCLS**SLOT RECIPROCAL (cycle 210)**

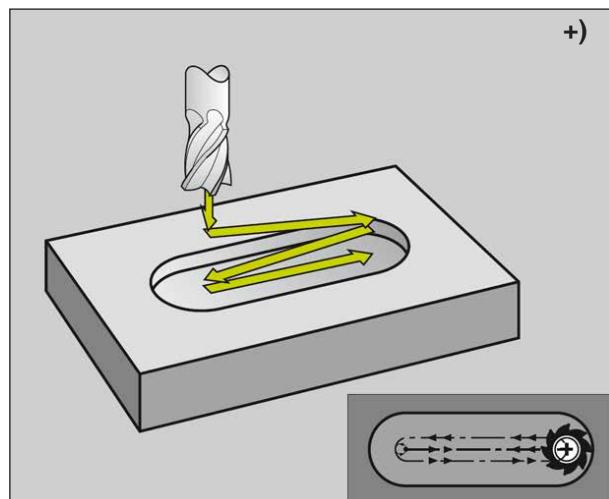
Parameter	Description	Unit
Q200	Safety distance (incremental): Distance between tool point (= starting position) and work piece surface.	mm
Q201	Depth (incremental): Distance work piece surface - slot bottom.	mm
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is in-fed in total during a reciprocal movement.	mm
Q215	Determine scope of machining: • 0 = Roughing and finishing • 1= Only roughing • 2= Only finishing	
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Z-coordinate, in which no collision between tool and work piece can occur.	mm
Q216	Centre 1st axis (absolute): Centre of slot in the main axis of the machining plane.	mm
Q217	Centre 2nd axis (absolute): Centre of slot in the minor axis of the machining plane.	mm
Q218	1. Side length: Value parallel to the main axis of the machining plane. Longer side of the slot.	mm
Q219	2nd Side length: Value parallel to the minor axis of the machining plane. Specify the width of the slot. If the slot width is specified as equal to the tool diameter, WinNC will only execute roughing (elongated hole milling).	mm

Parameter	Description	Unit
Q224	Angle of rotation (absolute): Angle, about which the whole slot is rotated. The centre of rotation lies in the slot centre.	degrees
Q238	Infeed for finishing (incremental): Dimension by which the tool is fed in on the spindle axis when finishing. With input 0: Finishing in one infeed.	mm
Q206	Feed for depth infeed: Traverse speed of the tool during movement to the depth. If plunging takes place into the material, a lower value must be entered, and when plunging takes place outside, a higher value must be entered.	mm/min

## Cycle description

### Roughing

- 1 WinNC automatically moves the tool in the spindle axis to the safety distance and subsequently into the centre of the left circle; from there, WinNC positions the tool at the safety distance above the work piece surface.
- 2 The tool moves at the milling feed to the work piece surface. From there, the milling cutter moves in longitudinal direction of the slot, diagonally plunging into the material, to the centre of the right circle.
- 3 Subsequently, the tool moves back to the centre of the left circle, plunging diagonally. These steps are repeated until the programmed cutting depth is reached.
- 4 At cutting depth, WinNC moves the tool for face milling to the other end of the slot and, after that, again to the centre of the slot.



### Finishing

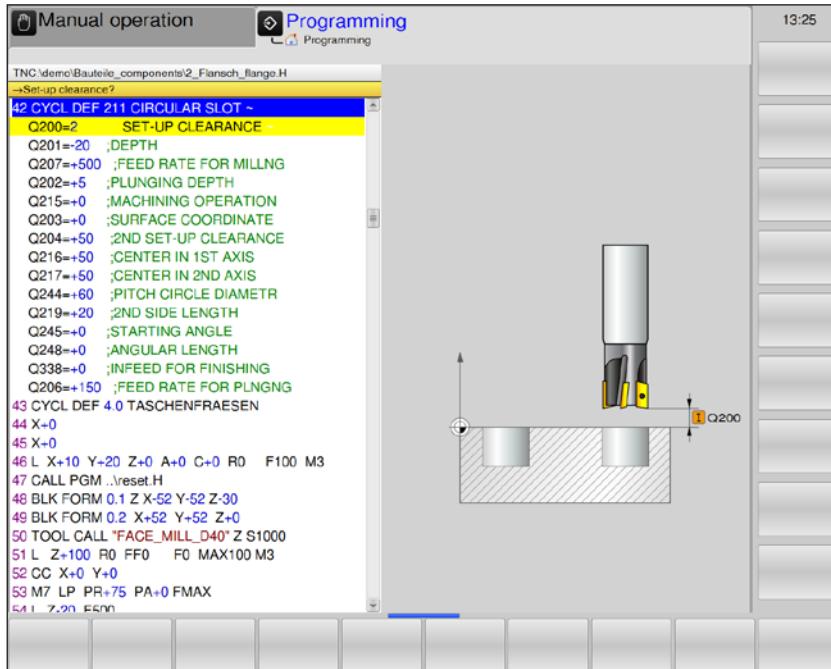
- 5 From the slot centre, WinNC approaches the finished contour with the tool tangentially, and then WinNC executes finishing in down-cut milling (with M3), if entered, also in multiple infeeds.
- 6 At the end of the contour, the tool tangentially retracts from the contour back to the centre of the slot.
- 7 At the end of the cycle, WinNC moves the tool at FMAX to the safety distance or, if entered, to the 2nd safety distance.

### Note:

Prior to programming, please take note:  
The sign of the cycle parameter depth determines the working direction.  
Select milling cutter diameter not larger than the slot width and not smaller than one third of the slot width.  
Do not select the milling cutter diameter to be smaller than half the slot length, otherwise WinNC cannot execute reciprocal plunging.



## ROUND SLOT (cycle 211)



Parameter	Description	Unit
Q200	Safety distance (incremental): Distance between tool point (= starting position) and work piece surface.	mm
Q201	Depth (incremental): Distance work piece surface - slot bottom.	mm
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q202	Infeed depth (incremental): Dimension by which the tool is in-fed in total during a reciprocal movement.	mm
Q215	Determine scope of machining: <ul style="list-style-type: none"> <li>• 0 = Roughing and finishing</li> <li>• 1= Only roughing</li> <li>• 2= Only finishing</li> </ul>	
Q203	Work piece surface coordinates (absolute)	mm
Q204	2nd Safety distance (incremental): Z-coordinate, in which no collision between tool and work piece can occur.	mm
Q216	Centre 1st axis (absolute): Centre of slot in the main axis of the machining plane.	mm
Q217	Centre 2nd axis (absolute): Centre of slot in the minor axis of the machining plane.	mm
Q244	Diameter of pitch circle	mm
Q219	2nd Side length: Width of the slot. If the slot width is specified as equal to the tool diameter, WinNC will execute roughing (elongated hole milling).	mm

Parameter	Description	Unit
Q245	Start angle (absolute): Polar angle of the starting point.	degrees
Q248	Opening angle of the groove (incremental)	mm
Q238	Infeed for finishing (incremental): Dimension by which the tool is fed in on the spindle axis when finishing. With input 0: Finishing in one infeed.	mm
Q206	Feed for depth infeed: Traverse speed of the tool during movement to the depth. If plunging takes place into the material, a lower value must be entered, and when plunging takes place outside, a higher value must be entered.	mm/min

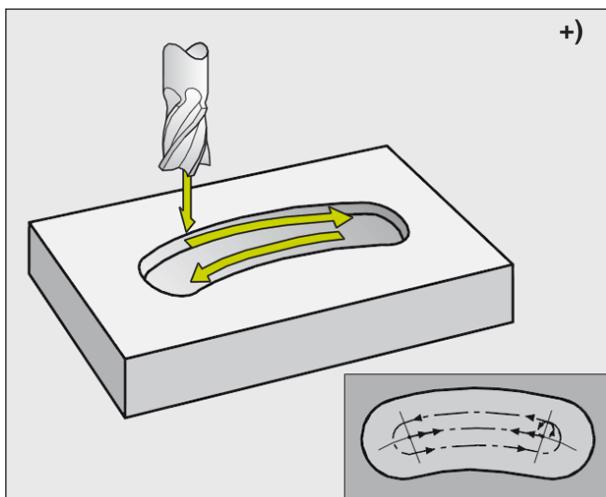
## Cycle description

### Roughing

- 1 WinNC moves the tool automatically in the spindle axis to the safety distance and subsequently to the centre of the right circle.
- 2 The tool moves at the milling feed onto the work piece surface and from there, the milling cutter plunges diagonally into the material and moves to the other end of the slot.
- 3 Subsequently, the tool again plunges diagonally and moves back to the starting point. The procedure 2 to 3 is repeated, until the programmed cutting depth is reached.
- 4 At cutting depth, WinNC moves the tool for face milling to the other end of the slot.

### Finishing

- 5 From the centre of the slot, WinNC moves the tool tangentially to the finished contour. After that, WinNC executes finishing of the contour with down-cut milling (at M3), if specified, also in multiple infeeds. The starting point for the finishing procedure lies here in the centre of the right circle.
- 6 At the end of the contour, the tool tangentially retracts from the contour.
- 7 At the end of the cycle, WinNC moves the tool at FMAX to the safety distance or, if entered, to the 2nd safety distance.

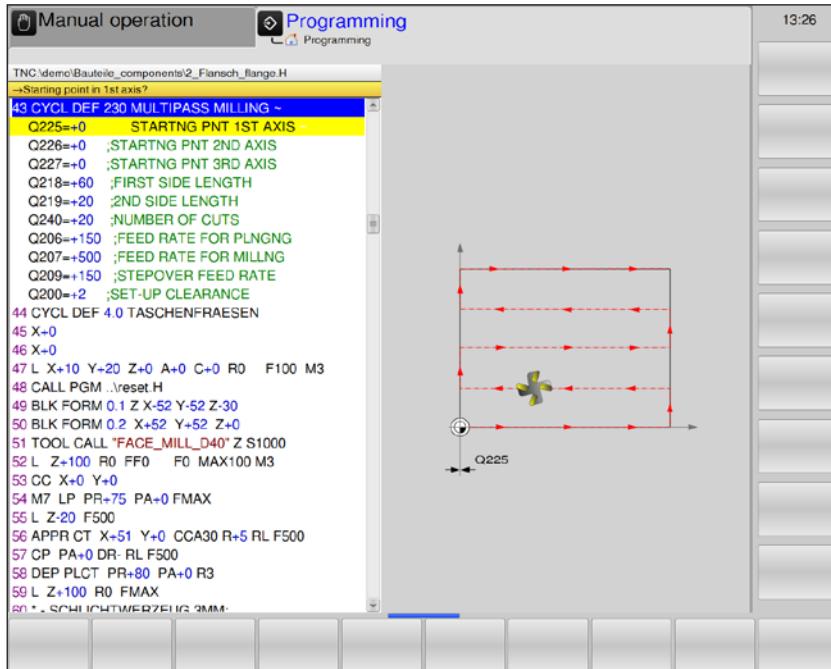


#### Note:

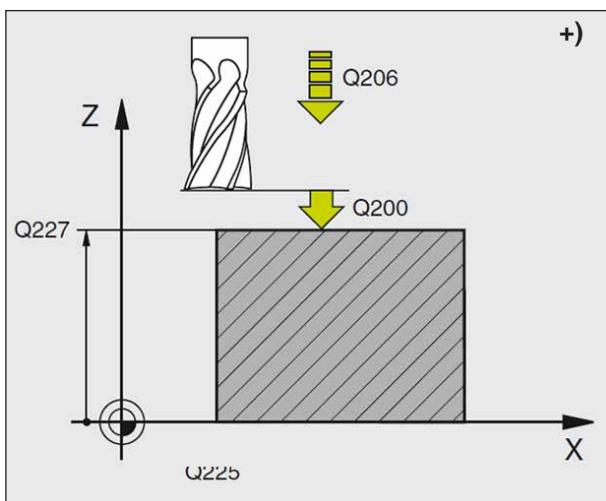
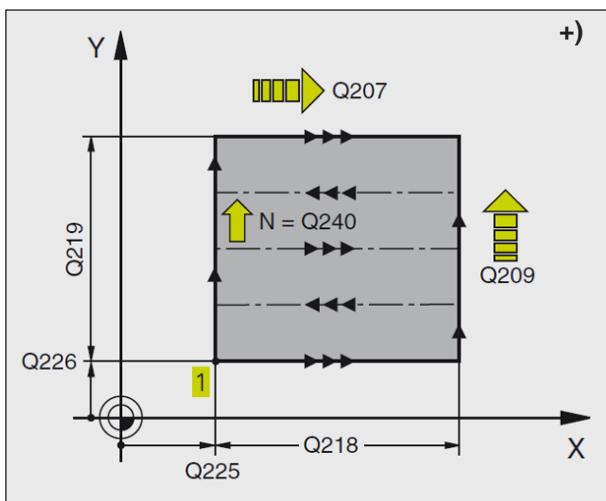
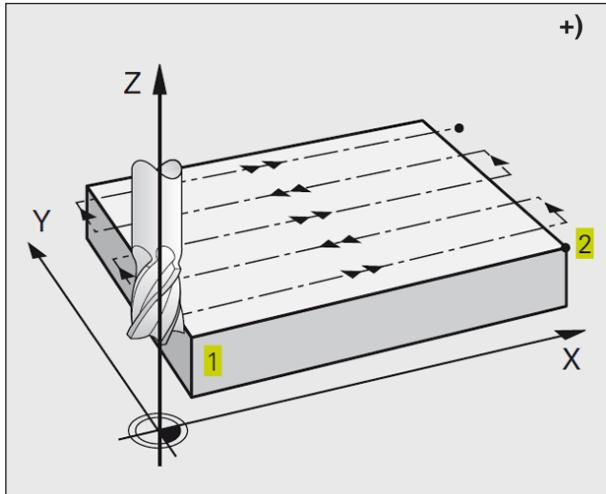
Prior to programming, please take note:  
The sign of the cycle parameter depth determines the working direction.  
Select milling cutter diameter not larger than the slot width and not smaller than one third of the slot width.  
Do not select the milling cutter diameter to be smaller than half the slot length, otherwise WinNC cannot execute reciprocal plunging.



## MULTIPASS MILLING (cycle 230)



Parameter	Description	Unit
Q225	Starting point 1st axis (absolute): Min-point-coordinate of the surface to be milled in the main axis of the machining plane.	mm
Q226	Starting point 2nd axis (absolute): Min-point-coordinate of the surface to be milled in the minor axis of the machining plane.	mm
Q227	Starting point 3rd axis (absolute): Height in the spindle axis, on which multipass milling is to be performed.	mm
Q218	1. Side length (incremental): Length of the surface to be machined by multipass milling in the main axis of the machining plane, related to the starting point 1st axis.	mm
Q219	2nd Side length (incremental): Length of the surface to be machined by multipass milling in the minor axis of the machining plane, related to the starting point 2nd axis.	mm
Q240	Number of cuts: Number of lines, on which WinNC should move the tool in width direction.	
Q206	Feed for depth infeed: Traverse speed of the tool when moving from the safety distance to the cutting depth.	mm/min
Q207	Milling feed: Tool traverse speed during milling	mm/min
Q209	Cross feed: Traverse speed of the tool during movement to the next line. If cross movement is performed in the material, Q209 must be specified smaller than Q207. If cross movement is performed outside, Q209 can be bigger than Q207.	mm/min
Q200	Safety distance (incremental): Distance from tool point to cutting depth for positioning at the start of the cycle and at the end of the cycle.	mm



### Cycle description

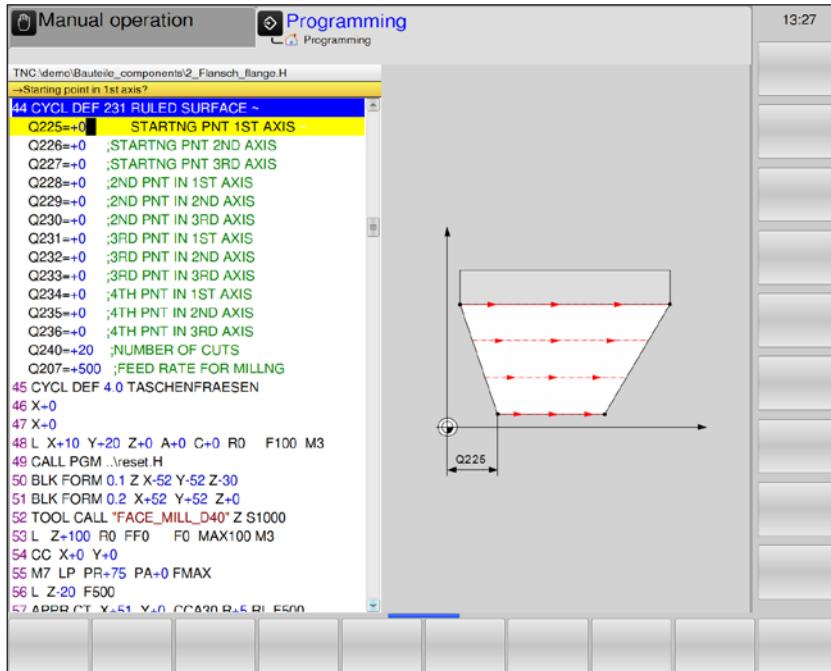
- 1 WinNC positions the tool at rapid traverse FMAX from the current position in the machining plane on the starting point **1**. In doing so, WinNC offsets the tool by the tool radius to the left and upwards.
- 2 Subsequently, the tool moves in the spindle axis at rapid traverse to the safety distance, and after that at the depth infeed feed to the programmed starting position in the spindle axis.
- 3 Then, the tool moves at the programmed milling feed to the end point **2**. WinNC calculates the end point using the programmed starting point, the programmed length and the tool radius.
- 4 WinNC offsets the tool in cross direction at the milling feed to the starting point of the next line. The offset is calculated by WinNC using the programmed width and the number of cuts.
- 5 After that, the tool moves back in negative direction of the 1st axis.
- 6 The multipass milling is repeated, until the specified surface is entirely machined.
- 7 At the end, WinNC returns at rapid traverse to the safety distance.

#### Note:

Prior to programming, please take note:  
Pre-position the tool in such a way, as to avoid any collision with the work piece or clamping device.

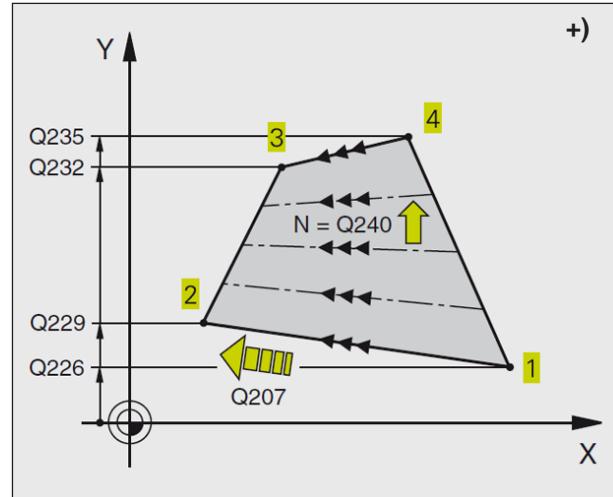
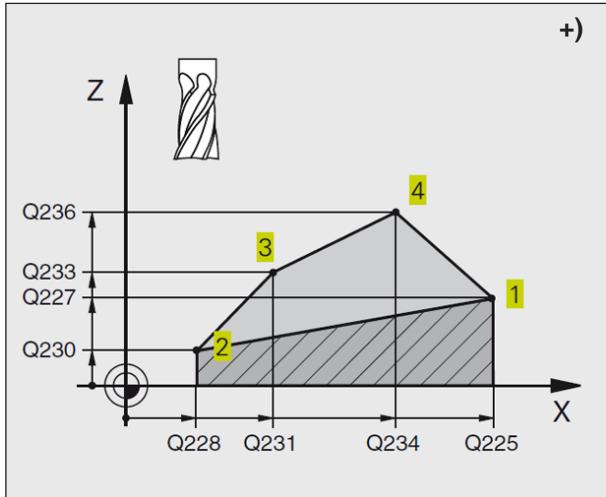


## RULED SURFACE (cycle 231)



Parameter	Description	Unit
Q225	Starting point 1st axis (absolute): Starting point coordinate of the surface to be milled in the main axis of the machining plane.	mm
Q226	Starting point 2nd axis (absolute): Starting point coordinate of the surface to be milled in the minor axis of the machining plane.	mm
Q227	Starting point 3rd axis (absolute): Starting point coordinate of the surface to be milled in the spindle axis.	mm
Q228	2nd Point 1st axis (absolute): End point coordinate of the surface to be milled in the main axis of the machining plane.	mm
Q229	2nd Point 2nd axis (absolute): End point coordinate of the surface to be milled in the minor axis of the machining plane.	mm
Q230	2nd Point 3rd axis (absolute): End point coordinate of the surface to be milled in the spindle axis.	mm
Q231	3. Point 1st axis (absolute): Coordinate of the point 3 in the main axis of the machining plane.	mm
Q232	3. Point 2nd axis (absolute): Coordinate of the point 3 in the minor axis of the machining plane.	mm
Q233	3. Point 3rd axis (absolute): Coordinate of the point 3 in the spindle axis.	mm
Q234	4. Point 1st axis (absolute): Coordinate of the point 4 in the main axis of the machining plane.	mm
Q235	4. Point 2nd axis (absolute): Coordinate of the point 4 in the minor axis of the machining plane.	mm
Q236	4. Point 3rd axis (absolute): Coordinate of the point 4 in the spindle axis.	mm

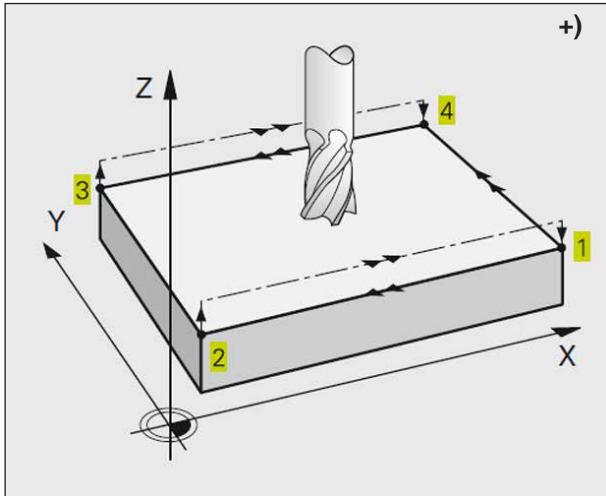
Parameter	Description	Unit
Q240	Number of cuts: Number of lines for the tool to traverse between point <b>1</b> and <b>4</b> and/or between point <b>2</b> and <b>3</b> .	mm
Q207	Milling feed: Tool traverse speed during milling. WinNC executes the first step with half of the programmed value.	mm/min



**Note:**

Prior to programming, please take note:  
Pre-position the tool in such a way, as to  
avoid any collision with the work piece or  
clamping devices.  
WinNC moves the tool with radius compensa-  
tion R0 between the entered positions.  
Possibly, use a centre-cut end mill (DIN 844).





### Cycle description

- 1 WinNC positions the tool from the current position with a 3D-straight line movement on the starting point **1**.
- 2 Subsequently, the tool moves at the programmed milling feed to the end point **2**.
- 3 There, WinNC moves the tool in rapid traverse by the tool diameter in positive spindle axis direction and, after that, again back to the starting point **1**.
- 4 At the starting point **1**, WinNC moves the tool again to the most recently traversed Z-value.
- 5 Subsequently, WinNC moves the tool in all three axes from point **1** to the next line in the direction of point **4**.
- 6 After that, WinNC moves the tool to the end point of this line. The end point is calculated by WinNC from point **2** and an offset in the direction of point **3**.
- 7 The multipass milling is repeated, until the specified surface is entirely machined.
- 8 At the end, WinNC positions the tool by the tool diameter above the highest entered point in the spindle axis.

### Cutting guide

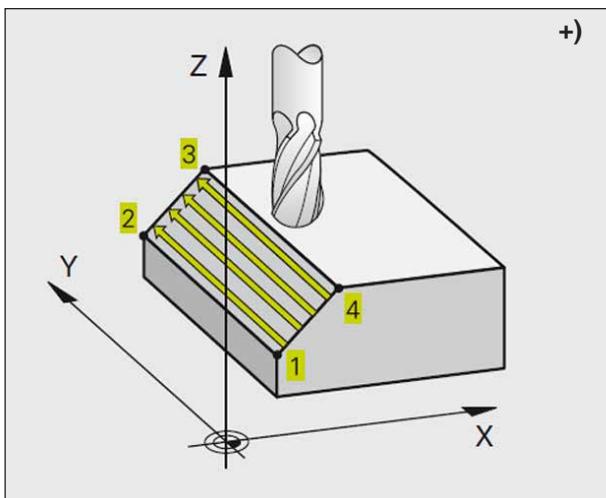
The starting point and thus the milling direction can be freely chosen, as WinNC always executes the individual cuts from point **1** to point **2** and the overall sequence runs from point **1 / 2** to point **3 / 4**. You can position point **1** at every corner of the surface to be machined.

You can optimise the surface quality when using end mills:

- By a pushing cut (spindle axis coordinate of point **1** bigger than the spindle axis coordinate of point **2**) with slightly inclined surfaces.
- By a pulling cut (spindle axis coordinate of point **1** smaller than the spindle axis coordinate of point **2**) with strongly inclined surfaces.
- When milling warped surfaces, program the main cutting direction (from point **1** to point **2**) in the direction of the stronger inclination.

You can optimise the surface quality when using radius cutters:

- When milling warped surfaces, program the main cutting direction (from point **1** to point **2**) perpendicular to the direction of the steepest inclination.



## Subprograms

### Labelling Subprograms and Program Section Repeats\*)

You can execute once programmed machining steps any number of times with subprograms and program section repeats.

#### Label

In the machining program, subprograms and program section repeats start with the marker LBL, an abbreviation of LABEL.

A LABEL receives a number between 1 and 65535. You are allowed to assign each LABEL number only once using LABEL SET.

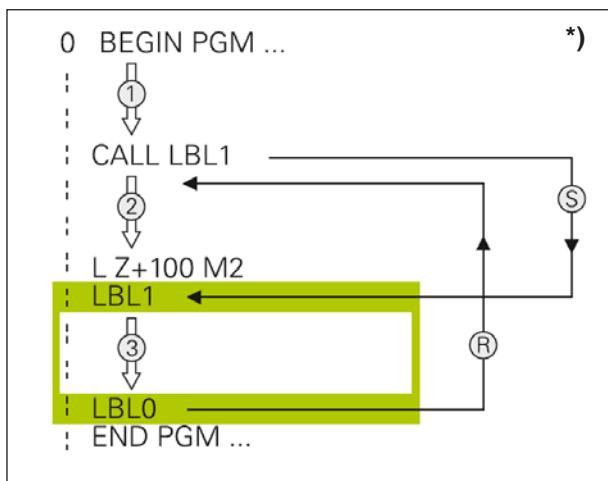
**Note:**

If you assign a LABEL number multiple times, WinNC will issue an error message, when the LBL SET block is finished.



LABEL 0 (LBL 0) marks the end of a subprogram and can be used any number of times.

## Subprograms \*)



### Method of working

- 1 WinNC executes the machining program to the point of a CALL LBL subprogram call.
- 2 From this point, WinNC executes the called subprogram right through to the end of the subprogram LBL 0.
- 3 After that, WinNC continues executing the machining program from the block that follows the CALL LBL subprogram call.

### Programming instructions

- One main program can contain up to 254 subprograms.
- Subprograms can be called any number of times in any desired order.
- A subprogram is not allowed to call up itself.
- Program the subprograms at the end of the main program (after the block with M2 and/or M30).
- If subprograms are positioned in the machining program prior to the block with M02 or M30, they are executed without call up at least once.

### Programming subprograms

- Mark the beginning: Press the LBL SET key and enter a label number.
- Enter a subprogram number.
- Mark the end: Press the LBL SET key and enter a "0" for label number.



### Call up subprogram



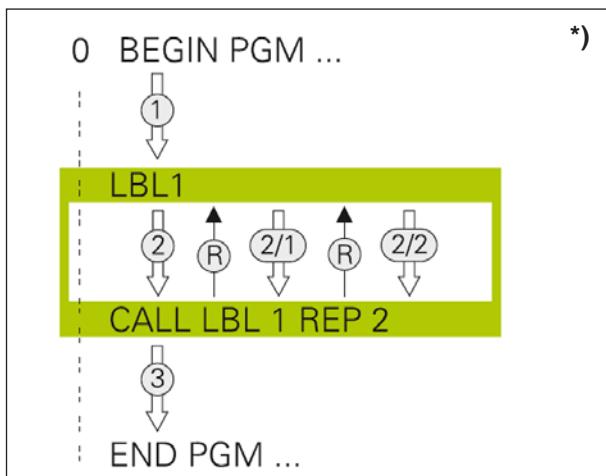
- Call up subprogram: Press LBL CALL key.
- **Label number:** Enter the label number of the subprogram to be called.
- **Repetitions REP:** Transfer the dialogue by pressing the NO ENT key. Use repetitions REP only for program section repeats.

#### Note:

CALL LBL 0 is not permissible, as it denotes the end of a subprogram.



## Program section repeats \*)



### Label LBL

Program section repeats start with the marker LBL (LABEL).

A program section repeat ends with CALL LBL / REP.

### Method of working

- 1 WinNC executes the machining program to the end of the program section (CALL LBL/REP). This means, WinNC executes the label one time without a separate call.
- 2 Subsequently, WinNC repeats the program section between the called up LABEL and the CALL LBL/REP label call the number of times you have specified under REP.
- 3 After that, WinNC continues to execute the machining program.

### Programming instructions

- You can repeat a program section up to 65 534 times in a row.
- Program sections will be executed by WinNC always one time more than the programmed number of repetitions.

### Programming program section repeats



- Mark the beginning: Press the LBL SET key and enter a LABEL number for the program section to be repeated.
- Enter the program section

### Calling program section repeats



- Press the LBL CALL key, enter the label number of the program section to be repeated and the number of repetitions REP.

## Any desired program as subprogram \*)

### Method of working

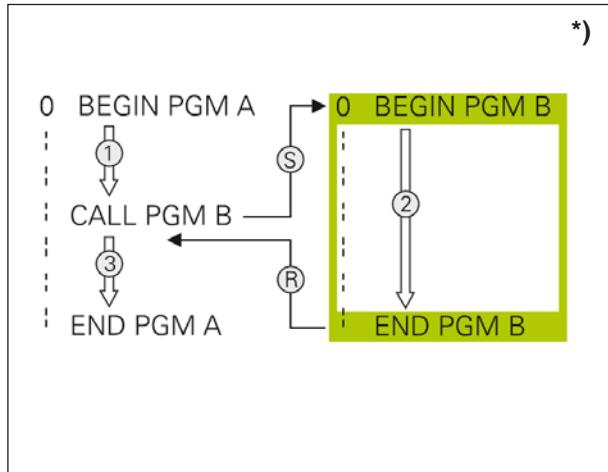
- 1 WinNC keeps executing the machining program, until you call a different program using CALL PGM.
- 2 Subsequently, WinNC executes the called program to its end.
- 3 After that, WinNC continues executing the (calling) machining program from the block that follows the program call.

### Programming instructions

- In order for WinNC to use any desired program as a subprogram, no LABELs are required.
- The called program must not contain any additional function M2 or M30.
- The called program must not contain any CALL PGM call into the calling program (infinite loop).

### Calling any desired program as subprogram

- Select function for program call: Press PGM CALL key.



### Note:

If you only enter the program name, the called program must be in the same directory as the calling program.  
If the called program is not listed in the same directory as the calling program, please specify the complete pathname, e. g.: TNC:\ZW35\SCHRUPP\PGM1.H



## Nesting

### Types of nesting

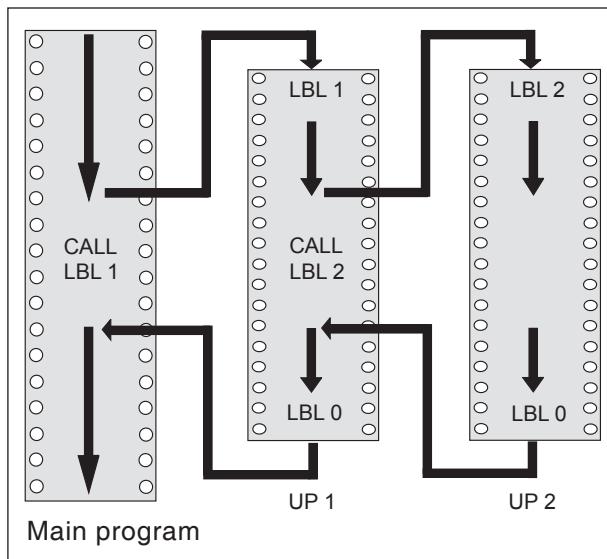
- Calling subprograms in a subprogram
- program section repeats in a program section repeat
- Subprogram call in program section repeats
- Program section repeats in a subprogram

### Nesting depth

The nesting depth determines, how often further subprograms or program section repeats may be contained in program sections or subprograms

- Maximum nesting depth for subprograms: 6
- Maximum nesting depth for main program calls: 4
- Program section repeats can be nested any number of times

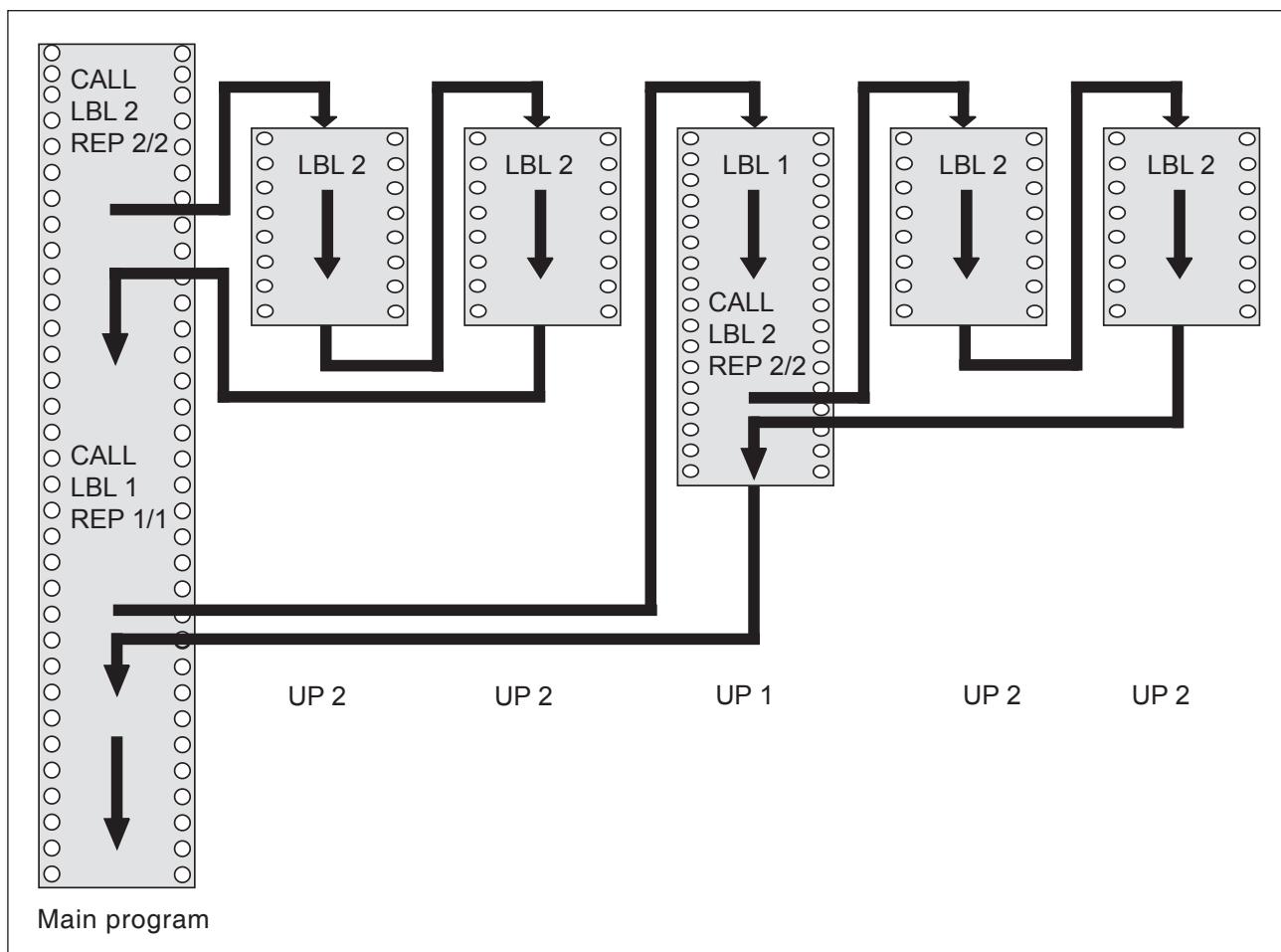
Example: Subprogram in a subprogram	Comment
0 BEGIN PGM UPGMS MM	Call up main program at LBL 1
...	
17 CALL LBL 1	Call up a subprogram at LBL 1
...	
35 L Z+100 R0 FMAX M2	Last program block of the main program (with M2)
36 LBL 1	Start of subprogram 1
...	
39 CALL LBL 2	
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Start of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM UPGMS MM	63 END PGM UPGMS MM



### Program execution

- 1 The main program UPGMS is executed up to block 17
- 2 Subprogram 1 is called up and executed up to block 39
- 3 Subprogram 2 is called up and executed up to block 62. End of subprogram 2 and jump back to the subprogram, from where it was called
- 4 Subprogram 1 is executed from block 40 to block 45. End of subprogram 1 and jump back into the main program UPGMS
- 5 Main program UPGMS is executed from block 18 to block 35. Jump back to block 0 and end of program

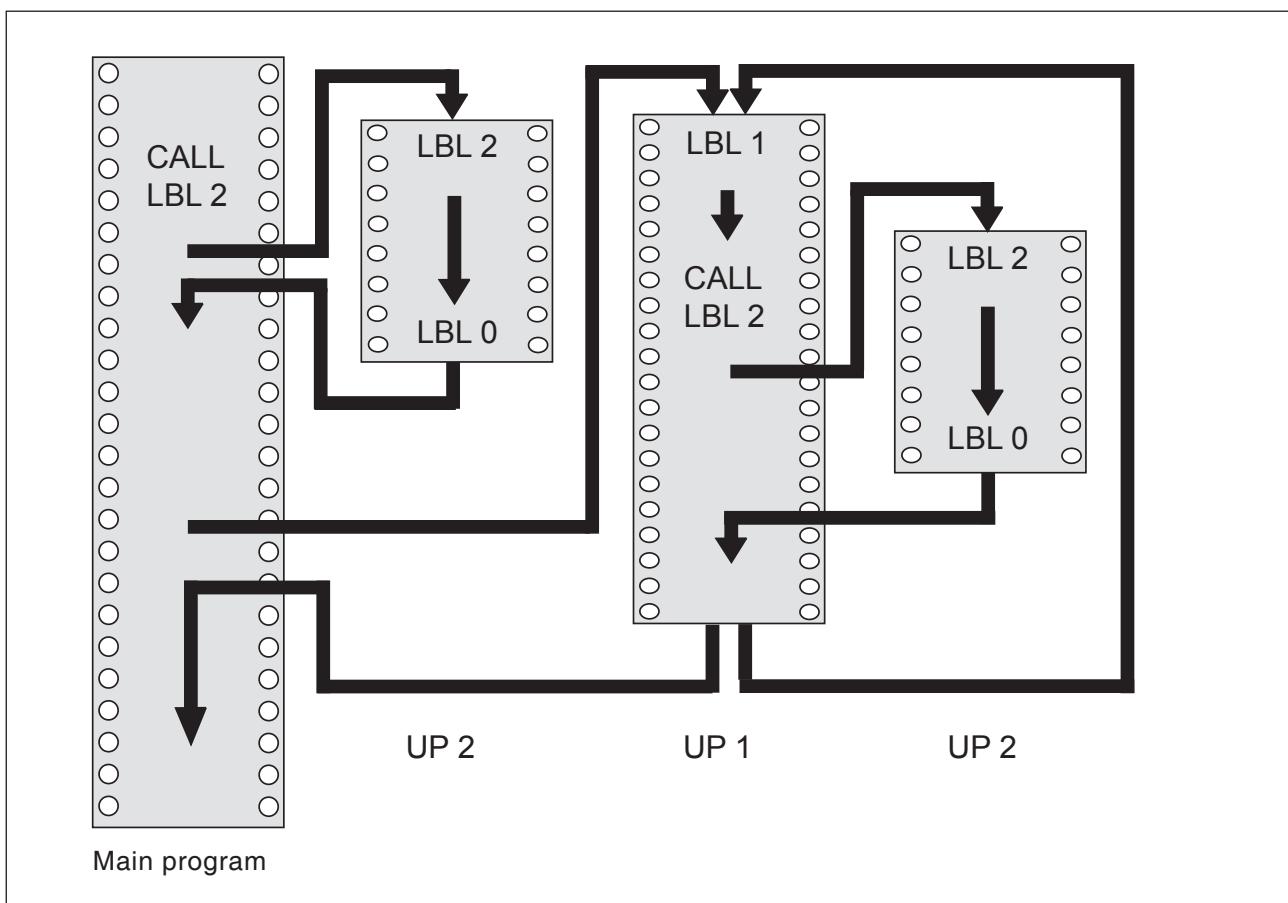
Example: Repeating program section repeats	Comment
0 BEGIN PGM UPGMS MM	
...	
15 LBL 1	Start of program section repeat 1
...	
20 LBL 2	Start of program section repeat 2
...	
27 CALL LBL 2 REP 2/2	Program section between this block and LBL 2 (block 20) is repeated 2 times
...	
35 CALL LBL 1 REP 1/1	Program section between this block and LBL 1 (block 15) is repeated 1 time
...	
50 END PGM REPS MM	



### Program execution

- 1 The main program REPS is executed up to block 27
- 2 Program section between block 20 and block 27 is repeated 2 times
- 3 Main program REPS is executed from block 28 to block 35
- 4 Program section between block 15 and block 35 is repeated 1 time (contains the program section repeat between block 20 and block 27)
- 5 Main program REPS is executed from block 36 to block 50 (end of program)

Example: Repeating a subprogram	Comment
0 BEGIN PGM UPGMS MM	
...	
10 LBL 1	Start of program section repeat 1
11 CALL LBL 2	Call subprogram
12 CALL LBL 1 REP 2/2	Program section between this block and LBL 1 (block 10) is repeated 2 time
...	
19 L Z+100 R0 FMAX M2	Last block of the main program with M2
20 LBL 2	Start of the subprogram
...	
28 LBL 0	End of the subprogram
29 END PGM UPGREP MM	



#### Program execution

- 1 The main program UPGREP is executed up to block 11
- 2 Subprogram 2 is called up and executed
- 3 Program section between block 10 and block 12 is repeated 2 times: subprogram 2 is repeated 2 times
- 4 Main program UPGREP is executed from block 13 to block 19; end of program

# E: Tool programming

## Tool related input data

### Feed rate F \*)

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

#### Input

You can specify the feed in the **TOOL CALL** block (tool call) and in every positioning block. (See "Creating program blocks via path function keys", Chapter D)

#### Rapid traverse

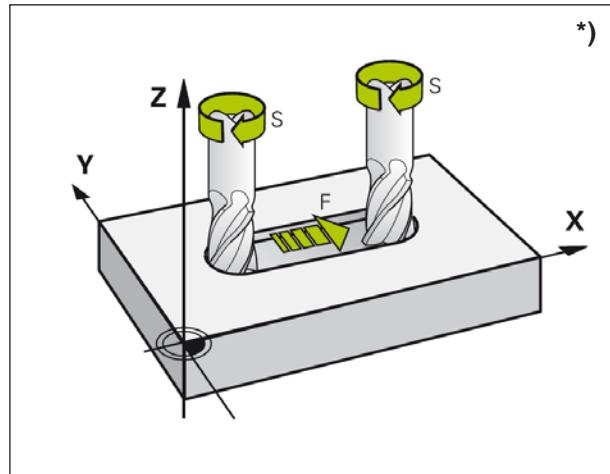
For rapid traverse, enter **F MAX** or **F9999**. For entering **F MAX**, press the dialogue question **Feed F= ?** the ENT key or the softkey **FMAX**.

#### Duration of validity

The feed programmed via a numerical value is valid up until the block where a new feed is programmed. **F MAX** is only valid for the block in which it has been programmed. After the block with **F MAX**, the last feed programmed via a numerical value is valid again. **F9999** is an auto-hold rapid traverse. It is deleted by the input of a numerical value feed.

#### Change during the program run

While the program run is active, you can change the feed via the override knob **F** for the feed.



Feed and spindle speed

## Spindle speed S\*)

You enter the spindle speed in revolutions per minute (rpm) in a **TOOL CALL** block.

### Programmed change

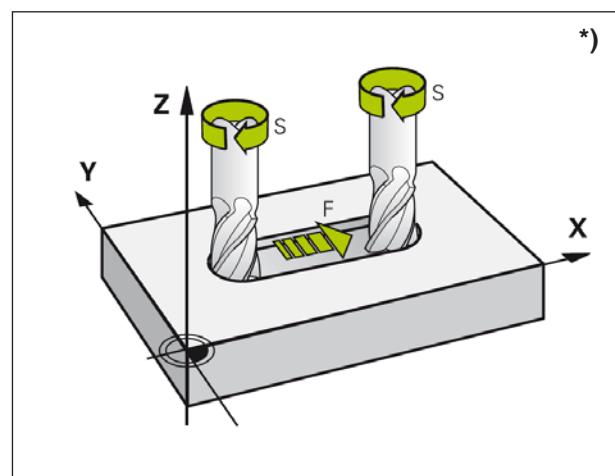
In the machining program, you can change the spindle speed via a TOOL CALL block by exclusively entering the new spindle speed:



- Programming the speed: Press the TOOL CALL key
- Dialogue **Tool number?** skip by pressing the NO ENT key
- Dialogue **Spindle axis parallel X/Y/Z ?** skip by pressing the NO ENT key
- In dialogue **Spindle speed S= ?** enter the new spindle speed, confirm with END key

### Change during the program run

While the program run is active, you can change the spindle speed via the override knob S for the spindle speed.

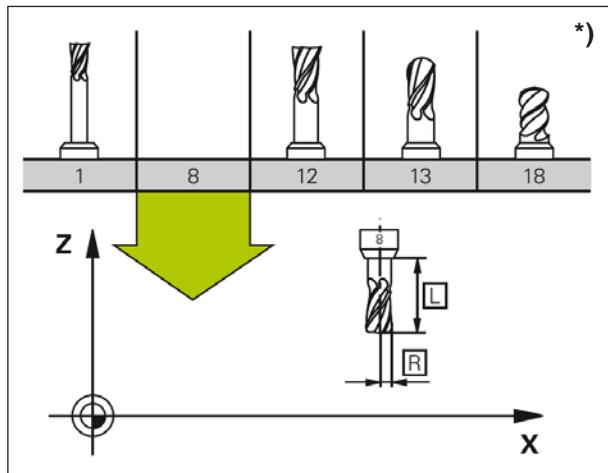


Feed and spindle speed

## Tool data

### Requirements for tool compensation \*)

Typically, you program the coordinates of the path movements according to the dimensions given in the work piece drawing. For WinNC to be able to calculate the tool centre and thus perform the tool compensation, you have to specify the length and the radius for every tool that is used.



Tool data

### Tool number, tool name

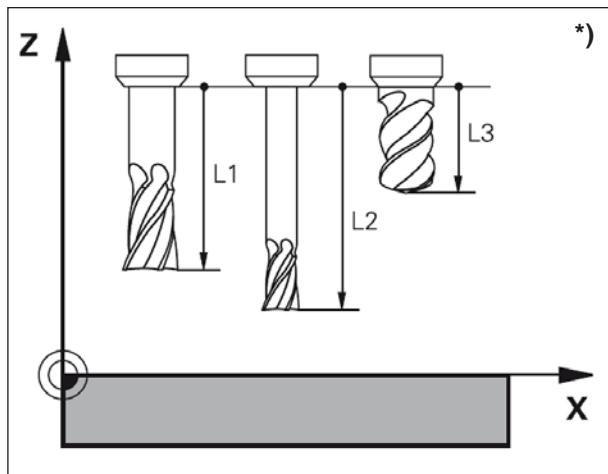
Every tool has a tool name. If you work with tool tables, you can use higher numbers and additionally assign tool names.

The tool with the number 0 is designated as zero tool having the length  $L=0$  and the radius  $R=0$ . The tool T0 cannot be called up. In the tool tables, you also should define the tool T0 with  $L=0$  and  $R=0$ .

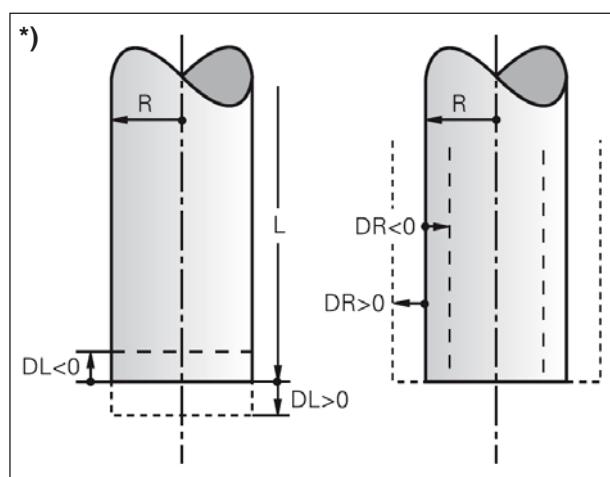
### Tool length L

You should fundamentally enter the tool length  $L$  as an absolute length value related to the tool reference point.

For multiple functions in conjunction with multi-axis machining, it is imperative for WinNC to have the total length of the tool.



Tool length



Tool radius

### Tool radius $R$ \*)

You enter the tool radius  $R$  directly.

### Delta values for lengths and radii

Delta values are deviations for the length and the radius of tools.

A positive delta value represents an allowance ( $DL, DR, DR2 > 0$ ). In case of a machining operation with an allowance, enter the value for the allowance when programming the tool call via TOOL CALL.

A negative delta value represents an undersize ( $DL, DR, DR2 < 0$ ). In the tool table, an undersize is specified for the wear of a tool.

You enter delta values as numerical values; in a TOOL CALL block, you can also transfer the value via a Q parameter.

Input scope: Delta values can be maximum  $\pm 99.999$  mm.

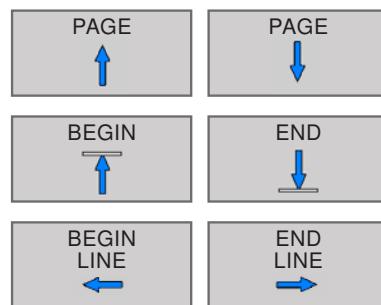
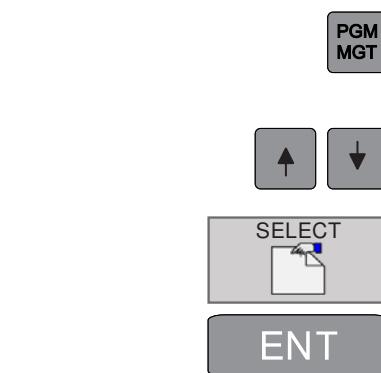
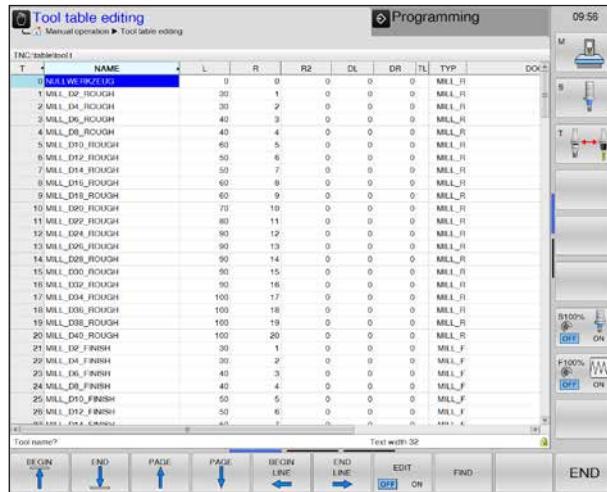
## Entering tool data in the table

In a tool table you can define tools and save the associated tool data.

You must use the tool tables, if indexed tools, such as step drills having multiple length compensations, are to be used.

### Tool table: Standard tool data

Abbr.	Enter	Dialogue
T	Number used in the program to call up the tool (e. g.: 5, indexed: 5.2)	—
NAME	Name used in the program to call up the tool	Tool name?
L	Compensation value for the tool length L	Tool length L?
R	Compensation value for the tool radius R	Tool radius R?
R2	Tool radius R2 for corner radius cutter (only for three-dimensional radius compensation or graphic display of the machining operation with radius cutter)	Tool radius R2?
DL	Delta value for tool length L	Allowance tool length?
DR	Delta value tool radius R	Allowance tool radius?
TL	Set tool lock (TL: for tool locked)	Tool locked? Yes= ENT / No= NO ENT
TYPE	Tool type: Press ENT key to edit the field. The GOTO key opens a window, where the tool type can be selected. Tool types are assigned, in order to be able to assign display filter settings such as to make only the selected type visible in the table.	Tool type?
DOC	Commentary regarding the tool	Tool commentary?
LCUTS	Cutting length of the tool for cycle 22	Cutting length in the tool axis?
ANGLE	Maximum plunging angle of the tool with reciprocal plunging movement for cycles 22 and 208	Maximum plunging angle?
CUT	Number of tool blades	Number of blades?
NMAX	Spindle speed limit for this tool. The programmed value (error message) as well as a speed increase via potentiometer are both monitored. Function inactive: - enter	Maximum speed [1/min]?
T-ANGLE	Tip angle of the tool. Is used by the centring cycle (cycle 240), in order to calculate the centring depth from the diameter entry.	Tip angle
PITCH	Thread pitch of the tool. Is used by the cycles for thread tapping (cycle 206, 207 and 209). A positive algebraic sign represents a right-hand thread.	Tool thread pitch?
3D TOOL NAME	Select a tool from a drop-down list	Tool from database for 3D simulation?
COLOR	Select the color for the tool	Tool color in 3D simulation?



## Editing tool tables

The tool table that is valid for the program run has the file name **TOOL.T**. **TOOL T** must be saved in the directory **TNC:\table** and it can only be edited in one machine mode of operation. Tool tables that are archived or used for program testing can have any other file name with the extension **.T**.

### Opening the tool table **TOOL.T**:

- 1 Select any machine mode of operation
- 2 Select the tool table: Press the **TOOL TABLE** softkey
- 3 Set softkey **EDIT** to „ON“.

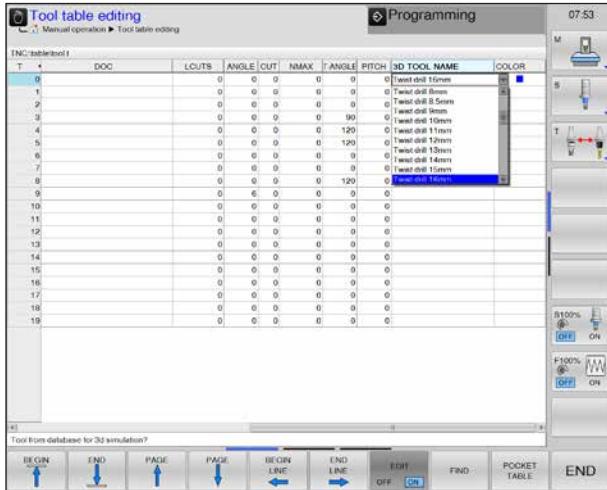
### Display only certain tool types:

- 1 Switch the softkey bar until **TABLE FILTER** appears.
- 2 Select desired tool type via softkey.
- 3 Cancel filter: Press softkey **SHOW ALL**

## Opening any other tool table

- 1 Call up file management
- 2 Move the illuminated field using the arrow keys or the arrow softkeys to the file that you want to select.
- 3 Select file: Press softkey **SELECT** or **ENT**.

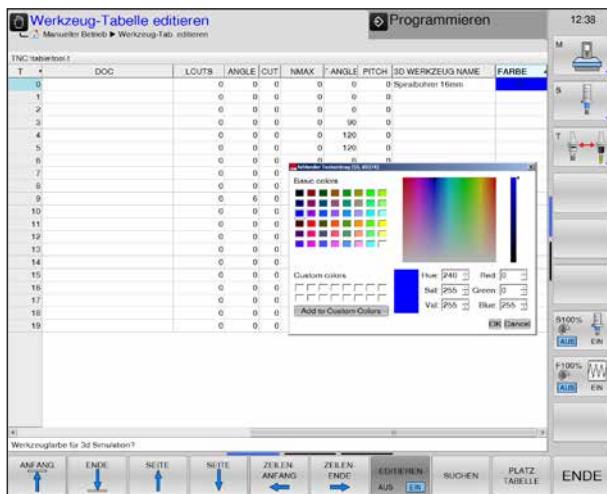
If you open a tool table for editing, you can move the illuminated field with the arrow keys or softkeys to any desired position. You can overwrite the saved values or enter new values on any desired position.



## 3D tool-name

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

- 1 Move the scroll bar to the right to display the 3D tools.
- 2 Double-click the 3D tool to activate the tool selection (drop down menu). Press the space bar to scroll in the tool selection.
- 3 To deactivate a tool, the blank line (at the very top) must be selected in the drop down menu.



## 3D tool-color

Colors can be assigned to tools in order to display them better in the simulation.

- 1 Move the scroll bar to the right to display the Color selection.
- 2 Double-click the color selection (alternatively use the space bar) to open the color selection window.
- 3 Predefined colors are stored as basic colors, user defined colors as custom colors.
  - Create Custom colors:  
Move the mouse in the colored area to the required color and click to select. Alternatively the values for R,G,B can be entered manually.
  - By clicking "Add to Custom Colors" a new color is added.
- 4 To deselect a color, select black.
- 5 Confirm with OK or quit with Cancel.

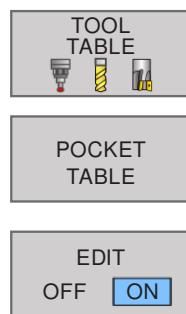
### Note:

3D tools and 3D colors are stored in a separate table **TOOL.3d**.

### Instructions:

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

## Pocket table for tool changer \*



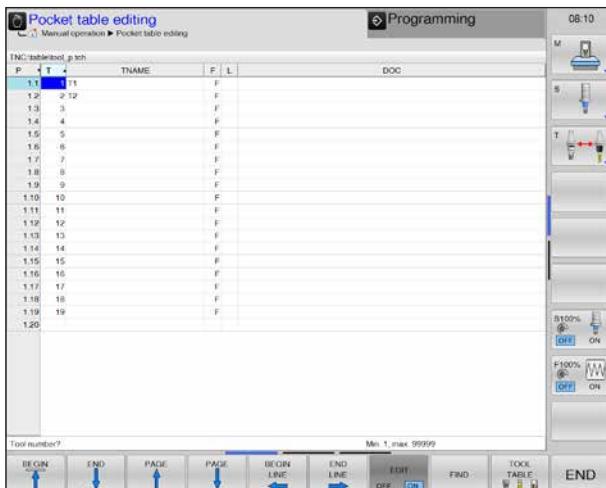
Open tool table TOOL.T:

- 1 Select any machine mode of operation.
- 2 Select the tool table: Press the TOOL TABLE softkey.

- 3 Press softkey to open POCKET TABLE.

- 4 Set softkey EDIT to „ON“.

Refer to the tool table to see functions for edit options of the pocket table.



For automatic tool changing you need the a pocket table.

You manage the assignment of your tool changer in the pocket table. The pocket table is in the **TNC:TABLE** directory.

The filename **TOOL\_P.TCH** is preset and cannot be changed for the machine area.

### Pocket-table: inputs

Abbr.	Enter	Dialogue
P	Pocket number of the tool in the tool magazine	—
T	Tool number	Tool number?
TName	Tool name	Tool name?
F	tool is always returned to the same pocket in the tool magazine	Fixed pocket? Yes = ENT / No = NO ENT
L	Locket pocket	Pocket locked Yes = ENT / No = NO ENT
DOC	Display of the comment to the tool from TOOL.T	Pocket comment?

#### Note:

F and L can't be edited at the moment, because chaotic tool management is not available in the current software version.

#### Note for TName and DOC:

These entries are coupled in the pocket table and the tool table and are therefore automatically transferred.

## Tool change via softkey

In the vertical softkey bar it is possible to change the tool via softkey.

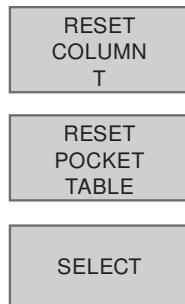


**1** Press softkey for tool change.

**2** Swivel in a new tool.

**3** Return to the pocket table.

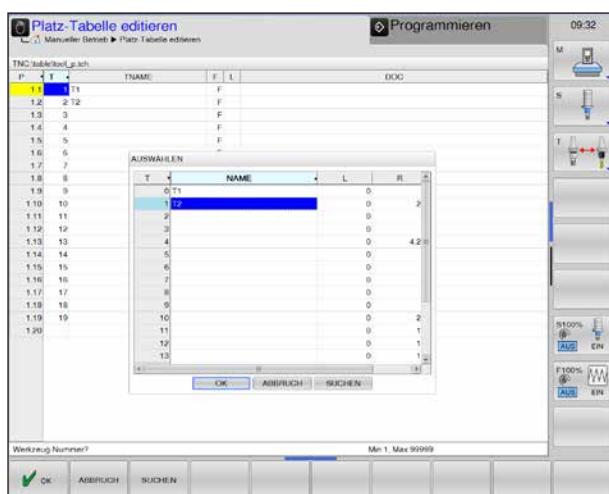
## More functions in the horizontal softkey bar



**1** Position the cursor to column T and press softkey to reset the column.

**2** Reset the entire table.

**3** Select a tool from the tool table.



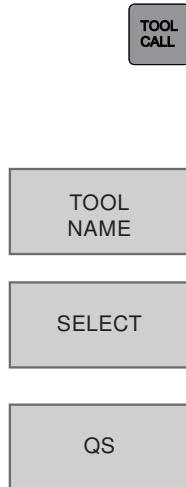
The content of the tool table is displayed. Select a tool by using the arrow keys. Press OK to transfer the selection to the pocket table.



**4** Complete input by pressing OK or CANCEL your input.

## Call tool data \*)

You have to program a **TOOL CALL** in the machining program by entering the following information:



- Select tool call by pressing the TOOL CALL key.
- **Tool number:** Enter name or number of the tool. The tool has been previously defined in a TOOL DEF block or in the tool table.
- Use the TOOL NAME softkey to enter the name.

Via the SELECT softkey, you can open a window via which you can directly select a tool defined in the tool table TOOL.T without entering the number or name of the tool.

- With the QS soft key you enter a string parameter.  
The WinNC automatically places the tool name in quotation marks.  
Names always refer to an entry in the active tool table TOOL .T.

- **Spindle axis parallel X/Y/Z :** Enter the tool axis.
- **Spindle speed S:** Enter spindle speed directly.
- **Feed F:** Enter feed directly. F remains valid until you program a new feed in a positioning block or a TOOL CALL block.
- **Allowance tool length DL:** Delta value for the tool length.
- **Allowance tool radius DR:** Delta value for the tool radius.

### Example: Tool call

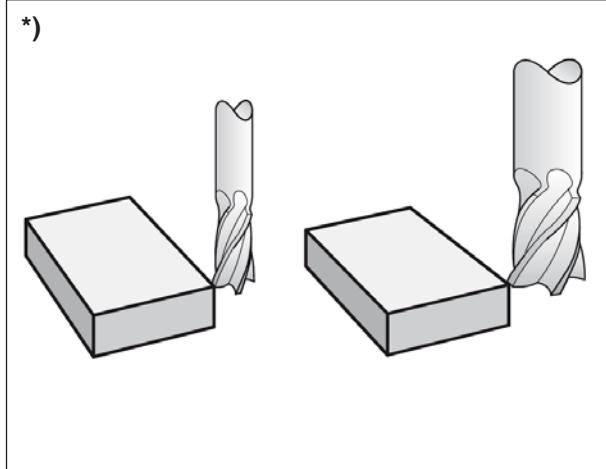
Tool number 5 in the tool axis Z with a spindle speed of 2500 rpm and a feed of 350 mm/min is called. The allowance for the tool length and the tool radius is 0.2, the undersize for the tool radius is 1 mm.

20 TOOL CALL 5 Z S2500 F350 DL+0.2 DR-1

The **D** before the **L** and **R** stands for delta value.

## Tool compensation

### Introduction \*)



Tool correction

### Tool length compensation

The tool compensation for the length is effective as soon as you call up a tool and move it in the spindle axis. It is inactivated, as soon as a tool with the length L=0 is called (TOOL CALL 0).



#### Danger of collision:

##### Caution, danger of collision:

If you inactivate a length compensation with a positive value via **TOOL CALL 0**, the distance between tool and work piece is reduced. After a **TOOL CALL**, the programmed path of the tool changes in the spindle axis by the length difference between the old and the new tool.

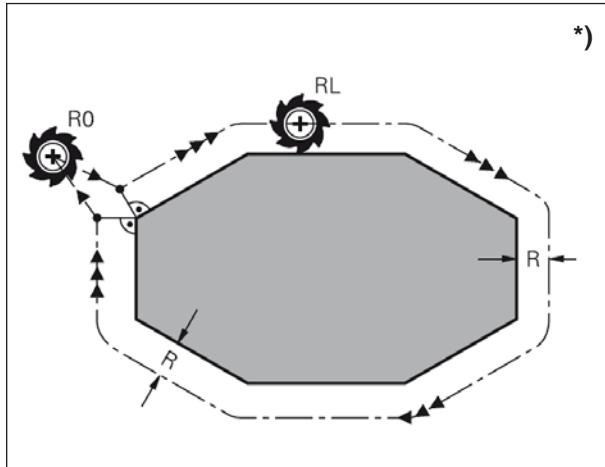
The length compensation takes delta values from the **TOOL CALL** block as well as from the tool table into account.

Compensation value =  $L + DL_{TOOL\ CALL} + DL_{TAB}$  with

**L:** tool length **L** from **TOOL DEF** block  
or from the tool table

**DL<sub>TOOL CALL</sub>:** Allowance  
**DL** for length from **TOOL CALL** block

**DL<sub>TAB</sub>:** Allowance **DL** for length from the tool table



Tool radius compensation

## Tool radius compensation \*)

The program block for a tool movement contains:

- **RL** or **RR** for a radius compensation
- **R0**, if no radius compensation is to be executed

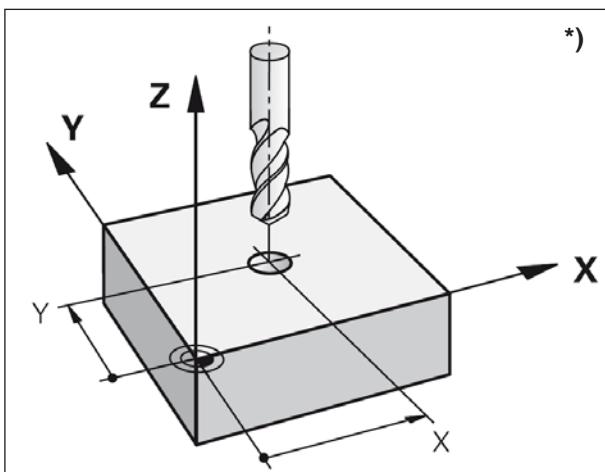
The radius compensation is effective, as soon as a tool is called up and moved in the machining plane with **RL** or **RR**.



### Note:

WinNC inactivates the radius compensation, if you:

- program a straight line block with **R0**
- move away from the contour with the **DEP** function
- program a **PGM CALL**
- select a new program with **PGM MGT**.



Tool radius compensation

The radius compensation takes delta values from the **TOOL CALL** block as well as from the tool table into account.

Compensation value =  $R + DR_{TOOL\ CALL} + DR_{TAB}$   
with

**R**: tool radius **R** from the **TOOL DEF** block or from the tool table

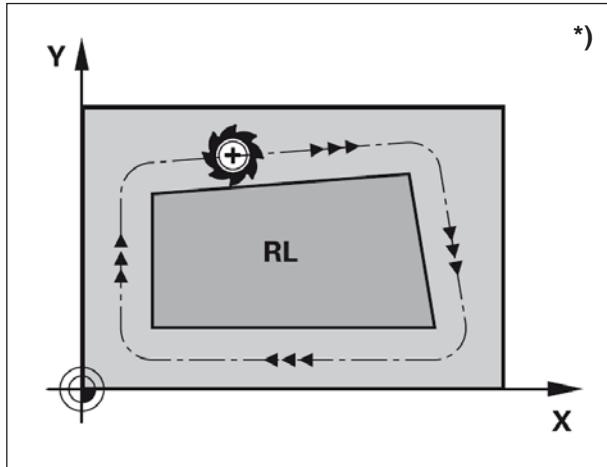
**DR<sub>TOOL CALL</sub>** : Allowance **DR** for radius from the **TOOL CALL** block

**DR<sub>TAB</sub>** : Allowance **DR** for radius from the tool table

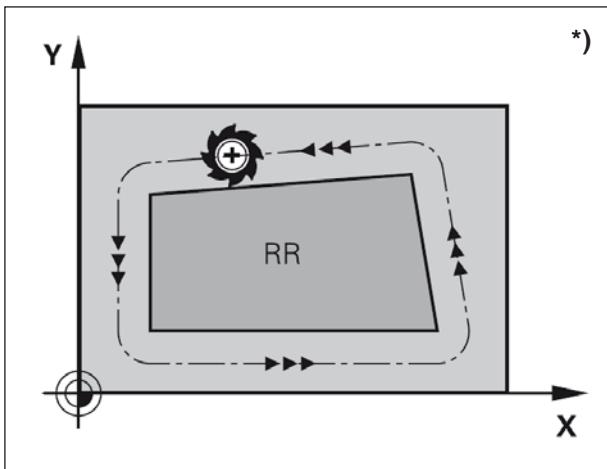
### Path movements without radius compensation: R0

The tool traverses in the machining plane with its centre point on the programmed path, and/or on the programmed coordinates.

Application: Drilling, pre-positioning.



Path movements



Path movements

### Contouring with radius compensation: RR and RL \*)

**RR** The tool moves on the right side of the contour  
**RL** The tool moves on the left side of the contour

In doing so, the tool centre point is at the distance of the tool radius from the programmed contour. "Right" and "left" refers to the position of the tool in movement direction along the Workpiece contour.



#### Note:

Between two program blocks with different radius compensation **RR** and **RL**, at least one movement block in the machining plane without radius compensation (with **R0**) must be inserted.

A radius compensation becomes effective towards the end of the block in which it has been programmed for the first time.

In case of the first block with radius compensation **RR/RL** and when cancelling with **R0**, WinNC will position the tool always vertically on the programmed starting or end point. Make sure to position the tool before the first contour point and/or behind the last contour point such as not to damage the contour.

### Entering the radius compensation



**ENT**



**RL**

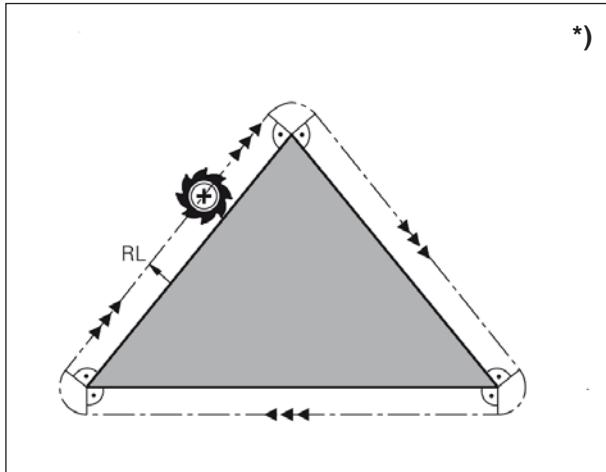


**RR**



**ENT**





Machining corners

### Radius compensation: Machining corners

- Outer corners:  
If you have programmed a radius compensation, WinNC will guide the tool at the outer corners on a transition circle. If required, WinNC reduces the feed at the outer corners, for example for significant direction changes.
- Inner corners:  
At inner corners, WinNC calculates the point of intersection of the paths, on which the tool centre is moved in a compensated manner. From this point onward, the tool moves along the closest contour element.. Thus, the work piece is not damaged at the inner corners. It follows from this that the tool radius for a certain contour cannot be selected in any desired size magnitude.

**Note:**

Do not position the starting point or end point of an inner machining operation on a contour corner point, as otherwise the contour could be damaged.



# F: Program sequence

## Preconditions

### **Set reference point or zero points cycle 7**

The zero points used must be measured and entered.

### **Tools**

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

### **Reference point**

The reference point must be approached in all axes.

### **Machine**

The machine must be ready for operation.  
The workpiece must be securely clamped.  
Loose parts (spanners, etc.) must be removed from the working area to avoid collisions.  
The machine doors must be closed at the start of the program.

### **Alarms**

There must be no alarms pending.

## Program start, program stop



Press the "Cycle Start" button.



Stop program with "Cycle Stop", continue with "Cycle Start".



Cancel program with "Reset".

## Modes of operation program run



In block sequence program run, the WinNC executes a program up to the end of program or up to a manual and/or programmed interruption. After an interruption, the program run can be resumed again.



In the Program run single block mode of operation you execute each block separately by pressing the machine START button.

The following functions are available:

- Interrupt program run
- Start the program run from a certain block
- Block skip
- Edit the tool table TOOL.T
- Checking and changing Q parameters



Select program run single bloc / block sequence

Select a program for execution.

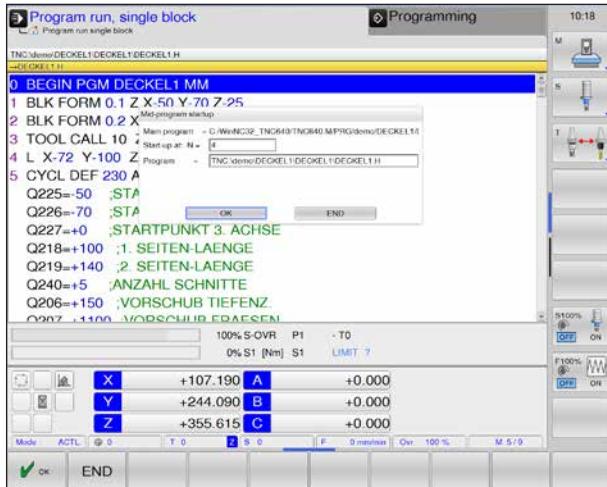
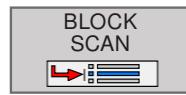


### Note:

Select the necessary tables and program files in the block sequence program mode and single block mode (status M).

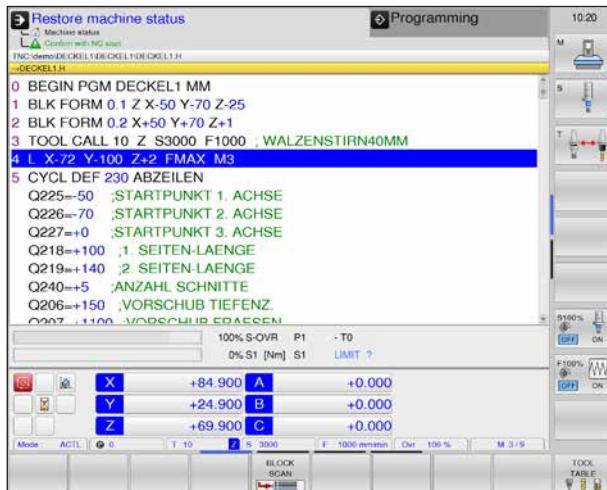
## Block scan

With the restore POS at N feature (block scan) you can start a part program at any block you desire.



- Open the dialog for BLOCK SCAN

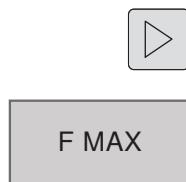
- **Start-up at N=** Enter the block number N at which the block scan should end.
- **Programm:** Shows the name of the program containing block N.



After executing block scan the following message appears: "Restore machine status".

Press NC-start to confirm restore machine status.

## F MAX \*)



- Switch the softkey bar until the function for F MAX appears.
- Press the softkey to open the dialogue für F MAX.

It is possible to reduce the feed rate when starting the NC program using the FMAX soft key.

The reduction applies to all rapid traverse and feed rate movements. The value you enter is no longer in effect after the machine has been turned off and on again. In order to re-establish the respectively defined maximum feed rate after switch-on, you need to re-enter the corresponding value.

The behavior of this function varies depending on the respective machine.

Refer to your machine manual.

## Optional block skip

In a test run or program run, the control can skip over blocks that begin with a slash "/".



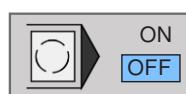
To run or test the program without the blocks preceded by a slash, set the soft key to ON.

To run or test the program with the blocks preceded by a slash, set the soft key to OFF.

## Optional program-run interruption

The WinNC optionally interrupts program run at blocks containing M1.

If you use M1 in the Program Run mode, the WinNC doesn't switch off the spindle or coolant.



Do not interrupt program run or test run at blocks containing M1: Set soft key to OFF

Interrupt program run or test run at blocks containing M1: Set soft key to ON.

# G: Flexible NC programming

## Q parameters

By means of Q parameters, an entire family of parts can be defined in one single machining program. For that, instead of numerical values, variable parameters are used, the Q parameters:

Among other things, Q parameters are used for

- Coordinate values
- Feeds
- Speeds
- Cycle data

Moreover, you can use Q parameters for programming contours that are defined via mathematical formulas.

Or you can use Q parameters for executing machining steps that depend on logical conditions.

Q parameters are characterised by letters and numbers. The letters define the parameter type, the numbers define the parameter scope.

### Instructions for programming:



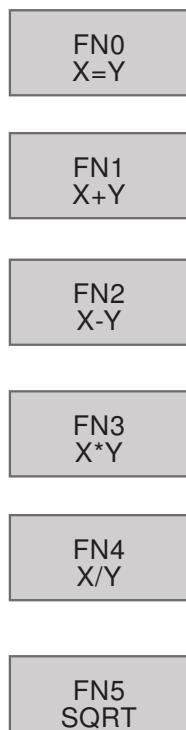
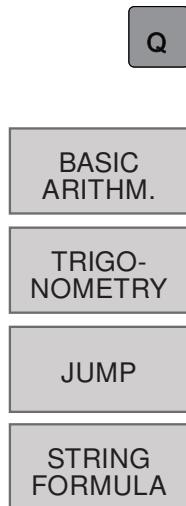
Q parameters and numerical values can be entered into the program in a mixed manner.

Permissible input scope:

Numerical values between -999 999 999 and +999 999 999. Maximum 16 digits are permitted, 9 of them in a pre-decimal position.

Type	Scope	Meaning
Q parameters		Parameters affect all programs in the WinNC memory
	0-30	Parameters for SL cycles
	31-99	Parameters for the user
	100-199	Parameters for special functions of WinNC
	200-1199	Parameters for Heidenhain cycles
	1200-1999	Parameters for the user
QL parameters		Parameters are only valid locally inside one program
	0-499	Parameters for the user
QR parameters		Parameters have a permanent effect on all programs in the WinNC memory, even beyond a power interruption
	0-499	Parameters for the user
QS parameters		
	0-1999	String parameters for the user

## Calling up Q parameters



Press the Q key during the input of a machining program.

The following functions are available:

**1 Basic mathematical functions**

**2 Trigonometric functions**

**3 If / then decisions, jumps**

**4 Function string formula**

Before string variables are used, they must be defined:

example: QS10="WERKSTÜCK"

## Basic mathematical functions

The following functions are available:

**1 FN 0: Assignment**

Directly assign a value  
e.g.: FN 0: Q5= +60

**2 FN 1: Addition**

Create the sum of two values and assign it  
e.g.: FN 1: Q1 = -Q2= +5

**3 FN 2: Subtraction**

Create the difference of two values and assign it  
e.g.: FN 2: Q1 = +10 - +5

**4 FN 3: Multiplication**

Create the product of two values and assign it  
e.g.: FN 3: Q2 = +3 \* +3

**5 FN 4: Division**

Create the quotient of two values and assign it  
e.g.: FN 4: Q4 = +8 DIV +Q2  
Dividing by 0 is not permitted!

**6 FN 5: Root**

Extract a root from a number and assign it  
e.g.: FN 5: Q20 = SQRT 4

Root of a negative numerical value is not permitted!

### Information:

On the right-hand side of the sign "=" you can enter:

- two numbers
- two Q parameters
- one number and one Q parameter
- Q-Parameters and numerical values in the equations can have algebraic signs.

## Trigonometric functions

The following functions are available:

FN6  
SIN(X)

FN7  
COS(X)

### 1 FN 6: SINE

Determine the sine of an angle in degrees ° and assign it

e.g.: FN 6: Q20= SIN-Q5

### 2 FN 7: COSINE

Determine the cosine of an angle in degrees ° and assign it

e.g.: FN 7: Q21 = -COS-Q5

Sine, cosine and tangent correspond to the aspect ratio of a rectangular triangle.

The following applies:

**Sine:** .....  $\sin \alpha = a / c$

**Cosine:** .....  $\cos \alpha = b / c$

**Tangent:** .....  $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

In this case:

- **c** is the side opposite the right angle
- **a** is the side opposite angle  $\alpha$
- **b** is the third side

From the tangent, WinNC can determine the angle:

$$\alpha = \arctan(a / b) = \arctan(\sin \alpha / \cos \alpha)$$

### Example

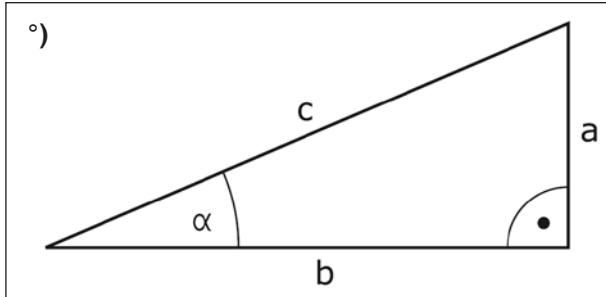
$$a = 25 \text{ mm}$$

$$b = 50 \text{ mm}$$

$$\alpha = \arctan(a / b) = \arctan(0.5) = 26.57^\circ$$

Additionally, the following applies:

$$a^2 + b^2 = c^2 \text{ (with } a^2 = a \times a\text{)}$$



## If/then decisions with Q parameters

WinNC compares a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, WinNC continues the machining program at the LABEL, which is programmed following the condition. If the condition is not fulfilled, WinNC executes the next block.

If you want to call up another program as a subprogram, program a PGM CALL following the LABEL.

### Unconditional jumps

Unconditional jumps are jumps, whose condition is always (=unconditionally) fulfilled, e. g.

FN9: IF+10 EQU+10 GOTO LBL1

### Programming if/then conditions

Press the JUMPS softkey, in order to display the if/then decisions:

#### 1 FN 9: If equal, jump

If both values or parameters are equal, jump to the specified label

e.g.: FN 9: IF +Q1 EQU +Q3 GOTO LBL 12

#### 2 FN 10: If not equal, jump

If both values or parameters are not equal, jump to the specified label

e.g.: FN 10: IF +10 NE -Q5 = GOTO LBL 10

#### 3 FN 11: If bigger, jump

If the first value or parameter is bigger than the second value or parameter, jump to the specified label

e.g.: FN 11: IF +Q1 GT +10 GOTO LBL 5

#### 4 FN 12: If smaller, jump

If the first value or parameter is smaller than the second value or parameter, jump to the specified label

e.g.: FN 12: IF +Q5 LT +0 GOTO LBL 8

JUMP

FN9  
IF X EQU Y  
GOTO

FN10  
IF X NE Y  
GOTO

FN11  
IF X GT Y  
GOTO

FN12  
IF X LT Y  
GOTO

# H: Alarms and Messages

## Machine Alarms 6000 - 7999

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

### PC MILL 50 / 55 / 100 / 105 / 125 / 155

#### Concept MILL 55 / 105 / 155

##### **6000: EMERGENCY OFF**

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

##### **6001: PLC-CYCLE TIME EXCEEDING**

Contact EMCO Service.

##### **6002: PLC - NO PROGRAM CHARGED**

Contact EMCO Service.

##### **6003: PLC - NO DATA UNIT**

Contact EMCO Service.

##### **6004: PLC - RAM MEMORY FAILURE**

Contact EMCO Service.

##### **6005: OVERHEAT BRAKEMODUL**

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

##### **6006: OVERLOAD BRAKE RESISTOR**

see 6005

##### **6007: SAFETY CIRCUIT FAULT**

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

##### **6008: MISSING CAN SUBSCRIBER**

Check fuses or EMCO customer service.  
Contact EMCO Service.

##### **6009: SAFETY CIRCUIT FAULT**

A running CNC program will be interrupted, the auxiliary drives will be stopped, the reference position will be lost.

##### **6010: DRIVE X-AXIS NOT READY**

The step motor board is defective or too hot, a fuse or cabling is defective.  
A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.  
Check fuses or contact EMCO service.

##### **6011: DRIVE Y-AXIS NOT READY**

see alarm 6010.

##### **6012: DRIVE Z-AXIS NOT READY**

see alarm 6010.

##### **6013: MAIN DRIVE NOT READY**

Main drive power supply defective, main drive too hot, fuse defective.  
A running program will be stopped, the auxilliary drives will be switched off.  
Check fuses or contact EMCO Service.

##### **6014: NO MAIN SPINDLE SPEED**

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

##### **6019: VICE TIME EXCEED**

The electric vice has not reached a stop position within 30 seconds.  
The control or the clamping device board are defective, the vice is stuck. Adjust the proximity switches of the stop position.

##### **6020: VICE FAILURE**

When the electric vice is closed, the signal "clamping device clamped" of the clamping device board has failed.  
The control, the clamping device board or the wiring are defective.

K 2017-07

**6022: CLAMPING DEVICE BOARD DEFECTIVE**

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

Replace the board.

**6024: MACHINE DOOR OPEN**

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

**6027: DOOR LIMIT SWITCH DEFECTIVE**

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

Contact EMCO service.

**6028: DOOR TIMEOUT**

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

**6030: NO PART CLAMPED**

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

**6040: TOOL TURRET INDEX FAILURE**

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

**6041: TOOL CHANGE TIMEOUT**

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

**6043-6046: TOOL DISK POSITION FAULT**

Position error of main drive, error of position supervising (inductive proximity switch defective or disadjusted, drum allowance), fuse defective, hardware defective.

The Z axis could have been slipped out of the toothings while the machine was switched off. A running CNC program will be stopped. Contact EMCO service.

**6047: TOOL DISK UNLOCKED**

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

A running CNC program will be interrupted. Contact EMCO service.

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

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

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

**6048: DIVIDING TIME EXCEEDED**

Dividing head stuck, insufficient pressured air supply, hardware defective.

Check for collision, check pressured air supply or contact EMCO service.

**6049: INTERLOCKING TIME EXCEEDED**

see alarm 6048

**6050: M25 AT RUNNING MAIN SPINDLE**

Cause: Programming mistake in NC program.

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: Correct NC program

**6064: DOOR AUTOMATIC NOT READY**

Cause: pressure failure automatic door  
automatic door sticks mechanically  
limit switch for open end position defective  
security print circuits defect  
cabling defective  
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic door

**6069: CLAMPING FOR TANI NOT OPEN**

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

**6070: PRESSURE SWITCH FOR TANI MISSING**

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

**6071: DIVIDING DEVICE NOT READY**

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

**6072: VICE NOT READY**

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

Vice sticks mechanically, insufficient compressed air supply, compressed air switch defective, fuse defective, hardware defective.

Check the fuses or contact EMCO service.

**6073: DIVIDING DEVICE NOT READY**

Cause: locking switch defective  
cabling defective  
fuses defective

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: service automatic dividing device  
lock the dividing device

**6074: DIVIDING TIME EXCEEDED**

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

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: Check for collision, check the compressed-air supply or contact the EMCO service.

**6075: M27 AT RUNNING MAIN SPINDLE**

Cause: Programming mistake in NC program.

A running program will be aborted.

The auxilliary drives will be switched off.

Remedy: Correct NC program

**6110: 5TH AXIS NOT CONNECTED**

Cause: 4th/5th axis was selected in EMConfig, but  
not connected electrically.

Remedy: Connect 4th/5th axis or deselect in EM-  
Config.

**6111: 5TH AXIS CONNECTED**

Cause: 4th/5th axis was deselected in EMConfig,  
but is electrically connected.

Remedy: Remove 4th/5th axis from machine or  
select in EMConfig.

**6112: MOTOR PROTECTION RELEASED**

Cause: A motor protection switch has triggered.  
A possibly active NC program is stopped  
immediately.

Remedy: Check the device connected to the motor  
protection switch and switch on again. If  
recurring, contact EMCO service department.

**7000: INVALID TOOL NUMBER PROGRAMMED**

The tool position was programmed larger than 10.  
The CNC program will be stopped.  
Interrupt program with RESET and correct the  
program.

**7001: NO M6 PROGRAMMED**

For an automatic tool change you also have to  
program a M6 after the T word.

**7007: FEED STOP!**

The axes have been stopped by the robotics in-  
terface (robotics entry FEEDHOLD).

**7016: SWITCH ON AUXILIARY DRIVES**

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

**7017: REFERENCE MACHINE**

Approach the reference point.

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

**7018: TURN KEY SWITCH**

With NC-Start the key switch was in position "set-  
ting 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 SIN-  
GLE block operation.

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

**7021: INITIALIZE TOOL TURRET**

The tool turret operating was interrupted.

No traversing operation is possible.

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

**7022:INITIALIZE TOOL TURRET!**

see 7021

**7023: WAITING TIME MAIN DRIVE!**

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

**7038: LUBRICATION SYSTEM FAULT**

The pressure switch is defective or gagged.

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

Contact EMCO service.

**7039: LUBRICATION SYSTEM FAULT**

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

**7040: MACHINE DOOR OPEN**

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

**7042: INITIALIZE MACHINE DOOR**

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

**7043: PIECE COUNT REACHED**

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

**7050: NO PART CLAMPED**

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

Traverse the vice manually on a valid end position.

**7051: DIVIDING HEAD NOT LOCKED!**

Either the dividing head is in an undefined position after the machine has been switched on, or the locking signal after a dividing process is missing.

Initiate the dividing process, check, respectively adjust the proximity switch for locking.

**7054: VICE OPEN**

Cause: the workpiece is not clamped

When switching on the main spindle with M3/M4 alarm 6072 (vice not ready) will be released.

Remedy: Clamp

**7055: OPEN TOOL CLAMPING SYSTEM**

A tool is clamped in the main spindle and the control does not recognize the corresponding T number.

Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and "1".

**7056: SETTING DATA INCORRECT**

An invalid tool number is stored in the setting data.

Delete the setting data in the machine directory xxxx.pls.

**7057: TOOLHOLDER OCCUPIED**

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

**7058: RETRACTING THE AXES**

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

**7087: MOTOR PROTECTION HYDRAULIC CLAMPING RELEASED!**

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

**7090: ELECTRICAL CABINET OVERRIDE SWITCH ACTIVE**

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

**7107: OTOR PROTECTION RELEASED**

A motor protection switch has triggered. A possibly running NC program will be completed. A new NC start will be prevented.  
Check the device connected to the motor protection switch and switch on again. If recurring, contact EMCO service department.

**7270: OFFSET COMPENSATION ACTIVE !**

Only with PC-MILL 105

Offset compensation activated by the following operation sequence.

- Reference point not active
  - Machine in reference mode
  - Key switch in manual operation
  - Press STRG (or CTRL) and simultaneously 4
- This must be carried out if prior to the tool change procedure spindle positioning is not completed (tolerance window too large)

**7271: COMPENSATION FINISHED, DATA SAVED !**

see 7270



**PC TURN 50 / 55 / 105 / 120 / 125 / 155**  
**Concept TURN 55 / 60 / 105 / 155 /**  
**250 / 260 / 460**  
**Concept MILL 250**  
**EMCOMAT E160**  
**EMCOMAT E200**  
**EMCOMILL C40**  
**EMCOMAT FB-450 / FB-600**

**6000: EMERGENCY OFF**

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

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

Check fuses or contact EMCO service.

**6001: PLC-CYCLE TIME EXCEEDING**

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

**6011: DRIVE Z-AXIS NOT READY**  
see 6010.**6012: DRIVE Z-AXIS NOT READY**  
see 6010.**6002: PLC - NO PROGRAM CHARGED**

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

**6013: MAIN DRIVE NOT READY**

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

A running program will be stopped, the auxilliary drives will be switched off.  
Check fuses or contact EMCO Service.

**6004: PLC - RAM MEMORY FAILURE**  
The auxiliary drives will be switched off.  
Contact EMCO Service.**6014: NO MAIN SPINDLE SPEED**

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

**6005: K2 OR K3 NOT DE-ENERGIZED**  
Turn machine on/off. Defective security board.**6015: NO DRIVEN TOOL SPINDLE SPEED**  
see 6014.**6007 SAFETY CIRCUIT FAULT**

**6008: MISSING CAN SUBSCRIBER**  
The PLC-CAN board is not identified by the control.  
Check the interface cable and the power supply of the CAN board.

**6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING****6009: SAFETY CIRCUIT FAULT**

**6010: DRIVE X-AXIS NOT READY**  
The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.

**6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING**

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

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

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

Turn machine on/off. Defective security board.

**6019: POWER SUPPLY MODULE NOT READY**

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

**6020: DRIVEN TOOL DRIVE MALFUNCTION**

The driven tool power supply defective or the driven tool drive is too hot, fuse defective, mains overvoltage or undervoltage. A running CNC program will be aborted, the auxiliary drives will be stopped. Check the fuses or contact EMCO service.

**6021: COLLET TIME OUT**

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

**6022: CLAMPING DEVICE BOARD DEFECTIVE**

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

**6023: COLLET PRESSURE MONITORING**

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

**6024: MACHINE DOOR OPEN**

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

**6025: GEARBOX COVER NOT CLOSED**

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

Close the cover to continue.

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

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

**6028: DOOR TIMEOUT**

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

**6029: TAILSTOCK QUILL TIME EXCEED**

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

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

**6030: NO PART CLAMPED**

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

**6031: QUILL FAILURE****6032: TOOL CHANGE TIMEOUT**

see alarm 6041.

**6033: TOOL TURRET SYNC ERROR**

Hardware defective.

Contact EMCO service.

**6037: CHUCK TIMEOUT**

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

**6039: CHUCK PRESSURE FAILURE**

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

**6040: TOOL TURRET INDEX FAILURE**

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

A running CNC program will be stopped.

Swivel the tool turret with the tool turret key, check fuses or contact EMCO service.

**6041: TOOL CHANGE TIMEOUT**

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

A running CNC program will be stopped.

Check for collisions, check fuses or contact EMCO service.

**6042: TOOL TURRET OVERHEAT**

Tool turret motor too hot.

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

**6043: TOOL CHANGE TIMEOUT**

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

A running CNC program will be stopped.

Check for collisions, check fuses or contact EMCO service.

**6044: BRAKING RESISTANCE - MAIN DRIVE OVERLOADED**

Reduce number of speed changes in the program.

**6045: TOOL TURRET SYNC MISSING**

Hardware defective.

Contact EMCO service.

**6046: TOOL TURRET ENCODER FAULT**

Fuse defective, hardware defective.

Check fuses or contact EMCO service.

**6048: CHUCK NOT READY**

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

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

Check fuses or contact EMCO service.

**6049: COLLET NOT READY**

see 6048

**6050: M25 DURING SPINDLE ROTATION**

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

**6055: NO PART CLAMPED**

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

**6056: QUILL NOT READY**

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

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

Check for collisions, check fuses or contact EMCO service.

**6057: M20/M21 DURING SPINDLE ROTATION**

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

**6058: M25/M26 DURING QUILL FORWARD**

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

**6059: C-AXIS SWING IN TIMEOUT**

C-axis does not swivel in within 4 seconds.

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

**6060: C-AXIS INDEX FAILURE**

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

Check pneumatics, mechanics and limit switch.

**6064: AUTOMATIC DOOR NOT READY**

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

Check for collisions, check fuses or contact EMCO service.

**6065: LOADER MAGAZINE FAILURE**

Loader not ready.

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

**6066: CLAMPING DEVICE FAILURE**

No compressed air at the clamping device

Check pneumatics and position of the clamping device proximity detectors.

**6067: NO COMPRESSED AIR**

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

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

Cause: The axis arrived in the tailstock sleeve.

Remedy: Drive the travel off the tailstock sleeve.

**6071: LIMIT SWITCH X AXIS ACTIVE**

Cause: The axis arrived to the end switch.

Remedy: Drive the axis off the end switch again.

**6072: LIMIT SWITCH Z AXIS ACTIVE**

see 6071

**6073: CHUCK GUARD OPEN**

Cause: The chuck guard is open.

Remedy: Close the chuck guard.

**6074: NO FEEDBACK FROM USB-PLC**

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

**6075: AXIS LIMIT SWITCH TRIGGERED**

see 6071

**6076: DRIVE Y-AXIS NOT READY**

see 6010

**6077 VICE NOT READY**

Cause: Loss of pressure in clamping system.

Remedy: Check pressurised air and air ducts.

**6078 MOTOR PROTECTION TOOL MAGAZINE RELEASED**

Cause: Swing intervals are too short.

Remedy: Raise swing intervals.

**6079 MOTOR PROTECTION TOOL CHANGER RELEASED**

see 6068

**6080 PRESSURE SWITCH FOR TANI MISSING**

Cause: The pressure switch fails to active when the clamping closes. No pressurised air or mechanical problem.

Remedy: Check pressurised air.

**6081 CLAMPING FOR TANI NOT OPEN**

see 6080

**6082 FAULT AS/SIGNAL**

Cause: Active Safety-Signal X/Y-controller is faulty.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

**6083 FAULT AS/SIGNAL**

Cause: Active Safety-Signal main spindle/Z-controller is faulty.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

**6084 FAULT AS/SIGNAL UE-MODUL**

Cause: Active Safety-Signal Uncontrolled power supply module is faulty.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

**6085 N=0 RELAY NOT DE-ENERGIZED**

Cause: Rotation zero relay did not drop.

Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO (replace relay).

**6086 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC**

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

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

**6087 DRIVE A-AXIS NOT READY**

see 6010

**6088 PROTECT SWITCH DOOR CONTROL UNIT RELEASED**

Cause: Door drive overload.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace motor, drive).

**6089 DRIVE B-AXIS NOT READY**

see 6010

**6090 CHIP CONVEYOR CONTACTOR NOT DE-ENERGIZED**

Cause: Chip conveyor guard not down.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace guard).

**6091 AUTOMATIC DOOR CONTACTOR NOT DE-ENERGIZED**

Cause: Automatic door guard not down.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace guard).

**6092 EMERGENCY-OFF EXTERNAL**

**6093 FAULT AS/SIGNAL A-AXIS**

Cause: Active Safety-Signal A control element faulty.

Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO.

**6095 OVERHEATING IN THE SWITCHGEAR CABINET**

Cause: Temperature monitoring responded.

Remedy: Check switchgear cabinet filter and fan, raise triggering temperature, switch machine on and off.

**6096 SWITCHGEAR CABINET DOOR OPEN**

Cause: Switchgear cabinet door opened without key switch release.

Remedy: Close switchgear cabinet door, switch machine off and on.

**6097 EMERGENCY-OFF TEST REQUIRED**

Cause: Functional test of Emergency off.  
 Remedy: Press EMERGENCY-OFF button on the control panel and unlock again. Press Rest button to acknowledge the emergency stop state.

**6098 FLOAT SWITCH HYDRAULIC MISSING**

Effect: Auxiliary units are switched off.  
 Meaning: The hydraulic float switch has released.  
 Remedy: Refill hydraulic oil.

**6099 PROX. SWITCH SPINDLE-BRAKE 1 MISSING**

Effect: Feed stop, Read in lock  
 Meaning: M10 Spindle brake on → Inductive proximity switch stays 0.  
           M11 Spindle brake off → Inductive proximity switch stays 1.  
 Remedy: Check inductive proximity switch, check magnetiv valve spindle brake.

**6100 LOW PRESSURE QUILL**

Effect: Auxiliary units are switched off.  
 Meaning: At the moment of a spindle start command the tailstock pressure was not built up or the pressure dropped while spindle run.  
 Remedy: Check clamping device pressure and pressure switch.  
 Check program.

**6101 QUILL -B3 OR -B4 MISSING**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for the movement of the quill was activated, but the switches –B3 and –B4 does not alter its status.  
 Remedy: Check switches, magnetic valves.

**6102 QUILL POSITION ALARM (PART MOVED?)**

Effect: Feed stop, Read in lock  
 Meaning: The tailstock target position was overtravelled in AUTOMATIC mode.  
 Remedy: Check tailstock target position, check technology (clamping device pressure higher, tailstock pressure lower)

**6103 QUILL NO BACKPOSITION**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for the tailstock was activated, but the switch for "tailstock back" stays 0.  
 Remedy: Check magnetic valve, check switch.

**6104 LOW PRESSURE CLAMPING EQUIPMENT 1**

Effect: Auxiliary units are switched off.  
 Meaning: At the moment of a spindle start command the clamping pressure was not built up or the clamping pressure dropped while spindle run.  
 Remedy: Check clamping device pressure and pressure switch.  
 Check program.

**6105 CLAMPING EQUIPMENT 1 NOT OPEN**

Effect: Feed stop, Read in lock  
 Meaning: The analogous proximity switch for clamping device 1 was not actuated.  
 Remedy: Adjust inductive proximity switch (see "Machine Description - chapter "C Programming and operation").

**6106 CLAMPING EQUIPMENT 1 NOT CLOSED**

Effect: Feed stop, Read in lock  
 Meaning: The pressure switch for "clamping device closed" does not switch.  
 Remedy: Check pressure switch

**6107 LIMIT SWITCH CLAMPING EQUIPMENT 1**

Effect: Auxiliary units are switched off.  
 Remedy: Adjust clamping device (do not clamp in end position of the clamping system - see "Machine Description, chapter C Programming and Operation").

**6108 PARTS CATCHER FORWARD MISSING**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for "collection tray forward/backward" was activated, but the switch for "collection tray forward/backward" does not alter its status.  
 Remedy: Check switches, magnetic valves.

**6109 PARTS CATCHER ROTATE IN MISSING**

Effect: Feed stop, Read in lock  
 Meaning: The magnetic valve for "collection tray in/out" was activated, but the switch for "collection tray in/out" does not alter its status.  
 Remedy: Check switch, magnetic valve.

**6900 USBPLC not available**

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

**6901 Error emergency-off relay USBPLC**

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

**6902 Error standstill monitoring X**

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

**6903 Error standstill monitoring Z**

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

**6904 Error alive circuit PLC**

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

**6906 Error overspeed spindle**

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

**6907 Error enable pulses ER-module**

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

**6908 Error standstill monitoring main drive**

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

**6909 Error main drive enable without spindle start**

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

**6910 Error standstill monitoring Y**

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

**6911 Error standstill axes**

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

**6912 Error overspeed axis**

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

**6913 Error overspeed X**

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

**6914 Error overspeed Y**

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

**6915 Error overspeed Z**

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

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

**6916 ERROR: X-INDUCTIVE PROXIMITY SWITCH DEFECT**

**Cause:** No signal is delivered by X axis Bero.

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

**6917 ERROR: Y-INDUCTIVE PROXIMITY SWITCH DEFECT**

**Cause:** No signal is delivered by Y axis Bero.

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

**6918 ERROR: Z-INDUCTIVE PROXIMITY SWITCH DEFECT**

**Cause:** No signal is delivered by Z axis Bero.

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

**6919 ERROR: SPINDLE-INDUCTIVE PROXIMITY SWITCH DEFECT**

**Cause:** No signal is delivered by main spindle Bero.

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

**6920 INVERSION OF X-DIRECTION TOO LONG "1"**

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

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

**6921 INVERSION OF Y-DIRECTION TOO LONG "1"**

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

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

**6922 INVERSION OF Z-DIRECTION TOO LONG "1"**

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

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

**6923 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC**

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

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

**6924 ERROR ENABLE PULSES MAIN DRIVE**

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

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

**6925 GRID PROTECTION ERROR!**

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

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

**6926 MOTOR PROTECTION ERROR!**

**Cause:** Motor protection drops out in current operating state.

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

**6927 EMERGENCY OFF ACTIVE ERROR!**

**Cause:** Emergency off button was pressed.

**Remedy:** Restart the machine.

**6928 TOOL CHANGER SHUTDOWN MONITORING ERROR**

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

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

**6929 MACHINE DOOR CLOSING/LOCKING ERROR**

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

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

### **6930 BEROS MAIN SPINDLE PLAUSIBILITY ERROR**

Cause: Beros main spindle signal different.  
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

### **6931 MAIN DRIVE QUICK STOP FUNCTION PLAUSIBILITY ERROR**

Cause: Main drive actuator does not confirm the quick stop function in the current operating state.  
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

### **6999 USB-EXTENSION FOR ROBOTIK NOT AVAILABLE**

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

### **7000: INVALID TOOL NUMBER PROGRAMMED**

The tool position was programmed larger than 8. The CNC program will be stopped.  
Interrupt program with RESET and correct the program.

### **7007: FEED HOLD**

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

### **7016: SWITCH ON AUXILIARY DRIVES**

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

### **7017: REFERENCE MACHINE**

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

### **7018: TURN KEY SWITCH**

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

### **7019: PNEUMATIC LUBRICATION MONITORING!**

Refill pneumatic oil

### **7020: SPECIAL OPERATION MODE ACTIVE**

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

### **7021: TOOL TURRET NOT LOCKED**

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

### **7022: COLLECTION DEVICE MONITORING**

Time exceed of the swivelling movement.  
Check the pneumatics, respectively whether the mechanical system is jammed (possibly a work-piece is jammed).

### **7023: ADJUST PRESSURE SWITCH !**

During opening and closing of the clamping device the pressure switch has to turn off and on once.  
Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

### **7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !**

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

### **7025 WAITING TIME MAIN DRIVE !**

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

### **7026 PROTECTION MAIN MOTOR FAN RELEASED!**

**7038: LUBRICATION SYSTEM FAULT**

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

**7039: LUBRICATION SYSTEM FAULT**

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

**7040: MACHINE DOOR OPEN**

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

**7041: GEARBOX COVER OPEN**

The main spindle cannot be switched on and NC start cannot be activated. Close the gearbox cover in order to start a CNC program.

**7042: INITIALIZE MACHINE DOOR**

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

**7043: PIECE COUNT REACHED**

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

**7048: CHUCK OPEN**

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

**7049: CHUCK - NO PART CLAMPED**

No part is clamped, the spindle can not be switched on.

**7050: COLLET OPEN**

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

**7051: COLLET - NO PART CLAMPED**

No part is clamped, the spindle can not be switched on.

**7052: QUILL IN UNDEFINED POSITION**

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

**7053: QUILL - NO PART CLAMPED**

The tailstock reached the front end position. Traverse the tailstock back to the back end position to continue.

**7054: NO PART CLAMPED**

No part clamped, switch-on of the spindle is locked.

**7055: CLAMPING DEVICE OPEN**

This message indicates that the clamping device is not in clamping state. It disappears as soon as a part is clamped.

**7060 RETRACT SLEEVE LIMIT SWITCH !**

The axis arrived in the tailstock sleeve. Drive the travel off the tailstock sleeve.

**7061 RETRACT X AXIS LIMIT SWITCH !**

The axis arrived to the end switch. Drive the axis off the end switch again.

**7062 RETRACT Z AXIS LIMIT SWITCH !**

see 7061

**7063 OIL LEVEL CENTRAL LUBRICATION !**

Low oil level in central lubrication. Refill oil as per maintenance instructions to the machine.

**7064 CHUCK GUARD OPEN !**

The chuck guard is open. Close the chuck guard.

**7065 MOTOR PROTECTION COOLANT PUMP RELEASED !**

Overheated coolant pump. Check the coolant pump for ease of motion and presence of dirt. Ensure sufficient amount of coolant fluid in the coolant facility.

**7066 CONFIRM TOOL !**

To confirm the tool change, press T after the change has been completed.

**7067 MANUAL OPERATING MODE**

The Special Operation key switch is in the Set position (manual).

**7068 X AXIS HANDWHEEL ACTIVE**

The safety wheel is locked for manual travel movement. The safety wheel locking is monitored by contactless switches. With the manual wheel locked, the axis feed cannot be switched on. For automatic processing of a program, the manual wheel must be released again.

**7069 Y AXIS HANDWHEEL ACTIVE**

see 7068

**7070 Z AXIS HANDWHEEL ACTIVE**

see 7068

**7071 VERTICAL TOOL CHANGE**

The sheath for manual clamping of the tool holder is monitored by a switch. The switch reports an unaccepted socket wrench or a sheath which was left open. Remove the socket wrench after clamping the tool and close the sheath.

**7072 HORIZONTAL TOOL CHANGE**

The turning knob for manual tool clamping on the horizontal spindle is monitored by a switch. The switch reports a tightened turning knob. The spindle gets locked. Release the turning knob after clamping the tool.

**7073 RETRACT Y AXIS LIMIT SWITCH !**

see 7061

**7074 CHANGE TOOL**

Clamp programmed tool.

**7076: SWIVEL UNIT VOR MILLING HEAD UNLOCKED**

The milling head is not fully swung. Fix the milling head mechanically (the end switch must be pushed).

**7077: ADJUST TOOL TURRET**

No valid machine data for tool change are available. Contact EMCO.

**7078: POCKET NOT IN HOME POSITION**

Cancel during tool change. Swing back tool recessed in setup operation.

**7079: TOOL ARM NOT IN HOME POSITION**

see 7079

**7080: INCORRECT TOOL CLAMPED !**

The tool cone lies beyond tolerance. The clamped tool is twisted by 180°. Bero tool clamping is displaced. Check the tool and clamp it again. If this problem occurs with more tools, contact EMCO.

**7082: MOTOR PROTECTION CHIP CONVEYOR RELEASED**

The scrap belt is overloaded. Check the conveyor belt for ease of motion and remove jammed scrap.

**7083: MAGAZINE IS ACTIVE !**

A tool has been removed from the non-chaotic tool administration from the main spindle. Fill the tool drum.

**7084: VICE OPEN !**

The vice is not clamped. Clamp the vice.

**7085 ROUNDAXIS A MOVE TO 0 DEGRE !**

Cause: The MOC only shuts down if the A Round axis is at 0°.

When 4.5. is present, a round axis must be made each time before the machine is switched off.

Remedy: Move round axis to 0°.

**7088 SWITCHGEAR CABINET OVERHEATING**

Cause: Temperature monitoring responded.

Remedy: Check switchgear cabinet filter and fan, raise trigger temperature.

**7089 SWITCHGEAR CABINET DOOR OPEN**

Cause: Switchgear cabinet door open.

Remedy: Close switchgear cabinet door.

**7091 WAIT FOR USB-I2C SPS**

Cause: Communication with the USB-I2C PLC could not be established.

Remedy: If the message does not self-extinguish, switch off and on the machine. Please contact EMCO service when the message occurs permanently even after power off.

**7092 TEST STOP ACTIVE**

Cause: Safety test for monitoring the safety functions is active.

Remedy: Wait until the safety test is finished.

**7093 SET REFERENCE POINT**

Cause: The reference point acknowledge mode was activated by the operator.

**7094 SET X-REFERENCE POINT**

Cause: The reference value of the X axis was adopted to the acc.esd file.

**7095 SET Y-REFERENCE POINT**

Cause: The reference value of the Y axis was adopted to the acc.esd file.

**7096 SET Z-REFERENCE POINT**

Cause: The reference value of the Z axis was adopted to the acc.esd file.

**7097 FEED OVERRIDE 0**

Cause: The override switch (feed control) was set by the operator on 0%

**7098 SPINDLE-BRAKE 1 ACTIVE**

Effect: spindle stop.

**7099 QUILL DRIVES FORWARD**

Effect: Feed stop, Read in lock

Meaning: M21 was programmed → pressure switch "tailstock in front position" is not yet 1.

Remedy: Disappears with pressure switch front position.

**7100 QUILL DRIVES BACKWARD**

Effect: Feed stop, Read in lock

Meaning: M20 was programmed → limit switch "tailstock back" is not yet 1

Remedy: Disappears with limit switch "back position".

**7101 REFERENCE POINT TOOL TURRET MISSING**

Effect: Feed stop, Read in lock

Meaning: At NC start the tool turret is not referenced.

Remedy: Reference tool turret in JOG mode with tool turret key.

**7102 TOOL TURRET IN MOTION**

Effect: Tool turret swivels to the programmed position.

**7103 LIMIT-SWITCH CLAMPING EQUIPMENT 1**

Effect: Locking of NC start and main drive start, spindle stop S1

Meaning: The pick-up for the analogue value detects the clamping position as end position.

Remedy: Change clamping range of the clamping device (see "Machine Description - chapter C Programming and Operation").

**7104 QUILL IN INTERMEDIATE POSITION**

Effect: Feed hold / read in stop.

**7105 SET AWZ REFERENCE POINT**

Effect: The reference value of the AWZ motor was transferred to the acc.msd file.

**7900 INITIALIZE EMERGENCY OFF!**

Cause: The emergency off button must be initialized.

Remedy: Press and then release emergency off button.

**7901 INITIALIZE MACHINE DOORS!**

Cause: The machine doors must be initialized.

Remedy: Open the machine doors and close them again.

**7906 SET A-REFERENCE POINT**

Cause: The reference value of the A axis was adopted to the acc.esd file.

## Inputunit alarms 1700 - 1899

These alarms and messages are raised by the control keyboard.

### 1701 Error in RS232

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

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

### 1703 Ext. keyboard not available

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

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

### 1704 Ext. keyboard: checksum error

Cause: Error in the transmission.

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

### 1705 Ext. keyboard: general error

Cause: The attached keyboard reported an error.

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

### 1706 General USB error

Cause: Error in the USB communication.

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

### 1707 Ext. Keyboard: no LEDs

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

Remedy: EMCO-Service kontaktieren.

### 1708 Ext. Keyboard: unknown command

Cause: Unknown command was sent to the keyboard.

Remedy: Contact EMCO Customer Service

### 1710 Installation of Easy2control is damaged!

Cause: Incorrect installation of Easy2control

Remedy: Reinstall software and/or contact EMCO Customer Service

### 1711 Initialization of Easy2Control failed!

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

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

### 1712 USB-Dongle for Easy2control could not be found!

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

Remedy: Connect USB-Dongle for Easy2control.

### 1801 Keytable not found!

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

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

### 1802 Connection to keyboard lost

Cause: Connection to the serial keyboard was interrupted.

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



## Axis Controller Alarms

### 8000 - 9000, 22000 - 23000, 200000 - 300000

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

Cause: Internal error

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

**8101 Fatal init error AC**

see 8101.

**8102 Fatal init error AC**

see 8101.

**8103 Fatal init error AC**

see 8101.

**8104 Fatal system error AC**

see 8101.

**8105 Fatal init error AC**

see 8101.

**8106 No PC-COM card found**

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

Remedy: Mount board, adjust other address with jumper

**8107 PC-COM card not working**

see 8106.

**8108 Fatal error on PC-COM card**

see 8106.

**8109 Fatal error on PC-COM card**

see 8106.

**8110 PC-COM init message missing**

Cause: Internal error

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

**8111 Wrong configuration of PC-COM**

see 8110.

**8113 Invalid data (pccom.hex)**

see 8110.

**8114 Programming error on PC-COM**

see 8110.

**8115 PC-COM packet acknowledge missing**

see 8110.

**8116 PC-COM startup error**

see 8110.

**8117 Fatal init data error (pccom.hex)**

see 8110.

**8118 Fatal init error AC**

see 8110, ev. insufficient RAM memory

**8119 PC interrupt no. not valid**

Cause: The PC interrupt number can not be used.

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

**8120 PC interrupt no. unmaskable**

see 8119

**8121 Invalid command to PC-COM**

Cause: Internal error or defective cable

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

**8122 Internal AC mailbox overrun**

Cause: Internal error

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

**8123 Open error on record file**

Cause: Internal error

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

**8124 Write error on record file**

Cause: Internal error

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

**8125 Invalid memory for record buffer**

Cause: Insufficient RAM, record time exceeding.

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

**8126 AC Interpolation overrun**

Cause: Ev. insufficient computer performance.

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

**8127 Insufficient memory**

Cause: Insufficient RAM

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

**8128 Invalid message to AC**

Cause: Internal error

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

**8129 Invalid MSD data - axisconfig.**

see 8128.

**8130 Internal init error AC**

see 8128.

**8130 Internal init error AC**

see 8128.

**8132 Axis accessed by multiple channels**  
see 8128.

**8133 Insufficient NC block memory AC**  
see 8128.

**8134 Too much center points programmed**  
see 8128.

**8135 No centerpoint programmed**  
see 8128.

**8136 Circle radius too small**  
see 8128.

**8137 Invalid for Helix specified**

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

Remedy: Program correction.

**8140 Maschine (ACIF) not responding**

Cause: Machine off or not connected.

Remedy: Switch on machine or connect.

**8141 Internal PC-COM error**

Cause: Internal error

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

**8142 ACIF Program error**

Cause: Internal error

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

**8143 ACIF packet acknowledge missing**

see 8142.

**8144 ACIF startup error**

see 8142.

**8145 Fatal init data error (acif.hex)**

see 8142.

**8146 Multiple request for axis**

see 8142.

**8147 Invalid PC-COM state (DPRAM)**

see 8142.

**8148 Invalid PC-COM command (CNo)**

see 8142.

**8149 Invalid PC-COM command (Len)**

see 8142.

**8150 Fatal ACIF error**

see 8142.

**8151 AC Init Error (missing RPG file)**

see 8142.

**8152 AC Init Error (RPG file format)**

see 8142.

**8153 FPGA program timeout on ACIF**

see 8142.

**8154 Invalid Command to PC-COM**

see 8142.

**8155 Invalid FPGA packet acknowledge**

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

**8156 Sync within 1.5 revol. not found**

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

**8157 Data record done**

see 8142.

**8158 Bero width too large (referencing)**

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

**8159 Function not implemented**

Bedeutung: In normal operation this function can not be executed

**8160 Axis synchronization lost axis 3..7**

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

Remedy: Approach reference point

**8161 X-Axis synchronization lost**

Step loss of the step motor. Causes:

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

**8162 Y-Axis synchronization lost**

see 8161

**8163 Z-Axis synchronization lost**

see 8161

**8164 Software limit switch max axis 3..7**

Cause: Axis is at traverse area end

Remedy: Retract axis

**8168 Software limit overtravel axis 3..7**

Cause: Axis is at traverse area end

Remedy: Retract axis

**8172 Communication error to machine**

Cause: Internal error

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

Check connection PC - machine, eventually eliminate distortion sources.

**8173 INC while NC program is running**

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

**8174 INC not allowed**

Cause: At the moment the axis is in motion.

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

**8175 MSD file could not be opened**

Cause: Internal error

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

**8176 PLS file could not be opened**

see 8175.

**8177 PLS file could not be accessed**  
see 8175.

**8178 PLS file could not be written**  
see 8175.

**8179 ACS file could not be opened**  
see 8175.

**8180 ACS file could not be accessed**  
see 8175.

**8181 ACS file could not be written**  
see 8175.

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

**8184 Invalid interpolaton command**

**8185 Forbidden MSD data change**  
see 8175.

**8186 MSD file could not be opened**  
see 8175.

**8187 PLC program error**  
see 8175.

**8188 Gear command invalid**  
see 8175.

**8189 Invalid channel assignement**  
see 8175.

**8190 Invalid channel within message**  
see 8175.

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

**8192 Invalid axis in command**  
see 8175.

**8193 Fatal PLC error**  
see 8175.

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

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

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

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

**8198 Internal error (to manny threads)**  
see 8175.

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

**8200 Thread without spindle on**  
Remedy: Switch on spindle

**8201 Internal thread error (IPO)**  
see 8199.

**8201 Internal thread error (IPO)**  
see 8199.

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

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

**8205 PLC Runtime exceeded**  
Cause: Insufficient computer performance

**8206 Invalid PLC M-group initialisation**  
see 8199.

**8207 Invalid PLC machine data**  
see 8199.

**8208 Invalid application message**  
see 8199.

**8212 Rotation axis not allowed**  
see 8199.

**8213 Circle and rotation axis can't be interpolated**

**8214 Thread and rotation axis can't be interpolated**

**8215 Invalid state**  
see 8199.

**8216 No rotation axis for rotation axis switch**  
see 8199.

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

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

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

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

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

**8222 New master spindle is not valid**

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

Remedy: Correct the spindle number.

**8224 Invalid stop mode**

see 8199.

**8225 Invalid parameter for BC\_MOVE\_TO\_IO!**

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

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

**8226 Rotary axis switch not valid (MSD data)!**

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

**8228 Rotary axis switch not allowed while axis move!**

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

Remedy: Stop the rotary axis before switching.

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

Cause: Transmit is not possible at this machine.

**8232 Axis configuration (MSD) for TRACYL not valid!**

Cause: Tracyl is not possible at this machine.

**8233 Axis not available while TRANSMIT/TRACYL is active!**

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

**8234 Axis control grant removed by PLC while axis interpolates!**

Cause: Internal error

Remedy: Delete error with reset and inform EMCO.

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

see 8234.

**8236 TRANSMIT/TRACYL activated while axis or spindle moves!**

see 8234.

**8237 Motion through pole in TRANSMIT!**

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

Remedy: Alter the traversing movement.

**8238 Speed limit in TRANSMIT exceeded!**

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

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

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

for CT155/CT325/CT450:

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

for CT250:

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

This applies for rapid traverse in transmit:

CT155/250/325: 4200 mm/min

CT450: 3,500 mm/min

**8239 DAU exceeded 10V limit!**

Cause: Internal error

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

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

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

**8241 TRANSMIT not enabled (MSD)!**

Cause: Transmit is not possible at this machine.

**8242 TRACYL not enabled (MSD)!**

Cause: Tracyl is not possible at this machine.

**8243 Round axis invalid during active transformation!**

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

**8245 TRACYL radius = 0!**

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

Remedy: Correct the radius.

**8246 Offset alignment not valid for this state!**

see 8239.

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

Cause: The communication with the machine keyboard is interrupted.

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

**8249 Axis motion check alarm!**

see 8239

**8250 Spindle must be rotation axis !**

see 8239

**8251 Lead for G331/G332 missing !**

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

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

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

Remedy: Program exactly one linear axis.

**8253 Speed value for G331/G332 and G96 missing !**

Cause: No cutting speed has been programmed.

Remedy: Program the cutting speed.

**8254 Value for thread starting point offset not valid!**

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

Remedy: Correct the thread starting point offset.

**8255 Reference point not in valid software limits!**

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

Remedy: Correct the reference points in WinConfig.

**8256 Spindle speed too low while executing G331/G332!**

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

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

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

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

Remedy: Report the error to EMCO.

**8258 Error allocating Linux data!**

see 8257.

**8259 Current thread in sequence not valid!**

Cause: One block of a thread in sequence has been programmed without thread G33.

Remedy: Correct the program.

**8260 Change of leading axis within the thread sequence !**

Cause: Occurs if the thread run-out of the longitudinal thread is set so that it's not possible to reach the target point. with the required stopping distance

Remedy: The thread run-out should be at least as large as the pitch. this error also occurs. if the pitch of a thread chain is too large when changing the leading axis.

**8261 Missing thread in sequence !**

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

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

**8262 Reference marks are not close enough !**

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

Remedy: Correct the settings. Contact EMCO.

**8263 Reference marks are too close together!**

see 8262.

**8265 No or wrong axis in axis switch command!**

Cause: Internal error.

Remedy: Please contact the EMCO after-sales service.

**8266 Invalid tool**

Cause: Programmed tool is not set in magazine.

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

**8267 Speed difference to high**

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

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

**8269 USBSPS and ACC speed values or override are different**

Cause: USBSPS and ACC have different rotations saved.

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

**8270 Reference switch defective**

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

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

**8271 Tool load in locked place not possible**

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

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

**8272 Old PLC version, update necessary**

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

Remedy: Update the PLC.

**8273 Spindle overload**

Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).  
 Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

**8274 Define tool before loading**

Cause: The tool must be defined in the tool list before it is possible to transfer the tool into the spindle.  
 Remedy: Create the tool in the tool list, then load.

**8275 Could not read position from absolute encoder**

Cause: The position of an absolute encoder could not be read.  
 Remedy: Turnoff and on the machine. Contact EMCO service if the error occurs multiple times

**8276 Physical axis left the valid travel range**

Cause: An axis with absolute encoder is outside the valid range for travel.  
 Remedy: Turnoff and on the machine. Contact EMCO service if the error occurs multiple times.

**8277 Sinamics drive package error active**

Cause: Error in Sinamics drive.  
 Remedy: Turnoff and on the machine. Contact EMCO service if the error occurs multiple times.

**8278 Control not compatible with ACpn**

Cause: The WinNC control is incompatible with the ACpn machine.  
 Remedy: Install a WinNC control which is compatible with the Acpn.

**8279 Connection to drive got lost**

Cause: the connection between Acpn and CU320 was lost.  
 Remedy: Turnoff and on the machine. Contact EMCO service if the error occurs multiple times

**8280 Reference point in setting data not equivalent with machine data, please check!**

Cause: The reference point saved in the AC setting data does not match the reference point in the machine data (ACC\_MSD).  
 Remedy: Measure the reference point of all axes anew and enter into EMCConfig.

**8704 Feed override absent, REPOS is not executed**

Cause: The REPOS command is not executed because the feed override is set to 0%.  
 Remedy: Change the feed override and restart REPOS.

**8705 Tool sorting active**

Cause: The tools will be re-sorted with random tool management to facilitate non-random operation (tool 1 at place 1, tool 2 at place 2, etc.).  
 Remedy: Wait until sorting has finished. The controller will delete the report independently.

**8706 Check new controller - tool table**

Cause: The controller was changed with random tool management active.  
 Remedy: Check the tool or place table to clear the alarm.

**8707 Ending with auxiliary drives switched on not possible**

Cause: An attempt was made to end the controller, although the auxiliary drives are still switched on.  
 Remedy: Switch off the auxiliary drives and then end the controller.

**8710 Initiating communication with drives**

Cause: Acpn connects to the Sinamics drives  
 Remedy: Wait until the connection was established.

**8712 JOG in X and C deactivated during Transmit**

Cause: JOG in X and C ist not possible at active Transmit.

**22000 Gear change not allowed**

Cause: Gear step change when the spindle is active.  
 Remedy: Stop the spindle and carry out a gear step change.

**22270 Feed too high (thread)**

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

**200000 to 300000 are specific to the drives and show up only in combination with the alarm # "8277 Sinamics error".****201699 - „(F) SI P1 (CU): Test of the shutdown path is necessary“**

Cause: A test of the shutdownpath is necessary.  
 The machine remains operational.  
 Remedy: The test is performed automatically during the next restart of the WinNC.

**235014 TM54F: Teststop necessary**

Cause: A teststop is necessary.  
 Remedy: Shutdown and restart the WinNC.  
 The test is performed automatically during restart.



## Axis Controller Messages

### 8700 Execute REPOS in all axes before program start

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

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

### 8701 No NCStop during offset align

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

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

### 8702 No NCStop during positioning after block search

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

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

### 8703 Data record done

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

### 8705 Feed override missing, REPOS will not be executed

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

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

### 8706 Tool sorting active

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

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

### 8707 New control - please check tool table

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

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

### 8708 Switch off auxiliary drives for shutdown

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

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

### 8709 Insert tool in spindle for loading

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

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

## Control alarms 2000 - 5999

The alarms are released by the software.

**Fagor 8055 TC/MC**

**Heidenhain TNC 426**

**CAMConcept**

**EASY CYCLE**

**Sinumerik OPERATE**

**Fanuc 31i**

**Heidenhain TNC 640**

### **2200 Syntax error in line %s, column %s**

Cause: Syntax error in the program code.

### **2001 Circle end point invalid**

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

Remedy: Correct circular arc point.

### **2300 tracyl without corresponding round-axis invalid**

Cause: Maybe the machine has no rotary axis.

### **3000 Traverse feed axis manually to position %s**

Remedy: Move the axis manually to the required position.

### **3001 Tool T.. change!**

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

Remedy: Clamp the required tool in the machine.

### **4001 slot width too small**

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

### **4002 slot length to small**

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

### **4003 length equal zero**

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

### **4004 slot width too big**

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

### **4005 depth equal zero**

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

### **4006 corner radius too big**

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

### **4007 diameter too big**

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

### **4008 diameter too small**

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

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

### **4009 length too small**

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

### **4010 diameter less equal zero**

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

### **4011 blank diameter too big**

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

### **4012 blank diameter too small**

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

**4013 start angle equal to end angle**

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

**4014 tool radius 0 not permitted**

Cause: Tool radius zero is not permitted.

Remedy: Select a valid tool.

**4015 no outer contour defined**

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

**4017 tool radius too big**

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

**4018 allowance must not be 0**

Cause: There were programmed finishing operations without finishing offset.

**4019 too many iterations**

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

Remedy: Simplify the contour.

**4020 illegal radian correction**

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

Remedy: Check the cycle parameters.

**4021 can't calculate parallel contour**

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

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

**4022 illegal contour definition**

Cause: The programmed contour is not suited to the selected machining.

Remedy: Check the programmed contour.

**4024 no contour definition**

Cause: The contour file being defined in the cycle has not been found.

**4025 internal calculation error**

Cause: An unexpected error has occurred during calculation of the cycle movements.

Remedy: Please inform the EMCO after-sales service.

**4026 allowance too big**

Cause: A part of the finishing offset (for several finishing passes) is larger than the total finishing offset.

Remedy: Correct the finishing offsets.

**4028 pitch 0 not permitted**

Cause: The thread was programmed with pitch zero.

**4029 undefined working mode**

Cause: Internal error (invalid machining type for the thread).

**4030 function not yet supported**

Cause: Roughing out with pockets is not implemented yet.

Remedy: Please inform the EMCO after-sales service.

**4031 value not permitted**

Cause: An invalid retracting direction was programmed during inside turning.

**4032 plunging must be defined**

Cause: For the programmed cycle no cutting depth has been programmed.

**4033 radius/chamfer too big**

Cause: The radius, respectively the chamfer, cannot be inserted in the programmed contour.

Remedy: Reduce the radius, respectively the chamfer.

**4034 diameter too big**

Cause: The programmed starting point and the machining diameter are contradictory.

**4035 diameter too small**

Cause: The programmed starting point and the machining diameter are contradictory.

**4036 unknown working direction**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4037 unknown working type**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4038 unknown sub cycle**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4039 rounding not possible**

Cause: The programmed radius contradicts the rest of the cycle parameters.

**4042 illegal tool width**

Cause: The tool width for the cutting-off cycle must be defined.

**4043 groove width too small**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4044 distance not defined**

Cause: The distance for the multiple grooving cycle must not be zero.

**4045 illegal allowance type**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4046 invalid speed**

Cause: The spindle speed must be nonzero.

**4047 invalid end point**

Cause: The programmed end point contradicts the rest of the cycle definition.

**4048 tool cut width too small**

Cause: The cutting edge is too small for the programmed cutting depth.

**4050 invalid distance**

Cause: The hole patterns do not tally with the selected distance.

**4052 working pattern not possible**

Cause: Error in the definition of the hole pattern. The number of bores is contradictory.

**4053 invalid start point**

Cause: Internal error.

Remedy: Please inform the EMCO after-sales service.

**4055 illegal working direction**

Cause: The machining direction is contradictory to the rest of the cycle definition.

**4057 plunging angle less equal zero**

Cause: The plunging angle must be between 0 and 90 degree.

**4058 chamfer too large**

Cause: The programmed chamfer is too large for the pocket cycle.

**4062 radius/chamfer too small**

Cause: The radius, respectively the chamfer, cannot be machined with the current tool radius.

**4066 invalid mill step**

Cause: The mill step must be greater than zero.

**4069 invalid angle**

Cause: An angle of zero degree is not permitted.

**4072 plunging too small**

Cause: For the cycle, a cutting depth has been selected that leads to extra-long machining time.

**4073 invalid clearance angle**

Cause: The clearance angle indicated for the tool cannot be machined.

Remedy: Correct the clearance angle for the tool.

**4074 contour-file not found**

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

Remedy: Please select the contour file for the cycle.

**4075 not machinable with selected tool**

Cause: The tool is too wide for the programmed groove.

**4076 reciprocating plunge cut impossible  
(initial move too short)**

Cause: The first movement of the contour is shorter than the double tool radius and cannot be therefore used for the swinging delivery.

Remedy: Extend the first movement of the contour.

**4077 wrong tool type in grooving or cut-off  
cycle**

Cause: The wrong tool type was used in the cutting cycle.

Remedy: Use only grooving and punch tools in the cutting cycles.

**4078 radius of helix too small**

Cause: The pitch of the helix is  $\leq 0$ .

Remedy: Program the radius  $> 0$ .

**4079 pitch of helix too small**

Cause: The radius of the helix is  $\leq 0$ .

Remedy: Program the pitch  $> 0$ .

**4080 radius of helix or tool too large**

Cause: The helical approach cannot be executed with the selected data for the helix and the current tool radius without a contour breach.

Remedy: Use a tool with a smaller radius or reduce the radius of the helix.

**4200 leaving movement is missing**

Cause: No movement after the tool radius compensation was deactivated in the current plane.

Remedy: Insert the departing movement in the current plane after having deactivated the tool radius compensation.

**4201 TPC off missing**

Cause: The tool radius compensation has not been deactivated.

Remedy: Deactivate the tool radius compensation.

**4202 TPC requires at least three movements**

Cause: The tool radius compensation requires at least 3 movements in the current plane in order to calculate the tool radius compensation.

**4203 approaching movement not possible**

Cause: It was not possible to calculate the approaching movement.

**4205 leaving movement not possible**

Cause: It was not possible to calculate the departing movement.

**4208 TPC curve could not be calculated**

Cause: It was not possible to calculate the tool radius compensation for the programmed contour.

**4209 switching the plane is not allowed when TPC is switched on**

Cause: The programmed plane must not be changed during the tool radius compensation.

Remedy: Remove the change of planes during the tool radius compensation.

**4210 tool path compensation already activated**

Cause: G41 is active and G42 was programmed or G42 is active and G41 was programmed.

Remedy: Switch tool radius compensation off with G40 before programming the radius compensation again.

**4211 Bottleneck detected**

Cause: In the radius correction calculation some parts of the contour were omitted, as too large a milling cutter tool was used.

Remedy: Use a smaller milling cutting tool to process the contour completely.

**4212 Infeed has been programmed twice during approach**

Cause: After the approach movement a second infeed has been programmed, without previously moving to the work plane.

Remedy: First program a movement to the work plane before programming a second infeed.

**5000 drill manually now****5001 contour has been adjusted to the programmed clearance angle**

Cause: The programmed contour was adapted to the programmed clearance angle. Maybe there will remain rest material that cannot be machined with this tool.

**5500 3D simulation: Internal error**

Cause: Internal error inside the 3D simulation.

Remedy: Restart the software or, if necessary, report the error to EMCO customer service.

**5502 3D simulation: Tool place invalid**

Cause: Tool place unavailable on the machine used.

Remedy: Correct tool call-up.

**5503 3D simulation: Chuck invalid owing to the unmachined part definition**

Cause: The distance from the front of the unmachined part is > the unmachined part length.

Remedy: Change the distance.

**5505 3D simulation: Unmachined part definition invalid**

Cause: Implausibility in the unmachined part geometry (e.g. expansion in one axis  $\leq 0$ , inside diameter > outside diameter, unmachined part contour not closed, etc.).

Remedy: Correct unmachined part geometry.

**5506 3D simulation: STL chuck file has auto-overcuts**

Cause: Error in the chuck description.

Remedy: Correct file.

**5507 3D simulation: Pole transit on TRANS-MIT!**

Cause: Travel comes too close to the X0 Y0 co-ordinates.

Remedy: Change travel.



# I: Heidenhain TNC 640 controller alarms

## Controller alarms 0001 - 88000

The controller triggers these alarms. These are the same alarms as would have occurred on the Heidenhain TNC 640 controller.

### 38011 CYCL DEF not defined

**Explanation:** You have programmed a cycle call without first defining the cycle, or you tried to call a DEF-active cycle.

**Remedy:** Define the cycle before calling it.

### 38027 Label number not found

**Explanation:** With LBL CALL (ISO: L x,x), you tried to call a label that does not exist.

**Remedy:** Change the number in the LBL CALL block or insert the missing label (LBL SET).

### 38038 Direction of rotation missing

**Explanation:** A circle was programmed without direction of rotation.

**Remedy:** Always program a direction of rotation (DR).

### 38044 Block format incorrect

**Explanation:** A syntactically incorrect NC block was programmed.

**Remedy:** Edit the NC program.

### 38060 CYCL DEF incomplete

**Explanation:** You programmed an incomplete cycle definition or inserted other NC blocks between cycle blocks.

**Remedy:** Edit the NC program.

### 38195 M function not permitted

**Explanation:** An M function was programmed with a number that is not permitted on this control.

**Remedy:** Correct the number of the M function.

### 38214 No technological data available for contour pocket machining

**Explanation:** A Cycle 20 must be programmed before every fixed cycle 21, 22, 23, or 24.

**Remedy:** Program a Cycle 20.

### 38275 Label is missing

**Explanation:** You used Cycle 14 to call a nonexistent label.

**Remedy:** Change the number or name in Cycle 14 or insert the missing label.

### 42087 No point angle defined

**Explanation:** In Cycle 240 Centering you defined parameter Q343 such that centering is done with respect to the diameter.

However, no point angle is defined for the active tool.

**Remedy:**

- Set parameter Q343=0 (centering with respect to entered depth).
- Define the point angle in the column T-ANGLE of the tool table TOOL.T.

### 42509 Tool not defined

**Explanation:** You have called a tool that is not defined in the tool table.

**Remedy:**

- Add the missing tool to the tool table.
- Use another tool.

### 42517 Tool locked

**Explanation:** The tool was locked (e.g. after breakage).

**Remedy:** Check the tool and, if necessary, change it or unlock it in the tool table.

### 43029 Incorrect pole axis for selected working plane

**Explanation:**

- Z component with XY plane
- X component with YZ plane
- Y component with ZX plane

**Remedy:** Edit the NC program.

### 43118 Rounding or chamfer cannot be calculated

**Explanation:** Connecting geometrical element is too small or has the same tangent.

**Remedy:** Edit the NC program.

#### **43163 Invalid tool technology data**

**Explanation:** Technology data of the tool used, such as ANGLE or LCUTS, are not registered in the tool database or have invalid values.

**Remedy:** Make or correct the corresponding entries:

ANGLE : Max. plunge angle in degrees,  $1.0 \leq \text{ANGLE} \leq 90.0$

LCUTS : Tooth length in mm,  $0.1 \leq \text{LCUTS} \leq \text{tool length.}$

#### **43225 Transformation not allowed**

**Explanation:** Possible causes:

- Define datum shifts only in X/Y cylinder surface coordinates.
- Contour definition with angle data: program a datum shift only within the contour definition.
- Rotations and scaling are allowed only with length dimensions and only cylinder surface coordinates.
- Define mirroring only in cylinder surface coordinates.
- You must not change presets, basic rotations or the tilt condition during cylinder surface machining.

**Remedy:** Edit the NC program.

# W: Accessory Functions

## Activating accessory functions

According to the machine (turn/mill) the following accessories can be taken into operation:

- automatic tailstock
- automatic vice/clamping device
- Air purge system
- Dividing attachment
- Robot interface
- Automatic doors
- Win3D view simulation software
- DNC interface

The accessories are activated with EMConfig.

## Robotic Interface

The robotic interface is used to connect the concept machines to an FMS/CIM system.

The most important functions of a concept machine can be automated via the inputs and outputs of an optional hardware module.

The following functions can be controlled via the robotic interface:

- Program START / STOP
- Door open / closed
- Quill clamp / back
- Clamping device open / closed
- Feed STOP

**Note:**

The functionality of the Win3D-View is depending on the control software that is used.

## Automatic doors

Preconditions for activation:

- The auxiliary drives must be switched on.
- The main spindle must be still (M05 or M00) - this also means that the run-out phase of the main spindle must be ended (program dwell time if required).
- The feed axes must be still.
- The tool changer must be still.

Behavior when automatic doors active:

### Opening door

The door can be opened manually, via the robot interface or DNC interface.

In addition, the door opens when the following commands are executed in the CNC program:

- M00
- M01
- M02
- M30

### Closing door:

The door can be closed by manually pressing the button via the robot interface. It is not possible to close the door via the DNC interface.

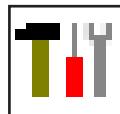
## Win3D View

Win3D View is a 3D simulation for turning and milling, which is offered as an additional option for the WinNC product. Graphical simulations of CNC controls are primarily designed for industrial practice. The Win3D View screen representation goes beyond the industrial standard. Tools, raw parts, clamping devices and the processing steps are represented extremely realistically. The programmed movement paths of the tool are checked by the system for a collision with clamping device and raw part. A warning message is issued when there is danger. This makes possible to have understanding and control of the manufacturing process already on the screen.

Win3D View is used to visualize and prevent costly collisions.

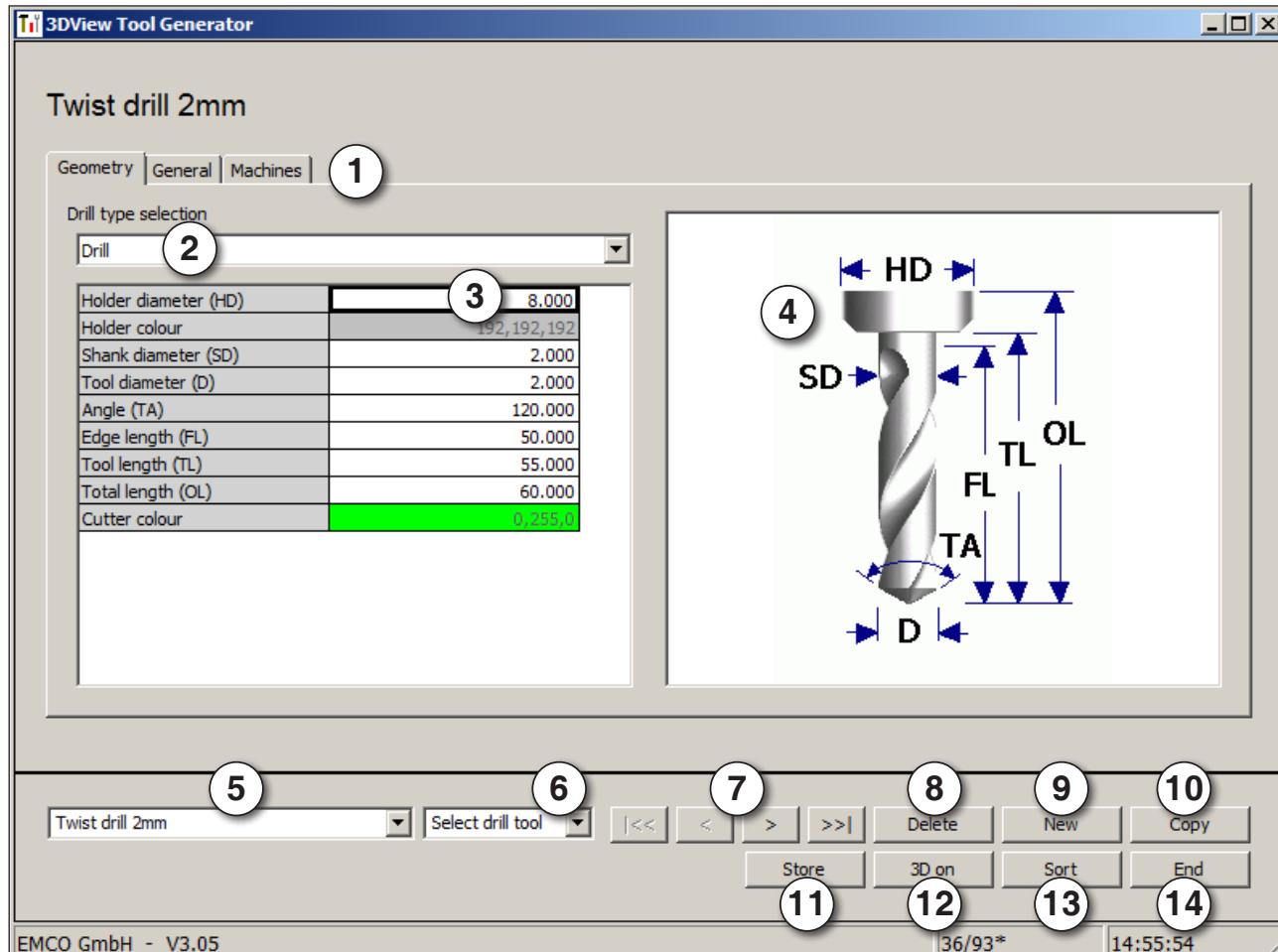
Win3D View offers the following advantages:

- Extremely realistic representation of workpiece
- Tool and clamping device collision control
- Cut representation
- Zoom functions and turning of views
- Representation as solid or wireframe model



## Tool modelling with the 3D-Tool Generator

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

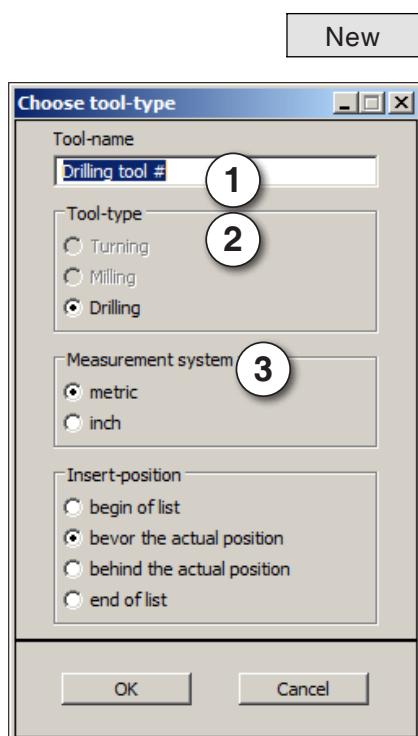


- 1 Register cards for "Geometry", "General" and "Machines" for drilling and milling tools and "Tip", "Holder", "General" and "Machines" for turning tools.
- 2 Selection of tool types
- 3 This window enables the input of tool dimensions.
- 4 Graphical support for the tool dimensioning
- 5 Choice of tools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

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

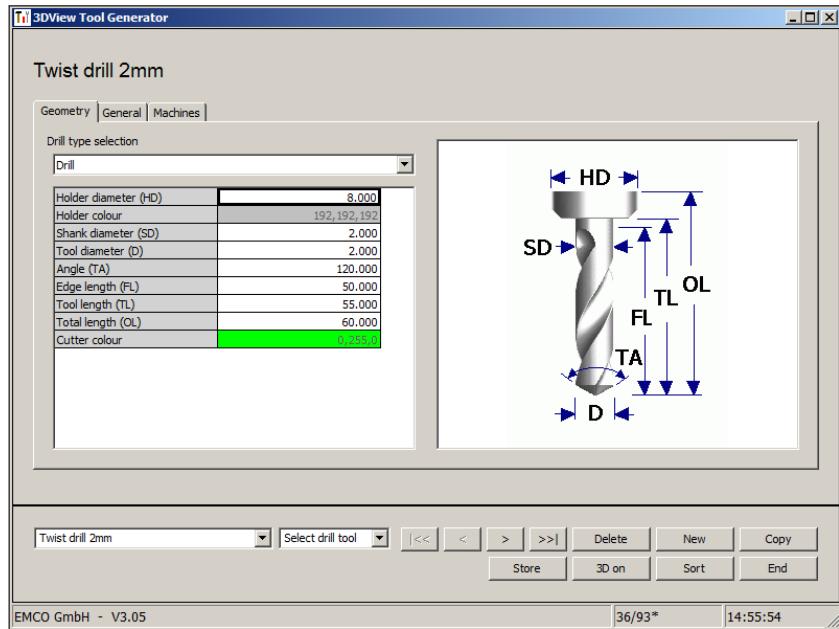
## Generating a new tool

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



- Confirm the entries with "OK".

- Define all tool dimensions.
- Define all tool colours (see "select tool colour").



- Confirm the entries with "Store".

### Copying a tool

**Copy**

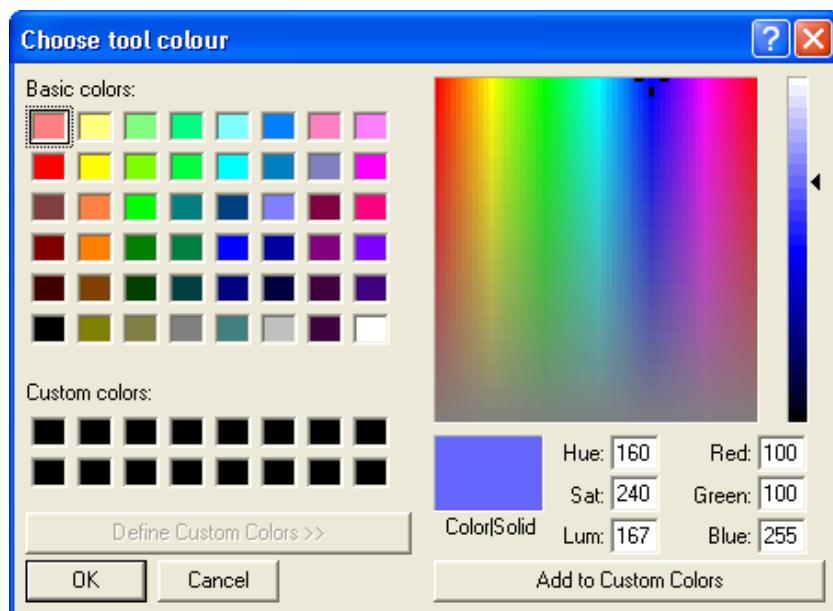
- Call the tool you wish to copy.
- Press the button to copy tools.
- Enter the new tool name.
- Confirm the inputs with "Save".

**Store**

### Changing an existing tool

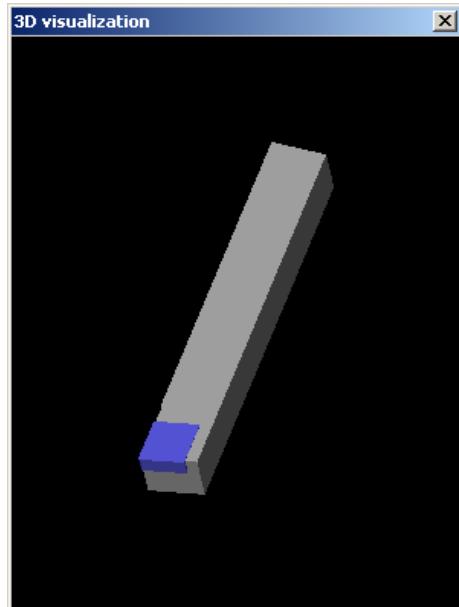
- Call the tool you wish to change.
- Change the values.
- Confirm the entries with "Store".

### Selecting a tool colour



**OK**

- Confirm the entries with "OK".



### Visualizing a tool

- Press the button for 3D visualization.

### Rotating image

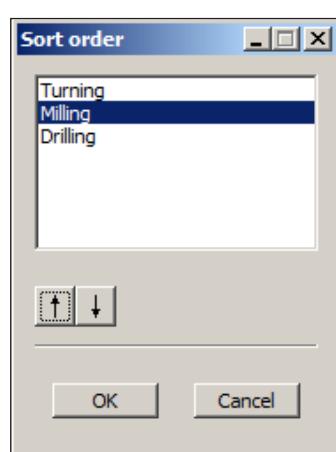
At any time you can rotate the simulation image in one plane as required by pressing and holding the left mouse button. For movements around the Z axis press "Shift" + left mouse button + mouse movement to the right or to the left.

### Zooming

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

### Shifting

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



### Sorting function

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

- Press the button for sorting.

- Define new sorting sequence.



- Confirm the entries with "OK".

## DNC interface

The DNC interface (Distributed Numerical Control) enables the control system (WinNC) to be controlled remotely via a software protocol.

The DNC interface is activated with EMConfig, by indicating TCP/IP or a serial interface for the DNC.

During the installation of the control software, the DNC interface is enabled and configured, and can be reconfigured with EMConfig later on.

The DNC interface creates a connection between a higher-level computer (production control computer, FMS computer, DNS host computer, etc.) and the control computer of an NC machine. After activation of the DNC drive the DNC computer (Master) takes over control of the NC machine (Client). The DNC computer takes over complete control of the manufacturing. The automation fittings such as door, chuck (collet), quill, coolant, etc. can be controlled from the DNC computer. The current status of the NC machine is visible on the DNC computer.

The following data can be transferred or loaded via the DNC interface:

- NC Start
- NC Stop
- NC programs \*)
- Zero point shifts \*)
- Tool data \*)
- RESET
- Approach reference point
- Periphery control
- Override data

The DNC interface can be operated with the following CNC control types:

- SINUMERIK Operate T and M
- FANUC 31i T and M

Further details of the functions and the DNC protocol can be found in the accompanying product documentation.

If the DNC interface is operated with TCP/IP, it will wait for incoming connections on port 5557.

\*) not for SINUMERIK Operate and FANUC 31i

# X: EMConfig

**Note:**

The settings which are available in EMConfig are depending on the machine and the control that is used.



## General

EMConfig is a configuration software for WinNC. EMConfig helps you to alter the settings of WinNC.

The most important settings are:

- Control language
- System of measurement mm - inch
- Activate accessories
- Selection of interface for control keyboard

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



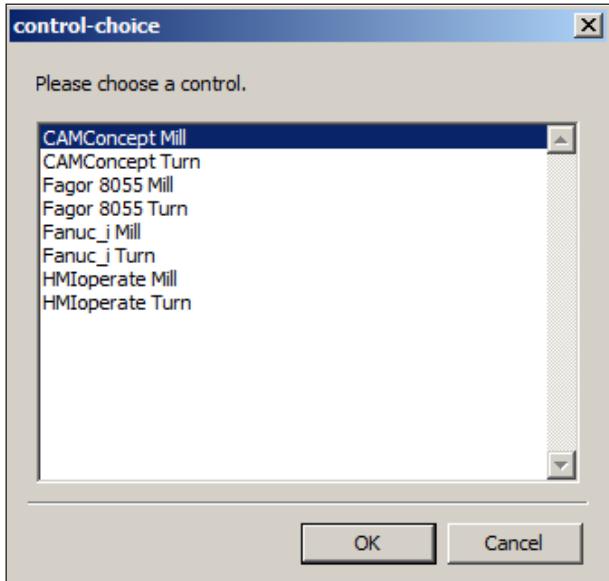
Safety-related parameters are protected by a password. They can only be activated by set-up technicians or by technical support representatives.



Icon for EMConfig

## How to start EMConfig

Open EMConfig.



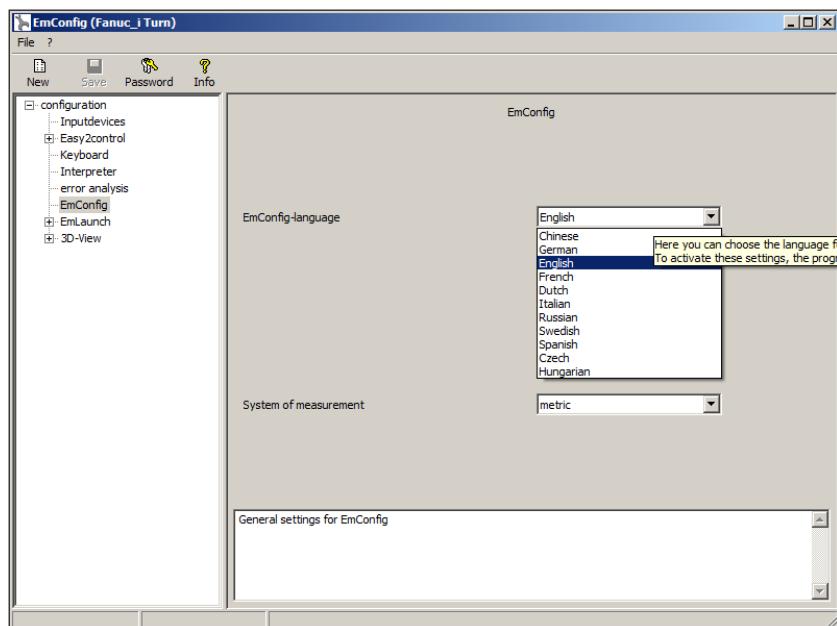
Selection box for control type

In case several control types are installed, a selection box will appear on the screen.

Select the required control type and click OK.

The following settings are only valid for the selected control type.

The window for EMConfig appears on the screen.



Change the language of EMConfig

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

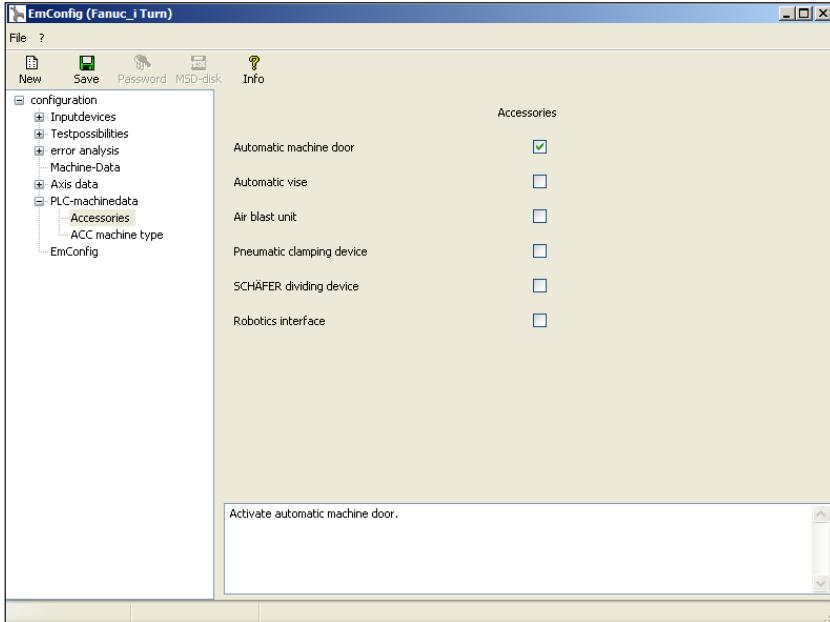
### Note:

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



## How to activate accessories

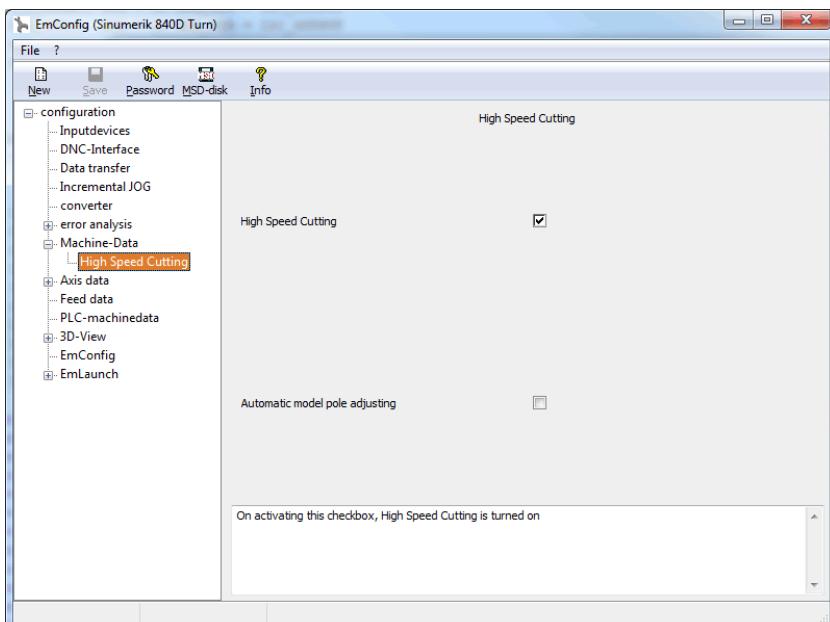
When you install accessories on your machine, you need to activate them here.



*Activate accessories*

## High Speed Cutting

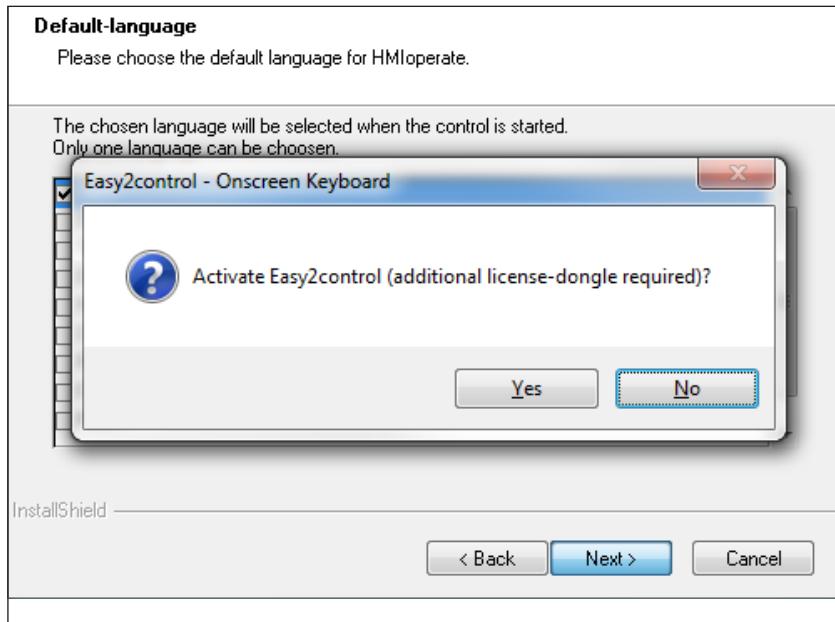
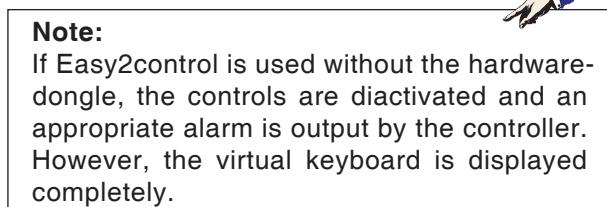
On activating this checkbox, High Speed Cutting is turned on.



*Activate High Speed Cutting*

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

If the feed is set up to higher than 2500 mm/min, it is automatically reset to the normal operating mode and sanded and rounded edges are created.

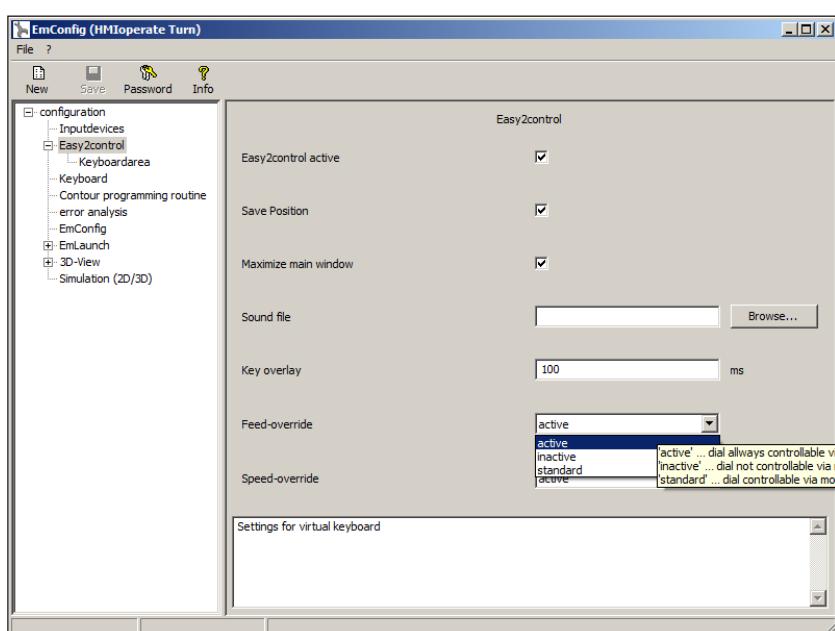


Activating Easy2control

## Easy2control on screen operation

Installation and activation using the example of WinNC for Sinumerik Operate.

When installing the software WinNC for Sinumerik Operate you will be prompted to activate Easy2control. In order to use the Software without restriction, the supplied dongle must be connected to a free USB port.



Easy2control settings

## Settings

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

### Dial feed-override and dial speed-override:

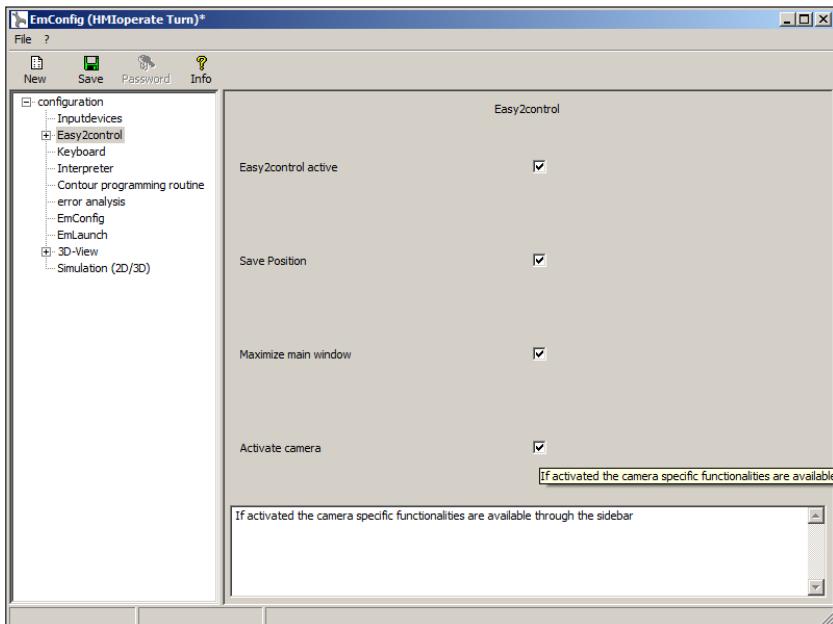
- **Aktive:** dial always controllable via mouse/touchscreen (even with available hardware-version).
- **Inaktive:** dial not controllable via mouse/touchscreen.
- **Standard:** dial controllable via mouse/touchscreen when no hardware-version is available.

**Danger:**

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

**Machine room camera**

The accessory machine room camera is available for all controls which support Easy2control.



*Activate camera*

Please refer to chapter Y "external input devices" for more information about the installation guide for the the camera.

**Attention:**

The camera may not be operated without the supplied waterproof case.

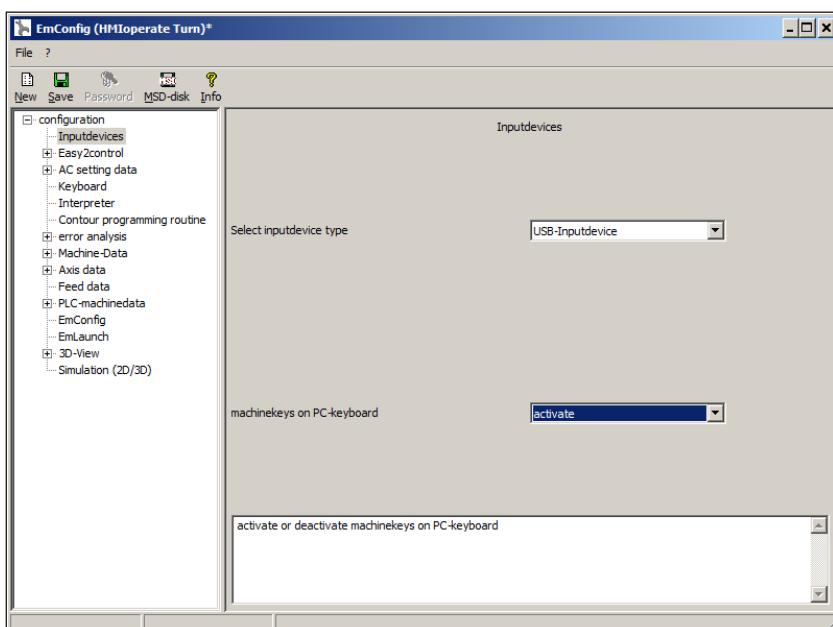
If the camera is operated without the waterproof case it may be damaged by coolant and chips.

**Machinekeys on PC keyboard**

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

The setting is available for the following controls:

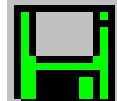
- Sinumerik Operate
- Fanuc 31i
- Emco WinNC for Heidenhain TNC 640



*Setting for the machinekeys*

## How to save changes

After the settings, the changes must be saved.



Select "Save" or click on the icon.

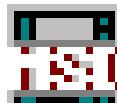


**Note:**

Input fields highlighted in red indicate inadmissible values. Inadmissible values are not saved in EMConfig.

After saving the changes, create a machine data floppy disk (MSD) or a machine data USB flash drive.

## How to create machine data floppy disk or machine data USB flash drive



After having changed the machine data, the machine data floppy disk or the machine data USB flash drive must be in the appropriate drive. Otherwise your changes cannot be saved and get lost.

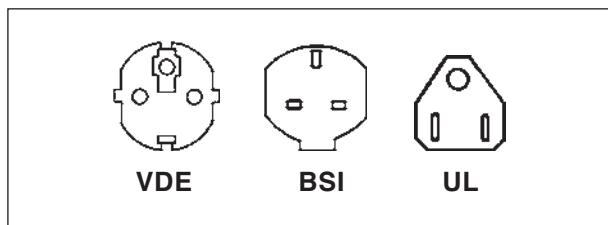
# Y: External Input Devices

## EMCO Control Keyboard USB

### Scope of supply

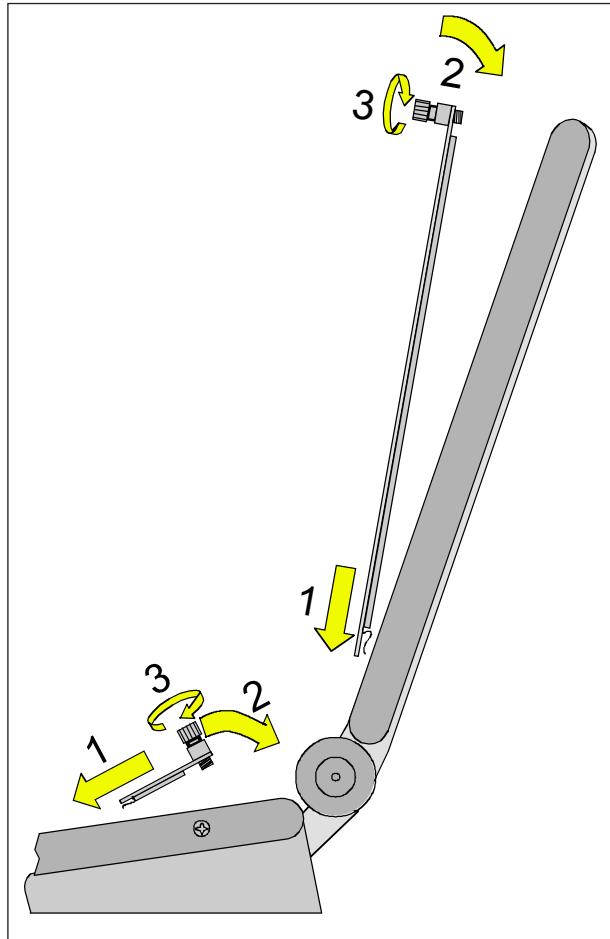
The scope of supply for a complete control keyboard consists of two parts:

- Basic case
- Key module



#### Ref. No. Description

X9B 000	Basic unit with USB cable	X9Z 426N	Key module HEIDENHAIN 426/430 2 key sheets with keys 1 package exchange keys
X9Z 600	TFT Display with screen cable and power supply unit	X9Z 060	Key module WinNC for SINUMERIK OPERATE 2 key sheets with keys
A4Z 010	Mains cable VDE	X9Z 030	Key module WinNC for FANUC 31i 2 key sheets with keys 1 package exchange keys
A4Z 030	Mains cable BSI	X9Z 640	Key module Emco WinNC for HEIDENHAIN TNC 640
A4Z 050	Mains cable UL		
X9Z 050N	Key module FAGOR 8055 TC 2 key sheets with keys		
X9Z 055N	Key module FAGOR 8055 MC 2 key sheets with keys		

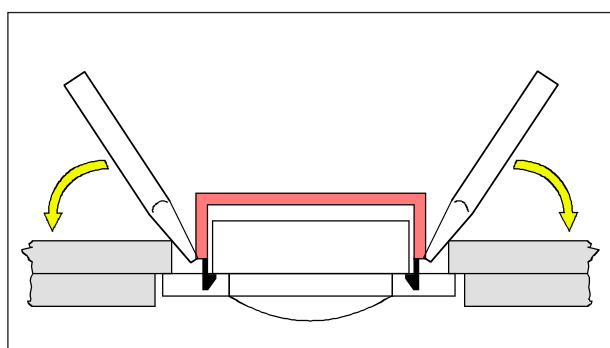


## Assembling

- Place the correseponding key sheet with the clips in the basic case (1).
- Pull the key sheet into the basic case, it must be insertet plainly (2).
- Fix the key sheet with the two knurled screws (3).

**Note:**

The key sheets must not be bended, otherwise the switching function can not be warranted.



## Exchange of single key caps

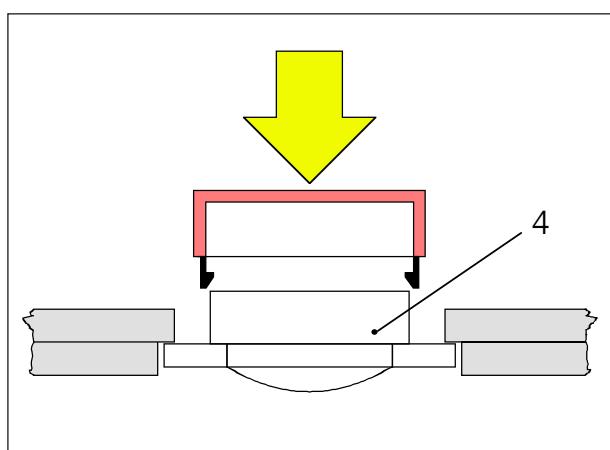
Off works the keyboards are equipped with the keys for turning.

The scope of supply includes a package of exchange key caps to equip the keyboard for milling.

If you want to use the control keyboard for milling, you have to exchange a part of the key caps. Exchange them as shown on the following pages.

**Note:**

For the control type Emco WinNC for HEIDENHAIN TNC 640 only a milling version is available.



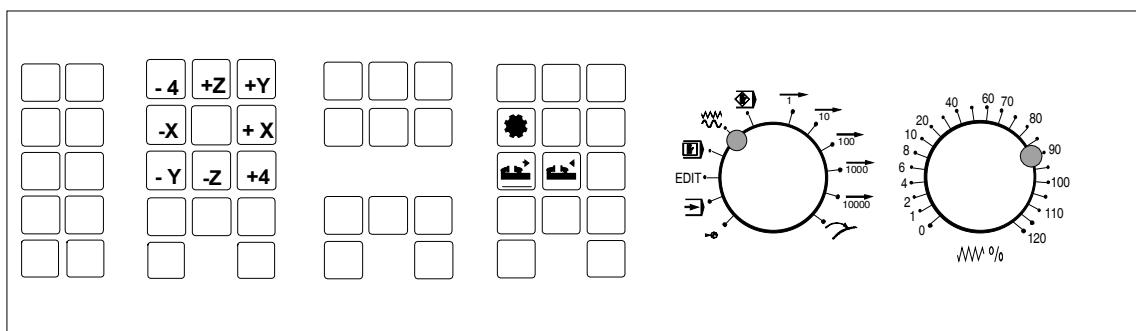
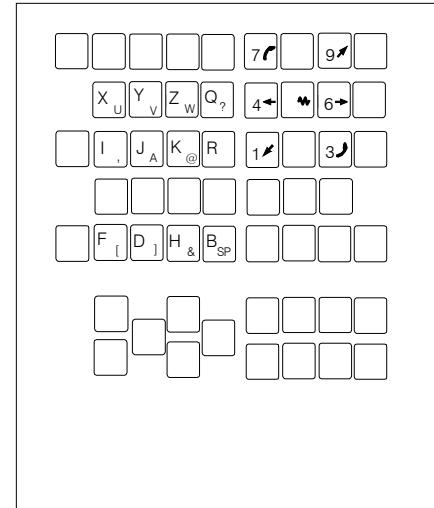
### Take off

Pull out carefully the key caps to be exchanged with a fine screw driver or a knife.

### Clip on

Move the key body in the middle of the recess. Push the key cap vertically down onto the key body, until the key cap snaps in tactily.

**FANUC 31iM**  
Exchange key caps  
for milling



## Connection to the PC

The control keyboard is connected via USB interface to the PC.

The connection cable USB taking over at the same time the energy supply of the control keyboard is situated at the rear side of the control keyboard.

## Settings at the PC software

### Setting during new installation of the PC software

During the installation indicate the control keyboard and the respective USB interface.

### Setting in case of PC software already installed

Select in EMConfig at the INI data settings the USB control keyboard as means of entry and the respective interface USB.

Furthermore, set the keyboard type to "New". Don't forget to memorize the settings.

## Easy2control On Screen operation

Easy2control adds a range of attractive applications to the successful interchangeable control system used in EMCO training machines. Suitable for use in machines and simulation workplaces alike, it displays additional control elements directly on the screen and, when used together with a touchscreen monitor, provides the ideal input interface.

### Scope of supply

The software for Easy2control is part of the control software.

The dongle for a workstation license is delivered:

**Ref. No.** X9C 111

Technical data for the screen:

16:9 Full-HD Monitor (1920x1080) at the minimum

Currently available controls (T and M):

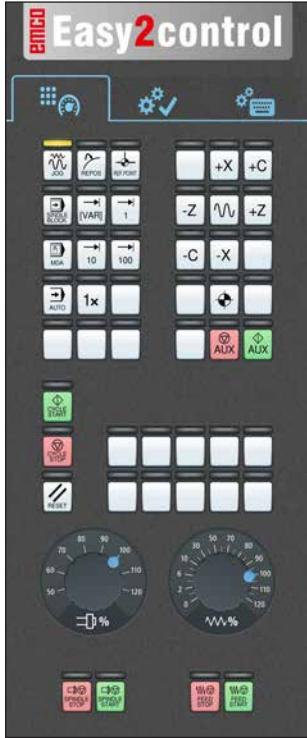
- Sinumerik Operate
- Fanuc 31i
- Emco WinNC for Heidenhain TNC 640 (M only)
- Emco WinNC for Heidenhain TNC 426 (M only)
- Fagor 8055

**Note:**

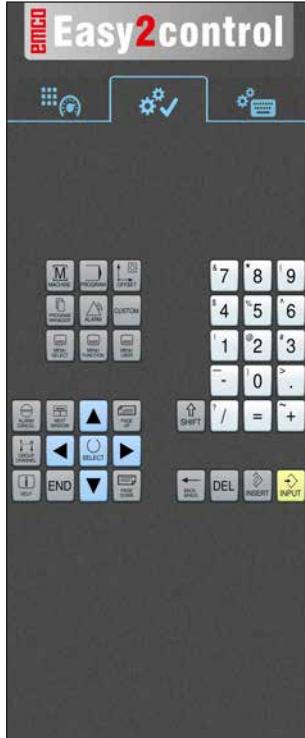
If a Full HD monitor is used without touch-screen function, the control is operated just with mouse and keyboard.

# Operating areas

## Sinumerik Operate



Machine control panel

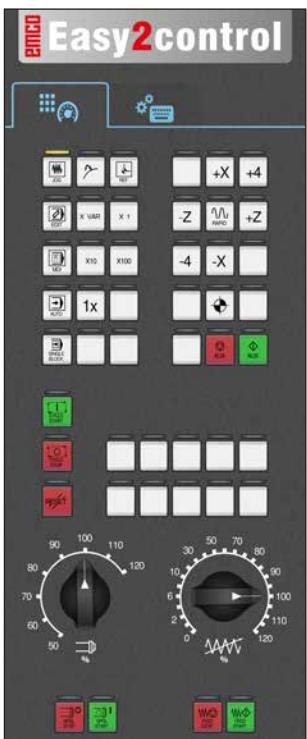


Controller-specific operation



Control operation complete

## Fanuc 31i



Machine control panel



Control operation complete

## Emco WinNC for Heidenhain TNC 640



Machine control panel

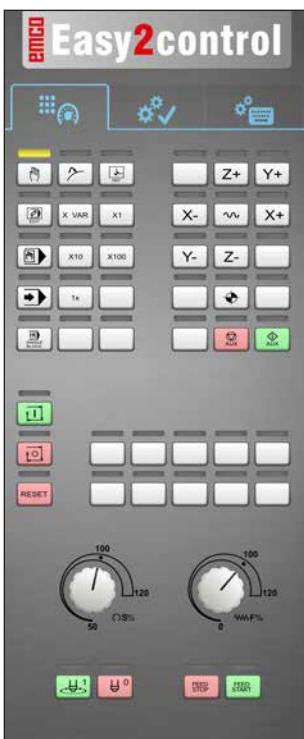


Controller-specific operation

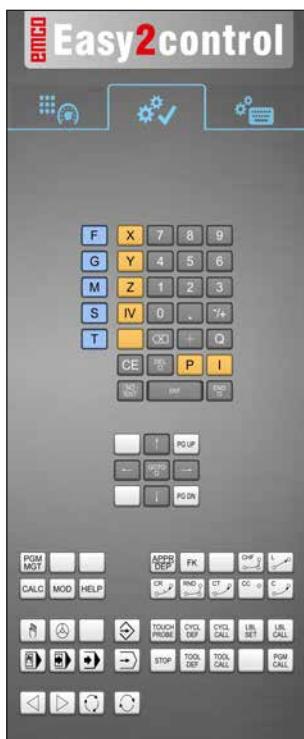


Control operation complete

## Heidenhain TNC 426



Machine control panel

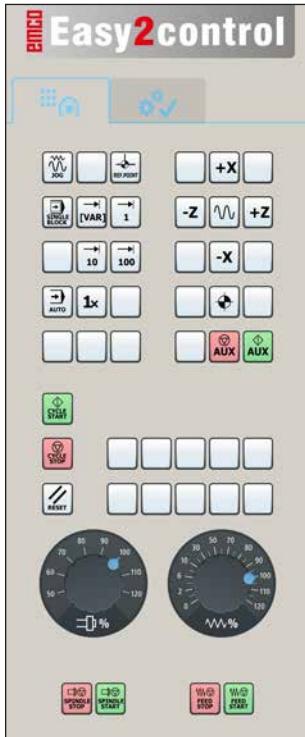


Controller-specific operation

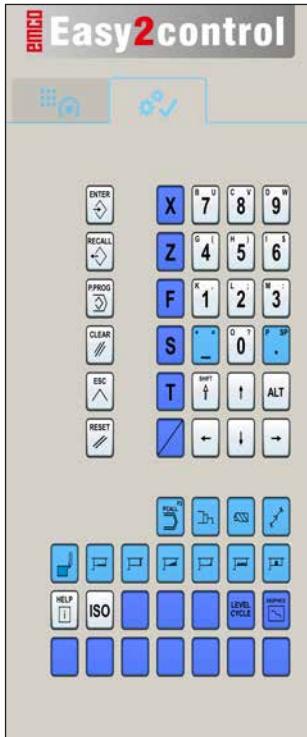


Control operation complete

## Fagor 8055



Machine control panel



Control operation complete

Refer to the chapter "Key Description" of the respective control description for operation and key function.

**Note:**

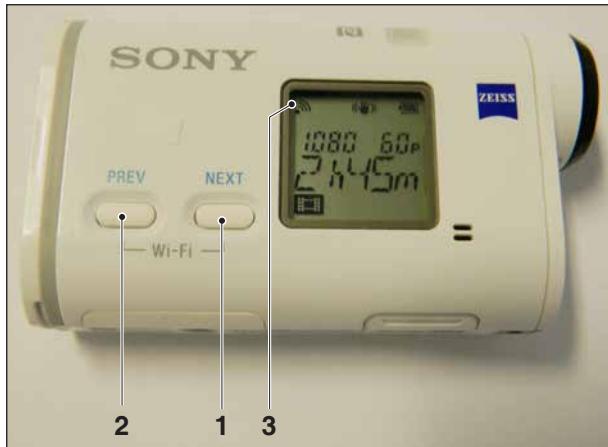
The screen display, based on customer-specific configurations, may look different.



## Machine room camera

This accessory can be ordered using the following order number:

Ref. No.: S4Z750



Activating WLAN



Connect WLAN

## Installing the camera

### Requirement

USB WLAN adapter for the machine.

### Setup WLAN

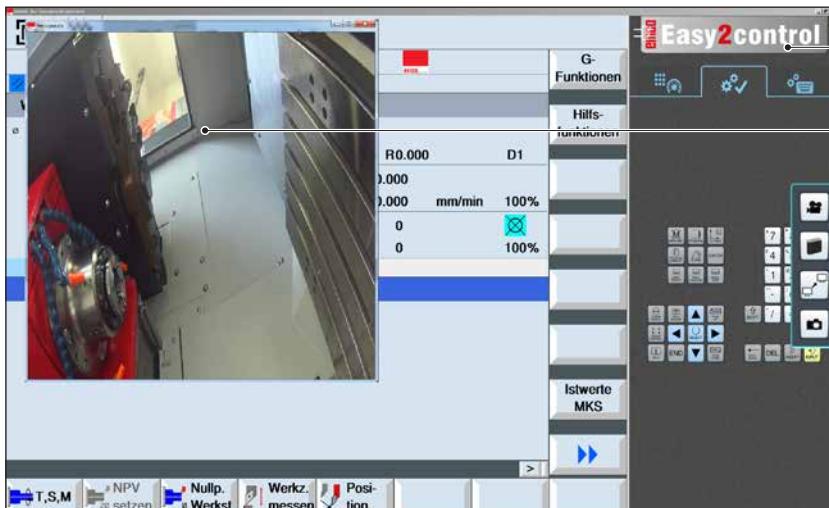
- press NEXT (1) or PREV (2) until an operating mode appears which supports WLAN, e.g. MOVIE. The symbol for WLAN (3) appears in the corner on top left.
- Open EMConfig and activate the camera.
- Connect the WLAN adapter to the USB port to the machine.
- Open windows network and sharing centre (4).
- Select the wireless network, enter the password and finish the connect setup.  
The name of the network (5) and the password are supplied together with the camera.
- Open the control software with activated Easy-2control.

## Operating the camera

- Click the Easy2control logo to open the sidebar.

### Functions of the sidebar

- Click the camera symbol to open the preview window (2).
- Open the operating manual of the control.
- Option for the second screen:
  - duplicate screen
  - screen extension to two monitors
- Creates a screenshot of the control in \*.png format.



Operating the machine room camera

#### Note:

The option for the second screen is only available for machines CT/CM 260 and 460.



#### Attention:

The camera may not be operated without the supplied waterproof case.

If the camera is operated without the waterproof case it may be damaged by coolant and chips.



# Z: Software Installation Windows

## System prerequisites

### Machine with integrated control PC

- All Concept machines
- Machines that were converted to ACC
- MOC with Windows 7 or higher (32 / 64 Bit)

### Machines with included control PC and programming stations

- Windows 7 or higher (32 / 64 Bit)
- free hard drive space 400 MB
- Programming station: 1\*USB, machine version: 2\*USB
- TCP/IP-capable network card for machine version)

### Recommended system prerequisites

- PC Dual Core
- Working memory 4 GB RAM
- free hard drive space 2 GB

## Software installation

- Start Windows
- Start the installation application on the USB stick or your download file.
- Follow the instructions from the installation guide.

For more informations regarding software installation and / or software update please refer to the documentation "short description for WinNC update installation".

### Note:

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

## Variants of WinNC

You can install EMCO WinNC for the following CNC control types:

- WinNC for SINUMERIK Operate T and M
- WinNC for FANUC 31i T and M
- HEIDENHAIN TNC 426
- Emco WinNC for HEIDENHAIN TNC 640
- FAGOR 8055 TC and MC
- CAMConcept T and M

In case there are several control types installed, a menu appears when starting EM Launch from which you can select the desired type.

The following versions can be installed from the WinNC variants:

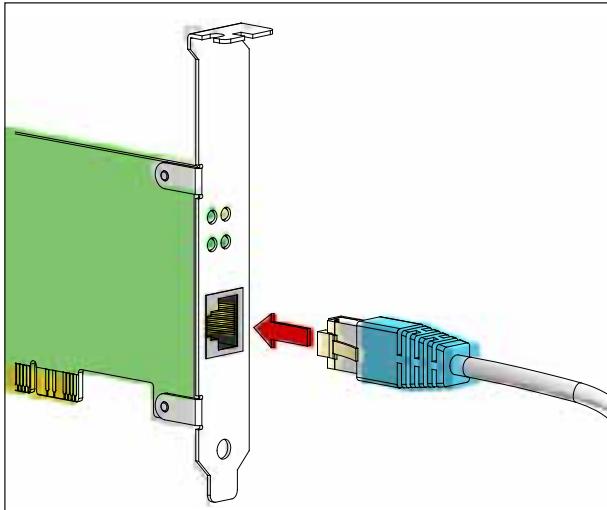
- Demo licence:  
The demo licence is valid for 30 days after the first use. 5 days before the demo licence expires, you can enter another valid licence key (see licence manager)
- Programming station:  
Programming and operation of the appropriate CNC control type is simulated by WinNC on your PC.
- Single user licence:  
Authorizes to external programming of CNC-controlled machine tools on one PC workstation.
- Multi-user licence:  
Authorizes to external programming of CNC-controlled machine tools. The multi-user licence can be installed on an unlimited number of PC workstations or in a network within the institute registered by the licensor.
- Educational licence version:  
Is a time-limited multi-liscence especially for schools and educational institutes.
- Machine licence:  
This licence allows to directly operate a PC-controlled machine (PC TURN, Concept TURN, PC MILL, Concept MILL) of WinNC as if it was operated by an ordinary CNC control.

**Danger:**

Removal and installation of the network card must only be carried by skilled personnel. The computer must be disconnected from the power supply (pull the power plug).

**Note:**

During a machine installation one network-card is reserved exclusively for the control of the machine.



*Connection of the machine to the PC*

**Network card (ACC)**

for:

Concept Turn 55  
Concept Mill 55  
Concept Turn 105  
Concept Mill 105  
Concept Turn 60

Only for machines with ACC kit:

PC Turn 50  
PC Mill 50  
PC Turn 100  
PC Mill 120

Network card type: TCP/IP compatible network card

Setting the network card for the local connection to the machine:

**IP address: 192.168.10.10  
Subnetmask 255.255.255.0**



In case of problems observe the instructions of your operating system (Windows help).

**Instructions:**

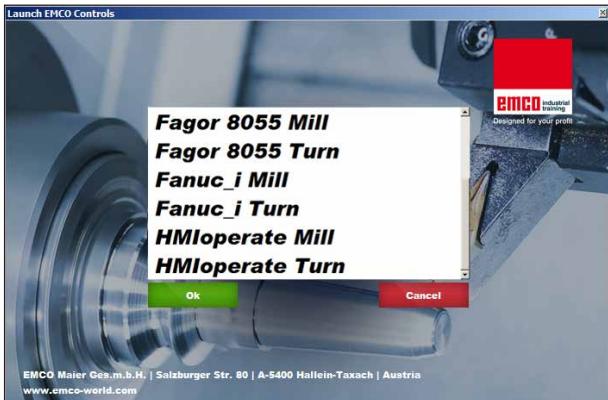
If the network connection to the machine could not be established at the start, the above adjustments are to be made.

## Starting WinNC

If you choose AUTO START YES during the installation of your machine version, WinNC starts automatically after switching on the PC.

Otherwise proceed as follows:

- 1 Switch the machine on.
- 2 Wait for 20 seconds to ensure that the machine operating system is running before the network connection to the PC is established. Otherwise it is possible that no connection can be established.
- 3 Switch the PC on and start Windows.
- 4 Click on the start symbol at the bottom.
- 5 Select program, EMCO and click on WinNC.
- 6 The start image will be shown on the screen. The licence holder is registered in the start screen.
- 7 If you have only installed one CNC control type, it starts immediately.
- 8 If you have installed several CNC control types, the selection menu appears.
- 9 Select the desired CNC control type (use cursor buttons or mouse) and press ENTER to start it.
- 10 If you use the control keyboard, you can select the desired CNC control type with the cursor buttons or mouse and start with the "NC-Start" button.



Selection menu EMLaunch

### Note:

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



## Terminating WinNC

- 1 Switch off auxiliary drive with AUX OFF. Only for machine places, not for programming stations.
- 2 By simultaneously pressing these buttons WinNC for Sinumerik Operate will be terminated specifically.  
This corresponds to Alt+F4 on the PC keyboard.

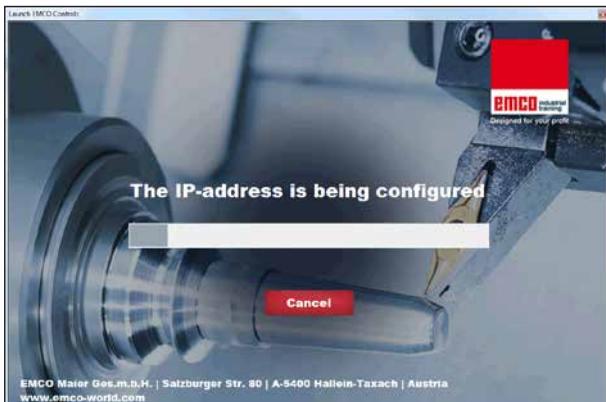
## Checks by EmLaunch

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

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



DHCP disabled



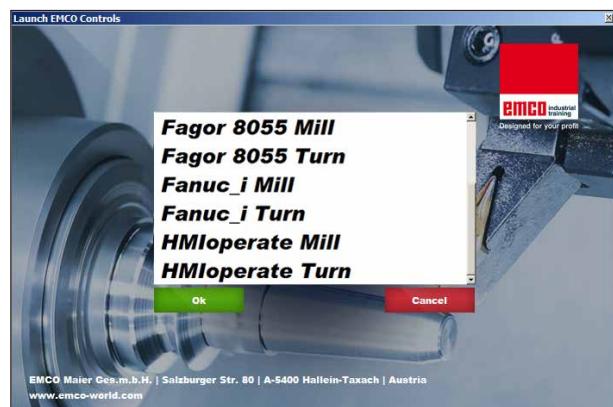
IP-address configuration



Setup the connection to the machine

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

The IP configuration is correct and the connection to the machine is checked. Once the machine is available, the selection of the available controls is displayed.

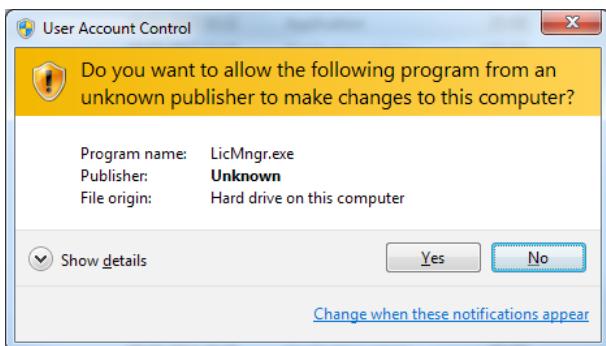


The connection to the machine is completed and the corresponding control can be started.

*Conncetion to the machine is OK*



Input window licence key enquiry



Run EMCO licence manager as an administrator



EMCO Licence Manager

## Licence input

After the installation of an EMCO software product, an input window appears during initial operation and asks for name, address and licence key. This input window appears for every software product that is installed. In case a demo licence is desired (see page Z1), please select "DEMO". Then the input window reappears only 5 days before the expiry date of the demo licence. A subsequent input of a licence key is also possible via the licence manager (see licence manager below).

## Licence manager

The query in the UAC dialog box must be confirmed with Yes in order to start the Licence Manager.

For the release of additional function groups of existing EMCO software products it is necessary to enter a new licence key (exception: demo licence).

The **EMCO Licence Manager** (see picture on the bottom on the left) enables the input of further new licence keys. For this purpose select the new product in the selection window and confirm the input.

The next time you start your control software an input window appears and asks you to enter name, address and licence key (see picture on the top left).

Please note that the licence key is asked for each software product individually. The picture on the left shows e.g. the input prompt for the licence key for the software product "Heidenhain TNC 426".

### Input licence key:

Start the WinNC with the option "Run as Administrator" right after installing the programm or launching the licence manager.