EMCO WinNC for Heidenhain TNC 640

Software description
Emco WinNC for Heidenhain TNC 640

Ref. no. EN 1844
Issue C 2019-06   **Rev 03**

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

Original operating instructions   **Software release effective from 01.00**

Note:

This manual can be used as a reference for the available scope of the described software (in the SW version to which the manual refers). All functions described in this document are implemented in the manner described here.

However, the scope of this manual doesn't necessarily include the entire functionality of the control software. Rather, emphasis is placed on presenting the important functions simply and clearly in order to achieve the greatest possible learning success.

Depending on the machine you are operating with this software (or have selected for the programming station), not all functions are available (example: the Concept machines TURN/MILL 55 has no position-controlled main spindle, therefore it's not possible to program spindle positions.

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

User's Manual Schulung NC-Programmierung, Schwenkbearbeitung iTNC 530 **)

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.

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

EMCO GmbH
Technical Documentation Department
A-5400 HALLEIN, Austria

emco

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.

emco

Table of contents

emco

E: Tool programming

F: Program sequence

G: Flexible NC programming

H: Alarms and Messages

I: Heidenhain TNC 640 controller alarms

W: Accessory Functions

X: EMConfig

Y: External Input Devices

Z: Software Installation Windows

A: Principles

Points on the machine

Machine reference points

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!
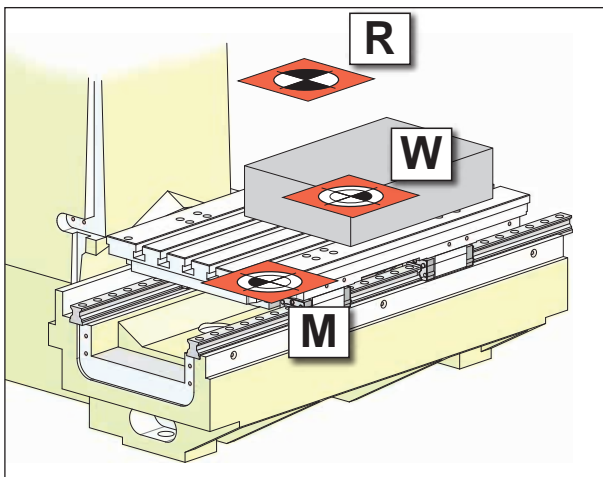
M = Machine zero point

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 pro-grammed by the user. By programming a work-piece 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.
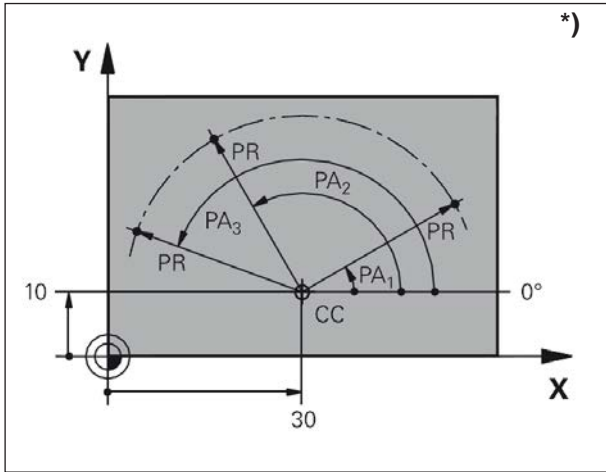
Polar coordinates *)

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)
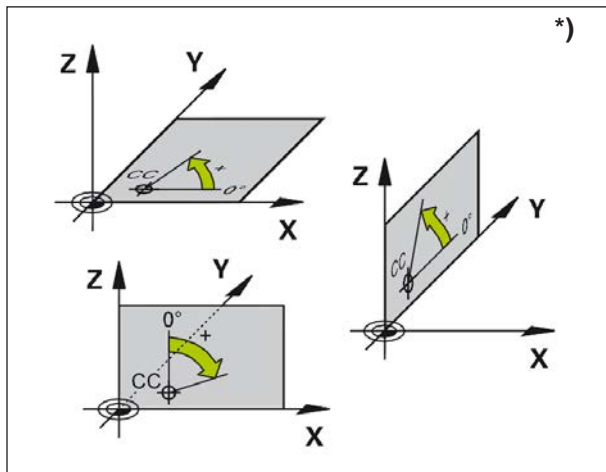
Determining the pole and the angle reference axis

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

emco

Absolute and incremental workpiece positions *)
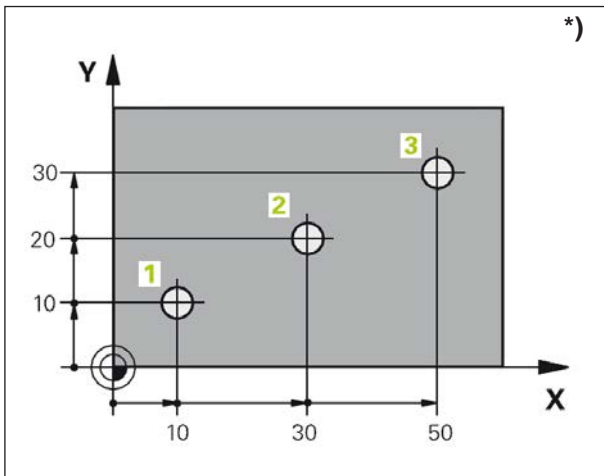
Absolute workpiece positions

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 of machine zero point M to tool zero point W

Zero point offset

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

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".

emco

Milling operations
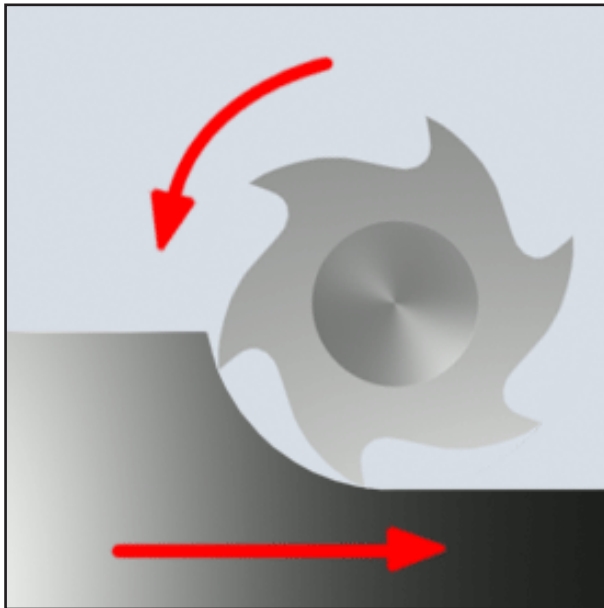
Down-cut Milling

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).
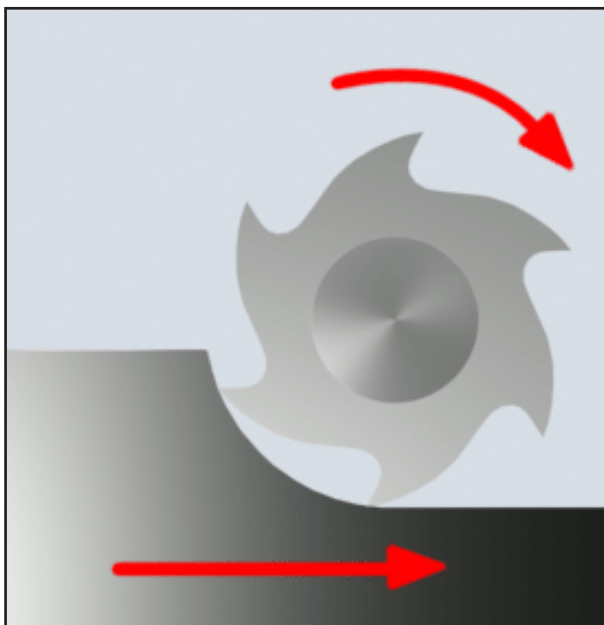
Down-cut Milling

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.

Up-cut milling

Down-cut/up-cut milling

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

Tool length
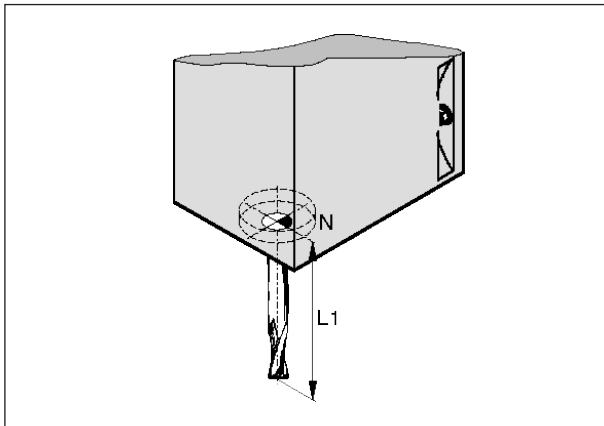
Tool data

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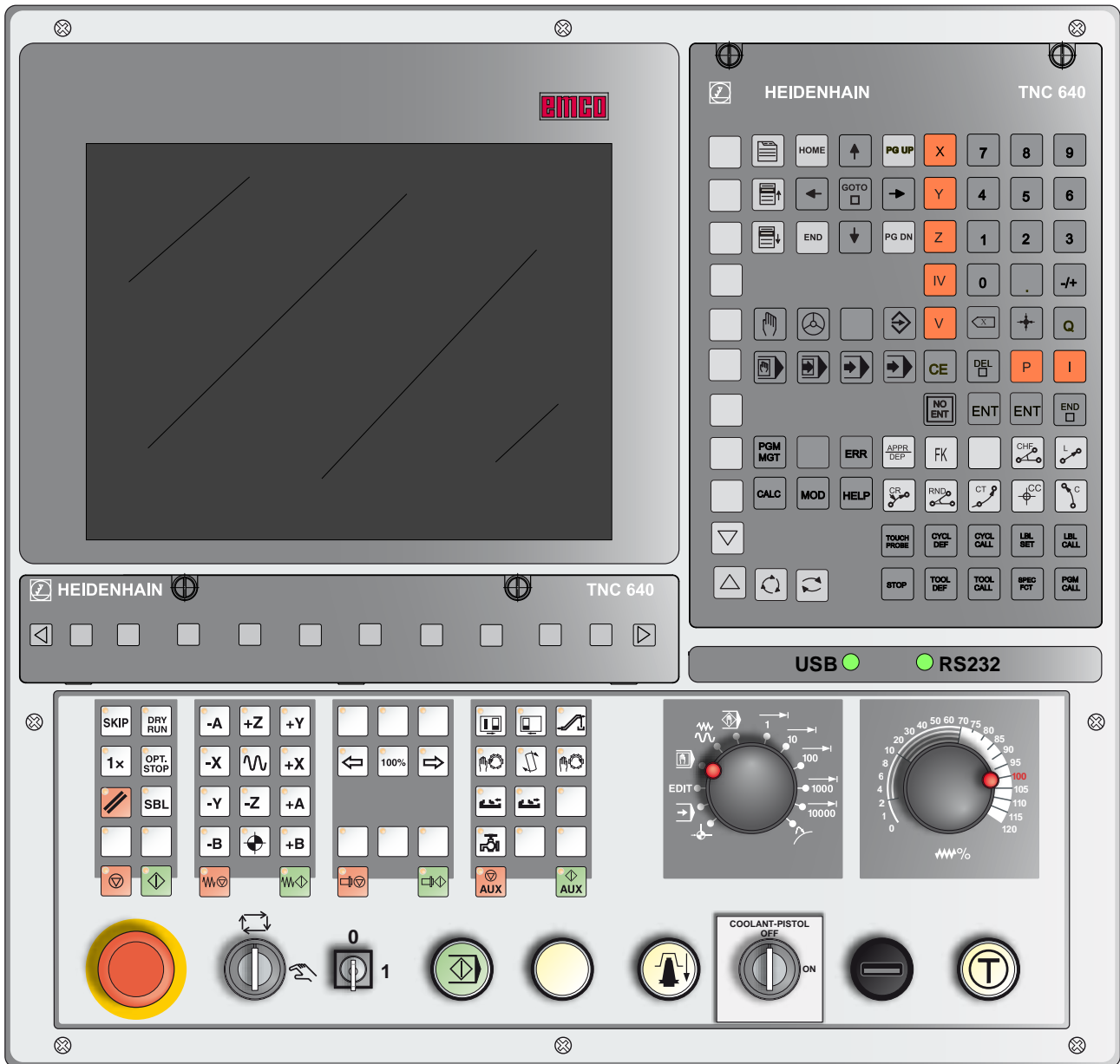
Note:
Depending on the machine you are operating with Emco WinNC for Heidenhain TNC 640, not all functions and machine keys may be available.

emco

EMCO WinNC for Heidenhain TNC 640 control keyboard variant with Easy2control and MOC-Touch
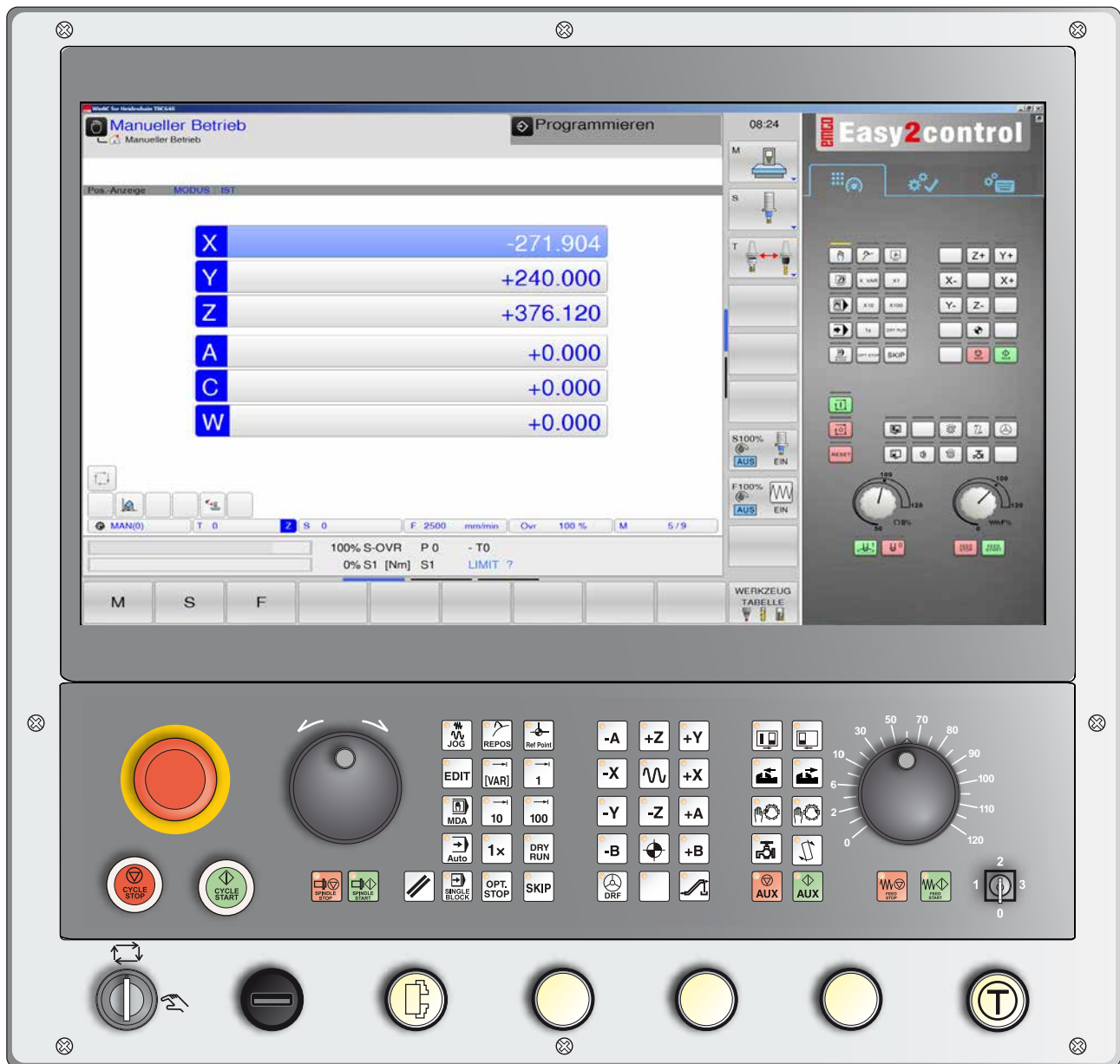
Note:
Depending on the machine you are operating with Emco WinNC for Heidenhain TNC 640, not all functions and machine keys may be available.

emco
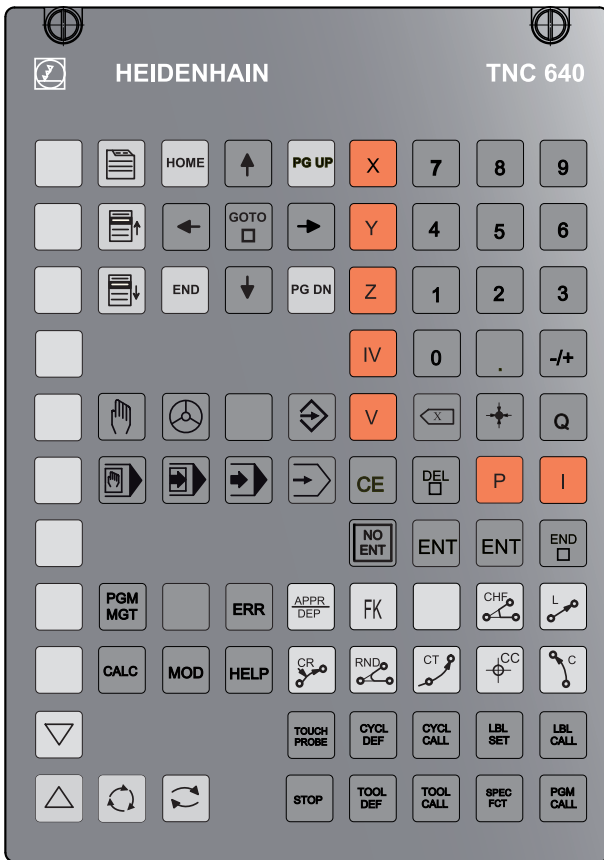
Address and number pad

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

PGM MGT — Select and delete programs/files
External data transfer

PGM CALL — Enter program call in a program

MOD — Select MOD function

HELP — Display context sensitive help

CALC — Display calculator

ERR — Displays all pending error messages

emco

Navigation keys

Shift illuminated field

GOTO 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

TOOL DEF Define tool data in the program

TOOL CALL Call up tool data

Program path movements

APPR DEP Approach / leave contour

FK 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
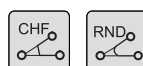
Chamfer / Corner rounding

Special functions

| SPEC FCT | Display special functions |

Select next tab in form

Dialogue or button next / back

Insert / edit coordinate axes and numerical values

X ... V   Select and/or enter coordinate axes in program

0 ... 9   Numerals

. ... -/+   Reverse decimal point and/or algebraic sign

P ... I   Enter polar coordinates / incremental values

Q   Q parameter programming / Q parameter status

             Adopt current position, values from calculator

NO ENT   Skip dialogue questions and delete words

ENT   Conclude input and continue dialogue

END   Conclude block, finish entry

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

DEL   Cancel dialogue, delete program section

emco

Screen layout manual operation
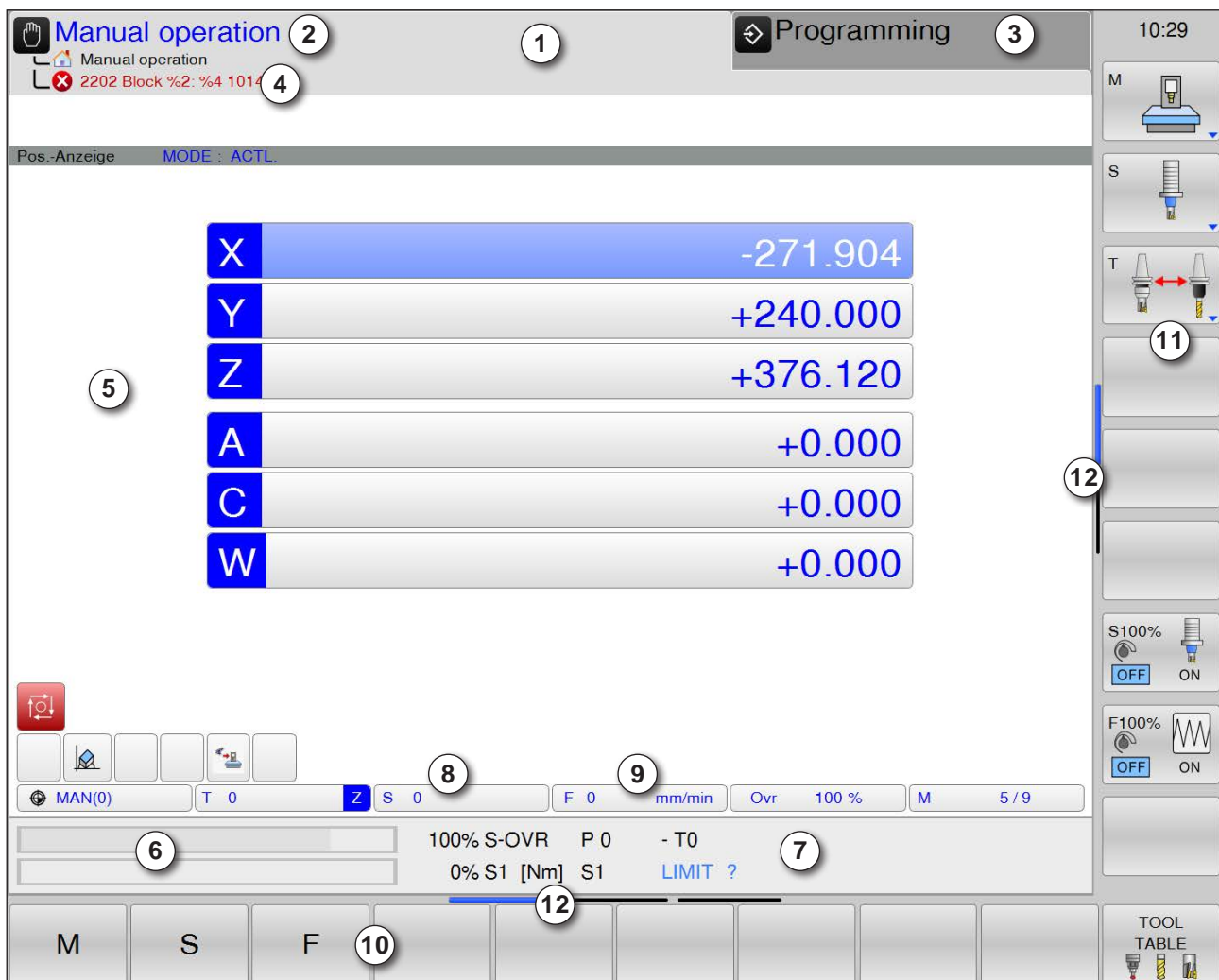
1   Header: Display of selected modes of operation:

2   Left tabs: Machine modes of operation

3   Right tabs: Programming modes of operation

4   Alarm and notification lines

5   Working window, NC displays

6   Power display

7   General status display informs about the current state of the machine:
S-OVR, S1, T0, LIMIT

8   Current spindle speed

9   Programmed feed

10   horizontal softkey bar

11   vertical softkey bar

12   narrow bars above the softkey bar show the number of softkey bars.
The current bar is displayed in blue colour

For detailed descriptions see chapter "C Operation"

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

emco

PC keyboard

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

Strg + Alt =

PC keyboard machinefunctions

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

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

emco

PC key	Control key	Function
F1 _ F10		Softkey keys
Strg F1	HELP	Display context sensitive help
Strg F11		Select screen layout
Strg F12		Switch screen between machine and programming mode of operation
F11	◁	Switch softkey bar
F12	▷	Switch softkey bar
Enter	ENT	Conclude input and continue dialogue
(arrow keys)	← ↑ ↓ →	Move mark
Strg Alt 1		Manual operation
Strg Alt 2		Electronic hand wheel
Strg Alt 4		Saving / editing program
Strg Alt APPR DEP 5	APPR DEP	Approach / leave contour
Strg Alt FK 6	FK	Free contour programming
Strg Alt CHF 8	CHF	Chamfer
Strg Alt 9	L	Straight line
Strg Alt Q		Positioning with manual input
Strg Alt W		Single block program run
Strg Alt E		Block sequence program run
Strg Alt R		Program test
Strg Alt T	CR	Circular path with radius

emco

PC key	Control key	Function
Strg Alt RND Z	RND	Corner rounding
Strg Alt CT U	CT	Circular path with tangential contact
Strg Alt CC I	CC	Circle centre / pole for polar coordinates
Strg Alt C O	C	Circular path around circle centre
Strg Alt PGM MGT P	PGM MGT	Select and delete programs/files, external data transfer
Strg Alt CYCL DEF S	CYCL DEF	Define cycles
Strg Alt CYCL CALL D	CYCL CALL	Call up cycles
Strg Alt LBL SET F	LBL SET	Enter subprograms and program section repeats
Strg Alt LBL CALL G	LBL CALL	Call up subprograms and program section repeats
Strg Alt HELP H	HELP	Display context sensitive help
Strg Alt ERR L	ERR	Display all pending error messages
Strg Alt STOP Y	STOP	Enter program stop in a program
Strg Alt TOOL DEF X	TOOL DEF	Define tool data in the program
Strg Alt TOOL CALL C	TOOL CALL	Call up tool data
Strg Alt PGM CALL B	PGM CALL	Define program call up, select zero point and point tables
Strg Alt CALC N	CALC	Display calculator
Strg Alt MOD M	MOD	Select MOD function
DEL Entf	DEL	Cancel dialogue, delete program section
END Ende	END	Conclude block, finish entry
Bild		Select next tab in form
Strg Alt ↑		Next dialogue or key

emco

Overview key assignment control keyboard

PC key	Control key	Function
Strg — Alt — ↓ / ▤↓	▤↓	Back to previous dialogue or key
Strg — Alt — ✛÷	✛	Adopt current position, values from calculator
Strg — Alt — •/X	•	Reverse decimal point
Strg — Alt — ⁻/+ / -	-/+	Reverse algebraic sign
Strg — Alt — GOTO▢ / 5	GOTO ▢	Directly select blocks, cycles and parameter functions
Strg — Alt — Q / 6	Q	Q parameter programming / Q parameter status
Strg — Alt — I / 3	I	Enter incremental values
Strg — Alt — P / 9	P	Enter polar coordinates
Strg — Alt — DEL▢ / Entf	DEL ▢	Cancel dialogue
Strg — Alt — NO ENT	NO ENT	Skip dialogue questions and delete words
Strg — Alt — ENT	ENT	Conclude entry, delete program section
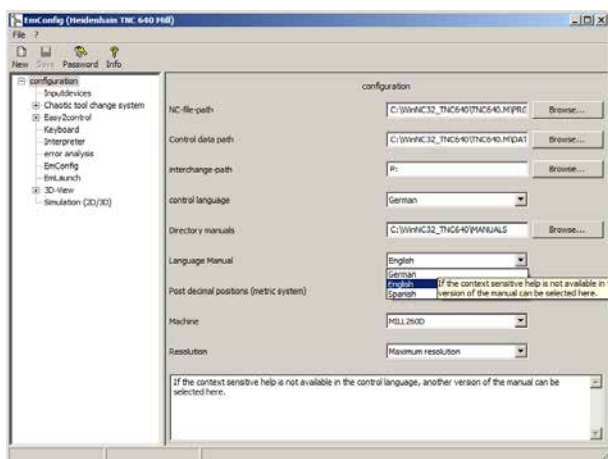
Language manual

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
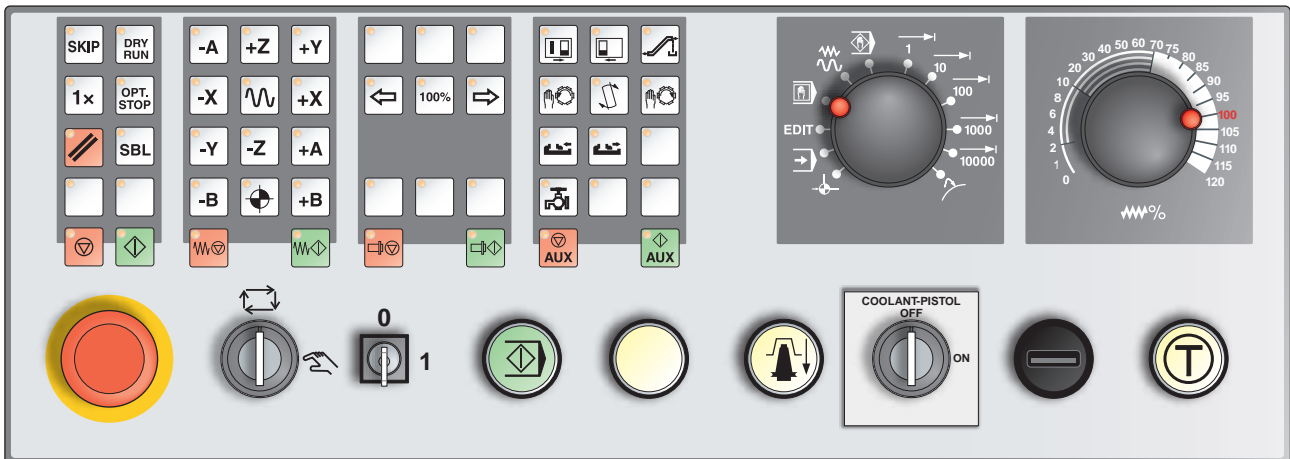
Depending on machine configuration, the control panel can differ slightly from what is shown here.
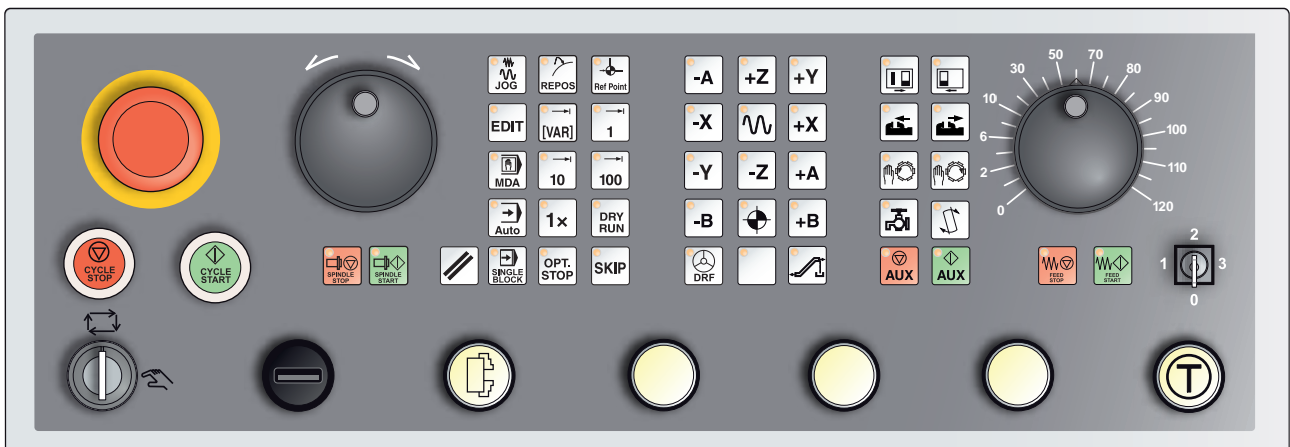
Machine control panel variant with Easy2Control and MOC-Touch

Key description

Skip (block mask)

SKIP

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

Dry run (test run feed)

DRY RUN

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.

emco

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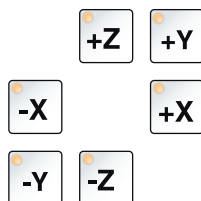
Use these keys to move the NC axes in JOG mode.

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 μinch

Inc 10 - Incremental feed

Move step by step a predefined distance of 10 increments.
Metric measurement system: Inc 10 corresponds to 10μm
Imperial (inch-based) measurement system: Inc 10 corresponds to 1 μinch

Inc 100 - Incremental feed

Move step by step a predefined distance of 100 increments.
Metric measurement system: Inc 100 corresponds to 100μm
Imperial (inch-based) measurement system: Inc 100 corresponds to 10 μinch

Inc [VAR]

Move in steps with a continuously variable increment.

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

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

- Metric is assigned in the imperial measuring system as follows:

  Feed:
  Millimetres in inches:
  mm/min => inches/min
  mm/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 [Jog] 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 [Jog] keys and then [↕].
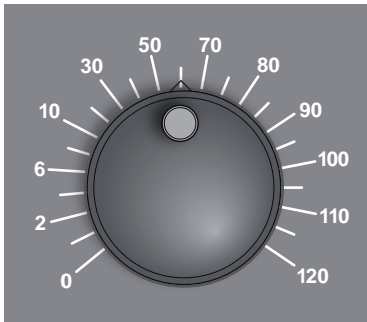
Override switch (feed rate override)

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
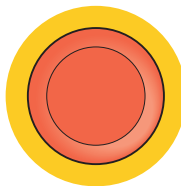
Press the red button only in an emergency.

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.
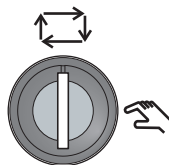
Key Switch Special Operations Mode

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 ....).

emco

Turn/press

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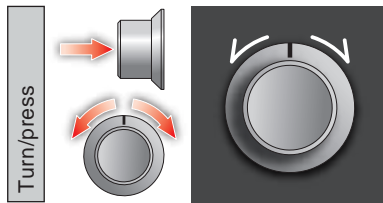
The multifunction control is designed as a rotary switch with a press feature.

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:
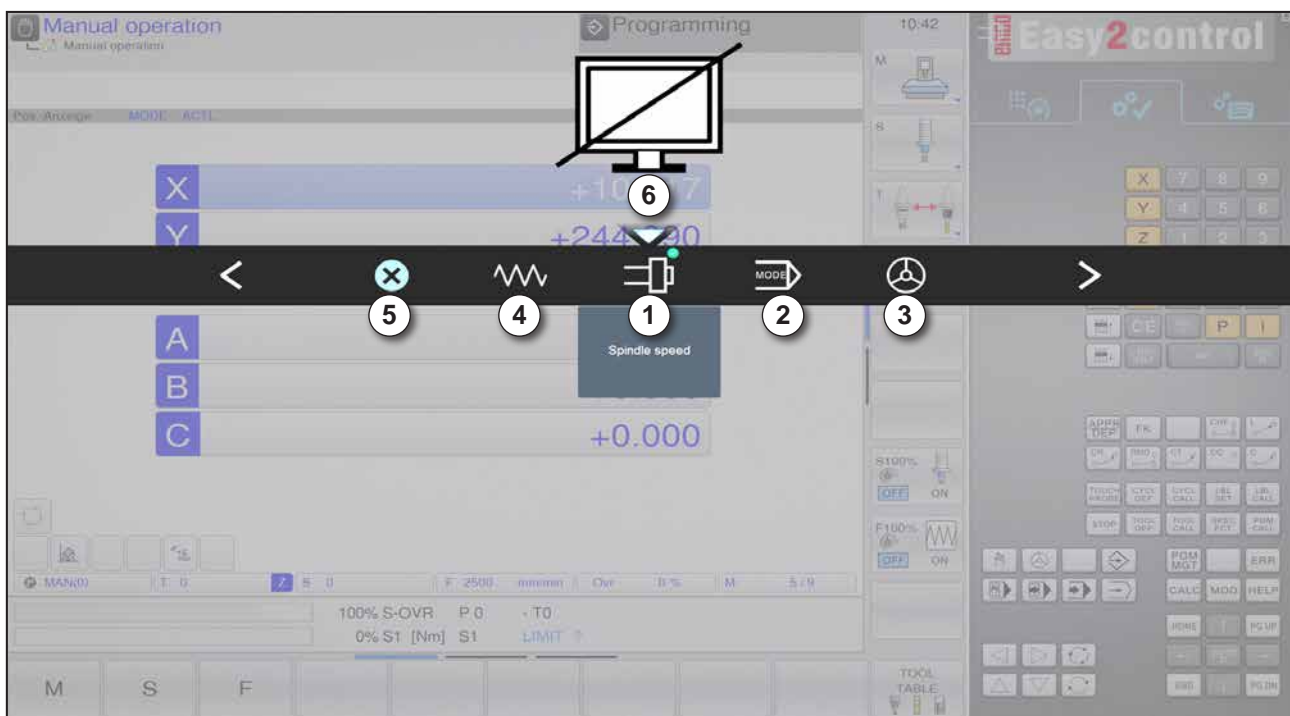
Function overview

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.
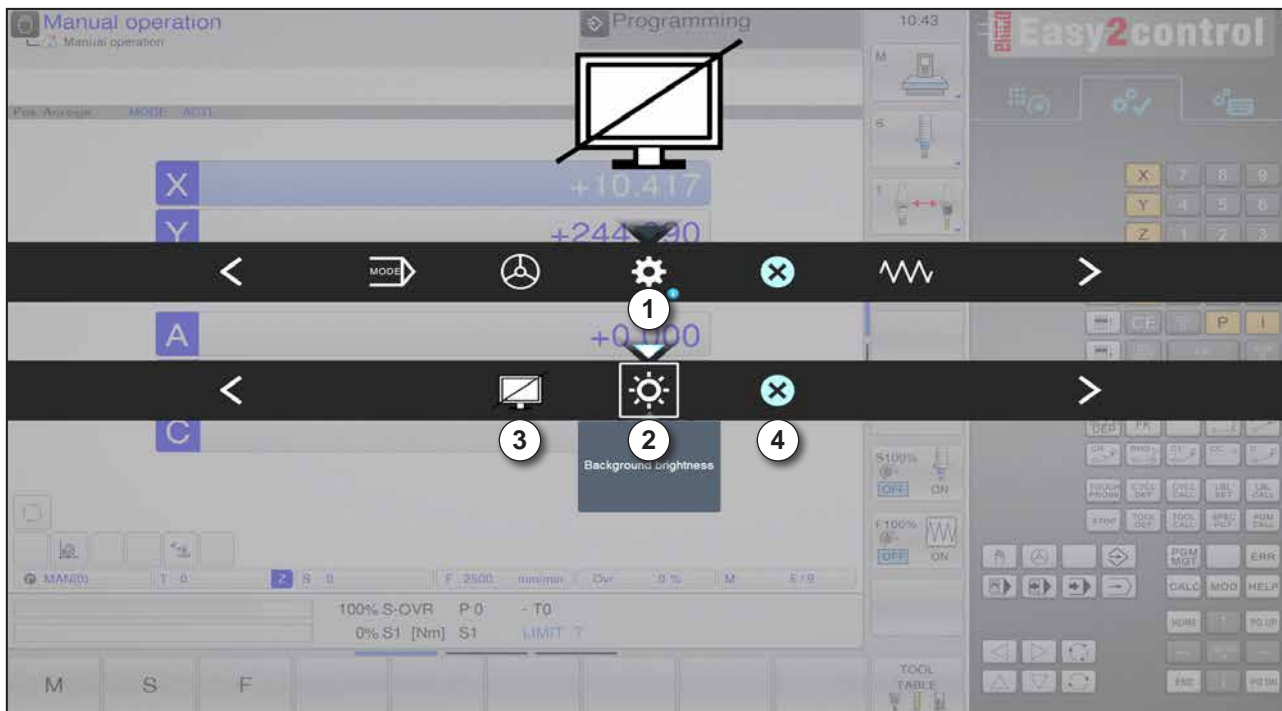
Background brightness settings

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.
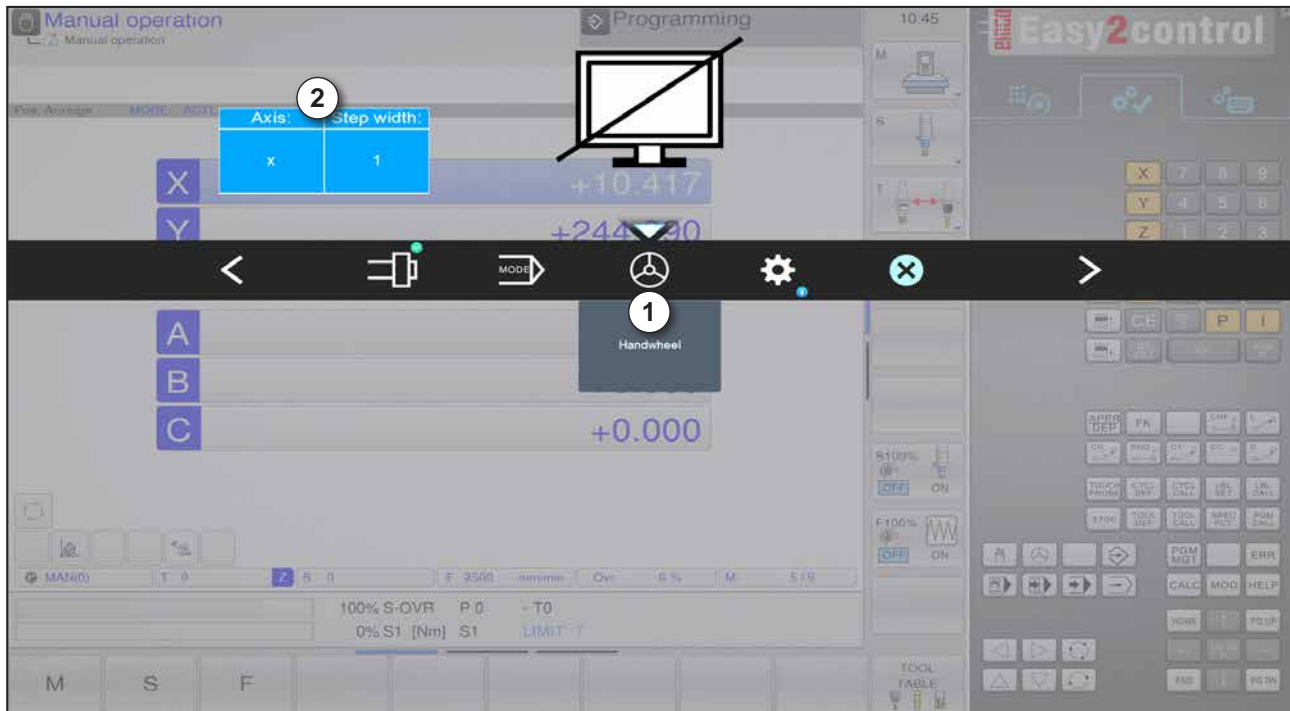
Hand wheel function

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".
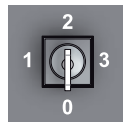
Key switch

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 and spindle speed

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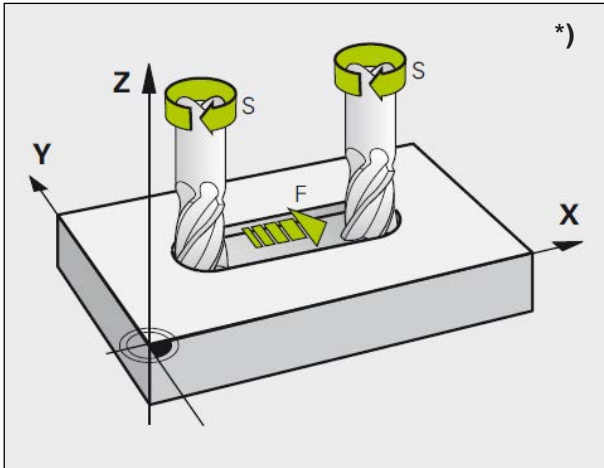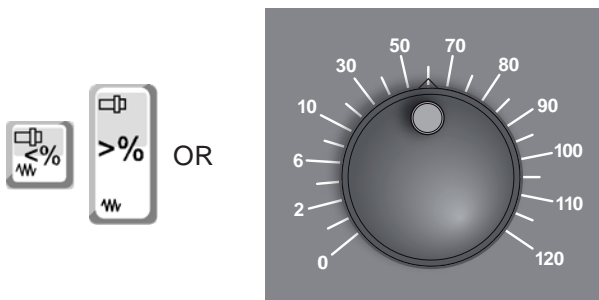
Feed influence
The feed value F programmed by you corresponds to 100%.
With this button or with the feed override, the set feed value F can be changed in %.
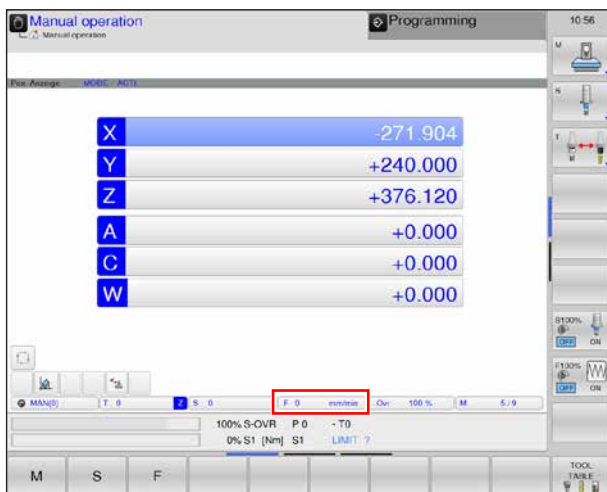
Feed

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.
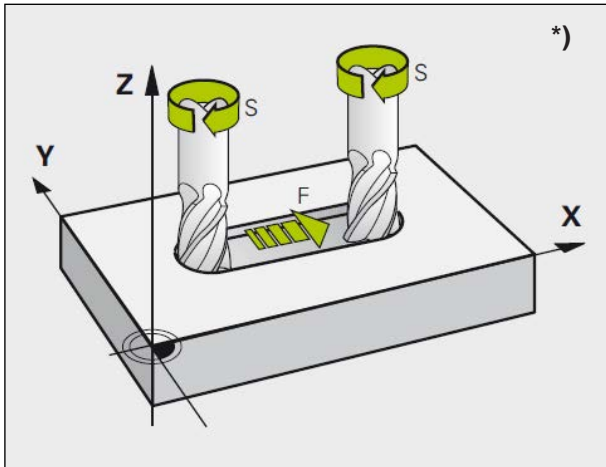
Feed and spindle speed

Spindle speed S [rev/min]

Specify the spindle speed S in revolutions per minute (rpm).
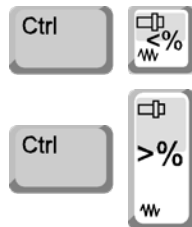
Spindle speed correction
The spindle speed S programmed by you corresponds to 100%.
These button combinations or the spindle speed override change the set spindle speed S in %.
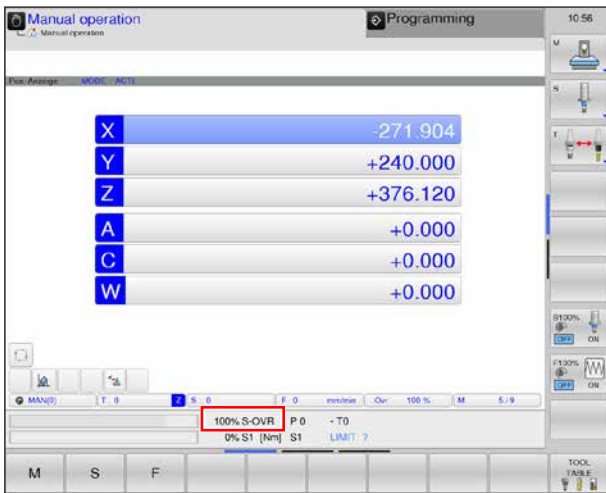
Spindle speed

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

emco

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

Strg   **Alt**   via the PC keyboard
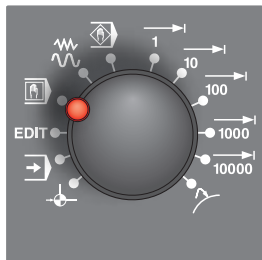
via the mode selector switch of the machine control panel

via the machine control panel variant Easy2Control with MOC-Touch
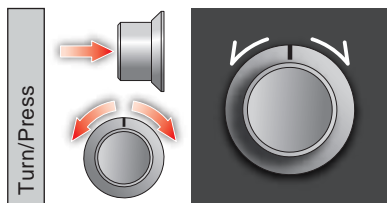
Turn/Press   via multifunction wheel of the Easy2Operate

Navigation in the menu window

F11 or ◁

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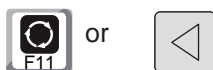
In the footer, further functions of the softkey bar are displayed. For orientation purposes, narrow bars directly above the softkey bar show the number of softkey bars, which can be selected via the black arrow keys arranged on the outside or via the F11 and/or F12-key. The active bar is displayed as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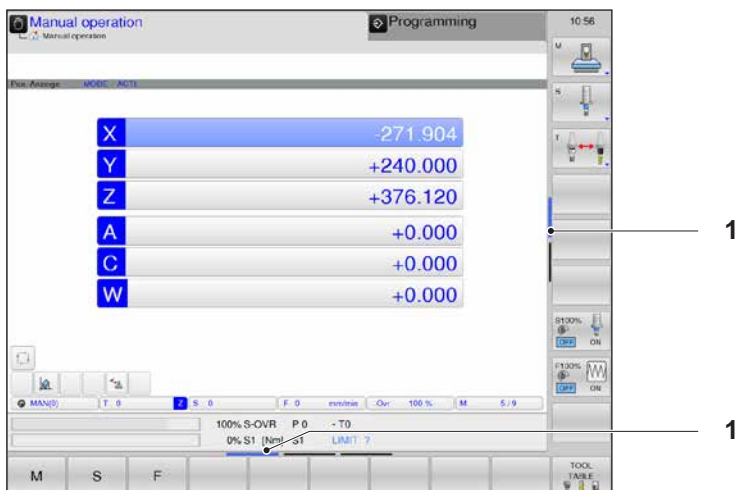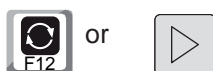
1

1

emco

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.

emco

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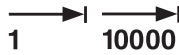
- The arrow keys move the axes in the corresponding direction by the set incremental distance each time the button is pressed.
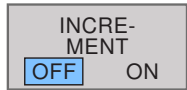
- Set the softkey INCREMENT to ON, in order to select incremental positioning.
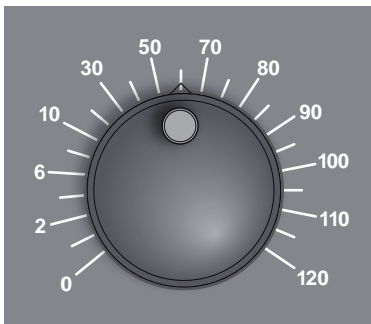
- The feed speed is set with the override switch.

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

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

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

The values entered correspond to "INC var" [VAR].

Confirm the entered value.

Complete the entry with OK.

Switch off increment positioning.

emco

Additional status display

The additional status displays contain detailed information on the program run. They can be called in all operating modes except for the Programming mode of operation.

Switch on the additional status display

Call up the softkey bar for the screen layout.

| POSITION + STATUS |
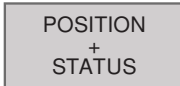
Select the screen layout with additional status display: In the right half of the screen, the WinNC shows the Overview (1) status form per default.
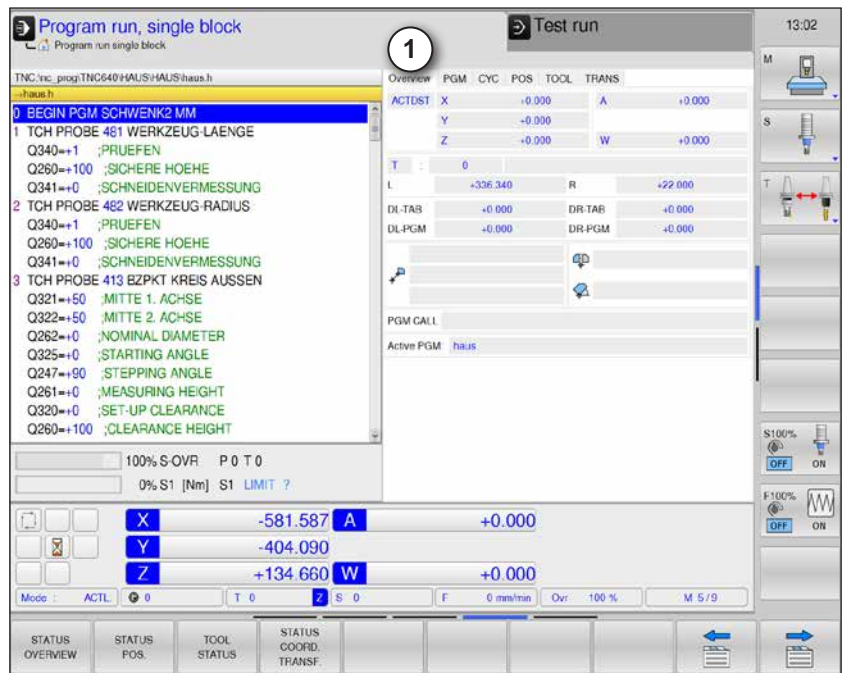
Select an additional status display

Switch the softkey bar until the function for STATUS appears.

| STATUS POS. |
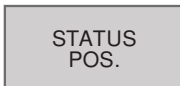
Either select the additional status display directly by softkey, e.g. positions and coordinates, or

Use the switch-over soft keys to select the desired view.

Overview

After switch-on, the WinNC displays the Overview status form, provided that you have selected the PROGRAM+STATUS screen layout (or POSITION + STATUS). The overview form contains a summary of the most important status information, which you can also find on the various detail forms.

The following softkeys are available

Status overview

STATUS
OVERVIEW

- Position display
- Tool information
- Active coordinate transformations
- Program called with PGM CALL

Positions and coordinates

STATUS
POS.

- Type of position display

Information on tools

TOOL
STATUS

- Display of active tool
- Tool length and tool radii
- Oversizes (delta values) from the tool table (TAB) and the TOOL CALL (PGM)

Coordinate transformations

STATUS
COORD.
TRANSF.

- Active datum number
- Mirrored axes
- Active rotation angl
- Active scaling factor/factors

emco

Administration of reference points in the preset table

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

PRESET
TABLE
⊕

- Press softkey to open the preset table.

🖐 OR ⊛

- The preset table can only be used in manual mode and El. Handwheel.

CHANGE
PRESET

- 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 \.
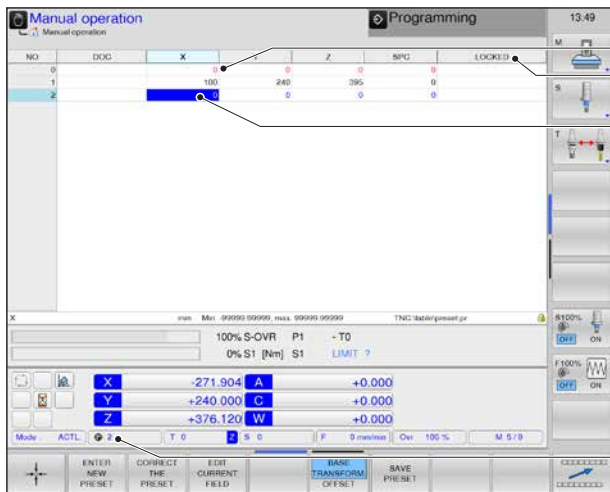
Structure of the preset table

Structure of the preset table

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.

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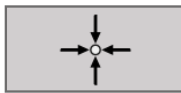
- Take over the actual tool position directly as a new reference point: The function stores the reference point only in the axis which is currently active.

ENTER
NEW
PRESET

- 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.

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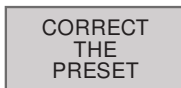
- Moves a stored reference point incrementally: The function stores the reference point only in the axis which is currently active. Enter the desired value in the pop-up window.

EDIT
CURRENT
FIELD

- 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.

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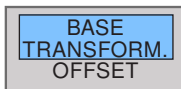
- Select BASE TRANSFORMATION/ OFFSET: In standard view base transformation, the columns X, Y and Z are displayed. SPA, SPB and SPC columns are displayed depending on the machine. The control software stores the basic rotation (in the case of tool axis Z, the control uses column SPC).

BASIS-
TRANSFORM.
OFFSET
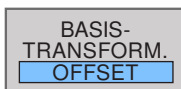
　The Offset view displays the offset values for the preset.

SAVE
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.

Activate preset

- Select manual operation.

PRESET
TABLE
⊕

- Shows Preset table

- Select reference point number to be activated.

ACTIVATE
PRESET

- Activate preset

EXECUTE
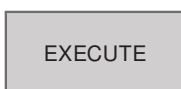
- Confirm activation. The WinNC sets the display and, if defined, the basic rotation.

emco

Edit functions for the table

APPEND
N LINES

• Append N lines at the end of the table.

COPY
FIELD

• Copy actual selected field.

PASTE
FIELD

• Paste field.

RESET
LINE

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

INSERT
LINE

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

DELETE
LINE

• Delete single line at the end of the table.

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

Probing functions
Touch system table
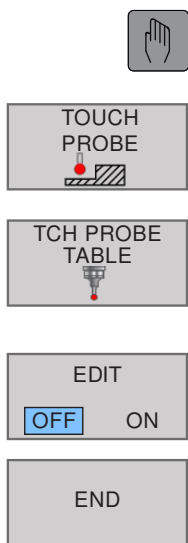
• Select manual mode of operation.

TOUCH PROBE

• Press softkey "TOUCH PROBE" in the softkey bar.

TCH PROBE TABLE

• Press softkey "TCH PROBE TABLE".

EDIT
OFF   ON

• Set softkey "EDIT" to ON.

Conduct desired changes in the table with and complete the editing by selecting "END".

END

Abbr.	Enter	Dialogue
NO	Touch system number.	—
TYPE	Fixed to PART (not changeable)	**Touch system selection?**
CAL_OF1	Offset from touch system axis to spindle axis in the main axis	**TS centre offset main axis? [mm]**
CAL_OF2	Offset from touch system axis to spindle axis in the secondary axis	**TS centre offset secondary axis? [mm]**
CAL_ANG	The WinNC orients the touch system before calibration and/or scanning on the orientation angle (if an orientation is feasible)	**Spindle angle for calibration?**
F	Infeed used by the TNC for the work piece	**Initial touch infeed? [mm/min]**
FMAX	Infeed the touch system uses to pre-position and/or positioning between the measuring points	**Rapid traverse in initial touch cycle? [mm/min]**
DIST	If the touch pin is not deflected within the value defined here, WinNC will output an error message	**Maximum measuring path? [mm]**
SET_UP	SETU_UP is used to determine the distance at which the WinNC should pre-position the touch system from the scanning point that was defined (or calculated by the cycle). The smaller the value here, the more precise the scanning position must be defined. Many touch system cycles also allow you to define a safety distance that is applied in addition to the machine parameter SET_UP.	**Safety distance? [mm]**
F_PREPOS	Determine the speed for pre-positioning: • Pre-positioning with speed from FMAX: FMAX_PROBE • Pre-positioning with rapid machine traverse: FMAX_MACHINE	**Pre-positioning with rapid traverse?** **ENT/NO ENT**

Set reference point on any axis

• Select manual mode of operation.

TOUCH
PROBE

• Press softkey "TOUCH PROBE" in the softkey bar.

PROBING
POS

• Press softkey "PROBING POS".
Position the touch system near the scanning point.

Z-
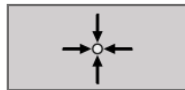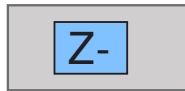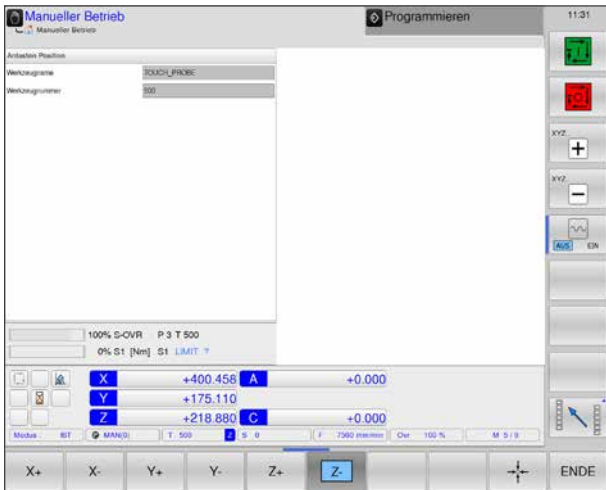
• Select scanning direction and axis.
e.g. press softkey for Z-

CYCLE
START

• Scanning by pressing the START key.

OR

• Directly copy the actual position of the tool as the new reference point: The function will only save the reference point in the axis corresponding to the illuminated field position.
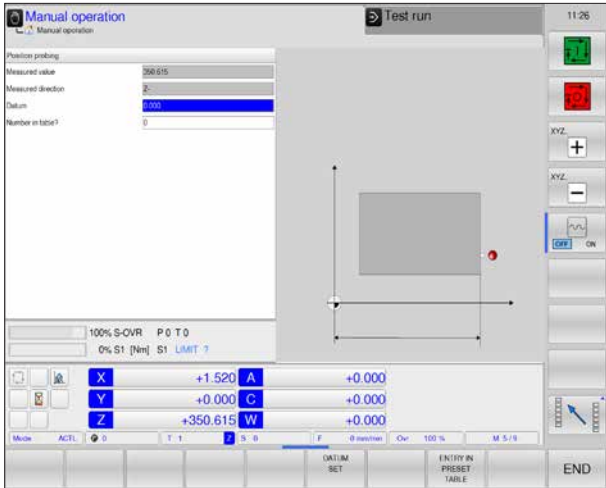
- After a successful measurement, the measuring values can be directly copied into the reference point and/or the preset table.

DATUM
SET

- Press softkey to set the reference point:

ENTRY IN
PRESET
TABLE

- Copying of the value into the preset table
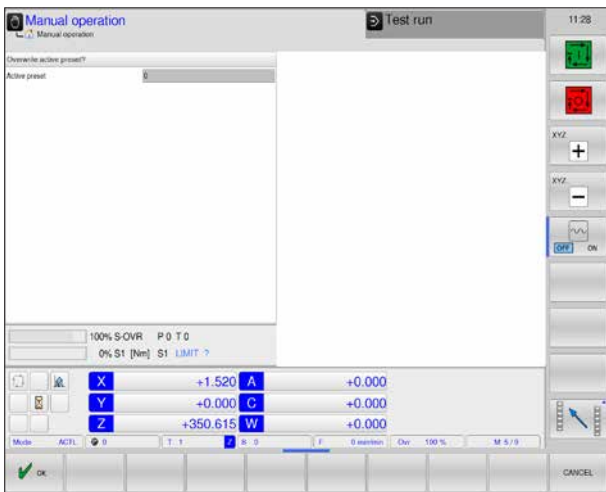
OK   CANCEL

- Overwrite active preset?
  Confirm the preset value with OK or
  end the input with "CANCEL".

END

- End scanning function with softkey "END".

emco

Measuring a basic rotation using a line

- Select manual mode of operation.

- Press softkey "TOUCH PROBE" in the softkey bar.

- Press softkey "PROBING ROT"
  Position the touch system near the 1st scanning point.
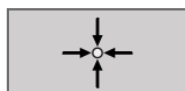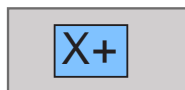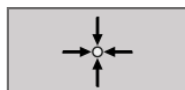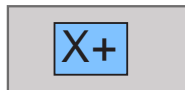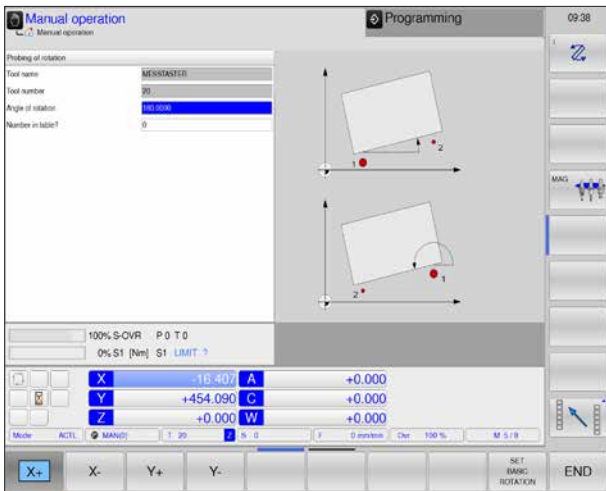
- Select scanning direction and axis.
  e.g. press softkey for X+

- Scanning by pressing the START key.
  OR

- Directly copy the actual position of the tool.

- Repositioning of the tool for the 2nd touch point.

- Select scanning direction and axis.
  e.g. press the softkey for X +

- Scanning by pressing the START key.
  OR

- Directly copy the actual position of the tool.

- The WinNC determines the basic rotation and displays the angle in the Rotation Angle dialog.
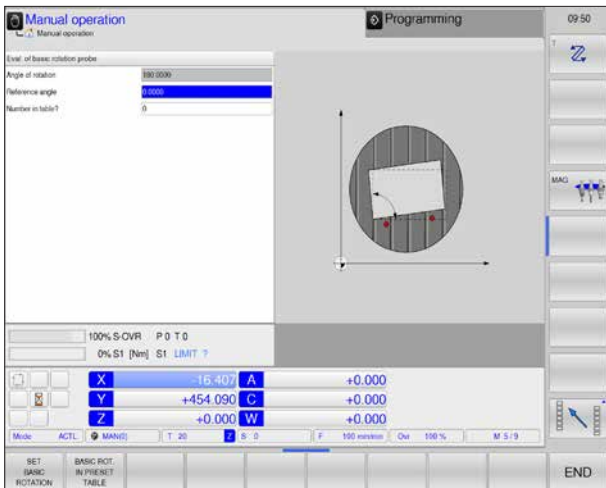
| SET BASIC ROTATION | • Press softkey to activate basic rotation. OR |

| BASIC ROT. IN PRESET TABLE | • Transferring the value to the preset table. |

| END | • End scanning function. |

Cancel basic rotation

• Enter 0 for angle.

| SET BASIC ROTATION | • Press softkey to accept the entry. |

| END | • End scanning function. |

Set circle centre as reference point: Hole

[hand icon]	• Select manual mode of operation.
TOUCH PROBE	• Press softkey "TOUCH PROBE" in the softkey bar.
PROBING CC	• Press softkey "PROBING CC". Position the touch system near the scanning point.
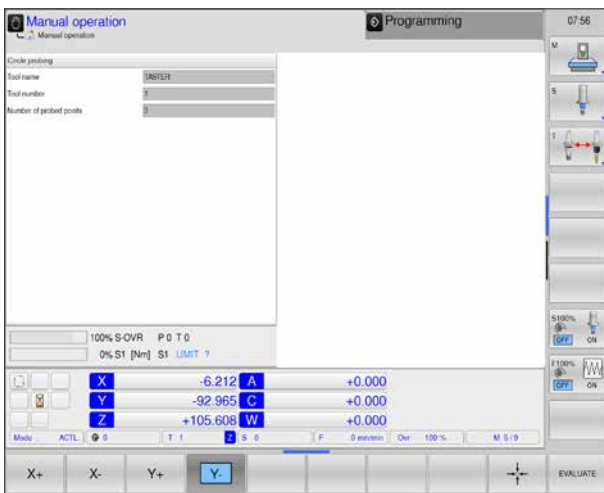
Procedure for manual scanning routine (for datum setting, hole and circular stud)
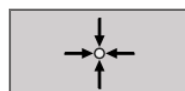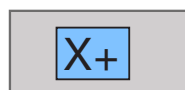
X+	• Select axis for manual scanning. After each measurement, the selected axis jumps forward by one axis.
CYCLE START	• Scanning by pressing the START key. OR
[take over icon]	• Take over measured value after each individual measurement.
EVALUATE	• After at least 3 readings, the measurement result can be evaluated. Press the "EVALUATE" softkey.

Procedure for automatic scanning routine
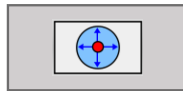
- Press softkey for automatic scanning routine (Hole inner circle).
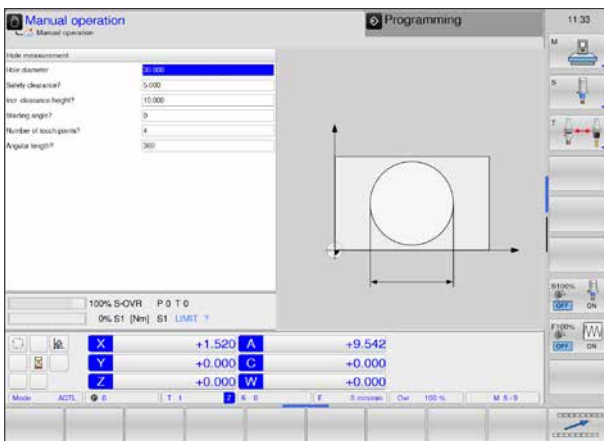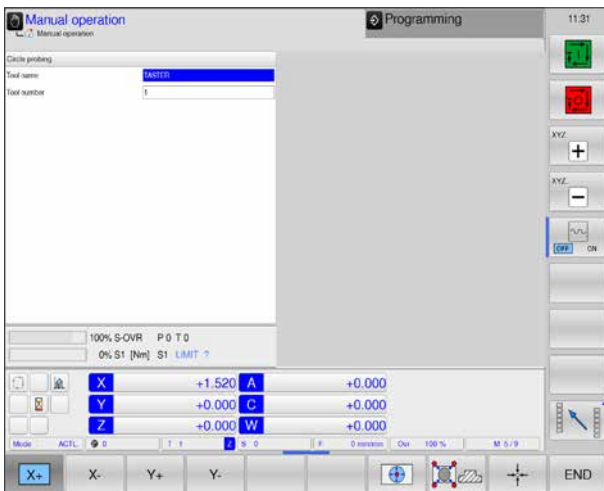
- Enter parameter for:

  Hole diameter
  Safety distance
  Safe height
  Start angle
  Number of scanning points
  Opening angle
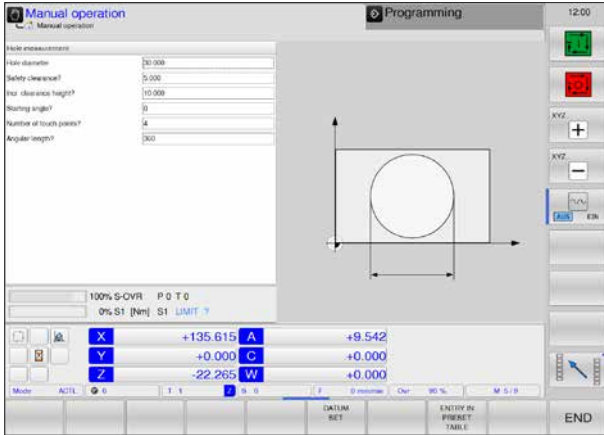
- Scanning by pressing the START key.

  The touch system scans the inner wall of the circle in the selected direction.
  After the third scanning sequence, the centre can be calculated. Four scanning points are recommended.

DATUM SET

- Enter both centre coordinates in the menu window. Press the softkey "DATUM SET" to copy the input. The reference point is the centre of the circle.

ENTRY IN PRESET TABLE

- Press softkey to copy the reference point into the preset table.

END

- End scanning function

Set circle centre as reference point: stud
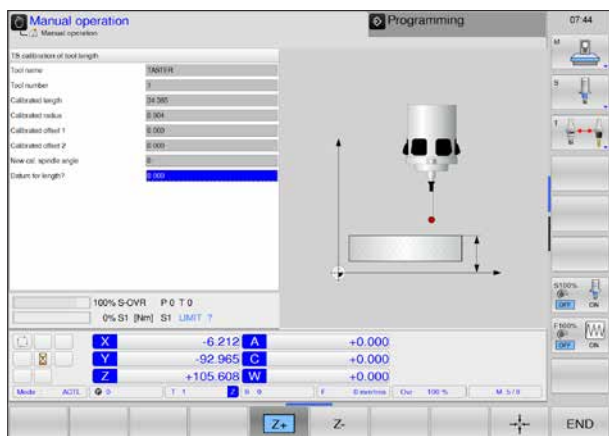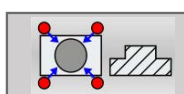
• Select manual mode of operation.

TOUCH PROBE

• Press softkey "SCANNING FUNCTION" in the softkey bar.

PROBING CC

• Press softkey "PROBING CC".
Position the touch system near the scanning point.

• Press the softkey for the automatic scanning of the cylindrical stud (outer circle).

• Enter parameter for:

Number of scanning points (min. 3, max. 8)
Radius
Measuring value X
Measuring value Y
Reference point X
Reference point Y
Number in table

• Scanning by pressing the START key.

CYCLE START

The touch system scans all required points in an automatic probing routine and calculates the effective ball tip radius.

DATUM SET

• Enter both centre coordinates in the menu window. Press the softkey "DATUM SET" to copy the input. The reference point is the centre of the circle.

ENTRY IN PRESET TABLE

• Press softkey to copy the reference point into the preset table.

END

• End scanning function

emco

Setting the centerline as datum

[hand icon] • Select manual mode of operation.
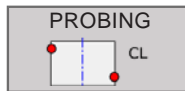
TOUCH
PROBE
[icon] • Press softkey "SCANNING FUNCTION" in the softkey bar.

PROBING
CL • Press softkey "PROBING CL".
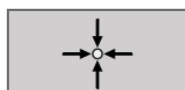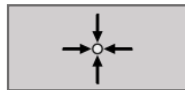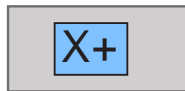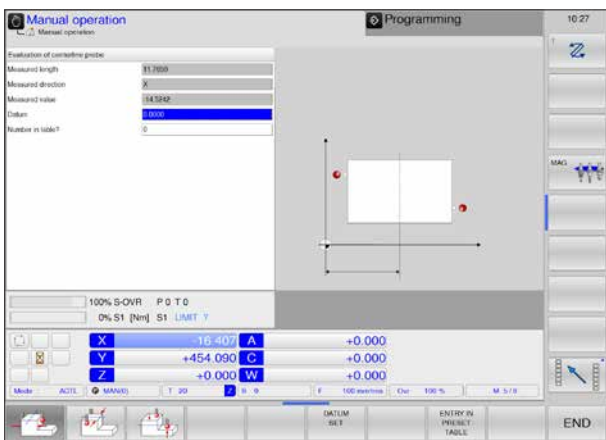
X+ • Position the touch probe near the first touch point.
• Select scanning direction and axis.

CYCLE
START • Scanning by pressing the START key.
OR

[icon] • Directly copy the actual position of the tool.
• Repositioning of the tool for the 2nd touch point.

X- • Select scanning direction and axis.
e.g. press the softkey for X -

CYCLE
START • Scanning by pressing the START key.
OR

[icon] • Directly copy the actual position of the tool.

emco
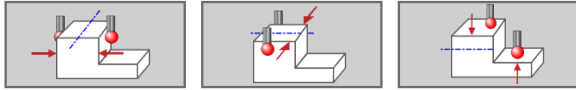
- After you have measured the second touch point, you can use the evaluation menu to change the direction of the centerline.
  You can choose by softkey whether the datum or zero point should be set in the reference axis, minor axis or tool axis.
  This can be necessary if, for example, you would like to save the measured position in the reference and minor.

DATUM SET

- Enter the coordinates of the datum in the menu window.
  Press the "DATUM SET" softkey to accept the entry.
  OR

ENTRY IN PRESET TABLE

- Press softkey to write the value to the preset table.

END

- End scanning function.

emco

Calibrating touch probe

General

In order to precisely specify the actual trigger point of a 3-D touch probe, you must calibrate the touch probe, otherwise the WinNC cannot provide precise measuring results.

During calibration, the WinNC finds the effective length of the stylus and the effective radius of the ball tip. To calibrate the 3-D touch probe, clamp a ring gauge or a stud of known height and known radius to the machine table.

Calibrating the effective length
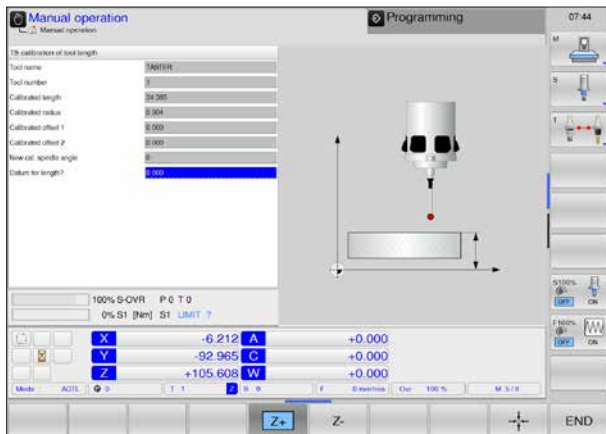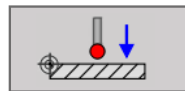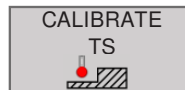
- Select manual mode of operation.

- Press softkey "CALIBRATE TS" in the softkey bar.

- Press softkey "CALIBRATE LENGTH".

- Datum for length: Enter the height of the ring gauge in the menu window.

- Move the touch probe to a position just above the ring gauge.

- To change the traverse direction (if necessary), press a soft key or an arrow key.

- To probe the upper surface of the ring gauge, press the machine START button.

- Check the results.

- Press the OK softkey for the values to take effect.

Calibrating the effective radius

When calibrating the ball tip radius, the TNC executes an automatic probing routine.
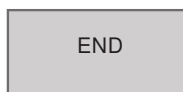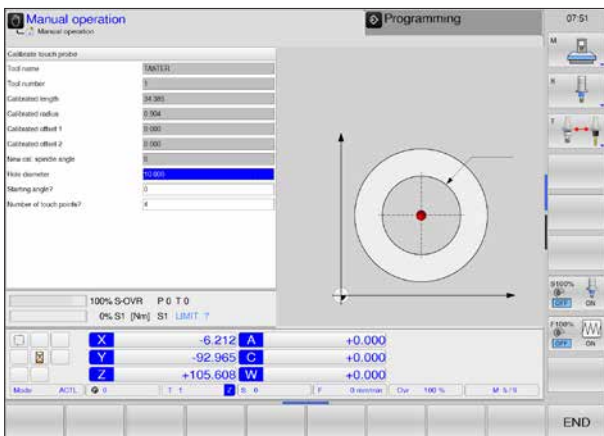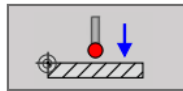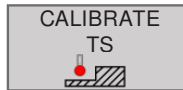
Calibration using a calibration ring

- Select manual mode of operation.

CALIBRATE
TS

- Press softkey "CALIBRATE TS" in the softkey bar.

- Press softkey "CAL. R".

- Enter the diameter of the ring gauge

- Enter the start angle

- Enter the number of touch points

- Start the probing procedure: Press the machine START button

- The 3-D touch probe probes all required touch points in an automatic probing routine and calculates the effective ball-tip radius.

- Press the OK softkey for the values to take effect.

END

- Press "END" to terminate the calibrating function.

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

OFF

SHUT
DOWN

4 SHUT DOWN or

RESTART

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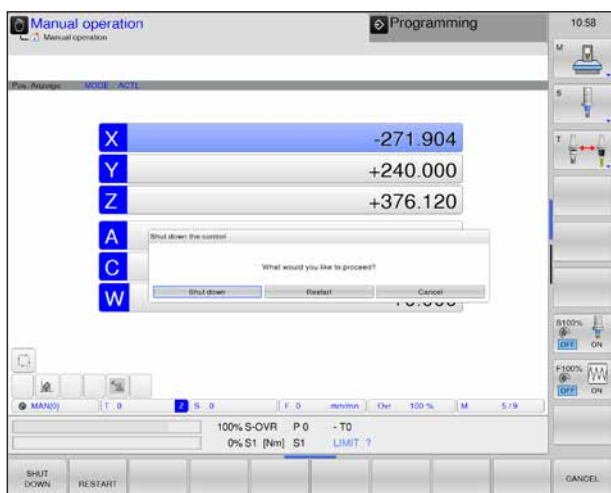
As soon as the control unit has been shut down, the supply voltage of the control unit can be switched off.

Switching off the WinNC randomly can lead to data loss!

Shut down

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).

emco

File management fundamentals

Files

Files in the control unit	Type
Programs	
in HEIDENHAIN format	.H
Tool-table	.T
Tool changer	.TCH
Preset-table	.PR
Point-table	.PNT

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

PGM
MGT

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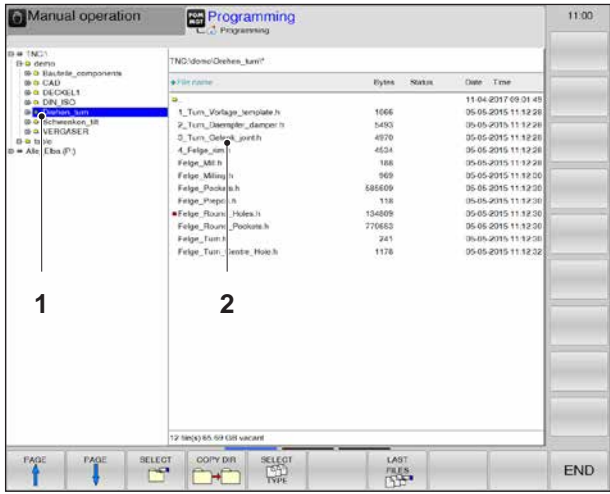
File management

The narrow window on the left-hand side shows the memory area on the data carrier (1) for the WinNC (NC programs, tables). Furthermore, a directory defined as interchange directory (3) is available. This can be set in EMConfig.
In addition, temporary drives (e.g. USB) are displayed.

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:
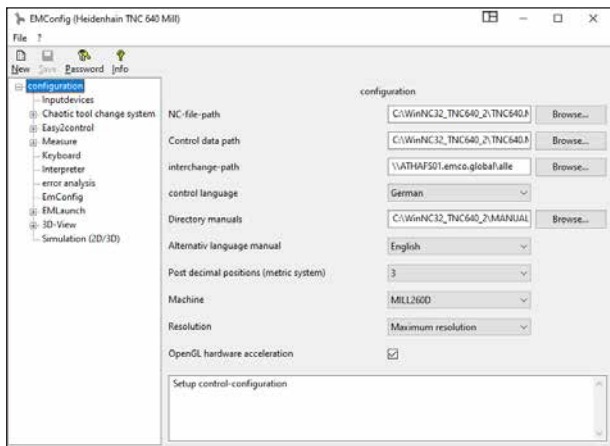
Define interchange-path

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

emco
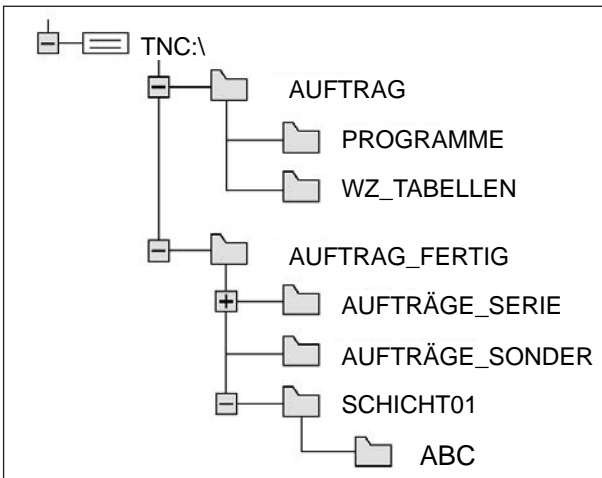
Paths

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

PGM MGT

1 Call up file management

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

OK   CANCEL

ENT

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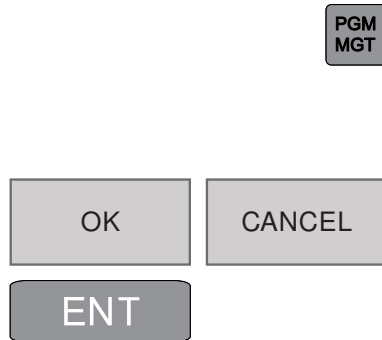
Creating a new file

PGM MGT

1 Call up file management

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

NEW FILE

2 Open the dialogue for creating a new file

3 Enter the new file name with the extension,

MM   INCH

4 Enter measurement system

OK   CANCEL

ENT

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

6 For * .H files (programs) the block form can be defined, see also "Special Functions" - Defining a raw part.
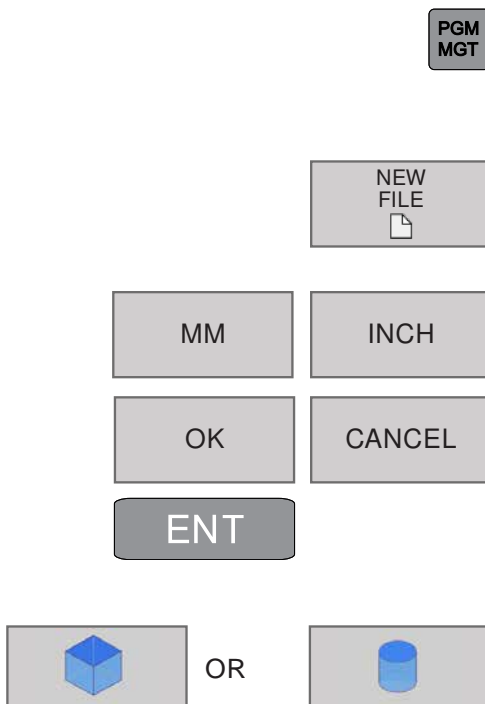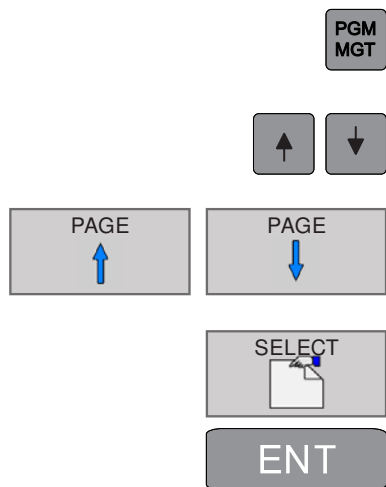
OR

• Block or
• Cylinder

Selecting a file

PGM MGT

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.

PAGE ↑   PAGE ↓

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

SELECT

ENT

4 Select file: Press softkey SELECT or ENT.

Delete file

PGM MGT
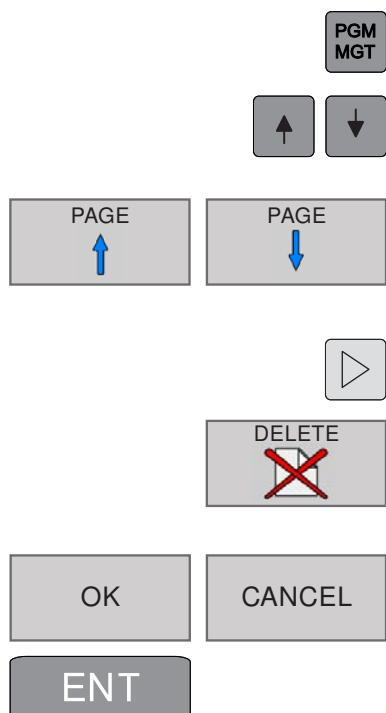
1 Call up file management.

↑ ↓

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

PAGE ↑   PAGE ↓

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

▷

4 Switch the softkey bar until the function for DE-LETE appears.

DELETE

5 Delete file: Press softkey DELETE.

OK   CANCEL

ENT

5 Confirm with OK or ENT or cancel with CANCEL.

Deleting a directory

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.

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

emco

Copying a file

PGM MGT

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.

PAGE ↑   PAGE ↓

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

COPY

4 Copy the file: Press softkey COPY.

5 Enter new file name.

OK   CANCEL

ENT

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.

✓ OK   CANCEL

ENT
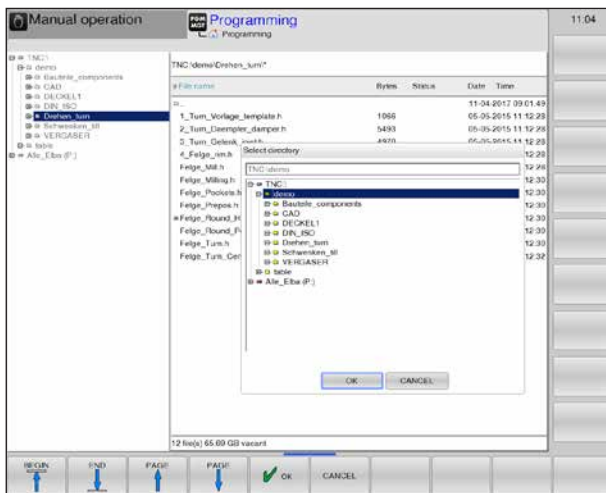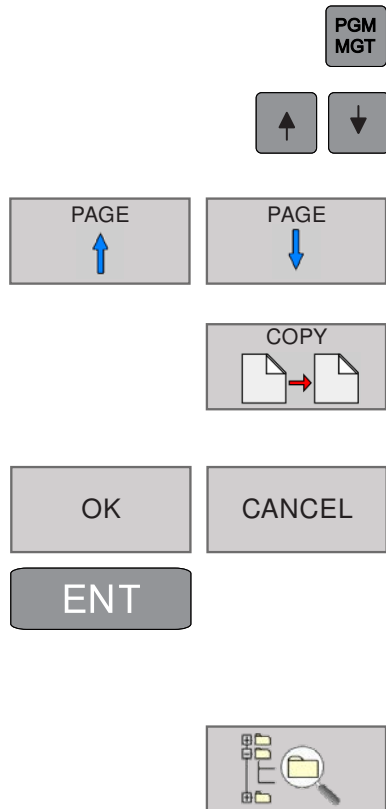
9 Complete the entry with OK or ENT.

> **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.

Tool table: replace fields

PGM
MGT

1 Call Programm Management

2 When overwriting a table with the same name, only the row contents (e.g. of existing tools) are replaced, the new ones added and the old ones retained.
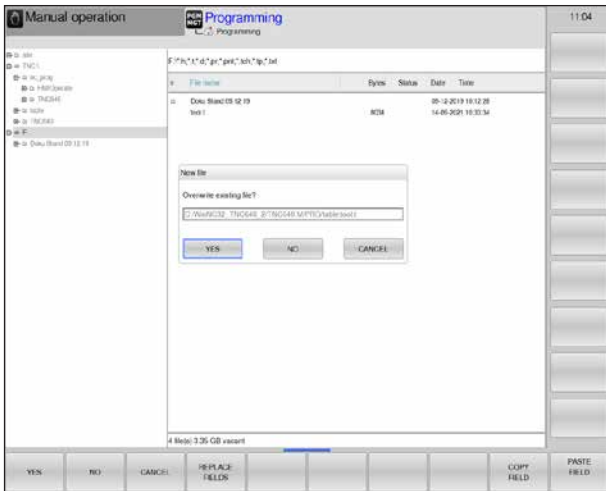
REPLACE
FIELDS

3 Complete the copying process with the "RE-PLACE FIELDS" softkey if row contents are to be replaced.

emco

Selecting one of the last 10 selected files

PGM MGT
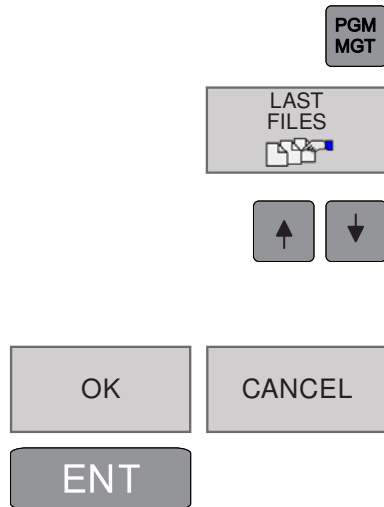
1 Call up file management

LAST FILES

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.

OK   CANCEL

ENT

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

Renaming files
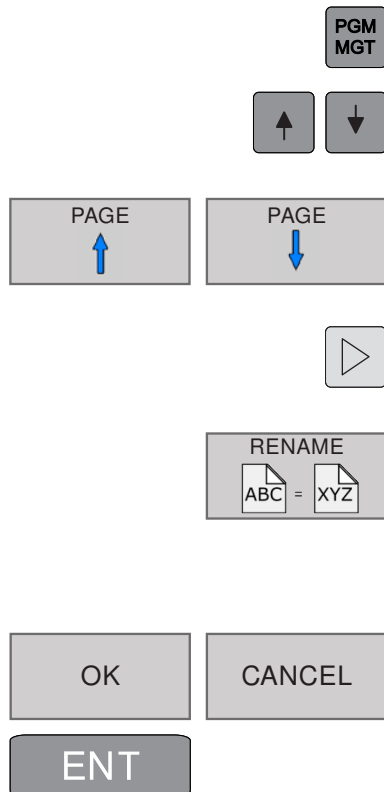
PGM MGT

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.

PAGE ↑   PAGE ↓

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

▷

4 Switch the softkey bar until the function for RE-NAME appears.

RENAME
ABC = XYZ

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.

OK   CANCEL

ENT

emco

Tagging files
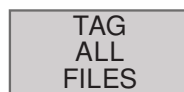
1 Call up file management

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
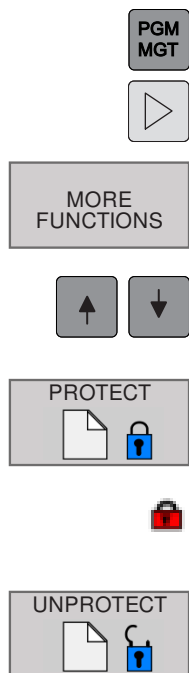
1 Call up file management

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
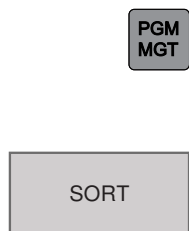
1 Call up file management

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

Opening and entering programs

Structure of an NC program in HEIDENHAIN plain text format

Block

10 L	X+15 Y+8 R0 F200 M3

Path function   Words
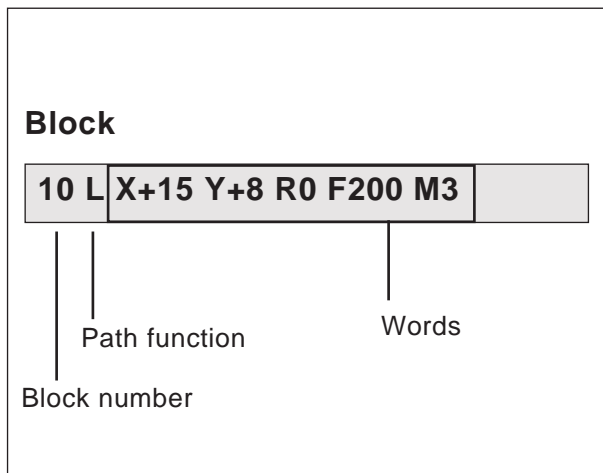
Block number

Elements of the program block

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.
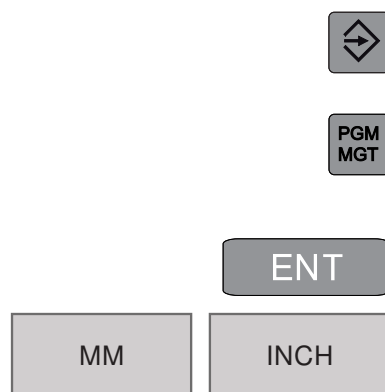
Opening a new machining program

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).
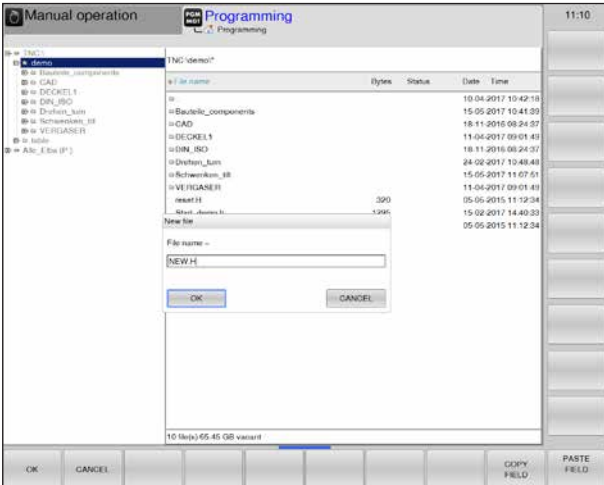
Entering a new program

6 select block form
- block or
- cylinder

Displaying BLK Form in the program

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.

Enter blank cylinder:
    Z: machining level in the graphic
    R: outer radius, L: length
    DIST: distance to zero point and
    RI: inner radius of the cylinder

For cylinder: X corresponds the the 4th axis

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	NO ENT
Prematurely skip dialogue question	END
Cancel and delete dialogue	DEL
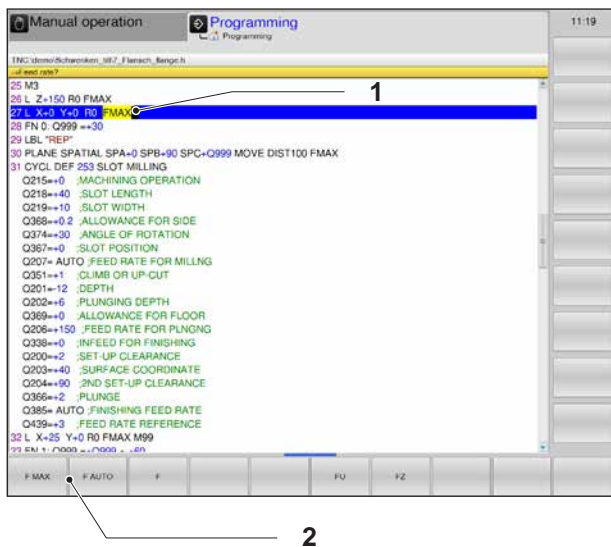
Additional softkeys for data entry

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

Functions for determining the feed

| F MAX | **1** Move in rapid traverse, effective per block. |

| F AUTO | **2** Automatically calculated feed. |

| F | **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. |

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

| FZ | **5** Define tooth feed: Unit mm/tooth. The number of teeth must be defined in the tool table in the CUT. column. |
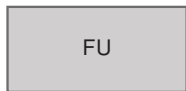
Editing program

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):
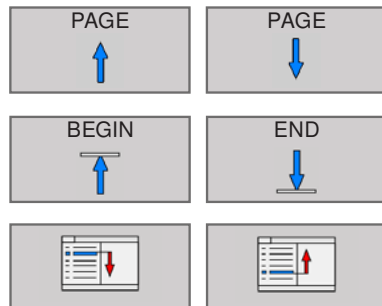
1 Scroll page up / down

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.

Save

1 In the Programming and Editing mode of operation, the following softkeys are available in the softkey toolbar: STORE and SAVE AS.

STORE

2 STORE: Can be used to save programs being processed. File name and path remain unchanged.

SAVE AS

3 SAVE AS: Opens a dialogue box with which the program name and the path of the program can be changed.
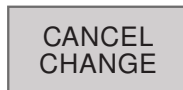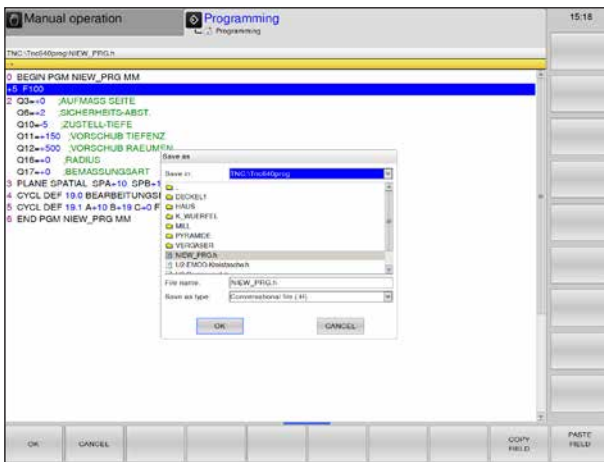
CANCEL CHANGE

4 CANCEL CHANGE: This softkey restores the last saved version.

emco

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.

SELECT BLOCK

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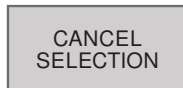
CANCEL SELECTION

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

COPY BLOCK

1 Press softkey COPY BLOCK.

CUT OUT BLOCK

2 Press softkey CUT OUT BLOCK to cut out the selected program section.
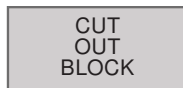
The WinNC stores the marked block.

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

INSERT BLOCK

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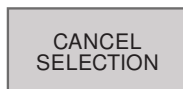
CANCEL SELECTION

4 Press softkey CANCEL SELECTION to end the marking function.

Search functions

FIND

FIND
CURRENT
WORD

X   -/+   5   2

FIND

FIND

END
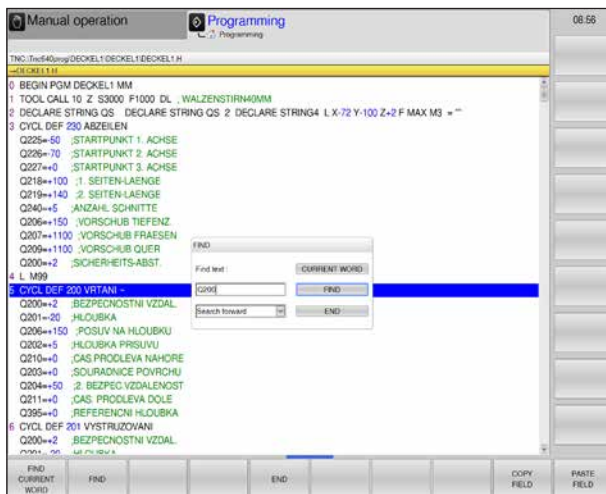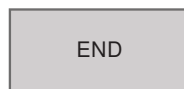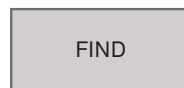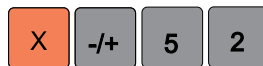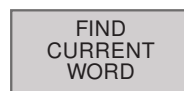
Search functions

The control searches for any texts within the program.

Search for any text

1 Select search function

The search window is displayed and the available search functions are displayed in the softkey bar.

2 Find current word:
The control takes over the first word of the current block. Press again to select the next word in the block.

3 Enter the text to search for. Pay attention to upper and lower case.

4 Start search: The control jumps to the next block in which the searched text is stored.

5 Repeat search: The control jumps to the next block in which the searched text is stored.

6 Exit the search function.

emco

MOD function

Configurations can be displayed and changed via MOD functions.

Selecting MOD functions

MOD

Press the MOD key and open the MOD screen.
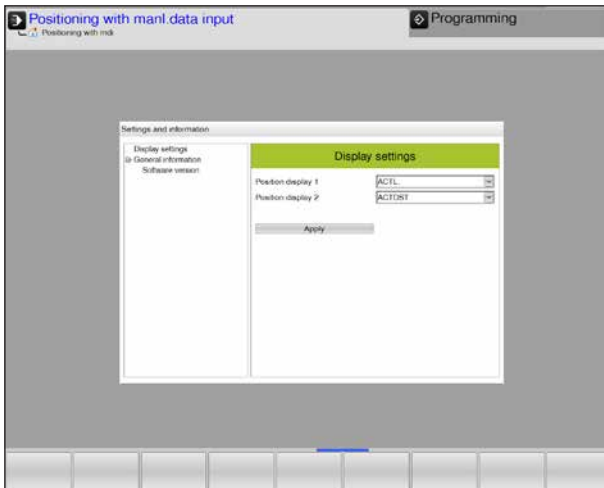
MOD functions

The following functions are available regardless of the selected operating mode
- Display settings:
  Position displays
- General information:
  Software version

Select the position display

In the Manual Operation mode and the Program Run, Full Sequence and Program Run, Single Block modes of operation, you can select the type of coordinates to be displayed:

The WinNC position displays can show the following coordinates:
- ACTL: Actual position; current tool position
- REF ACTL: Reference position; the actual position relative to the machine datum
- ACTDST: Distance remaining to the programmed position in the input system; difference between actual and target positions
- REFDST: Distance remaining to the programmed position with reference to the machine datum; difference between reference and target positions

With the MOD function Position display 1, you can select the position display in the status display.

With the MOD function Position display 2, you can select the position display in the additional status display.

Exiting MOD functions

MOD or CANCEL

Press the MOD key again or press the softkey CANCEL.

> **Note:**
> The available options and functions depend on the WinNC software version.

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
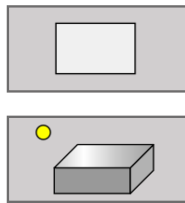
Testing the program

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

emco

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.
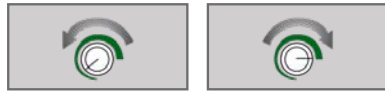
Incrementally lower the simulation speed

Test the program at maximum possible speed (predefined setting)

Simulation start / reset

START — Start simulation
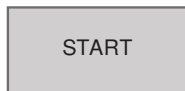
RESET + START — Reset and start simulation

START SINGLE — Start simulation single block

CE — Simulation reset

Selecting views

VIEWS — Volume view

VIEWS — Volume view and tool paths

VIEWS — Tool paths

emco

3D display

Functions for rotating, enlarging and reducing
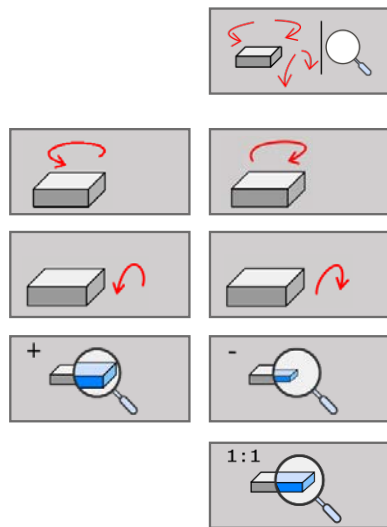
Rotate picture in 5° steps

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
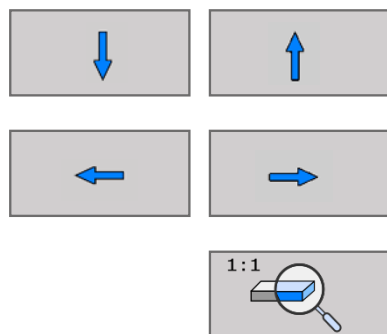
Shift the picture to the right and to the left

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
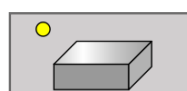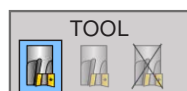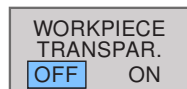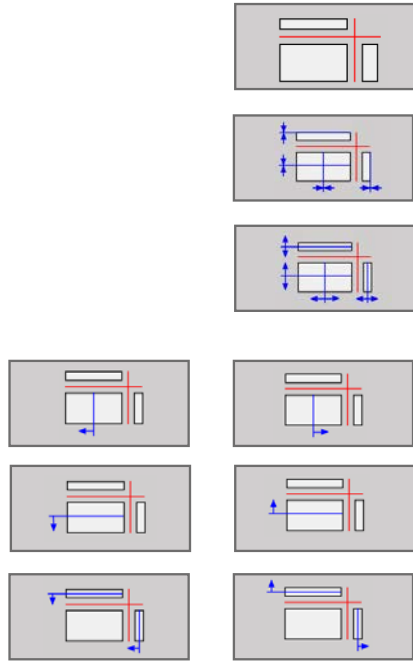
- Display workpiece in in 3D three-quarter view of the raw piece.

- This function is used to reset the sectional planes.

- This function is used to move the sectional planes. The following softkeys are displayed:

- Move the vertical sectional plane to the right or left.

- Move the vertical sectional plane forward or backward.

- Move the vertical sectional plane up or down.
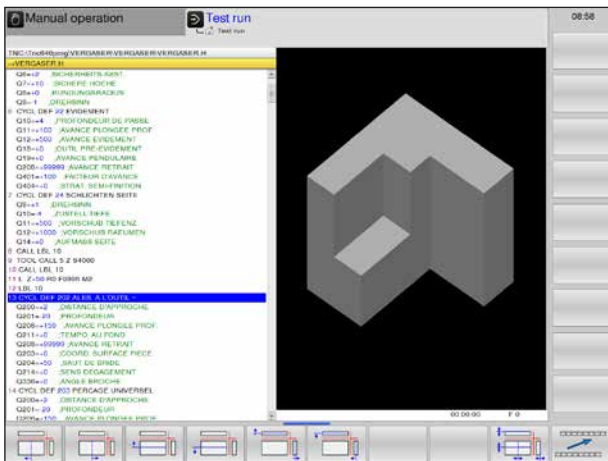
The default setting of the cutting plane is selected so that it is located in the machining plane at the center of the workpiece blank and in the tool axis at the bottom edge of the blank.

Note:
The clamping device is not shown in the raw piece display!

3D VIEW
CONFIG.

• Press softkeys to start 3D view configuration.
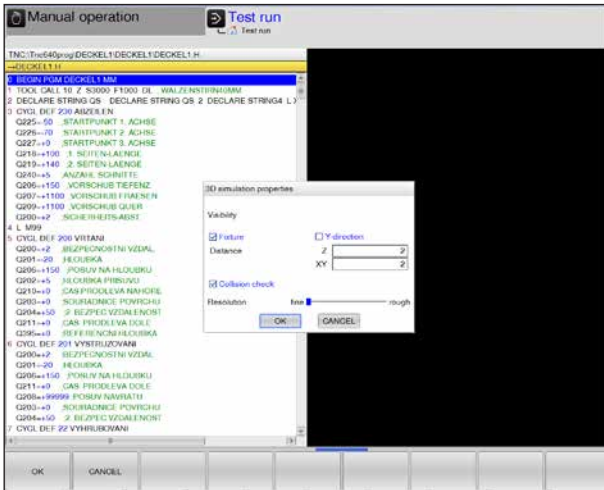
Input distance rawpiece to clamping device

• in Z-direction (1)
• in XY-direction (2)
  depending on the clamping situation

• Collision check on or off
• Set resolution: set from fine to rough.
• Display for processing time (3) and feed (4).

Note:
The choice of setting options available depends on whether a 3D view license is present or not.
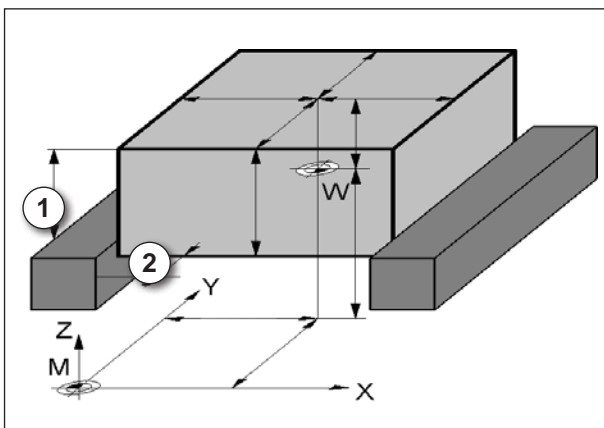
Distance rawpiece to clamping device
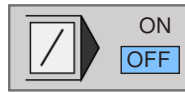
Skipping blocks

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

Overview

M-commands

M00   Programmed stop
M01   Optional stop (program stop only at OPT. STOP)

M02   End of program

M03   Spindle ON clockwise
M04   Spindle ON counterclockwise
M05   Spindle stop

M06   Perform tool change

M08   Coolant ON
M09   Coolant OFF

M13   M3 + M8
M14   M4 + M8
M15   M5 + M9

M19   always positions the spindle at 0 degrees
M20   positions the spindle to the value programmed in CYCLE 13.

M30   End of main program

M91   Move in machine coordinates

M89   Cycle call modal
M99   Block wise cycle call

M109   Feed rate for circular arcs:
       the control keeps the feed of circular arcs at the tool cutting edge constant during internal and external machining t
M110   the control keeps the feed of circular arcs on the tool cutting edge constant only during internal machining
M111   Reset M109 and M110

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.

CALC

Scientific

+	-	*	/	7	8	9
(	)	CE	=	4	5	6
ARC	SIN	COS	TAN	1	2	3
x^y	SQRT	1/x	PI	0	.	±
M+	MS	MR	MC			
LN	LOG	e^x	SGN			
ABS	INT	FRAC	MOD			
DEG	RAD	MM	INCHES			

GET CURRENT VALUE

- Transfer current value from the program to the calculator.

CONFIRM VALUE

- 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^Y	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^x	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

ERR
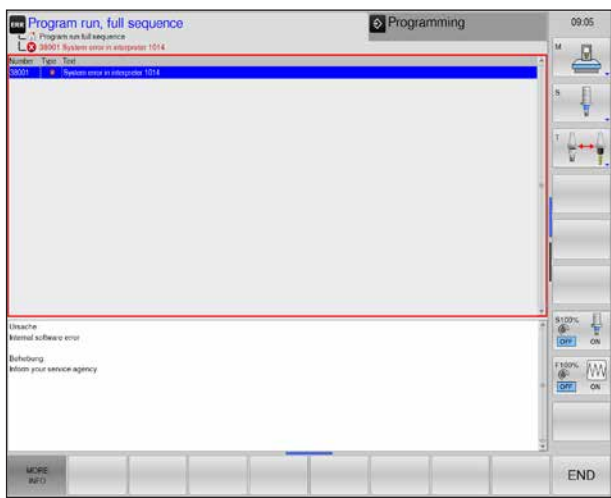
- retrieving error window with ERR.

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.

MORE INFO

- The "MORE INFO" softkey can be used to display additional information on the alarms.

CE

- Delete the error message with the CE button.

Note:
Detailed information on the alarms and messages can be found in chapters "H" and "I" of these operating instructions.

emco

Programming graphics

Generate / do not generate graphics during programing

While you are writing the part program, you can have the WinNC generate a 2-D pencil-trace graphic of the programmed contour.

Switch the screen layout to displaying program blocks to the left and graphics to the right.
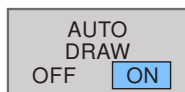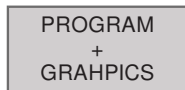
Press softkey "PROGRAM + GRAPHICS".

PROGRAM
+
GRAHPICS

Set softkey "AUTO DRAW" to ON. While you are entering the program lines, the WinNC generates each programmed path contour in the graphics window in the right screen half.
Use the arrow keys to select the block up to which you want the graphic to be generated
After editing or adding an element, the graphic is displayed.
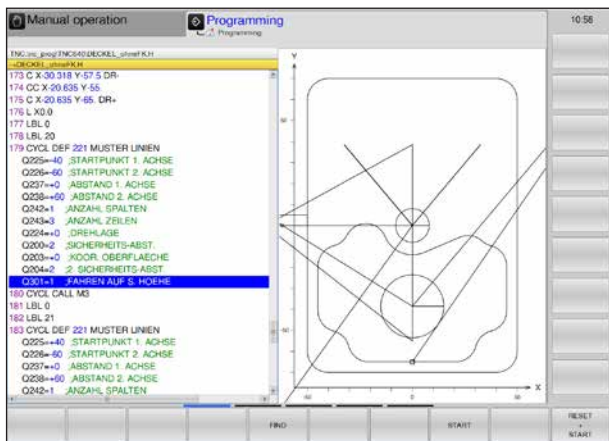
AUTO
DRAW
OFF   ON

If you do not wish to have graphics generated during programming, set the "AUTO DRAW" soft key to OFF.

Note:
If AUTO DRAW is set to ON, during generation of the 2-D line graphic the control does not consider:
• Program section repeats
• Jump commands
• M-functions such as M2 or M30
• Cycle calls

Only use automatic drawing during contour programming.
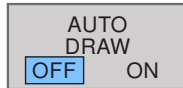
AUTO DRAW
OFF   ON

Set softkey "AUTO DRAW" to OFF.

The programme is completely simulated including cycles and jumps. This is only possible if no programme is running in the test run. A reset of the simulation can be triggered via the CE key.
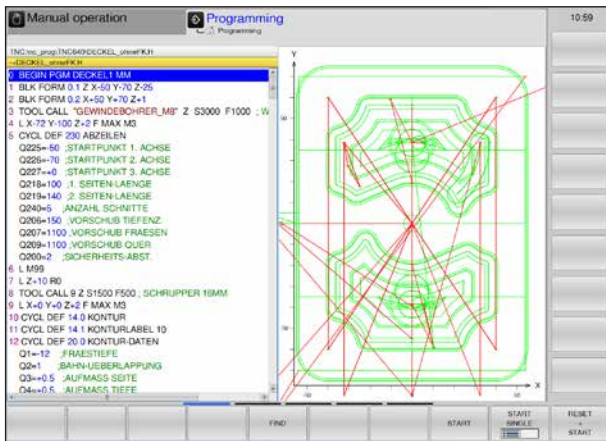
Generating a graphic for an existing program

RESET + START

Generate complete programming graphic.

START

Generate complete programming graphic or complete after RESET + START.

STOP

Stop the programming graphics.
This softkey only appears while the WinNC is generating the programming graphics.
This softkey is only available if "AUTO DRAW" is set to OFF.

START SINGLE

"START SINGLE" is only available if nur "AUTO DRAW" is set to OFF.

emco
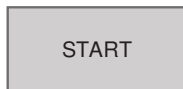
Switch softkey bar.
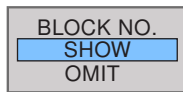
Show block numbers
Set softkey BLOCK NO. OMIT to SHOW.
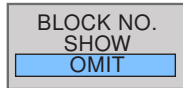
Hide block numbers
Set softkey BLOCK NO. SHOW to OMIT.

Show/hide block numbers only works with automatic drawing ON!

Delete graphic

Press softkey "CLEAR GRAPHICS" to delete graphics.

Show and hide grid lines

Set softkey "GRID LINES" to ON to show the grid lines.

Adjust section

You can set the view for a graphic yourself.

Switch softkey bar
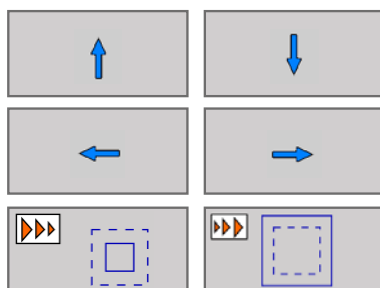
Press softkey to move the section.

Press softkey to reduce or enlarge the section.

Use the softkey "RESET BLK FORM" to restore the original section.

Mouse operation:
The section can be zoomed with the mouse wheel and moved by holding down the middle or right mouse button.

Structuring programs

Displaying the program structure window / Changing the active window
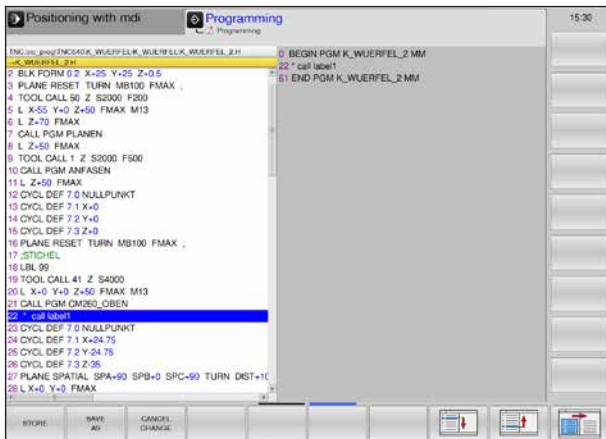
This WinNC function enables you to comment part programs in structuring blocks. Structuring blocks are short texts with up to 252 characters and are used as comments or headlines for the subsequent program lines.

With the aid of appropriate structuring blocks, you can organize long and complex programs in a clear and comprehensible manner.
This function is particularly convenient if you want to change the program later. Structuring blocks can be inserted into the part program at any point.

Select the block after which the structuring block is to be inserted:
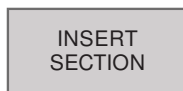
Press SPEC FCT kley.

Press the softkey INSERT SECTION or the * key on the ASCII-keyboard.

Insert section text.

They can also be displayed in a separate window. Use the appropriate screen layout for this.
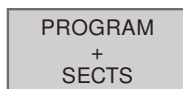
Call up the soft-key row for screen splitting to go to screen splitting.
Left: Programme, right: Programme outline

Press softkey PROGRAM + SECTS to open the structuring window.

Switch the active window:
Press the CHANGE WINDOW soft key.

Select blocks in the structure window.
If you are scrolling through the program structure window block by block, the WinNC at the same time automatically moves the corresponding NC blocks in the program window.
This way you can quickly skip large program sections.

emco

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 *)

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

Example:
L X+80 Y+0 Z-10

Movements parallel to the machine axes

Movements in the main planes

Three-dimensional movement
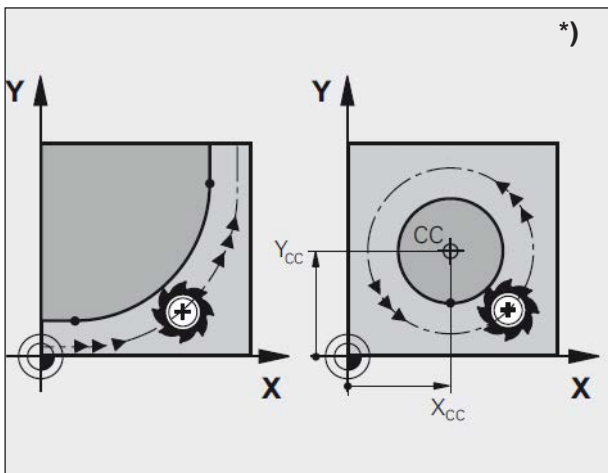
Circular movements

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
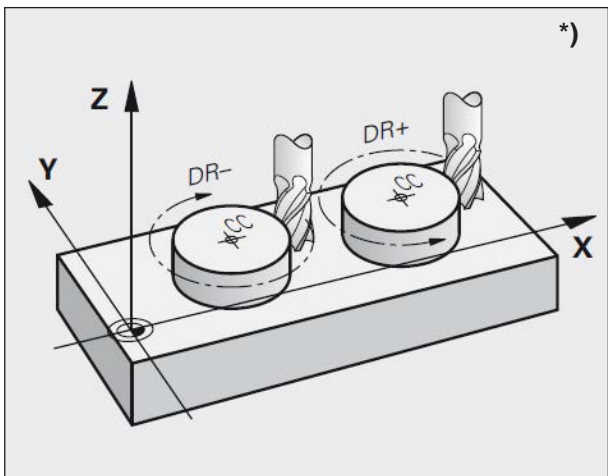
Direction of rotation of the circular movements

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

X 10

Y 5   ENT

| R0 | RL | RR |

100   ENT

FMAX

3   END
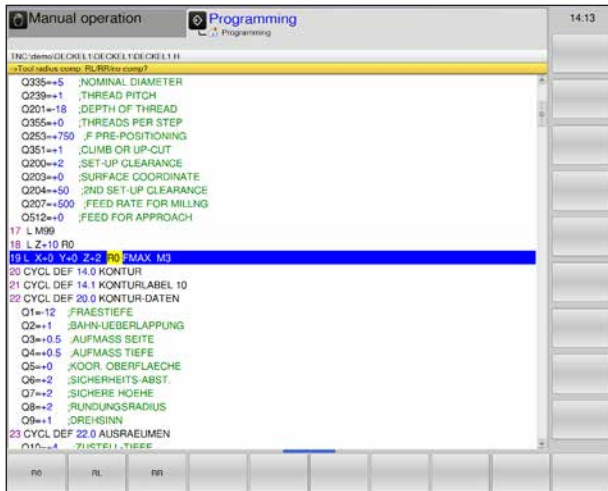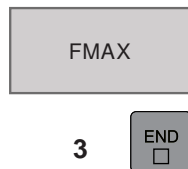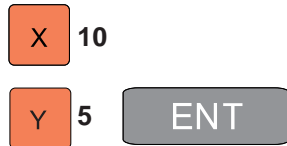
Radius compensation

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:

APPR LT	DEP LT
APPR LN	DEP LN
APPR CT	DEP CT
APPR LCT	DEP LCT
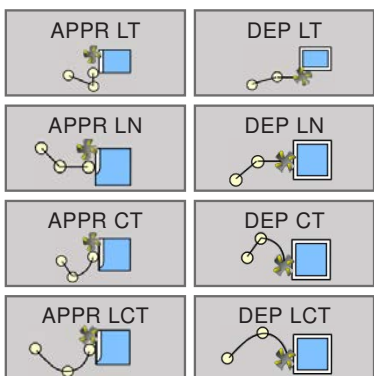
- Straight line with tangential contact **approach / leave**

- 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.

Abbreviations	Meaning
APPR	**APPR** = approach
DEP	**DEP** = departure
L	**L** = straight line
C	**C** = circle
T	**T**angential: continuous, smooth transition
N	**N**ormal (perpendicular)
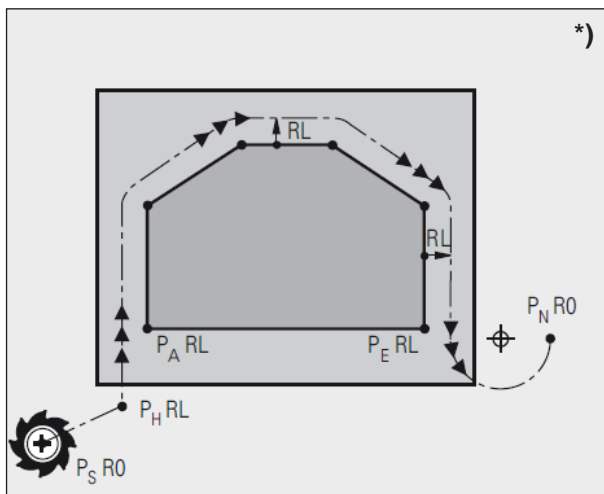
Important positions for approach and departure *)

- 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!

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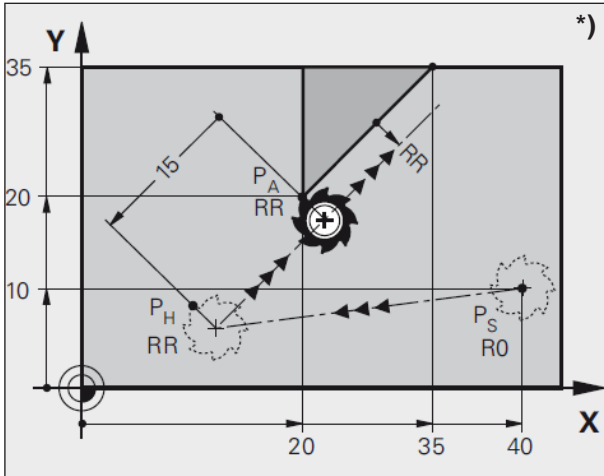
Approaching on a straight line with tangential connection: APPR LT   *)

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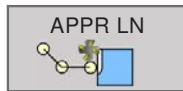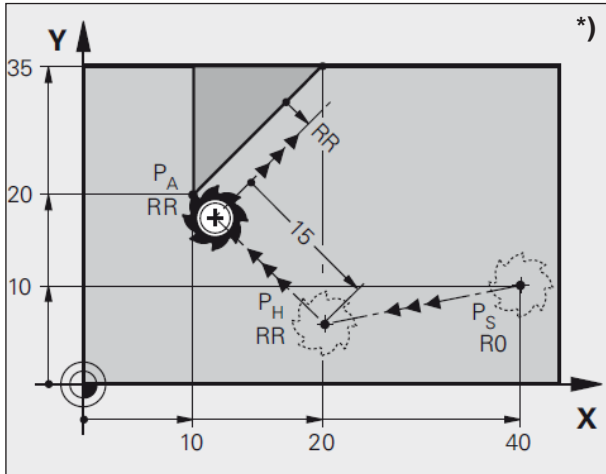
Approaching on a straight line perpendicular to the first contour point: APPR LN *)

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 + 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 alsways 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

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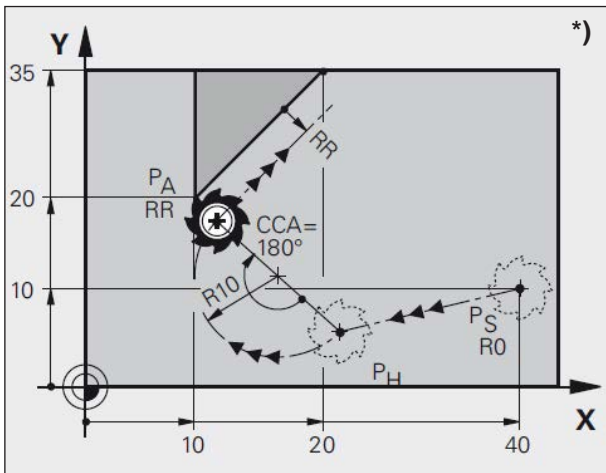
Approaching on a circular path with tangential connection: APPR CT *)

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.
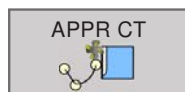
- Any path function: Approach starting point P_S.
- Open dialogue using the key APPR/DEP and softkey APPR CT:
- Coordinates of the first contour point P_A.
- Radius R of the circular path
- Approaching the tool from the side, which is defined by the radius compensation:
  Enter R positive.
- Approaching from the tool side:
  Enter R negative.
- Centre angle CCA of the circular path
- Enter CCA only positive
- Maximum input value 360°
- 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
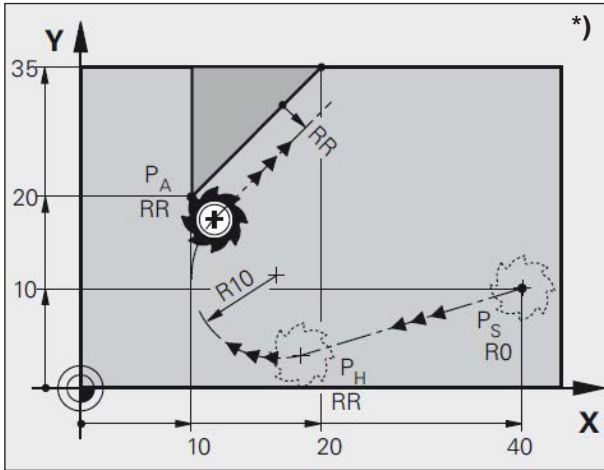
Approaching on a circular path with tangential connection from a straight line to the contour: APPR LCT *)

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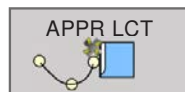
- Any path function: Approach starting point P_S
- Open dialogue using the key APPR/DEP and softkey APPR LCT:
- Coordinates of the first contour point P_A
- Radius R of the circular path. Specify R positive
- Radius compensation RR/RL for machining

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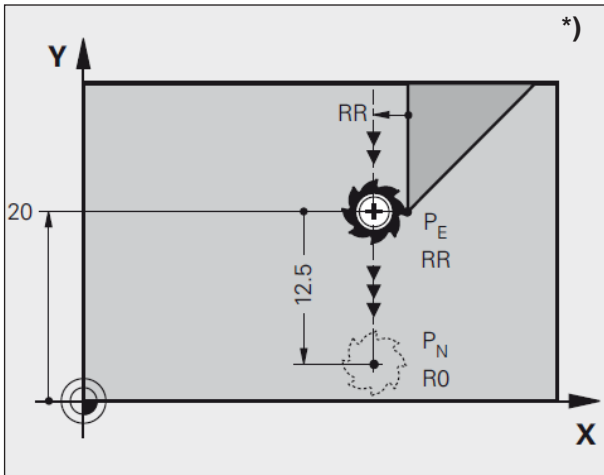
Departing in a straight line with tangential connection: DEP LT *)

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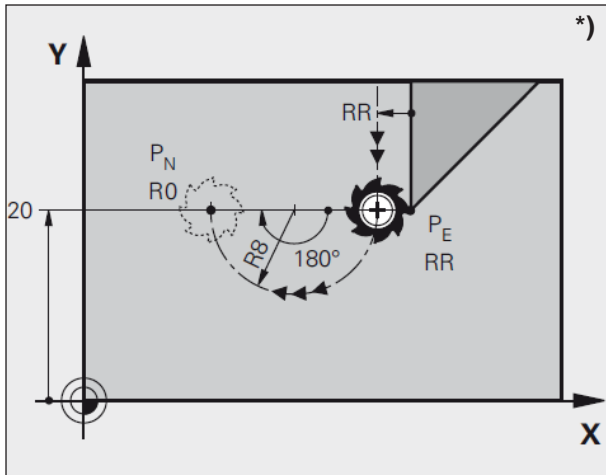
Departing on a circular path with tangential connection: DEP CT *)

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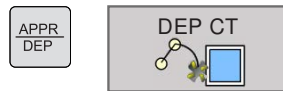
- Programming the last contour element with the end point P_E and radius compensation
- Open dialogue using the key APPR/DEP and softkey DEP CT:
- Centre angle CCA of the circular path
- Radius R of the circular path
- The tool is to depart from the work piece to the side, which is determined by the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

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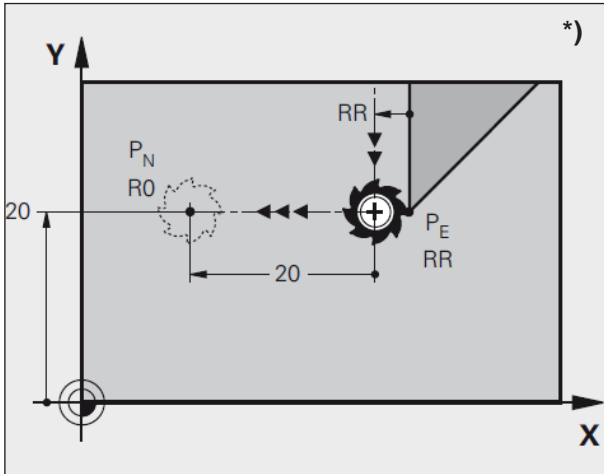
Departing in a straight line perpendicular to the last contour point: DEP LN *)

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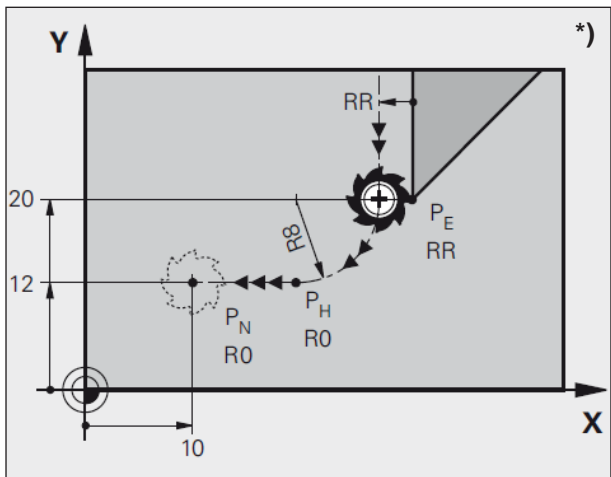
Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT *)

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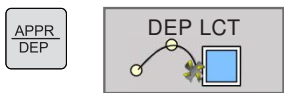
- Programming the last contour element with the end point P_E and radius compensation
- Open dialogue using the key APPR/DEP and softkey DEP LCT:
- Enter the coordinates of the end point P_N
- Radius R of the circular path. Enter R positive

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

emco

Path movements - rectangular coordinates

Overview of path functions

Function	Path functions key	Tool movement	Required inputs
Straight line **L** Line		Straight line	Coordinates of the of the straight line end point
Chamfer **CHF** **CH**am**F**er		Chamfer between two straight lines	Chamfer length
Circle centre **CC** **C**ircle **C**entre		No	coordinates of the circle centre and/or pole
Arc **C** **C**ircle		Circular path around the circle centre CC to the arc end point	Circle path end point coordinates, direction of rotation
Arc **CR** **C**ircle by **R**adius		Circular path with a specific radius	Circle path end point coordinates, circle radius, direction of rotation
Circle Tangential **CT** **C**ircle **T**angential		Circular arc with tangential connection to the preceding and subsequent contour element	Coordinates of the arc endpoint
Corner rounding **RND** **R**ou**ND**ing of Corner		Circular path with tangential contact with the preceding and the following element	Corner radius R
FK Free contourr-programming	FK	Straight line or circular path with any connection to the preceding contour element	See FK Free contourr-programming
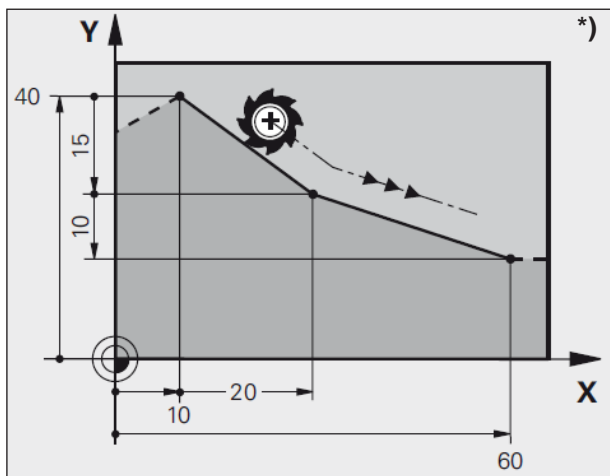
Straight line L

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, <u>after</u> 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.

*)
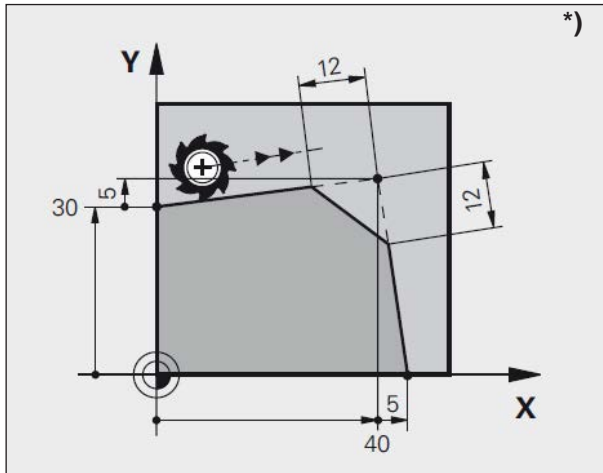
Insert a chamfer CHF between two straight lines

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.
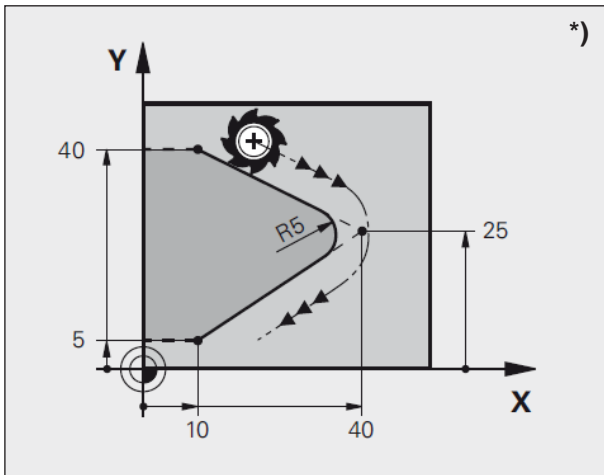
Corner roundings RND

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.
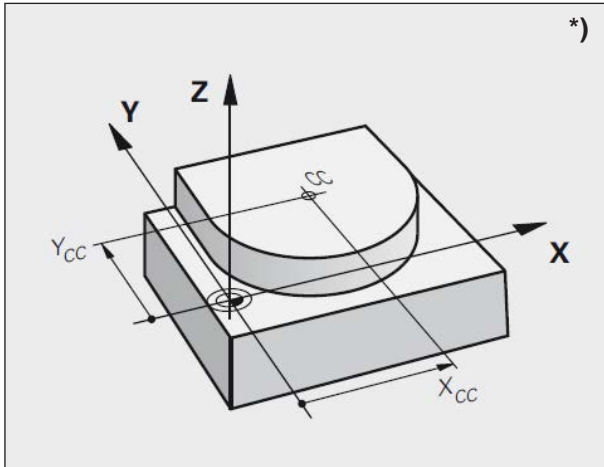
Circle centre CC *)

*)

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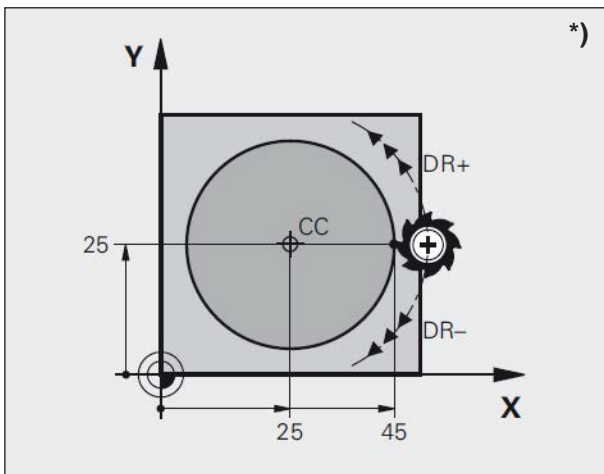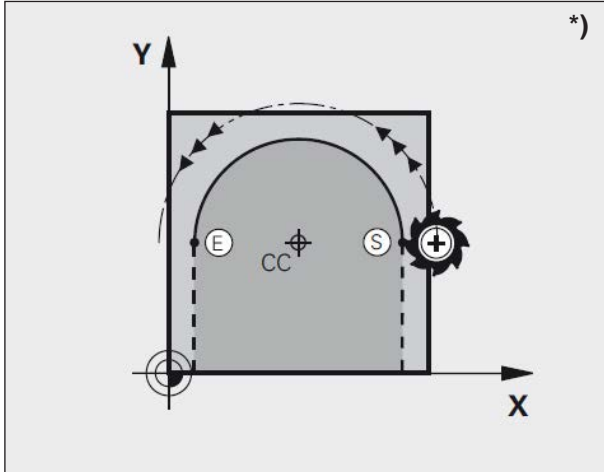
Example
5 CC X+25 Y+25
6 L X+45 Y+25 RR F200 M3
7 C X+45 Y+25 DR+

> **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**
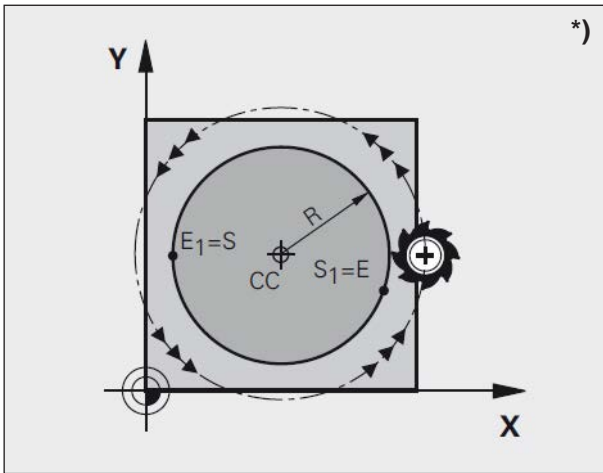
Full circle

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°
Radius has a positive sign R>0

Larger arc: CCA>180°
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)
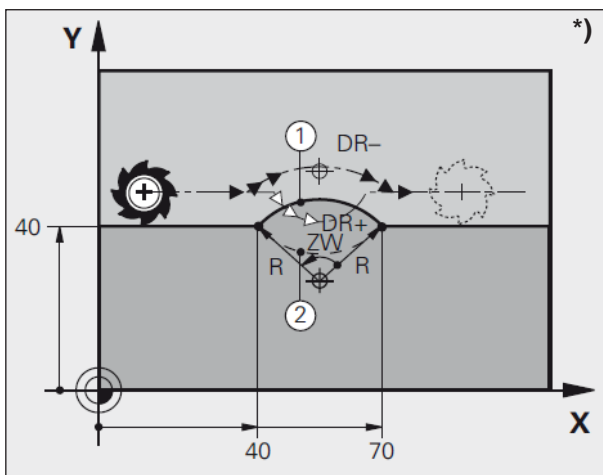
Example
11 CR X+70 Y+40 R-20 DR- (arc 3)
or
11 CR X+70 Y+40 R-20 DR+ (arc 4)
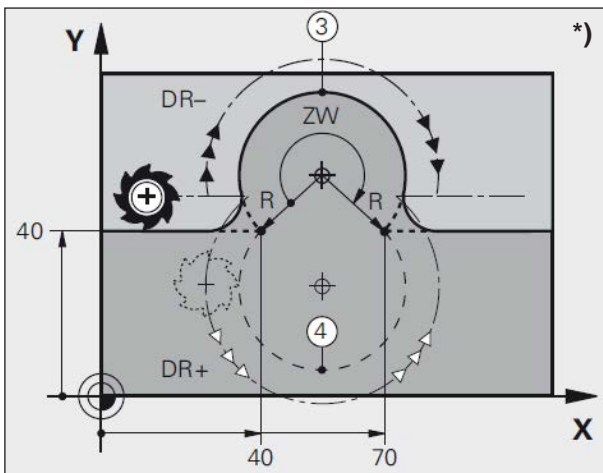
> **Note:**
> The distance between the starting point and the end point of the circle diameter must not exceed the circle diameter.
> The maximum radius is 99.9999 m.

Circular path CT with tangential connection *)

CT

*)
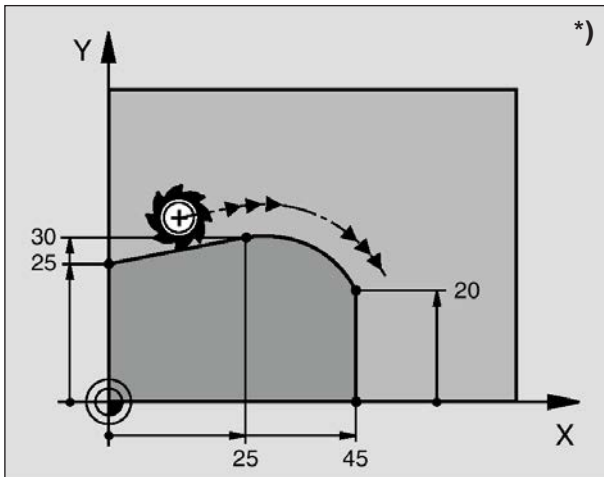
The tool moves on a circular arc that tangentially connects to the previously programmed contour element.

A transition is called "tangential" when there is no kink or corner at the intersection of the contour elements - the transition is smooth.

The contour element to which the arc tangentially connects must be programmed directly before the CT block. This requires at least two positioning blocks.

- **Coordinates** of the end point of the circular arc

Further entries, if necessary:

- **Feed rate F**
- **Miscellaneous function M**

Example
7 L X+0 Y+25 RL F300 M3
8 L X+25 Y+30
9 CT X+45 Y+20
10 L Y+0

Note:
The CT block and the previously programmed contour element should include both coordinates of the plane in which the circular arc is executed!

emco

Path contours - Polar co-ordinates *)

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**	L + P	Straight line	Polar radius, polar angle of the straight line end point
Arc **CP**	C + P	Circular path around the circle centre pole CC to the arc end point	Polar angle of the circle end point, direction of rotation
Arc **CTP**	CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helix	C + P	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
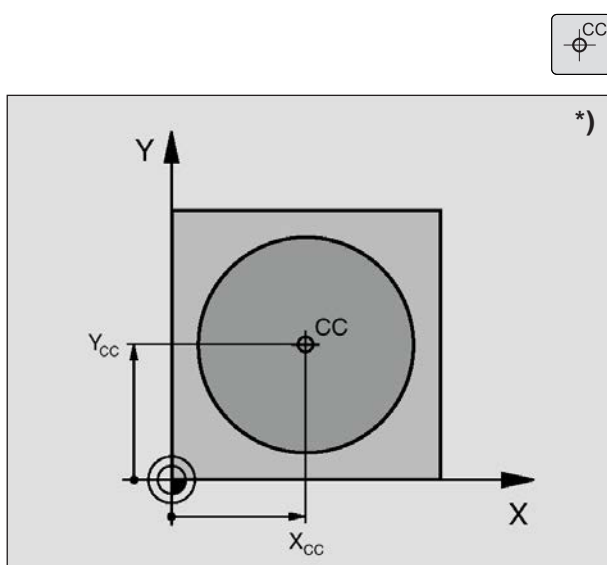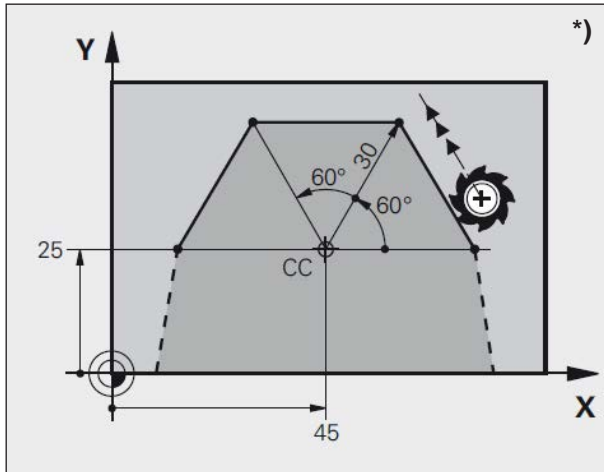
Straight line LP

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° and +360°

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
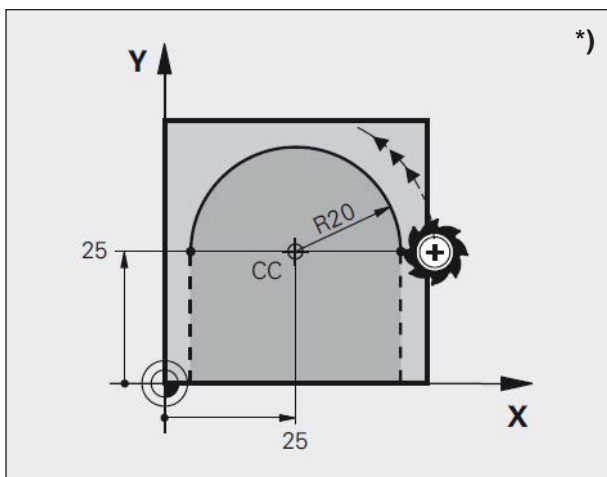
Circular path CP around pole CC

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° and +5400°
- **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.

emco

Circular path CTP with tangential connection

CT + P
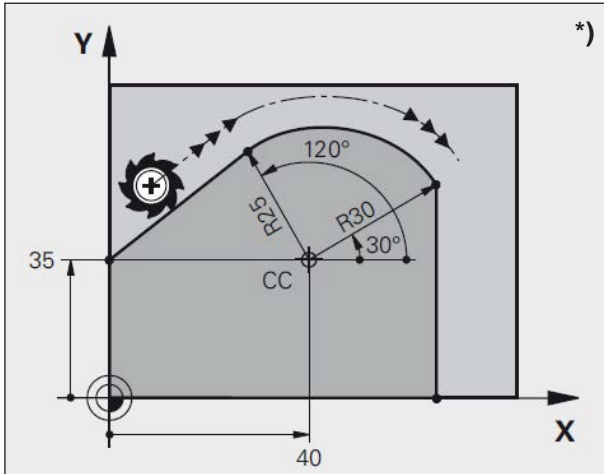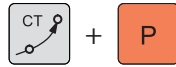
The tool moves on a circular path that tangentially connects to a preceding contour element.

- **Polar coordinate radius PR:** Distance between the circular-path end point and the pole CC
- **Polar coordinate angle PA:** Angular position of the circular-path end point

Example
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0

Note:
The pole CC is **not** the center of the contour arc!
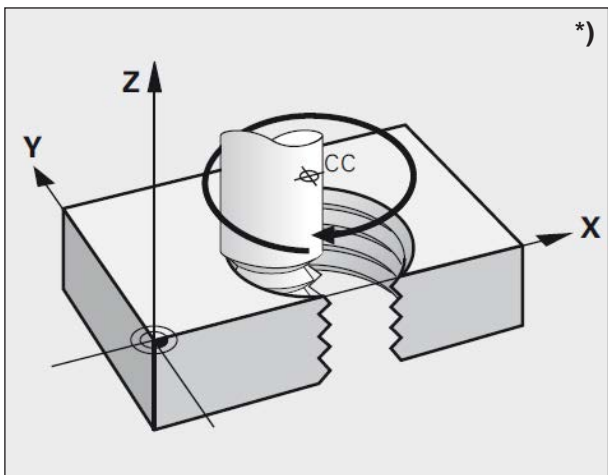
Helix

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° and +5400°. If the thread has more than 15 convolutions, you should program the helix in a repetitions of a program part.

Path contours – FK Free contour programming

Fundamentals
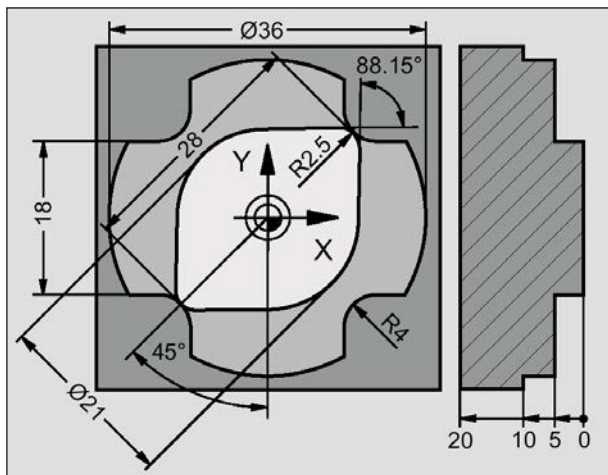
Workpiece drawings that are not dimensioned for NC often contain coordinate data that cannot be entered with standard path-functions. You may, for example, have only the following data on a specific contour element:

- Known coordinates lie on the contour element or in its proximity
- Coordinate data are referenced to another contour element or
- Directional data and data regarding the course of the contour

You program such data directly by using the FK free contour programming function. The WinNC derives the contour from the known coordinate data and supports the programming dialogue with the interactive FK graphics. The illustration on the left shows a workpiece drawing for which FK programming is the most convenient programming method.

If you want to run FK programs on old TNC controls, use the conversion function.

Pay attention to the following prerequisites for FK programming

- With the FK free contour programming you can only program contour elements that lie in the working plane.
- Enter all available data for every contour element. Even data that do not change must be entered in every block. Data that are not programmed will not be recognized!
- Q parameters are permissible in all FK elements.
- If you enter both conventional and free contour blocks in a program, every section of the FK contour must be fully defined.

- The WinNC needs a fixed point from which it can calculate the contour elements. Use the gray dialogue keys to program a position that contains both coordinates of the working plane, directly before programming the FK contour. Do not program any Q parameters in this block.
- If the first block of an FK contour is a FCT or FLT block, you must program at least two NC blocks with the gray dialogue keys to fully define the direction of contour approach.
- An FK contour must not start directly after an LBL label.

> **Note:**
> The solution selections with free contour programming (FSELECT) is different from the original control. Free contour programming is not fully implemented, the focus is placed on training relevance.

Graphics during FK programming
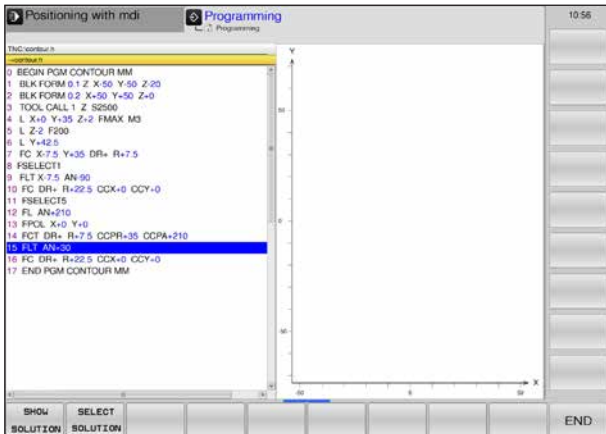
Call up the softkey bar for the screen layout.
.

PROGRAM + GRAPHICS

Press the PROGRAM + GRAPHICS softkey.

Incomplete coordinate data are often not sufficient to fully define a workpiece contour. In this case, the WinNC indicates the possible solutions in the FK graphic. You can then select the contour that matches the drawing. The FK graphic displays the elements of the workpiece contour in different colours:

black   The contour element is fully defined.

green   The entered data permit several solutions; you select the correct one.

red   The entered data are not sufficient to determine the contour element; enter further data

blue   The corresponding line of the contour element in the editor is selected.

If the entered data permit several solutions and the contour element is displayed in green, select the correct contour element as follows:

SHOW SOLUTION

SELECT SOLUTION

- Press the SHOW SOLUTION soft key repeatedly until the correct contour element is displayed.
- If the displayed contour element matches the drawing, select it with the SELECT SOLUTION soft key.

Select the green contour elements with the FSELECT soft key as soon as possible so as to reduce the ambiguity of subsequent contour elements.

If you do not yet wish to select a green contour element, press the EDIT soft key to continue the FK dialogue.

Initiating the FK dialog

FK

If you press the FK button, the WinNC displays the soft keys you can use to initiate an FK dialog, see the following table. Press the FK button a second time to deselect the softkeys.

If you initiate the FK dialog with one of these soft keys, the WinNC shows additional softkey rows that you can use for entering known coordinates, directional data and data regarding the course of the contour.
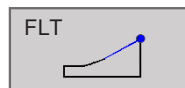
FLT

• Straight line with tangential connection
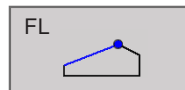
FL

• Straight line without tangential connection
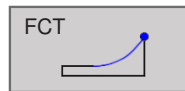
FCT

• Circular arc with tangential connection
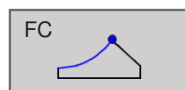
FC

• Circular arc without tangential connection
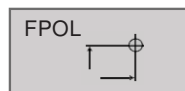
FPOL

• Pole for FK programming

Note:
The pole for FK programming remains active until you define a new one using FPOL.

emco

Free straight line programming

Straight line with tangential connection
If the straight line connects tangentially to another contour element, initiate the dialog with the FLT soft key.
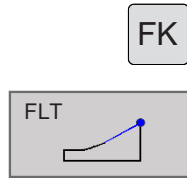
FK

FLT

- To display the soft keys for free contour programming, press the FK key.
- To initiate the dialog, press the FLT soft key.
- Enter all known data in the block by using the soft keys.

Straight line without tangential connection

FK

FL

- To display the soft keys for free contour programming, press the FK key.
- To initiate the dialog for free programming of straight lines, press the FL soft key. The WinNC displays additional soft keys.
- Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green.
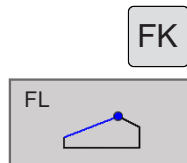
Note:
If AN is used, LEN also must be used, PR belongs to PA (polar coordinates).
In case of a mixture, the error message "FK programming: Inconsistent input in the contour" appears!

Free circular path programming

Circular arc with tangential connection
If the circular arc connects tangentially to another contour element, initiate the dialog with the FCT soft key:

FK
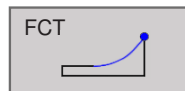
FCT

- To display the soft keys for free contour programming, press the FK key.
- To initiate the dialog, press the FCT soft key.
- Enter all known data in the block by using the soft keys.
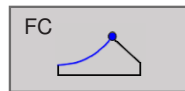
Circular arc without tangential connection

FK

FC

- To display the soft keys for free contour programming, press the FK key.
- To initiate the dialog for free programming of circular arcs, press the FC soft key. The WinNC displays soft keys with which you can directly enter data on the circular arc or the circle center.
- Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green.

emco

Input options

End point coordinates
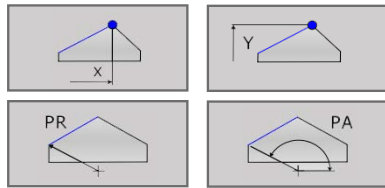
- Cartesian coordinates X and Y

- Polar coordinates referenced to FPOL
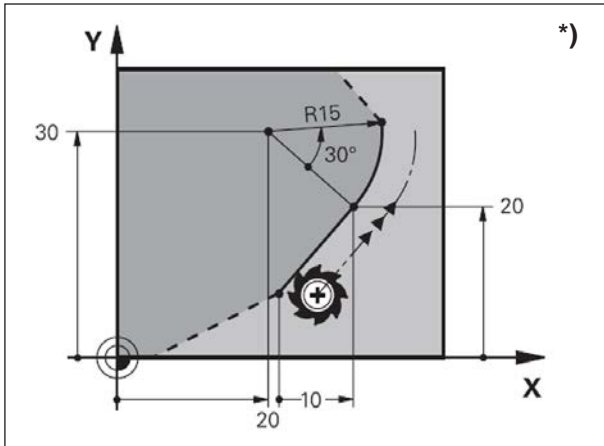
NC example blocks
7 FPOL X+20 Y+30
8 FL IX+10 Y+20 RR F100
9 FCT PR+15 IPA+30 DR+ R15

*)

Direction and length of contour elements
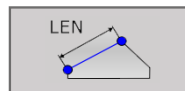
- Length of a straight line (only with FT and FLT)
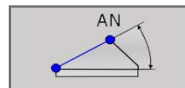
- Gradient angle of a straight line
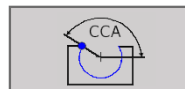
- Center angle of an arc
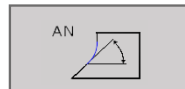
- Gradient angle AN of an entry tangent

Circle center CC, radius and direction of rotation in the FC/FCT block

The TNC calculates a circle center for free-programmed arcs from the data you enter. This makes it possible to program full circles in an FK program block.

If you wish to define the circle center in polar coordinates you must use FPOL, not CC, to define the pole. FPOL is entered in Cartesian coordinates and remains in effect until the control encounters a block in which another FPOL is defined.
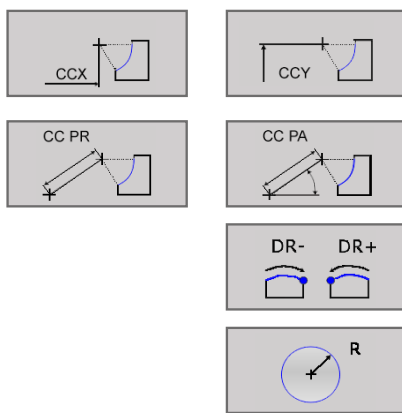
- Circle center in Cartesian coordinates

- Center point in polar coordinates

- Rotational direction of the arc

- Radius of an arc
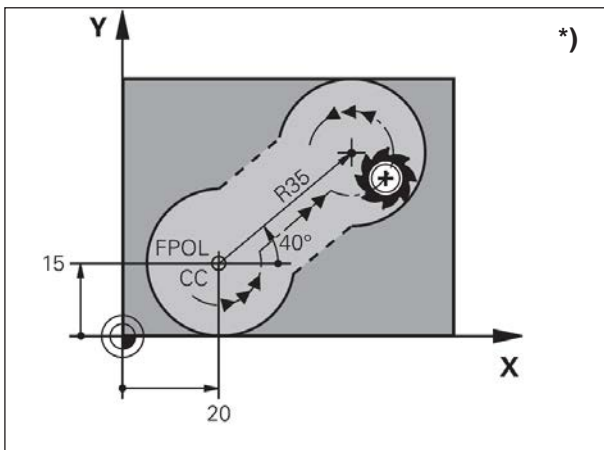
Note:

A circle center that was calculated or programmed conventionally is then no longer valid as a pole or circle center for the new FK contour: If you enter conventional polar coordinates that refer to a pole from a CC block you have defined previously, then you must enter the pole again in a CC block after the FK contour.

NC example blocks
10 FC CCX+20 CCY+15 DR+ R15
11 FPOL X+20 Y+15
12 FL AN+40
13 FC DR+ R15 CCPR+35 CCPA+40
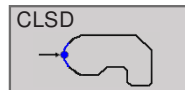
Closed contours

- You can identify the beginning and end of a closed contour with the CLSD soft key. This reduces the number of possible solutions for the last contour element.

  Enter CLSD as an addition to another contour data entry in the first and last blocks of an FK section.

- Beginning of contour: CLSD+
- End of contour: CLSD-
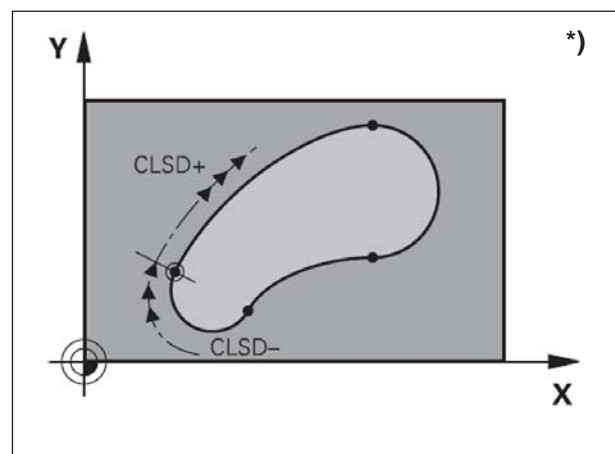
NC example blocks
12 L X+5 Y+35 RL F500 M3
13 FC DR- R15 CLSD+ CCX+20 CCY+35
...
17 FCT DR- R+15 CLSD-

Special functions

Defining a raw part: BLK FORM

After opening a new program, an unmachined work piece is defined.

For subsequent definition of the raw part proceed as follows:
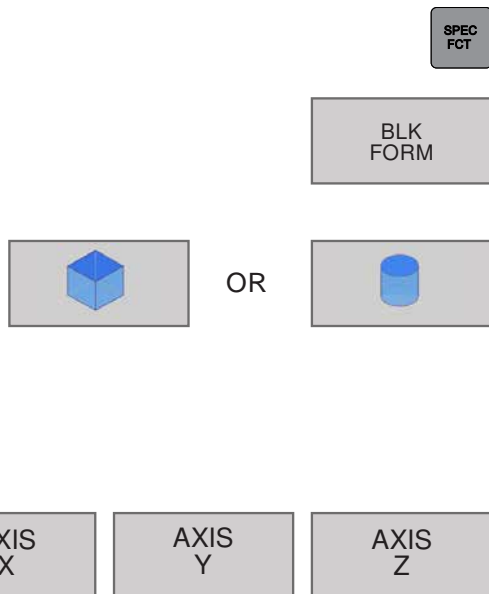
1 Call up Special Functions

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

3 For defining the block form 2 selections are available:

- block or
- cylinder

Input raw piece block:
definded only in machining plane XY (for Z-axis)

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

1 MIN-Point: smallest X-,Y- and Z-coordinate of the cuboid; enter absolute values.
2 MAX-Point: biggest X-,Y- and Z-coordinate of the cuboid.
Enter absolute or incremental values.

Enter blank cylinder:
Z: machining level in the graphic
R: outer radius, L: length

DIST: distance to zero point and
RI: inner radius of the cylinder

For cylinder: X corresponds the the 4th axis

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

Assigning string parameters

SPEC
FCT

1 Call up Special Functions

DECLARE
STRING

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"
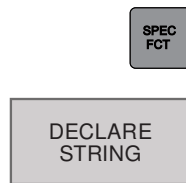
Insert comment

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

SPEC
FCT

1 Call up Special Functions

INSERT
COMMENT

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.
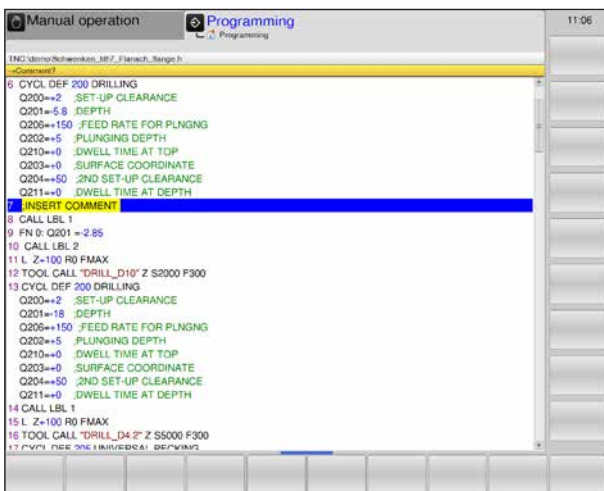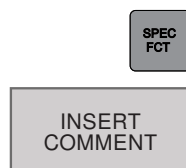
Insert comment

Insert section

SPEC FCT
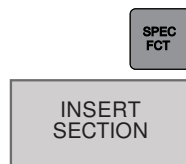
INSERT SECTION

1 Call up Special Functions

2 Press the INSERT SECTION soft key or the *
   key on the ASCII keyboard.

   Enter the structuring text.

   See also page D7 "structuring programs".
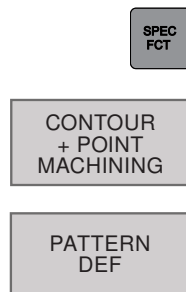
Contour and point machining

SPEC FCT

CONTOUR + POINT MACHINING

PATTERN DEF

1 Display Special Functions

2 Press the CONTOUR + POINT MACHINING
   softkey.

The PATTERN DEF function defines machining
patterns that can later be displayed using the
CYCL CALL PAT function.

A machining pattern remains active until a new
one is defined or a point table has been selected
with the SEL PATTERN function.
Using the block scan, any point can be selected at
which the machining can be started or continued.

The following patterns are available

- Point
- Row
- Pattern
- Frame
- Circle
- Pitch circle

emco
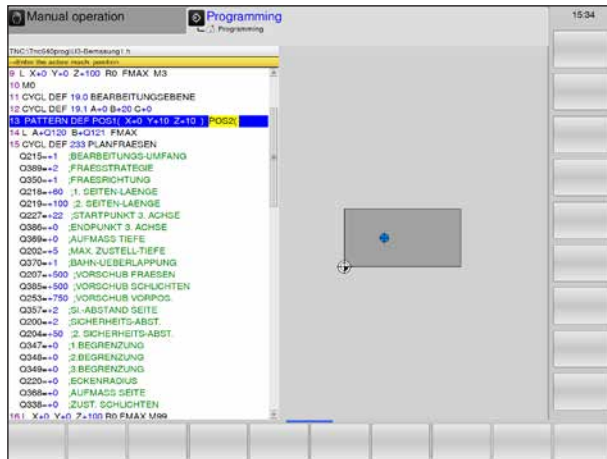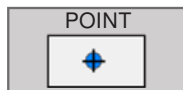
Pattern DEF

Point

A maximum of 9 points can be defined.
Confirm the entry by pressing ENT.
If a workpiece surface is defined in Z other than 0, this value will act in addition to the workpiece surface Q203 defined in the machining cycle.

Coordinate input

1   Enter the X coordinate of the machining position (absolute)
2   Enter the Y coordinate of the machining position (absolute)
3   Enter the Z coordinate of the workpiece surface (absolute). Z coordinate where the machining operation starts.
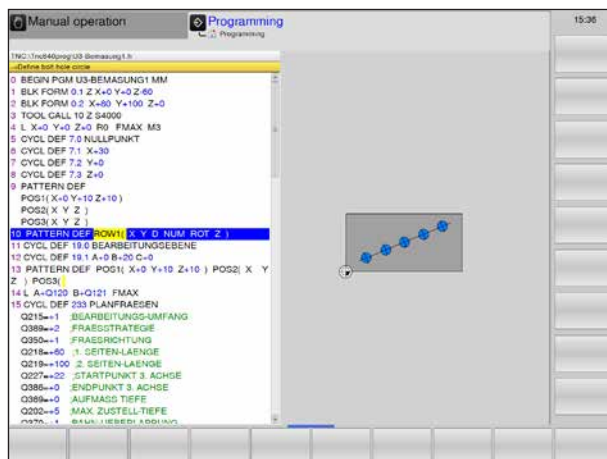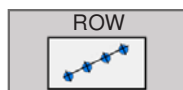
Row

If a workpiece surface is defined in Z other than 0, this value will act in addition to the workpiece surface Q203 defined in the machining cycle.

Coordinate input

1   Starting point X (absolute): Enter the coordinate of the row starting point in the X axis.
2   Starting point Y (absolute): Enter the coordinate of the row starting point in the Y axis.
3   Enter the distance between the machining positions (incremental). Positive value.
4   Number of machining operations: Total number of machining positions
5   Rotational position of the entire pattern (absolute): Angle of rotation around the starting point specified. Reference axis: Main axis of the active machining plane. Positive or negative value.
6   Workpiece surface coordinate (absolute): Z coordinate where the machining operation starts.
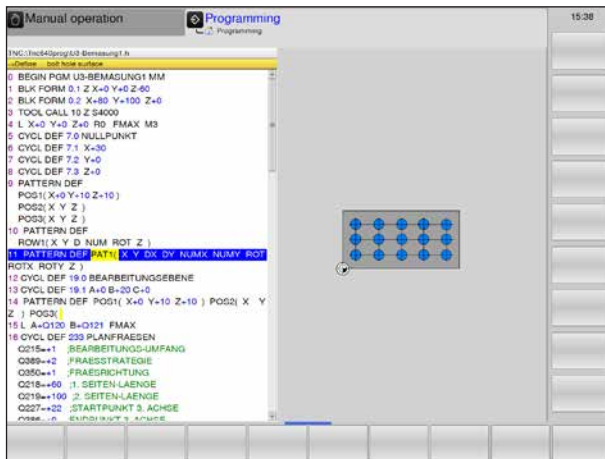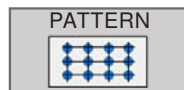
Pattern

If a workpiece surface is defined in Z other than 0, this value will act in addition to the workpiece surface Q203 defined in the machining cycle.

The main axis rotational position and minor axis rotational position parameters have an additive effect on a previously performed rotational position of the entire pattern.

Coordinate input

1 Starting point X (absolute): Enter the coordinate of the pattern starting point in the X axis.

2 Starting point Y (absolute): Enter the coordinate of the pattern starting point in the Y axis.

3 Distance between machining positions X (incremental): Distance between the machining positions in X direction. Positive or negative value.

4 Distance between machining positions Y (incremental):
Distance between the machining positions in the Y direction. Positive or negative value.

5 Number of columns: Total number of columns in the pattern.

6 Number of lines: Total number of lines of the pattern.

7 Rotational position of the entire pattern (absolute):
Rotation angle, by which the entire pattern is rotated around the starting point specified. Reference axis: Main axis of the active machining plane (e.g. X for tool axis Z). Positive or negative value.

8 Rotation position of main axis: Angle of rotation, by which only the reference axis of the machining plane is skewed with respect to the starting point specified. Positive or negative value.

9 Rotational position of minor axis: Angle of rotation, by which only the minor axis of the machining plane is skewed with respect to the starting point specified. Positive or negative value.

10 Coordinates of workpiece surface (absolute): Enter the Z coordinate, at which the machining is to start.

FRAME

Frame

If a workpiece surface is defined in Z other than 0, this value will act in addition to the workpiece surface Q203 defined in the machining cycle.

The main axis rotational position and minor axis rotational position parameters have an additive effect on a previously performed rotational position of the entire pattern.

Coordinate input

1 Starting point X (absolute): Enter the coordinate of the frame starting point in the X axis.

2 Starting point Y (absolute): Enter the coordinate of the frame starting point in the Y axis.

3 Distance between machining positions X (incremental):
Distance between the machining positions in X direction. Positive or negative value.

4 Distance between machining positions Y (incremental):
Distance between the machining positions in the Y direction. Positive or negative value.

5 Number of columns: Total number of columns in the pattern.

6 Number of lines: Total number of lines of the pattern.

7 Rotational position of the entire pattern (absolute):
Rotation angle, by which the entire pattern is rotated around the starting point specified. Reference axis: Main axis of the active machining plane (e.g. X for tool axis Z). Positive or negative value.

8 Rotation position of main axis: Angle of rotation, by which only the reference axis of the machining plane is skewed with respect to the starting point specified. Positive or negative value.

9 Rotational position of minor axis: Angle of rotation, by which only the minor axis of the machining plane is skewed with respect to the starting point specified. Positive or negative value.

10 Coordinates of workpiece surface (absolute): Enter the Z coordinate, at which the machining is to start.

Circle

CIRCLE

If a workpiece surface is defined in Z other than 0, this value will act in addition to the workpiece surface Q203 defined in the machining cycle.

Coordinate input

1 Bolt hole circle centre X (absolute): Enter the coordinate of the circle centre in the X axis.
2 Bolt hole circle centre Y (absolute): Enter the coordinate of the circle centre in the Y axis.
3 Bolt hole circle diameter
4 Start angle: Polar angle of the first machining position.
Reference axis: Main axis of the active machining plane (e.g. X for tool axis Z). Enter a positive or negative value.
5 Number of machining operations: Total number of machining positions on the circle.
6 Coordinates of workpiece surface (absolute): Enter the Z coordinate, at which the machining is to start.

Pitch circle

PITCH CIR

If a workpiece surface is defined in Z other than 0, this value will act in addition to the workpiece surface Q203 defined in the machining cycle.

Coordinate input

1 Bolt hole circle centre X (absolute): Enter the coordinate of the circle centre in the X axis.
2 Bolt hole circle centre Y (absolute): Enter the coordinate of the circle centre in the Y axis.
3 Bolt hole circle diameter
4 Start angle: Polar angle of the first machining position.
Reference axis: Main axis of the active machining plane (e.g. X for tool axis Z). Enter a positive or negative value.
5 Angular increments/end angle: Incremental polar angle between two machining positions. Enter a positive or negative value.
Alternatively, the end angle can be entered (switch using softkey)
6 Number of machining operations: Total number of machining positions on the circle.
7 Coordinates of workpiece surface (absolute): Enter the Z coordinate, at which the machining is to start.

STEP   STOP

emco

Select pattern

SPEC FCT

CONTOUR + POINT MACHINING

SEL PATTERN

SELECT FILE

✔ OK
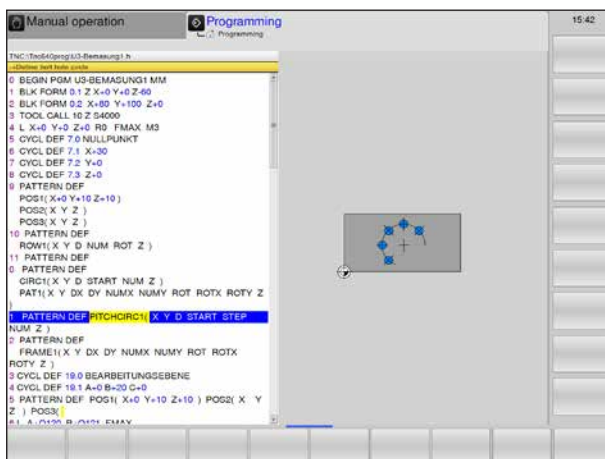
1 Display Special Functions

2 Press the CONTOUR + POINT MACHINING softkey.

3 Select a point table using the SEL PATTERN softkey.
4 As soon as the name of the table is entered, the SELECT FILE softkey appears.
5 When the softkey is pressed, a window opens to search for existing point tables.
6 Select file and confirm with OK
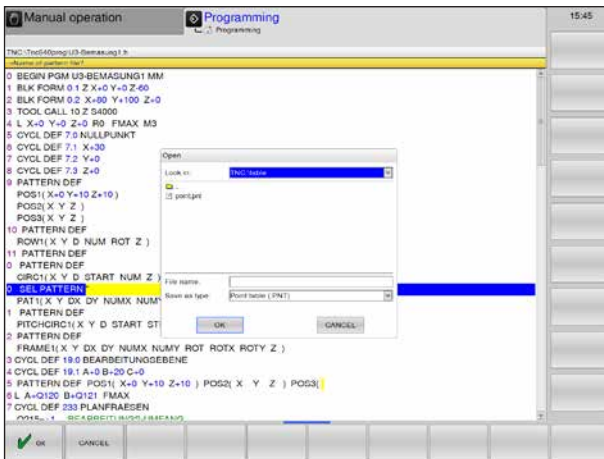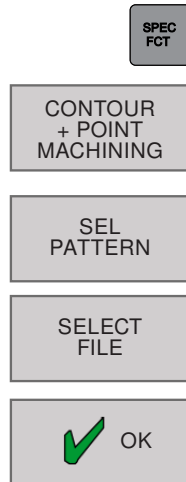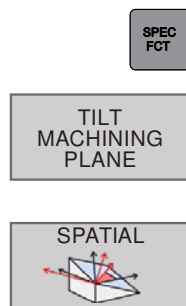
emco

Tilting the machining plane *)

SPEC FCT

1 Display Special Functions

TILT MACHINING PLANE

2 Press the TILT MACHINING PLANE softkey.

SPATIAL

3 Press the SPATIAL softkey to activate the cycle.

Application

Spatial angles define a machining plane by up to three rotations around a coordinate system, with two views existing for this purpose, which always lead to the same result.

Rotations around the machine-fixed coordinate system:
The sequence of rotations is first around the machine axis A, then around the machine axis B, and then around the machine axis C.

Rotations around the respective tilted coordinate system:
The sequence of rotations is first around the machine axis C, then around the rotated axis B, and then around the rotated axis A. This view is usually easier to understand because the rotations of the coordinate system are easier to follow when one rotation axis is fixed.

Note:
Prior to programming, please note:
All three spatial angles SPA, SPB and SPC must always be defined, even if one of the angles is 0.

The mode of operation corresponds to that of cycle 19.

PLANE SPATIAL is not allowed if cycle 8 MIRRORING is active.

---

emco

Parameter input

1 Axis angle A: Axis angle to which the A axis should be tilted.
If entered incrementally: the angle by which the A axis is to be tilted further from the current position.

2 Axis angle B: Axis angle to which the B axis is to be tilted.
If entered incrementally: the angle by which the A axis is to be tilted further from the current position.

3 Axis angle C: Axis angle to which the C axis should be tilted.
If entered incrementally: the angle by which the A axis is to be tilted further from the current position.

4 MOVE:
The PLANE function should automatically tilt the rotation axes to the calculated axis values, whereby the relative position between workpiece and tool does not change. WinNC executes a compensation movement in the linear axes.

4a Distance of tool tip from pivot point (incremental): WinNC tilts the tool (the table) around the tool tip. The DIST parameter is used to shift the pivot point of the tilting movement relative to the current position of the tool tip.

4b Feed F=: Path speed at which the tool should tilt.

5 TURN:
The PLANE function should automatically tilt the rotation axes to the calculated axis values, whereby only the rotation axes are positioned. WinNC does not execute a compensation movement in the linear axes.

5a Retraction length in the tool axis:
Retraction path MB, acts incrementally from the current tool position in the active tool axis direction, which the WinNC approaches before the tilt operation. MB MAX moves the tool to just before the software limit switch.

5b Feed F=: Path speed at which the tool should tilt.

6 STAY:
The rotation axes are tilted in a subsequent, separate positioning block.

MOVE

F MAX	F AUTO	F

TURN

F MAX	F AUTO	F

STAY
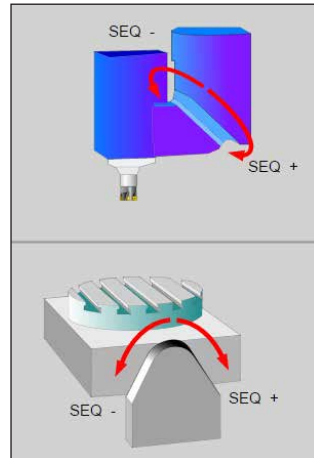
7 SEQ +/-
SEQ+ positions the master axis at a positive angle.
The master axis is the 1st rotation based on the tool or the last rotation axis based on the table
SEQ- positions the master axis so that it has a negative angle.

Recommended procedure to tilt with spatial angle
For targeted tilting with a defined orientation of the coordinate plane, you must first rotate the X axis to the bent edge with the spatial angle C. Afterwards, the tilting can be performed via the spatial angle A onto the surface to be machined.

This procedure makes the tilting programs more uniform and thus also clearer. Aligning the X axis via the spatial angle C before tilting in with the spatial angle A eliminates the need to align the coordinate plane, which would be necessary after turning in with Cycle 10.

By tilting in the same way onto the machining planes, an identical alignment of the coordinate system is guaranteed. The identical alignment makes it possible to start similar machining operations on the individual planes from a subprogram without any adjustments.

Positioning in the tilted system
With the additional function M130, it is also possible to move to positions in the tilted system that relate to the non-tilted coordinate system.
Positioning with straight-line blocks that relate to the machine coordinate system (blocks with M91 or M92) can be performed with a tilted machining plane.
Restrictions:

- Positioning occurs without length correction
- Positioning occurs without machine geometry correction
- Tool radius correction is not allowed.

No rotation in the tilted system!

Tilting in with more than one spatial angle 4
Example: Tilting in to spatial angle A+45 B+0 C+90 **)
The home position for both procedures is the machine-fixed coordinate system (MK).

Rotation sequence ABC

Home position
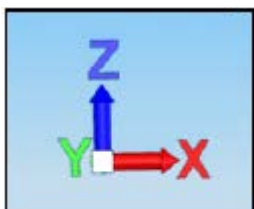
A+45° (Based on MK around X)
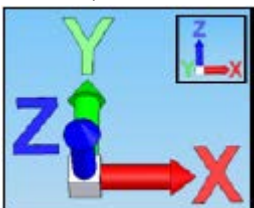
B+0° (Based on MK around Y)
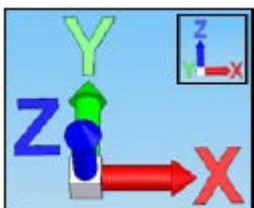
C+90° (Based on MK around Z)
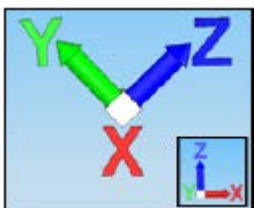
Rotation sequence CBA

Home position
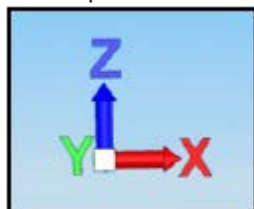
C+90° (Based on MK around Z)
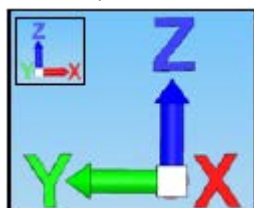
B+0° (Relative to the last tilt around Y)
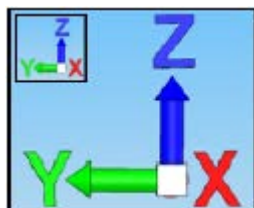
A+45° (Relative to the last tilt around X)
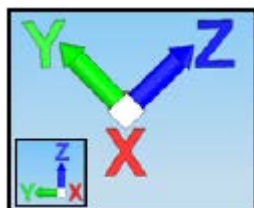
The result is identical for both approaches, but the reference for the respective subsequent tilting differs.

Programming example

```
0   BEGIN PGM U3-DIMENSIONING1 MM
1   BLK FORM 0.1 Z X+0 Y+0 Z-60
2   BLK FORM 0.2  X+80  Y+100  Z+0
3   TOOL CALL 10 Z S4000
4   L  X+0  Y+0  Z+0  R0   FMAX  M3
5   CYCL DEF 7.0 ZERO POINT
6   CYCL DEF 7.1  X+30
7   CYCL DEF 7.2  Y+0
8   CYCL DEF 7.3  Z+0
9   L  X+0  Y+0  Z+100  R0  FMAX  M3
10  M0
11  CYCL DEF 19.0 MACHINING PLANE
12  CYCL DEF 19.1 A+0 B+20 C+0
13  L A+Q120 C+Q122 FMAX
14  CYCL DEF 233 FACE MILLING~
    Q215=+1    ;SCOPE OF MACHINING ~
    Q389=+2    ;MILLING STRATEGY ~
    Q350=+1    ;MILLING DIRECTION ~
    Q218=+60   ;1. SIDE-LENGTH ~
    Q219=+100  ;2. SIDE-LENGTH ~
    Q227=+22   ;START POINT 3. AXIS ~
    Q386=+0    ;END POINT 3. AXIS ~
    Q369=+0    ;DEPTH MEASUREMENT ~
    Q202=+5    ;MAX. INFEED DEPTH~
    Q370=+1    ;PATH OVERLAP~
    Q207=+500  ;INFEED MILLING~
    Q385=+500  ;INFEED FINISHING ~
    Q253=+750  ;INFEED FORW. POS. ~
    Q357=+2    ;SAFETY DIST. SIDE ~
    Q200=+2    ;SAFETY DISTANCE ~
    Q204=+50   ;2. SAFETY DISTANCE ~
    Q347=+0    ;1ST LIMIT ~
    Q348=+0    ;2ND LIMIT ~
    Q349=+0    ;3RD LIMIT ~
    Q220=+0    ;CORNER RADIUS ~
    Q368=+0    ;SIDE MEASUREMENT ~
    Q338=+0 ;INFEED FINISHING
15 L X+0  Y+0  Z+100 R0 FMAX M99
16 L  Z+100 R0 FMAX
17 M0
18 PLANE RESET TURN FMAX
19 TOOL CALL 3 Z S6000
20 L  Z+100 R0 FMAX M3
21  PLANE  SPATIAL SPA+0  SPB+20  SPC+0
TURN FMAX SEQ-
22 CYCL DEF 200 DRILLING ~
    Q200=+2    ;SAFETY DISTANCE ~
    Q201=-20   ;DEPTH ~
    Q206=+150  ;PLNGNG FEED RATE. ~
    Q202=+5    ;INFEED DEPTH ~
    Q210=+0    ;DWELL TIME UP~
    Q203=+0    ;COOR. SURFACE ~
    Q204=+50   ;2. SAFETY DISTANCE ~
    Q211=+0    ;DWELL TIME DOWN~
    Q395=+0    ;REFERENCE DEPTH
23 L  X+20  Y+25  Z+100 R0 FMAX M99
24 M0
25 L  Y+75 R0 FMAX M99
26 L  X+50 R0 FMAX M99
27 L  Y+25 R0 FMAX M99
28 L  Z+100 R0 FMAX
29 PLANE RESET TURN FMAX
30 CYCL DEF 7.0 ZERO POINT
31 CYCL DEF 7.1  X+0
32 CYCL DEF 7.2  Y+0
33 CYCL DEF 7.3  Z+0
34 L  Z+100 R0 FMAX M30
35 END PGM U3-DIMENSIONING1 MM
```

emco

RESET tilting the machining plane *)

SPEC FCT

TILT MACHINING PLANE

RESET

1 Display Special Functions

2 Press the TILT MACHINING PLANE softkey.

3 Press the RESET softkey to activate the cycle.

Reset function: This internally resets the PLANE function; nothing changes at the current axis positions.

The PLANE RESET function completely resets the active PLANE function – or an active cycle 19 (angle = 0 and function inactive). A multiple definition is not necessary.

MOVE   **TURN**   **STAY**

4 Define whether the WinNC should move the tilted axes automatically to the home position (MOVE or TURN) or not (STAY).

END

End Special Functions.

Measuring a work piece and tool

- 400 basic rotation
- 408 datum slot center
- 409 datum ridge center
- 412 inner circle reference point
- 413 outer circle reference point
- 419 individual axis reference point
- 461 calibrate scanning system length
- 462 calibrate inner scanning system radius
- 480 calibrate scanning system TT
- 481 measure tool length
- 482 measure tool radius

TOUCH PROBE

emco

Basic information on tool and work piece measurement

The WinNC must be prepared for the use of 3D scanning systems by the machine manufacturer.

Reference point and axis scanning system

Die WinNC sets the reference point in the machining plane according to the scanning system axis defined in the measuring programme.

Axis assignment	
Active scanning system axis	Set reference point in axis
Z	X and Y

The tools are automatically measured with the table scanning system and the tool measuring cycles of the controls:

The WinNC stores the correction values for the length and radius in the central tool memory TOOL.T and automatically considered at the end of the scanning cycle.

The following measuring methods are available:

- Tool measurement with idle tool
- Tool measurement with rotating tool

The measurement cycles only work when the central tool memory TOOL.T is active.
Before it is possible to work with the measuring cycles, all data required for the measurement must be entered into the central tool memory. The tool to be measured must be called up with TOOL CALL.

Measuring a tool
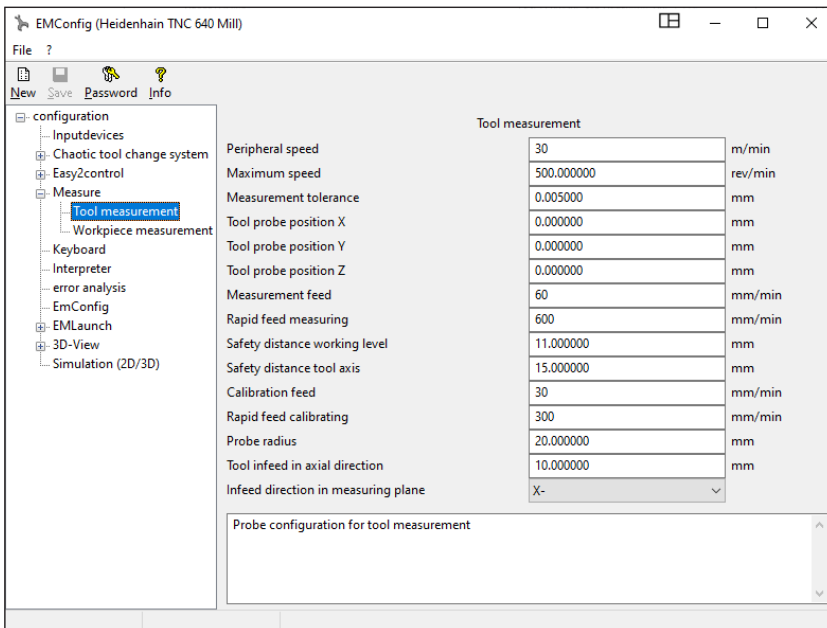
This interface lets you input the configuration data of the measuring scanner for tool measurements.

Measuring a work piece
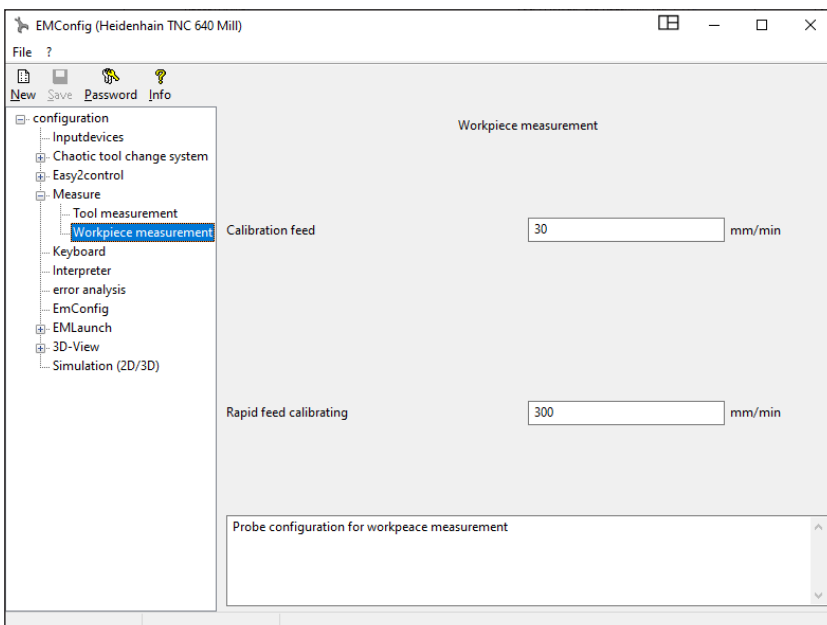
This interface lets you input the configuration data of the measuring scanner for work piece measurements.
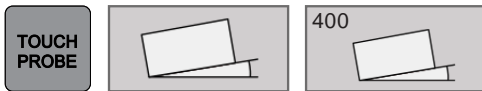
TCH PROBE BASIC ROTATION (cycle 400)
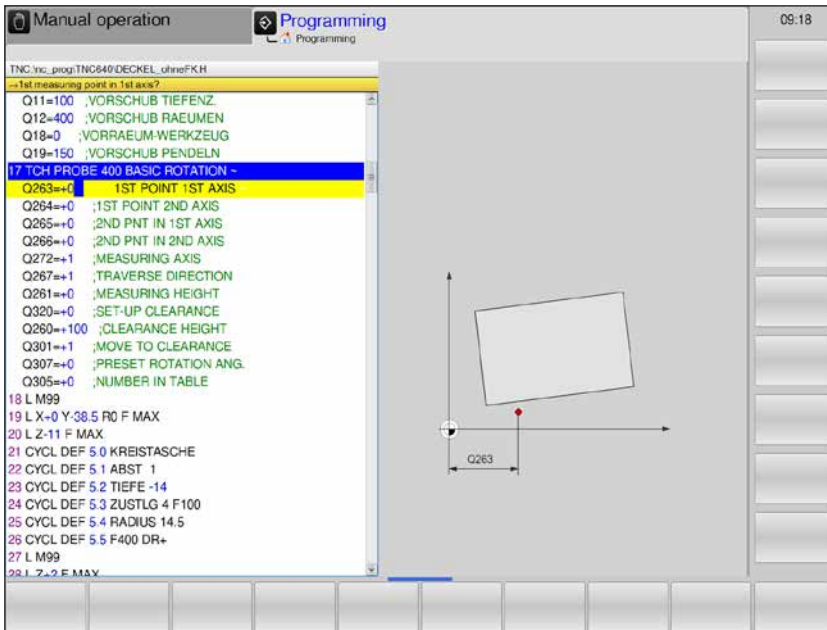
Parameters	Description	Unit
Q263	1st meas. point 1st axis (absolute): Coordinate of the first touch point in the reference axis of the working plane	mm
Q264	1st meas. point 2nd axis (absolut): Coordinate of the first touch point in the minor axis of the working plane.	mm
Q265	2nd meas. point 1st axis (absolute): Coordinate of the second touch point in the reference axis of the working plane.	mm
Q266	2nd meas. point 2nd axis (absolute): Coordinate of the second touch point in the minor axis of the working plane.	mm
Q272	Measuring axis (absolut): Axis in the working plane in which the measurement is to be made: • 1 = Principal axis = measuring axis • 2 = Secondary axis = measuring axis	mm
Q267	Traverse direction 1: Direction in which the probe is to approach the workpiece: • -1 = Negative traverse direction • +1 = Positive traverse direction	
Q261	Measuring height in the touch probe axis (absolut): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.	mm
Q320	Set-up clearance (absolute): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table).	mm

Parameters	Description	Unit
Q260	Clearance height (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur.	mm
Q301	Traversing to clearance height (absolute): definition of how the touch probe is to move between the measuring points: • 0: Move at measuring height between measuring points. • 1: Move at clearance height between measuring points.	mm
Q307	Preset value for rotation angle (absolute): If the misalignment is to be measured against a straight line other than the reference axis, enter the angle of this reference line. The WinNC will then calculate the difference between the value measured and the angle of the reference line for the basic rotation.	degree
Q305	Preset number in table: Enter the preset number in the table in which the WinNC is to save the determined basic rotation. If you enter Q305=0, the WinNC automatically places the determined basic rotation in the ROT menu of the Manual Operation mode.	

emco
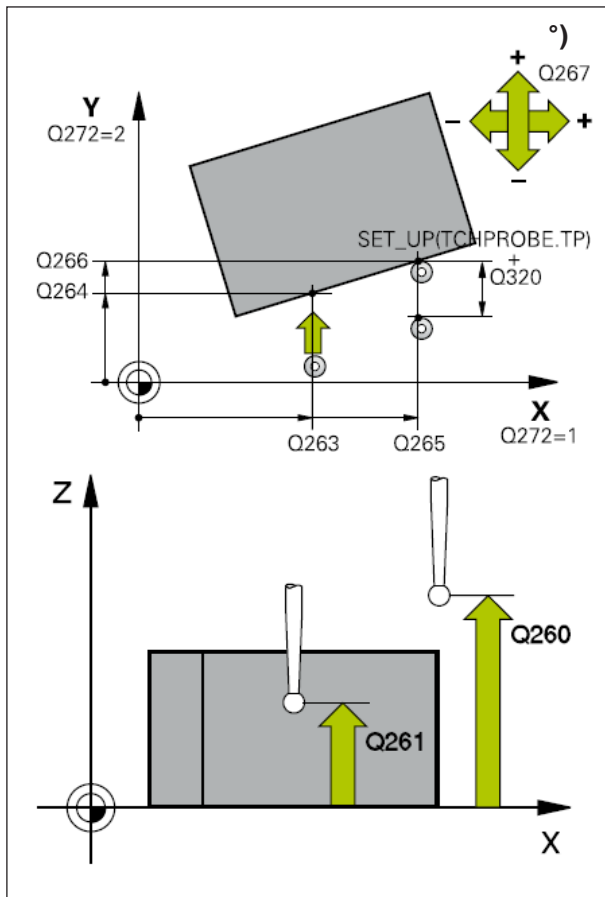
Cycle description

Touch probe cycle 400 determines a workpiece misalignment by measuring two points, which must lie on a straight surface. With the basic rotation function the WinNC compensates the measured value.

1. Following the positioning logic, the WinNC positions the touch probe at rapid traverse (value from FMAX column) to the programmed touch point 1. The WinNC offsets the touch probe by the safety clearance in the direction opposite to the defined traverse direction.
2. Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F).
3. Then the touch probe moves to the next starting position 2 and probes the second position.
4. The WinNC returns the touch probe to the clearance height and performs the basic rotation.

> **Note:**
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.
> The WinNC will reset an active basic rotation at the beginning of the cycle.

emco
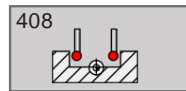
TCH PROBE DATUM SLOT CENTER (cycle 408)
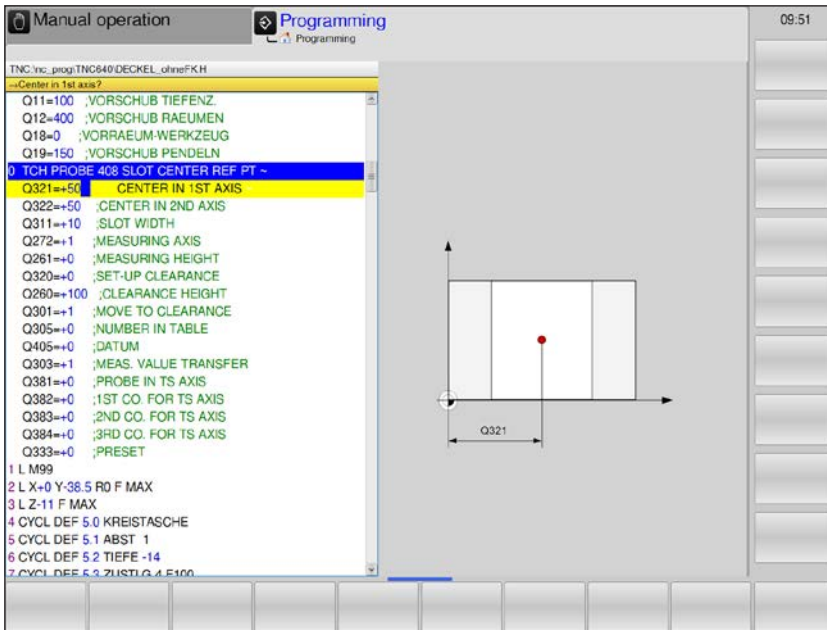
Parameters	Description	Unit
Q321	Center in 1st axis (absolute): Center of the slot in the reference axis of the working plane.	mm
Q322	Center in 2nd axis (absolute): Center of the slot in the minor axis of the working plane.	mm
Q311	Width of slot (incremental): Width of the slot, regardless of its position in the working plane.	mm
Q372	Measuring axis (absolute): Axis in the working plane in which the measurement is to be made: • 1 = Principal axis = measuring axis • 2 = Secondary axis = measuring axis	
Q261	Measuring height in the touch probe axis (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.	
Q320	Set-up clearance (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table).	mm
Q260	Clearance height (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur.	mm
Q301	Traversing to clearance height: definition of how the touch probe is to move between the measuring points: • 0 = Move at measuring height between measuring points. • 1 = Move at clearance height between measuring points.	

Parameters	Description	Unit
Q305	Number in table: Enter the number in the datum/preset table in which the WinNC is to save the coordinates of the slot center. If Q303=1: If you enter Q305=0, the WinNC automatically sets the display so that the new datum is on the slot center. If Q303=0: If you enter Q305=0, the WinNC writes to line 0 of the datum table.	
Q405	New datum (absolute): Coordinate in the measuring axis at which the WinNC should set the calculated slot center.	mm
Q303	Measured value transfer: Specify if the determined datum is to be saved in the datum table or in the preset table: • 0: Write the measured datum as datum shift active datum table. The reference system is the active workpiece coordinate system. • 1: Write the measured datum into the preset table. The reference system is the machine coordinate system.	
Q381	Probe in TS axis: Specify whether the WinNC should also set the datum in the touch probe axis: • 0: Do not set the datum in the touch probe axis. • 1: Set the datum in the touch probe axis.	
Q382	Probe TS axis: Coordinate 1st axis (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis.	mm
Q383	Probe TS axis: Coordinate 2nd axis Q383 (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1	mm
Q384	Probe TS axis: Coordinate 3rd axis (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1	mm
Q333	New datum in TS axis (absolute): Coordinate in the touch probe axis at which the WinNC should set the datum. Default setting = 0	mm

Cycle description

Touch Probe Cycle 408 finds the center of a slot and defines its center as datum. If desired, the WinNC can also enter the coordinates into a datum table or the preset table.

1　Following the positioning logic, the WinNC positions the touch probe at rapid traverse (value from FMAX column) to touch point 1. The WinNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.

2　Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F).

3　Then the touch probe moves either paraxially at measuring height or at clearance height to the next starting point 2 and probes the second touch point.

4　Finally the WinNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 and saves the actual values in the Q parameters listed below.

5　If desired, the WinNC subsequently measures the datum in the touch probe axis in a separate probing.

Note:

To prevent a collision between touch probe and workpiece, enter a low estimate for the slot width.

If the slot width and the safety clearance do not permit pre-positioning in the proximity of the touch points, the WinNC always starts probing from the center of the slot. In this case the touch probe does not return to the clearance height between the two measuring points.

Before a cycle definition you must have programmed a tool call to define the touch probe axis.

If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.
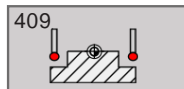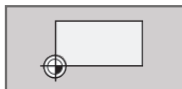
TCH PROBE DATUM RIDGE CENTER (cycle 409)

Parameters	Description	Unit
Q321	Center in 1st axis (absolute): Center of the ridge in the reference axis of the working plane.	mm
Q322	Center in 2nd axis (absolute): Center of the ridge in the minor axis of the working plane.	mm
Q311	Width of ridge (incremental): Width of the ridge, regardless of its position in the working plane.	mm
Q372	Measuring axis (absolut): Axis in the working plane in which the measurement is to be made: • 1 = Principal axis = measuring axis • 2 = Secondary axis = measuring axis	
Q261	Measuring height in the touch probe axis (absolute): Coordinate of the ball tip center (= touch point) in the touch probe axis in which the measurement is to be made.	
Q320	Set-up clearance (incremental): Additional distance between measuring point and ball tip. Q320 is added to SET_UP (touch probe table).	mm
Q260	Clearance height (absolute): Coordinate in the touch probe axis at which no collision between touch probe and workpiece (fixtures) can occur.	mm
Q305	Number in table: Enter the number in the datum/preset table in which the WinNC is to save the coordinates of the ridge center. If Q303=1: If you enter Q305=0, the TNC automatically sets the display so that the new datum is on the ridge center. If Q303=0: If you enter Q305=0, the WinNC writes to line 0 of the datum table:	

Parameters	Description	Unit
Q405	New datum (absolute): Coordinate in the measuring axis at which the WinNC should set the calculated ridge center. Default setting = 0	mm
Q303	Measured value transfer: Specify if the determined datum is to be saved in the datum table or in the preset table: • 0: Write the measured datum as datum shift active datum table. The reference system is the active workpiece coordinate system. • 1: Write the measured datum into the preset table. The reference system is the machine coordinate system.	
Q381	Probe in TS axis: Specify whether the WinNC should also set the datum in the touch probe axis: • 0: Do not set the datum in the touch probe axis. • 1: Set the datum in the touch probe axis.	
Q382	Probe TS axis: Coordinate 1st axis (absolute): Coordinate of the probe point in the reference axis of the working plane at which point the datum is to be set in the touch probe axis.	mm
Q383	Probe TS axis: Coordinate 2nd axis (absolute): Coordinate of the probe point in the minor axis of the working plane at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1	mm
Q384	Probe TS axis: Coordinate 3rd axis (absolute): Coordinate of the probe point in the touch probe axis, at which point the datum is to be set in the touch probe axis. Only effective if Q381 = 1.	mm
Q333	New datum in TS axis (absolute): Coordinate in the touch probe axis at which the WinNC should set the datum. Default setting = 0.	mm

Cycle description

Touch Probe Cycle 409 finds the center of a ridge and defines its center as datum. If desired, the WinNC can also enter the coordinates into a datum table or the preset table.

1 Following the positioning logic, the WinNC positions the touch probe at rapid traverse (value from FMAX column) to touch point 1. The WinNC calculates the touch points from the data in the cycle and the safety clearance from the SET_UP column of the touch probe table.
2 Then the touch probe moves to the entered measuring height and runs the first probing process at the probing feed rate (column F).
3 Then the touch probe moves at clearance height to the next touch point 2 and probes the second touch point.
4 Finally the WinNC returns the touch probe to the clearance height and processes the determined datum depending on the cycle parameters Q303 and Q305 and saves the actual values in the Q parameters listed below.
5 If desired, the WinNC subsequently measures the datum in the touch probe axis in a separate probing.

> **Note:**
> To prevent a collision between touch probe and workpiece, enter a high estimate for the ridge width.
>
> Before a cycle definition you must have programmed a tool call to define the touch probe axis.
>
> If you set a datum (Q303 = 0) with the touch probe cycle and also use probe in TS axis (Q381 = 1), then no coordinate transformation must be active.

emco

TOUCH PROBE   412
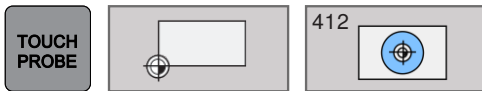
TCH PROBE INNER CIRCLE REFERENCE POINT (cycle 412)

Parameters	Description	Unit
Q321	Centre 1st Axis (absolute): Centre of pocket in the main axis of the machining plane	mm
Q322	Centre 2nd Axis (absolute): Centre of pocket in the minor axis of the machining plane. If Q322=0, then WinNC will align the bore centre with the positive Y axis. If Q322 is not equal to 0, then WinNC will align the bore centre with the nominal position.	mm
Q262	Nominal diameter: Approximate diameter of the circular pocket. Preferably select a small value to enter.	mm
Q325	Start angle (absolute): Angle between the main axis of the machining plane and the first scanning point.	Degrees
Q247	Angle step (incremental): Angle between 2 measuring point. The sign of the angle step determines the rotational direction the scanning system uses to travel to the next measuring point. If you are measuring arcs, programme the angle step smaller than 90°.	Degrees
Q261	Measuring height in the scanning system axis (absolute): Ball centre coordinate (touch point) in the scanning system axis where the measurement takes place.	mm
Q320	Safety distance (incremental): Additional distance between the measuring point and the scanning system ball. Acts additionally to SET_UP of the scanning system table	mm
Q260	Safe height (absolute): Coordinate of the scanning system axis, in which no collision between scanning system and work piece (clamping device) can occur.	mm
Q301	Travel to safe height: determines how the scanning system is supposed to travel between the measuring points • Travel between measuring points at measuring height • Travel between measuring points at safe height	mm

Parameters	Description	Unit
Q305	Zero position number in table: Input the number in the preset table where the WinNC stores the coordinate for the pocket centre. If Q303=1, then upon an input of Q305=0 the following applies: WinNC automatically sets the display to show the new reference point in the centre of the pocket.	
Q331	New reference point on main axis (absolute): Coordinate on the main axis where the controls are to place the determined pocket centre. Default setting: =0.	
Q332	New reference point on secondary axis (absolute): Coordinate on the secondary axis where the controls are to place the determined pocket centre. Default setting: =0.	
Q303	Measuring value transfer: determines whether the determined reference point should be stores in the zero position (or preset) table. • -1= do not use! Is input by WinNC. • 1= write the determined reference point into the preset table. The reference system is the machine coordinate system.	
Q381	Scanning in TS axis: determines whether WinNC is also to set the reference point in the scanning system axis. • 0= Do not set reference point in the scanning system axis • 1= Set reference point in the scanning system axis	
Q382	1. Coordinate for TS axis (absolute): Coordinate of the scanning point in the main axis of the machining plane where the reference point in the scanning system axis should be set. Only applies when Q381=1.	mm
Q383	2. Coordinate for TS axis (absolute): Coordinate of the scanning point in the secondary axis of the machining plane where the reference point in the scanning system axis should be set. Only applies when Q381=1.	mm
Q384	3. Coordinate for TS axis (absolute): Coordinate of the scanning point in the scanning system axis of the machining plane where the reference point in the scanning system axis should be set. Only applies when Q381=1.	mm
Q333	New reference point on TS axis (absolute): Coordinate in the scanning system axis where the WinNC is to set the reference point. Default setting =0.	
Q423	Number of measuring points: determines whether the WinNC is to measure the stud with 4 or 3 scans. • 4= Use 4 measuring points (default setting) • 3= Use 3 measuring points	
Q365	Type of travel: determines the path function the tool should use to travel between the measuring points when travel at a safe height is activated. • 0= Travel on a straight line between the machining operations • 1= Travel circularly on the pitch circle diameter between the machining operations.	

°)

Cycle description

The scanning system cycle 412 determines the centre of a circular pocket and sets it as a reference point.
The controls can write the centre either into a zero position or preset table.

1 The WinNC positions the scanning system in rapid traverse and with positioning logic at the scanning point **1**. WinNC calculates three scanning points from the information in the cycle and the safety distance from the scanning system table.
2 The scanning system travels to the input measuring height and executes the first scanning process with scanning infeed. WinNC automatically determines the scanning direction depending on the programmed starting angle.
3 The scanning system travels to the next scanning point **2** (either at measuring height or at safe height) and executes the second scanning sequence there.
4 WinNC positions the scanning system at the scanning point **3** and **4** and executes the third and/or fourth scanning sequence there.
5 Subsequently, the WinNC positions the scanning system at the safe height and processes the determined reference point depending on the parameters Q303 and Q305 and save the actual values in the listed Q parameters.
6 If desired, WinNC can determine the reference point in the scanning system axis in a separate scanning sequence

Q151: Actual value in the centre of the main axis
Q152: Actual value in the centre of the secondary axis
Q153: Actual diameter value

Note:

Attention: danger of collision:
In order to prevent any collision of the scanning system and the work piece, it is advised to input a slightly too small nominal diameter of the pocket.

If the pocked dimensions and the safety distance do not allow any prepositioning near the scanning points, the WinNC will always scan from the centre of the pocket. The scanning system does not move to the safe height between the four measuring points.

The smaller you programme the angle step Q247, the less accuracy WinNC displace in the calculation of the reference point. Minimum input value = 5°.

Before the cycle definition, a tool call for the definition of the scanning system axis must be programmed.

If a reference point (Q303=0) is set with the key system cycle, and scanning on the TS axis (Q381=1) is used as well, no coordinate conversion must be active!

TCH PROBE OUTER CIRCLE REFERENCE POINT (cycle 413)

Parameters	Description	Unit
Q321	Centre 1st Axis (absolute): Centre of stud in the main axis of the machining plane	mm
Q322	Centre 2nd Axis (absolute): Centre of stud in the minor axis of the machining plane. If Q322=0, then WinNC will align the bore centre with the positive Y axis. If Q322 is not equal to 0, then WinNC will align the bore centre with the nominal position.	mm
Q262	Nominal diameter: Approximate diameter of the stud. Preferably select a large value to enter.	mm
Q325	Start angle (absolute): Angle between the main axis of the machining plane and the first scanning point.	Degrees
Q247	Angle step (incremental): Angle between 2 measuring point. The sign of the angle step determines the rotational direction the scanning system uses to travel to the next measuring point. If you are measuring arcs, programme the angle step smaller than 90°.	Degrees
Q261	Measuring height in the scanning system axis (absolute): Ball centre coordinate (touch point) in the scanning system axis where the measurement takes place.	mm
Q320	Safety distance (incremental): Additional distance between the measuring point and the scanning system ball. Acts additionally to SET_UP of the scanning system table	mm
Q260	Safety height (absolute): Coordinate of the scanning system axis, in which no collision between scanning system and work piece (clamping device) can occur.	mm
Q301	Travel to safe height: determines how the scanning system is supposed to travel between the measuring points • 0= Travel between measuring points at measuring height • 1= Travel between measuring points at safe height	mm

Parameters	Description	Unit
Q305	Zero position number in table: Input the number in the preset table where the WinNC stores the coordinate for the stud centre. If Q303=1, then upon an input of Q305=0 the following applies: WinNC automatically sets the display to show the new reference point in the centre of the stud.	
Q331	New reference point on main axis (absolute): Coordinate on the main axis where the controls are to place the determined stud centre. Default setting: =0.	
Q332	New reference point on secondary axis (absolute): Coordinate on the secondary axis where the controls are to place the determined stud centre. Default setting: =0.	
Q303	Measuring value transfer: determines whether the determined reference point should be stores in the zero position (or preset) table. • -1= do not use! Is input by WinNC. • 1= write the determined reference point into the preset table. The reference system is the machine coordinate system.	
Q381	Scanning in TS axis: determines whether WinNC is also to set the reference point in the scanning system axis. • 0= Do not set reference point in the scanning system axis. • 1= Set reference point in the scanning system axis	
Q382	1. Coordinate for TS axis (absolute): Coordinate of the scanning point in the main axis of the machining plane where the reference point in the scanning system axis should be set. Only applies when Q381=1.	mm
Q383	2. Coordinate for TS axis (absolute): Coordinate of the scanning point in the secondary axis of the machining plane where the reference point in the scanning system axis should be set. Only applies when Q381=1.	mm
Q384	3. Coordinate for TS axis (absolute): Coordinate of the scanning point in the scanning system axis of the machining plane where the reference point in the scanning system axis should be set. Only applies when Q381=1.	mm
Q333	New reference point on TS axis (absolute): Coordinate in the scanning system axis where the WinNC is to set the reference point. Default setting =0.	
Q423	Number of measuring points: determines whether the WinNC is to measure the stud with 4 or 3 scans. • 4= Use 4 measuring points (default setting) • 3= Use 3 measuring points	
Q365	Type of travel: determines the path function the tool should use to travel between the measuring points when travel at a safe height is activated. • 0= Travel on a straight line between the machining operations • 1= Travel circularly on the pitch circle diameter between the machining operations.	

emco

Note:

Attention: danger of collision:

In order to prevent any collision of the scanning system and the work piece, it is advised to input a slightly too large nominal diameter of the pocket.

Before the cycle definition, a tool call for the definition of the scanning system axis must be programmed.

The smaller you programme the angle step Q247, the less accuracy WinNC displace in the calculation of the reference point. Minimum input value = 5°.

If a reference point (Q303=0) is set with the key system cycle, and scanning on the TS axis (Q381=1) is used as well, no coordinate conversion must be active!

Cycle description

The scanning system cycle 413 determines the centre of a circular stud and sets it as a reference point.

The controls can write the centre either into a zero position or preset table.

1 The WinNC positions the scanning system in rapid traverse and with positioning logic at the scanning point **1**. WinNC calculates three scanning points from the information in the cycle and the safety distance from the scanning system table.

2 The scanning system travels to the input measuring height and executes the first scanning process with scanning infeed. WinNC automatically determines the scanning direction depending on the programmed starting angle.

3 The scanning system travels to the next scanning point **2** (either at measuring height or at safe height) and executes the second scanning sequence there.

4 WinNC positions the scanning system at the scanning point **3** and **4** and executes the third and/or fourth scanning sequence there.

5 Subsequently, the WinNC positions the scanning system at the safe height and processes the determined reference point depending on the parameters Q303 and Q305 and save the actual values in the listed Q parameters.

6 If desired, WinNC can determine the reference point in the scanning system axis in a separate scanning sequence.

Q151: Actual value in the centre of the main axis
Q152: Actual value in the centre of the secondary axis
Q153: Actual diameter value

TCH PROBE INDIVIDUAL AXIS REFERENCE POINT (cycle 419)

Parameters	Description	Unit
Q263	Measuring point 1. Axis (absolute): Coordinate of the first scanning point in the main axis of the machining plane.	mm
Q264	Measuring point 2. Axis (absolute): Coordinate of the first scanning point in the secondary axis of the machining plane.	mm
Q261	Measuring height in the scanning system axis (absolute): Ball centre coordinate (touch point) in the scanning system axis where the measurement takes place.	mm
Q320	Safety distance (incremental): Additional distance between the measuring point and the scanning system ball. Acts additionally to SET_UP of the scanning system table.	Degrees
Q260	Safe height (absolute): Coordinate of the scanning system axis, in which no collision between scanning system and work piece (clamping device) can occur.	Degrees
Q272	Measuring axis (absolute): Axis on which the measurement should take place: • 1 = Main axis is measuring axis • 2 = Secondary axis is measuring axis • 3 = Scanning system axis is measuring axis	

Axis assignment

Active scanning system axis: Q272 = 3	Respective main axis: Q272 = 1	Respective secondary axis: Q272 = 2
Z	X	Y

Parameters	Description	Unit
Q267	Travel direction 1: Direction in which the scanning system is supposed to approach the work piece: • -1 Negative travel direction • +1 Positive travel direction	
Q305	Zero point number in table: Input the number in the preset table where the WinNC stores the coordinate. For Q303=1, the following applies: If Q303=0, WinNC automatically sets the display to show the new reference point on the scanned plane.	
Q333	New reference value (absolute): Coordinate where the WinNC is to set the reference point.	mm
Q303	Measuring value transfer: determines whether the determined reference point should be stored in the zero position (or preset) table. • -1: Do not use! Is used by the WinNC when old programmes are input. • 1: the determined reference point is written into the preset table. The active machine coordinate system is the reference system	
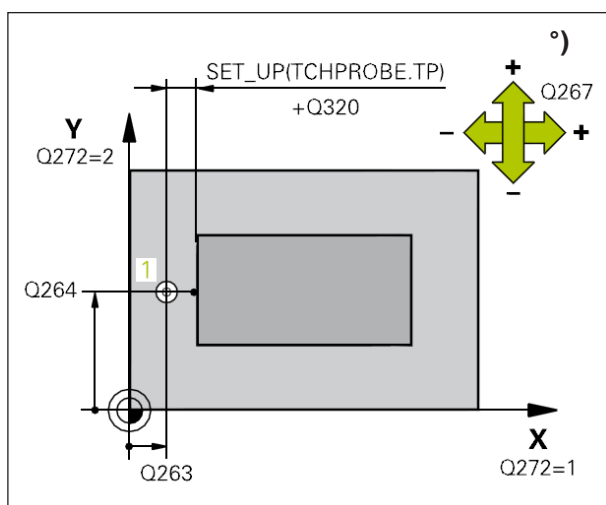
Cycle description

The scanning system cycle 419 measures any coordinate in an available axis and sets this coordinate as the reference point. Optionally, the WinNC can also write the measured coordinate into a zero position or preset table.

1. The WinNC positions the scanning system in rapid traverse and with positioning logic at the programmed scanning point **1**. For this, the WinNC will offset the scanning system by the safety distance from the programmed scanning direction.
2. The scanning system travels to a set measuring height and determines the actual position by simple scanning.
3. Subsequently, the WinNC positions the scanning system at the safe height and processes the determined reference point depending on the parameters Q303 and Q305.

Note:
Before the cycle definition, a tool call for the definition of the scanning system axis must be programmed.

If the cycle 419 is used repeatedly and consecutively in order to save the reference point in the preset table in several axes, you have to activate the preset number after every execution of cycle 419 into which cycle 419 wrote before.
This is not necessary if you are overwriting the active preset value.

TCH PROBE CALIBRATING THE SCANNING SYSTEM LENGTH (cycle 461)
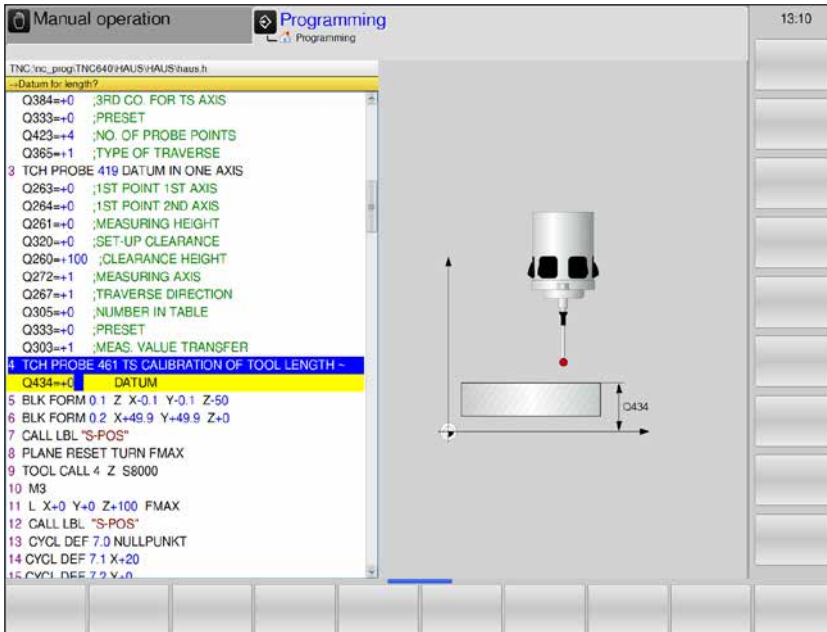
Parameters	Description	Unit
Q434	Reference point (absolute): Reference for the length (e.g. height of the adjustment ring)	mm

Cycle description

Before the calibration cycle is started, the reference point in the spindle axis must be set in such a way that Z=0 on the machine table and the scanning system is prepositioned above the calibration ring.

1 The WinNC orients the scanning system on the angle CAL_ANG from the scanning system table (only if TS can be oriented).
2 From the current position, the WinNC scans with a scanning infeed in the negative spindle direction.
3 After that, WinNC positions the scanning system back to the starting point in rapid traverse.

°)
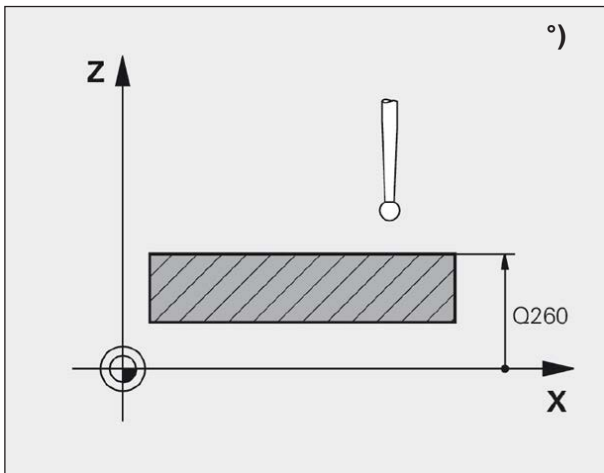
> **Note:**
> The effective length of the scanning system always refers to the took reference point. Usually, the machine manufacturer will set the tool reference point on the spindle nose.
>
> Before the cycle definition, a tool call for the definition of the scanning system axis must be programmed.

emco

TCH PROBE CALIBRATING THE INNER RADIUS OF THE SCANNING SYSTEM (cycle 462)
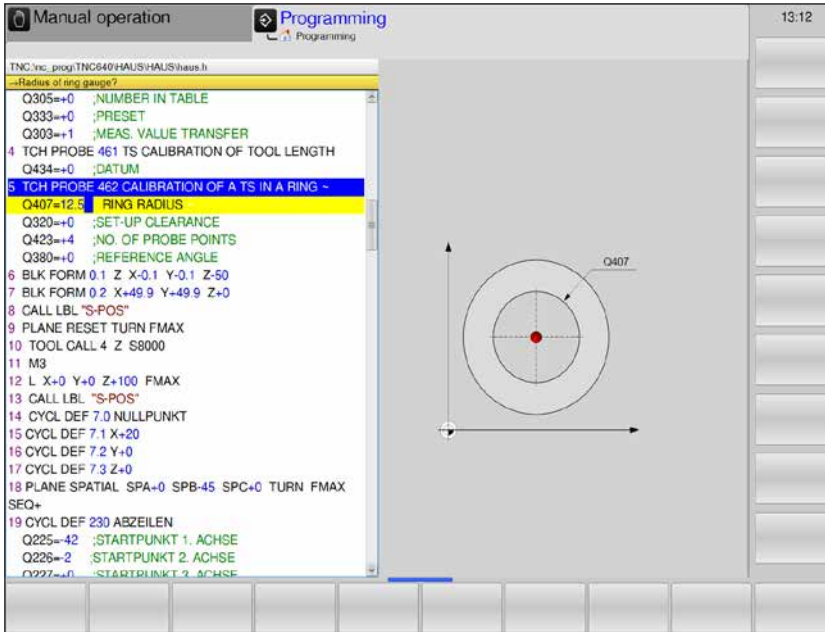
Parameters	Description	Unit
Q407	Ring radius: Adjustment ring diameter.	mm
Q320	Safety distance (incremental): Additional distance between the measuring point and the scanning system ball. Acts additionally to SET_UP of the scanning system table.	mm
Q423	Number of scans (absolute): Number of measuring points on the diameter.	
Q380	Reference angle (absolute): Angle between the main axis of the machining plane and the first scanning point.	Degrees
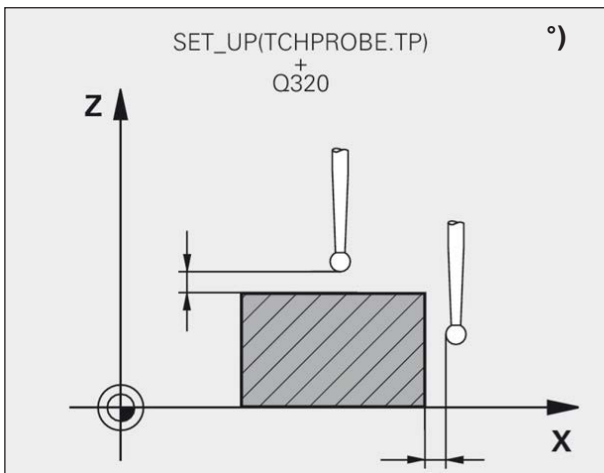
Cycle description

When calibrating the scanning ball radius, WinNC will execute an automatic scanning routine.

Note:
Before the cycle definition, a tool call for the definition of the scanning system axis must be programmed.

D84

TCH PROBE CALIBRATING THE SCANNING SYSTEM TT (cycle 480)

Parameters	Description	Unit
Q260	Safe height: Enter a position on the spindle axis at which any collision with work pieces or clamping devices cannot occur. Safe height refers to the active work piece reference point. If the safe height is set to be so small that the tool tip would be below the upper plate rim, the WinNC will automatically position the calibration tool above the plate.	mm

Note:
Before calibration, write the exact radius and the exact length of the calibration tool into the tool table TOOL.T.

Cycle description

The measuring system TT is calibrated with the cycle TCH PROBE 480.

The calibration process takes place automatically.

Use an exactly cylindrical part (cylinder pin) as the calibration tool. The WinNC stores the calibration values and takes them into account for subsequent tool measurements.

TOUCH PROBE

481

TCH PROBE MEASURING THE TOOL LENGTH (cycle 481)

Parameters	Description	Unit
Q340	Measuring / inspecting a tool: • 0 = measure • 1 = inspect Determine whether the tool is measured for the first time or whether you plan to inspect a previously measured tool. For any first measuring, WinNC overwrites the tool length L in the tool table TOOL.T and sets the delta value DL to 0. If you plan to inspect a tool, the length measured is compared to the tool length L from TOOL.T. The controls then calculate the deviation with the correct sign and input it in TOOL.T as the delta value DL. Additionally, the deviation is also available in the Q parameter Q115.	mm
Q260	Safe height: Enter a position on the spindle axis at which any collision with work pieces or clamping devices cannot occur. Safe height refers to the active work piece reference point. If the safe height is set to be so small that the tool tip would be below the upper plate rim, the WinNC will automatically position the calibration tool above the plate.	mm
Q341	Blade measurement: determines whether an individual blade measurement is conducted. • 0 = No	

Note:
Before tools are measured for the first time, an approximate radius, length and the cutting direction of the respective tool must be input into the tool table TOOL.T.

Cycle description

If the tool diameter is larger than the diameter of the measuring plane of the measuring scanner TT, measure with a rotating tool.

If the tool diameter is smaller than the diameter of the measuring plane of the measuring scanner TT, or if you are determining the length of drills or radius cutters, then measure with an idle tool.

Procedure for a measurement with rotating tool:
In order to determine the longest blade, the tool to be measured is driven with an offset to the scanning system centre and rotates on the measuring plane of the TT. Programme the offset into the tool table in the tool offset field: Radius (TT:R_OFS).

Procedure for a measurement with idle tool:
The tool to be measured is driven to the centre of the measuring plane. Then it is moved to the measuring plane of the TT with an upright spindle. For this measurement, enter the tool offset: Radius (TT:R_OFF) as 0 in the tool table.

TCH PROBE MEASURING THE TOOL RADIUS (cycle 482)

Parameters	Description	Unit
Q340	Measuring / inspecting a tool: • 0 = measure • 1 = inspect Determine whether the tool is measured for the first time or whether you plan to inspect a previously measured tool. For any first measuring, WinNC overwrites the tool radius R in the tool table TOOL.T and sets the delta value DR to 0. If you plan to inspect a tool, the radius measured is compared to the tool radius R from TOOL.T. The controls then calculate the deviation with the correct sign and input it in TOOL.T as the delta value DR. Additionally, the deviation is also available in the Q parameter Q116. Parameter no. for the result: • 0 = Tool within the tolerance limits • 1 = Tool is worn (RTOL exceeded) • 2 = Tool is broken (RBREAK exceeded)	mm
Q260	Safe height: Enter a position on the spindle axis at which any collision with work pieces or clamping devices cannot occur. Safe height refers to the active work piece reference point. If the safe height is set to be so small that the tool tip would be below the upper plate rim, the WinNC will automatically position the calibration tool above the plate.	mm
Q341	Blade measurement: determines whether an individual blade measurement is conducted. • 0 = No	

Note:
Before tools are measured for the first time, an approximate radius, length and the cutting direction of the respective tool must be input into the tool table TOOL.T.
Cylinder-shaped tools with diamond surfaces can be measured with the spindle upright.

Cycle description

- Measurement with rotating tool

The WinNC positions the tool to be measured at the side of the scanning head. The cutter front is below the upper scanning head edge as determined in offsetToolAxis. WinNC scans radially with rotating tool.

emco

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
- 204 Back boring
- 205 Universal deep hole drilling
- 208 Drill milling
- 206 Thread tapping new
- 207 Rigid tapping
- 209 Thread tapping chips crushing
- 233 Face milling
- 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

- 251 Rectangular pocket
- 252 Circular pocket
- 253 Slot milling
- 254 Circular slot
- 256 Rectangular stud
- 257 Circular stud
- 233 Face milling

Converting the coordinates

COORD. TRANSF.

- 7 Zero offset
- 8 Mirroring
- 10 Rotation
- 11 Scaling
- 19 Working plane
- 247 Datum setting

SL Cycles

SL CYCLES

- 14 Contour geometry
- 20 Contour data
- 21 Pilot drilling
- 22 Rough-out
- 23 Floor finishing
- 24 Side finishing
- 25 Contour train
- 27 Cylinder surface

emco

Dot patterns

PATTERN

- 220 Pattern on circle
- 221 Pattern on line

Special cycles

SPECIAL
CYCLES

- 9 Dwell time
- 12 PGM CALL
- 13 Orientation
- 225 Engraving

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

emco
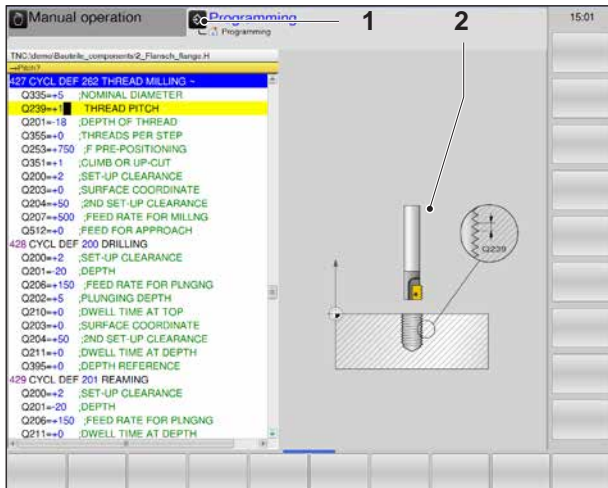
Working with cycles

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
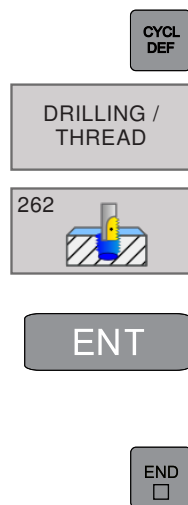
- Retrieve the softkey bar for the different cycle groups.
- Select the cycle group: DRILLING/THREAD
- Select cycle: 262 THREAD MILLING.
  WinNC opens a dialogue with the query for all input values from (1). On the right side of the screen, the control unit displays a diagram (2), where the parameters 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 ;2. 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
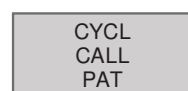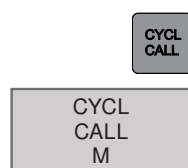
Cycle call with CYCL CALL PAT

*   The CYCL CALL PAT function calls the last defined machining cycle at all positions that were defined in a PATTERN DEF pattern definition or in a point table.

Cycle activation with CYCL CALL POS

```
CYCL
CALL
POS
```

- The CYCL CALL POS function activates the last defined cycle once. The starting point of the cycle is the position, which has been defined in the CYCL CALL POS block.
  WinNC moves to the position specified in the CYCL CALL POS block with positioning logic:

- If the current tool position in the tool axis is greater than the upper edge of the workpiece (Q203), then WinNC first positions the tool in the machining plane to the programmed position and then in the tool axis.

- If the current tool position in the tool axis is below the upper edge of the workpiece (Q203), then WinNC first positions the tool in the tool axis to the safe height and then in the machining plane to the programmed position.

> **Note:**
> Three coordinate axes must always be programmed in the CYCL CALL POS block. The start position can be easily changed via the coordinate in the tool axis. This acts as an additional zero point shift.
> The in-feed defined in the CYCL CALL POS block is only valid for moving to the starting position programmed in that block.
> WinNC always moves to the position defined in the CYCL CALL POS block with inactive radius correction (R0).
> If you use CYCL CALL POS to activate a cycle, in which a starting position is defined (e.g. cycle 212), the position defined in the cycle acts as an additional shift to the position defined in the CYCL CALL POS block. The start position to be defined in the cycle should therefore always be defined as 0.

Drilling / thread

DRILLING /
THREAD

- 200 Drilling
- 201 Grinding
- 202 Turning out
- 203 Universal drilling
- 204 Back boring
- 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

emco
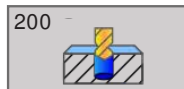
| CYCL DEF | DRILLING / THREAD | 200 | | **DRILLING (cycle 200)** |

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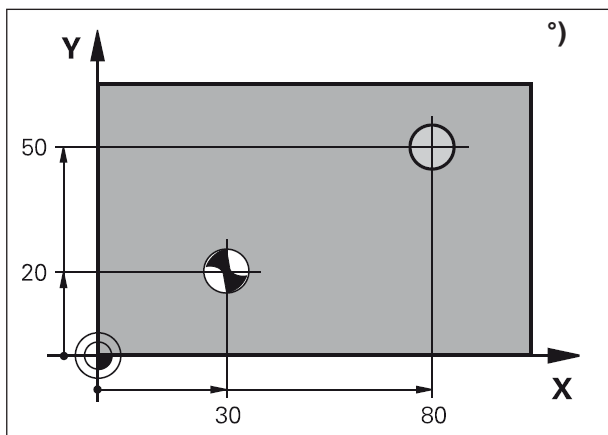
Note:
Prior to programming, please take note:
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.
A negative sign means:
Machining in the direction of the negative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

| CYCL DEF | DRILLING / THREAD | 201 | **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.
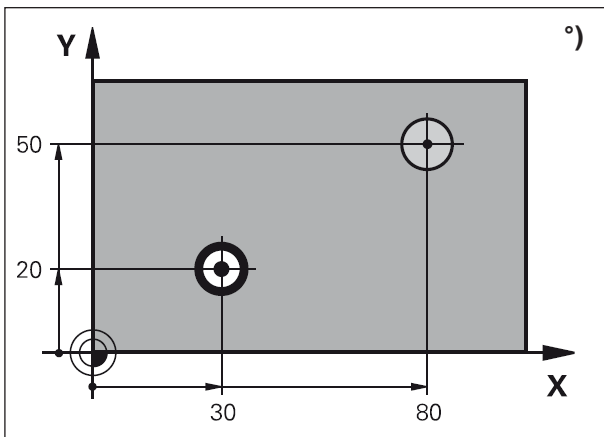
Note:
Prior to programming, please take note:
Program the positioning block on the starting point (hole centre) of the machining plane with radius compensation R0. The sign of the cycle parameter depth determines the working direction.
A negative sign means:
Machining in the direction of the negative spindle axis.
If depth = 0 is programmed, the cycle will not be executed.

emco

| CYCL DEF | DRILLING / THREAD | 202 | **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) • **0 =** Do not clear tool • **1 =** Clear tool in minus direction of the main axis • **2 =** Clear tool in minus direction of the minor axis • **3 =** Clear tool in plus direction of the main axis • **4 =** Clear tool in plus direction of the minor axis	
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
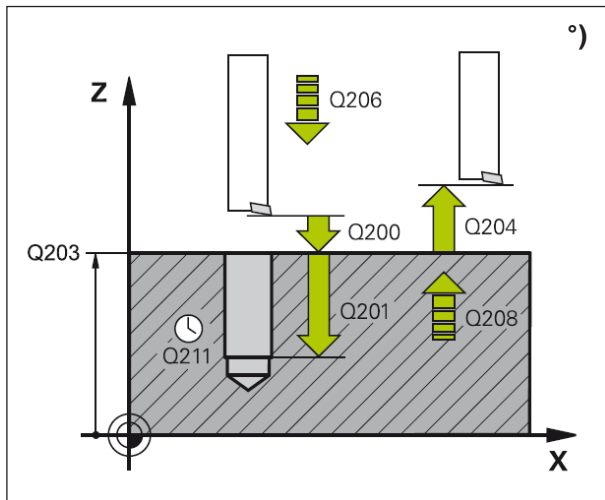
Cycle description

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.
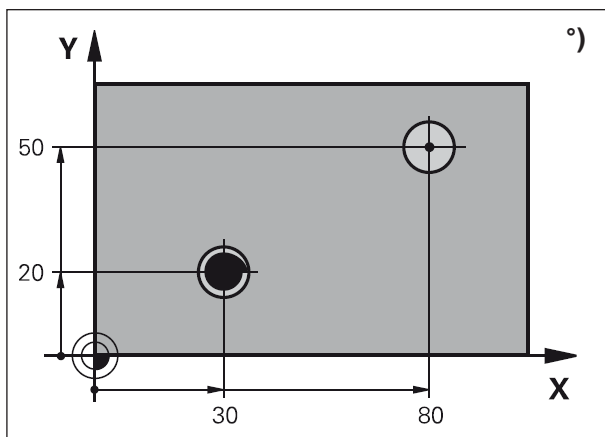
Danger of collision:

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.
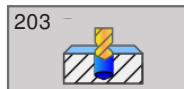
| CYCL DEF | DRILLING / THREAD | 203 | **UNIVERSAL DRILLING (cycle 203)** |

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: • infeed depth and depth are identical • infeed depth is bigger than depth	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.

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.

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.
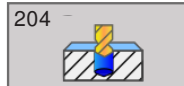
Parameter	Description	Unit
Q200	Set-up clearance (incremental): Distance between tool tip and work-piece surface	mm
Q249	Depth of counterbore (incremental): Distance between underside of workpiece and the top of the hole. A positive sign means the hole will be bored in the positive spindle axis direction	mm
Q250	Material thickness (incremental): Thickness of the workpiece	mm
Q251	Off-center distance (inkremental): Off-center distance for the boring bar (value from tool data sheet)	mm
Q252	Schneidenhöhe (incremental): Distance between the underside of the boring bar and the main cutting tooth (value from tool data sheet)	mm
Q253	Feed rate for pre-positioning: raversing speed of the tool in mm/min when plunging into the workpiece, or when retracting from the work-piece, alternatively FMAX, FAUTO	mm/min
Q254	Feed rate for back boring: Traversing speed of the tool during back boring in mm/min. alternatively FAUTO, FU	mm/min
Q255	Dwell time: Dwell time in seconds at the top of the bore hole	s
Q203	Coordinate of workpiece surface (absolute)	
Q204	2nd set-up clearance (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur	mm

Parameter	Description	Unit
Q214	Disengaging direction (1/2/3/4): Determine the direction in which the TNC displaces the tool by the off-center distance (after spindle orientation); programming 0 is not allowed. • 1 = Retract the tool in minus direction of the principle axis • 2 = Retract the tool in minus direction of the minor axis • 3 = Retract the tool in plus direction of the principle axis • 4 = Retract the tool in plus direction of the minor axis	
Q336	Angle for spindle orientation (absolute): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole.	degrees
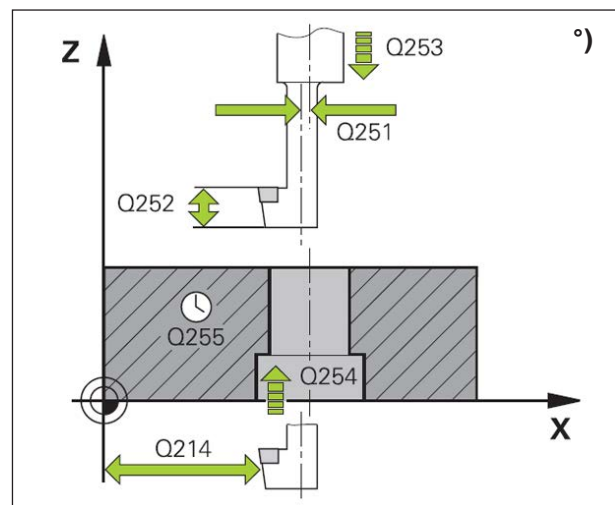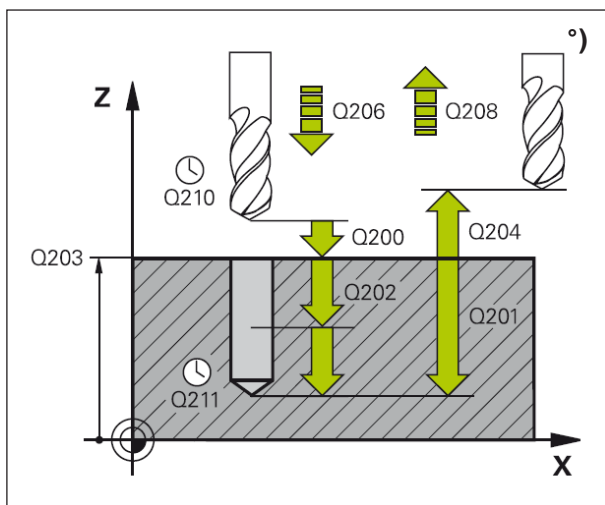
Cycle description

This cycle allows holes to be bored from the underside of theworkpiece.

1. The TNC positions the tool in the tool axis at rapid traverse FMAX to set-up clearance above the workpiece surface.
2. The WinNC then orients the spindle to the 0° position with an oriented spindle stop and displaces the tool by the off-center distance.
3. The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached set-up clearance on the underside of the workpiece.
4. The WinNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.
5. If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. The WinNC carries out another oriented spindle stop and the

tool is once again displaced by the off-center distance.

6. The tool then retracts to set-up clearance at the feed rate for pre-positioning, and from there (if programmed) to the 2nd set-up clearance at FMAX.

Note:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.
The entered tool length is the total length to the underside of the boring bar and not just to the tooth. When calculating the starting point for boring, the WinNC considers the tooth length of the boring bar and the thickness of the material.

emco

CYCL DEF	DRILLING / THREAD	205
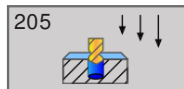
UNIVERSAL DEEP HOLE DRILLING (cycle 205)

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: • infeed depth and depth are identical • infeed depth is bigger than depth	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 WinNC pulls out the tool with feed Q206.	mm/min
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.	
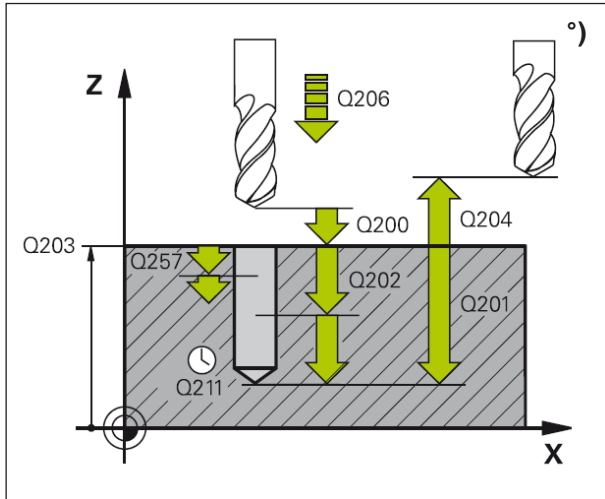
Cycle description

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 with feed for retraction Q208 (or Q208=0) 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.

emco
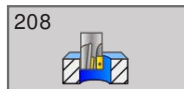
| CYCL DEF | DRILLING / THREAD | 208 | **DRILL MILLING (cycle 208)** |

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	
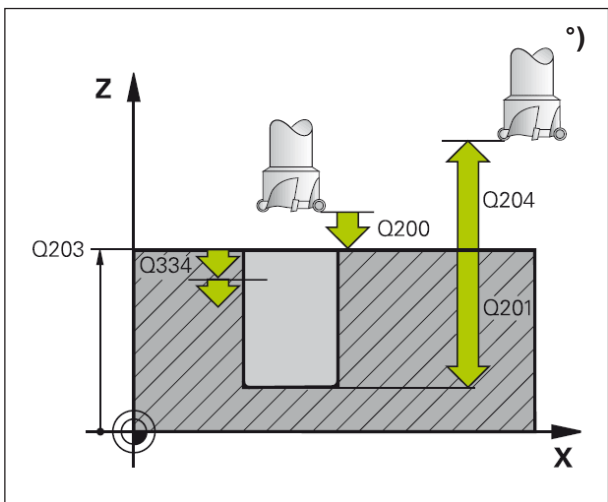
Cycle description

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.

CYCL DEF	DRILLING / THREAD	206	

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.
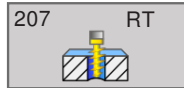
CYCL DEF	DRILLING / THREAD	207 RT	RIGID TAPPING (cycle 207)

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.

emco

CYCL DEF | DRILLING / THREAD | 209 RT

THREAD TAPPING CHIPS CRUSHING (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.	
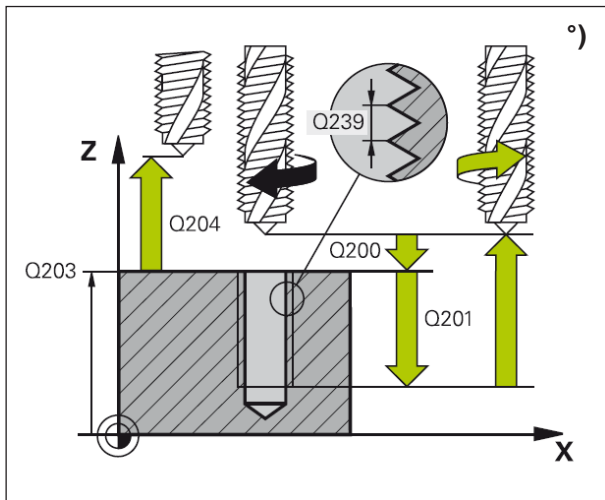
Cycle description

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.
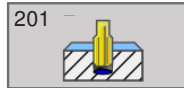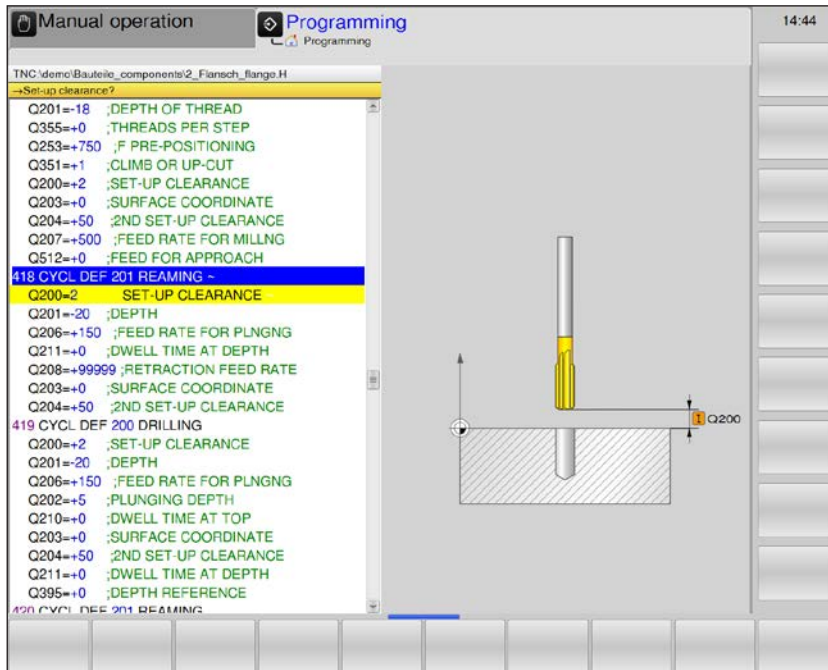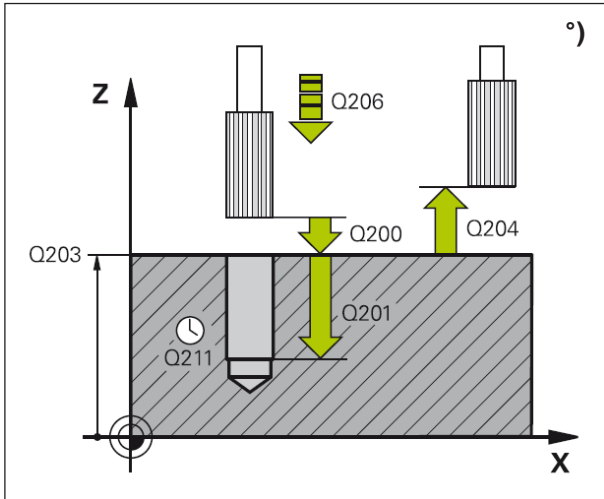
emco
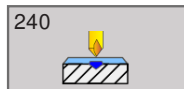
| CYCL DEF | DRILLING / THREAD | 240 | **CENTRING (cycle 240)** |

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. • 0 = centring relative to the entered depth • 1 = centring relative to the entered diameter. 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.
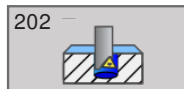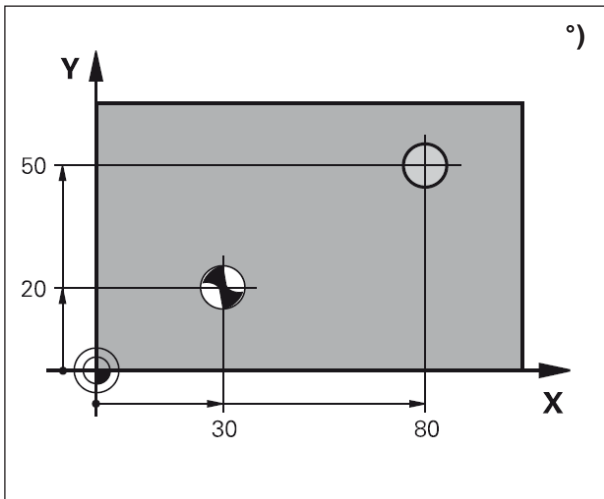
Note:

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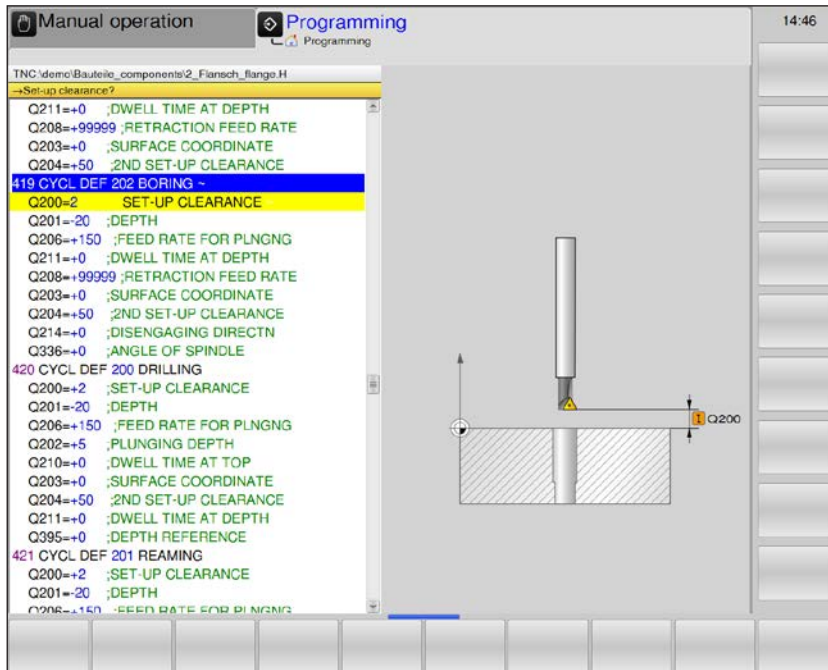A negative sign means:
Machining in the direction of the negative spindle axis.
If depth = 0 is programmed, the cycle will not be executed.

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.

CYCL DEF	DRILLING / THREAD	262

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.

CYCL DEF	DRILLING / THREAD	263

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.

emco

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 semicircle 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.

| CYCL DEF | DRILLING / THREAD | 264 | | 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.

emco

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.

CYCL DEF	DRILLING / THREAD	265

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: • + = Right-hand thread • - = Left-hand thread	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 • 0 = prior to the thread machining • 1 = after the thread machining	
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

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 semicircle 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.

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.

| CYCL DEF | DRILLING / THREAD | 267 | **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.
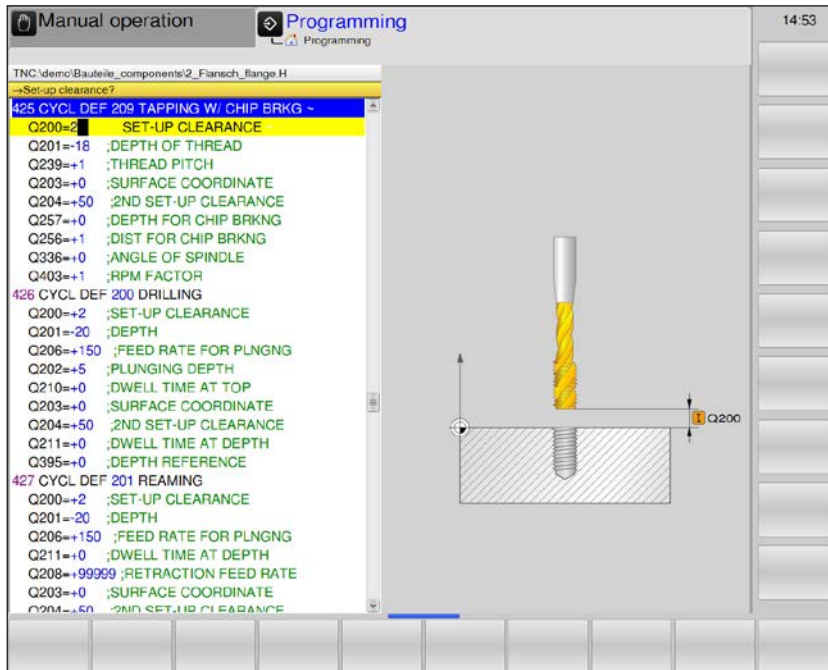The working direction is determined according to the following order:
1. Thread depth
2nd Front side depth
If depth = 0 is programmed, the cycle will not be executed.
The sign of the cycle parameter thread depth determines the working direction.
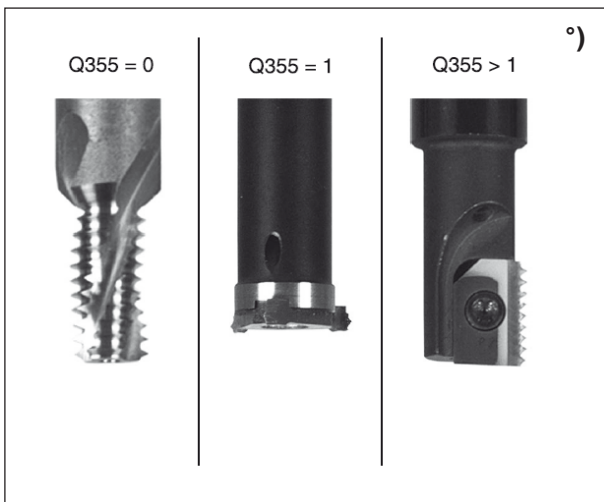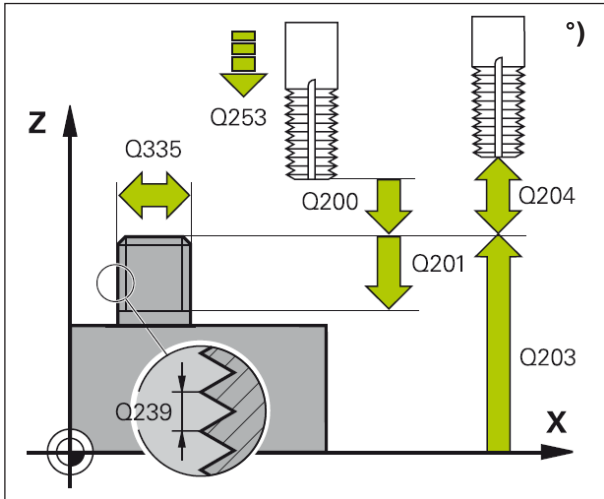
emco

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.

POCKETS /
STUDS /
SLOTS

Pockets / studs / slots

- 251 Rectangular pocket
- 252 Circular pocket
- 253 Slot milling
- 254 Circular slot
- 256 Rectangular stud
- 257 Circular stud
- 233 Face milling

emco

CYCL DEF	POCKETS / STUDS / SLOTS	251
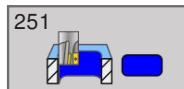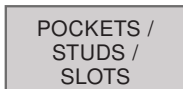
RECTANGULAR POCKET
(cycle 251)

Parameter	Description	Unit
Q215	Scope of machining: • 0 = Roughing and finishing • 1 = Roughing only • 2 = Finishing only	mm
Q218	1. Side length: Length of pocket, parallel to the main axis of the machining plane (incremental)	mm
Q219	2. Side length: Length of pocket, parallel to the minor axis of the machining plane	mm
Q220	Corner radius of the pocket corner. With input 0: Corner radius = tool radius	mm
Q368	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q224	Turning position (absolute): Angle, about which the whole machining is turned. The centre of rotation is in the position, in which the tool is when the cycle is initiated.	
Q367	Pocket position: relative to the position of the tool when the cycle is initiated: • 0 = Pocket centre • 1 = Lower left corner • 2 = Lower right corner • 3 = Upper right corner • 4 = Upper left corner	
Q207	Milling feed: Tool traverse speed during milling. Alternatively: F AUTO, FU, FZ	mm/min

Parameter	Description	Unit
Q351	Type of milling: Milling machining at M3: • +1 = Down-cut milling • –1 = Up-cut milling	
Q201	Depth (incremental): Distance work piece surface - pocket bottom.	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed. Enter positive value.	mm
Q369	Finishing allowance for depth: (incremental)	mm
Q206	Feed for depth infeed. Traverse speed during movements into the depth. Alternatively: F AUTO, FU, FZ	mm/min
Q338	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
Q200	Safety distance (incremental): between tip of the tool and the work-piece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2. Safety distance (incremental): Coordinate of the spindle axis, in which no collision between the tool and the work piece can occur.	mm
Q370	Path overlap factor: Tool radius gives the side infeed k. Input range from 0.1 to 1.414	mm
Q366	Plunging strategy: • 0 = Plunge vertically, regardless of the angle defined in the tool table • 1 = Plunge in a helical motion. The plunging angle for the active tool must not be equal to 0 in the tool table. • 2 = Plunge in a reciprocating motion. The plunging angle for the active tool must not be equal to 0 in the tool table. The reciprocation length is independent of the plunging angle; WinNC uses the double tool diameter as the minimum value.	
Q385	Traverse speed for side and depth finishing. Alternatively: F AUTO, FU, FZ	mm/min
Q439	Feed reference: determines, what the programmed feed refers to: • 0 = Feed refers to the midpoint path of the tool. • 1 = Feed refers to the tool cutting edge only for side finishing, otherwise, it refers to the midpoint path. • 2 = Feed refers to the tool cutting edge only for side finishing and depth finishing, otherwise, it refers to the midpoint path. • 3 = Feed always refers to the tool cutting edge.	

Note:
If the tool table is inactive, plunging is always vertical, because no plunging angle can be defined.

Pre-position the tool at the starting position in the machining position with R0.

Observe Q367 (position).

WinNC automatically positions the tool before the 2nd safety distance.

The sign of the cycle parameter for depth determines the working direction. If depth = 0, the cycle will not be executed.

At the end of the cycle, the tool is moved back to the start position.

At the end of a roughing operation, the tool is moved to the pocket center in rapid traverse. The tool is one safety clearance above the current infeed depth. Enter the safety distance such that the tool cannot jam with removed chips during traverse.

When plunging with a helix, WinNC issues an error message if the calculated helix has a diameter that is less than two diameters of the tool.

WinNC reduces the infeed depth to the LCUTS cutting length defined in the tool table, for the case that the cutting length is shorter than the Q202 infeed depth set for the cycle.

Cycle description

Cycle 251 has the following machining alternatives:
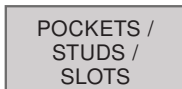
- Complete machining: Roughing, depth finishing, side finishing
- Roughing only
- Side and depth finishing only
- Depth finishing only
- Side finishing only

Roughing

1 The tool plunges into the work piece in the pocket depth and moves to the first infeed depth. The plunging strategy is determined using parameter Q366.
2 WinNC clears the pocket from the inside out, taking into account overlap factor Q370 and finishing allowances Q368 and Q369.
3 At the end of the roughing operation, WinNC moves away from the pocket wall, traversing by the safety distance over the current infeed depth, and from there back to the pocket center in rapid traverse.
4 The operation is repeated until the programmed pocket depth is achieved.

Finishing

5 If finishing allowances are defined, the tool cuts into the work piece in the pocket centre and then moves to the finishing infeed depth. WinNC finishes the pocket walls first - in several infeeds, if specified. In doing so, te pocket wall is approached tangentially.
6 Finally, WinNC finishes the pocket floor from the inside out. In doing so, the pocket floor is approached tangentially.
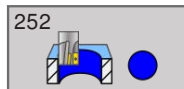
CIRCULAR POCKET (cycle 252)
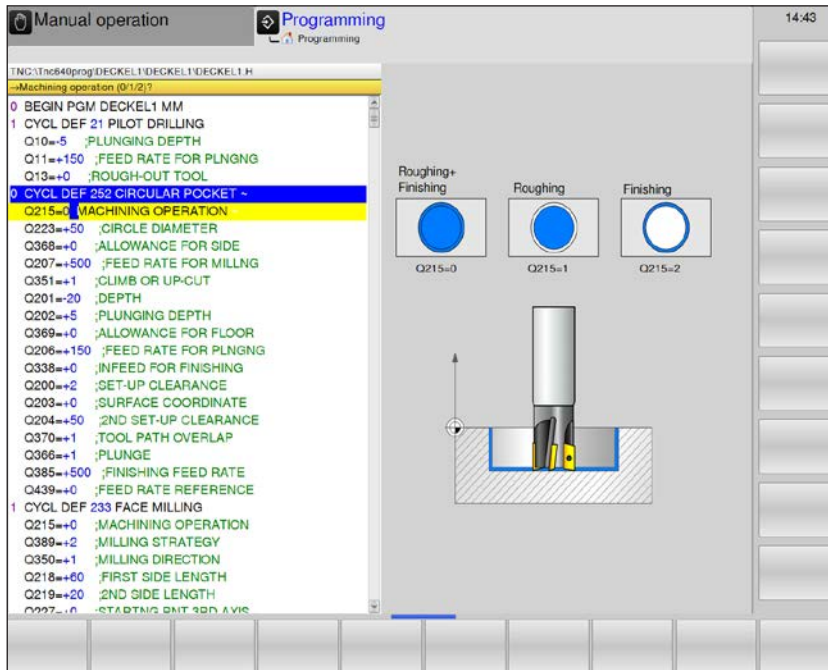
Parameter	Description	Unit
Q215	Scope of machining: • 0 = Roughing and finishing • 1 = Roughing only • 2 = Finishing only	
Q223	Circle diameter of the finished pocket	mm
Q368	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q207	Milling feed: Tool traverse speed during milling. Alternatively: F AUTO, FU, FZ	mm/min
Q351	Type of milling: Milling machining at M3: • +1 = Down-cut milling • -1 = Up-cut milling	
Q201	Depth (incremental): Distance work piece surface - pocket bottom.	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed. Enter positive value.	mm
Q369	Finishing allowance for depth: (incremental)	mm
Q206	Feed for depth infeed. Traverse speed during movements into the depth. Alternatively: F AUTO, FU, FZ	mm/min
Q338	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
Q200	Safety distance (incremental): between tip of the tool and the work-piece surface.	mm

Parameter	Description	Unit
Q203	Work piece surface coordinates (absolute)	mm
Q204	2. Safety distance (incremental): Coordinate of the spindle axis, in which no collision between the tool and the work piece can occur.	mm
Q370	Path overlap factor: Tool radius gives the side infeed k. Input range from 0.1 to 1.414	mm
Q366	Plunging strategy: • 0 = Plunge vertically, regardless of the angle defined in the tool table. The plunging angle for the active tool must be set to a value of 0 or 90 in the tool table. • 1 = Plunge in a helical motion. The plunging angle for the active tool must not be equal to 0 in the tool table.	
Q385	Finishing feed: Traverse speed for side and depth finishing. Alternatively: F AUTO, FU, FZ	mm/min
Q439	Feed reference: determines, what the programmed feed refers to: • 0 = Feed refers to the midpoint path of the tool. • 1 = Feed refers to the tool cutting edge only for side finishing, otherwise, it refers to the midpoint path. • 2 = Feed refers to the tool cutting edge only for side finishing and depth finishing, otherwise, it refers to the midpoint path. • 3 = Feed always refers to the tool cutting edge.	

Note:
If the tool table is inactive, plunging is always vertical, because no plunging angle can be defined. Q366=0
Pre-position the tool at the starting position (= circle centre) in the machining position with R0.
WinNC automatically positions the tool before the 2nd Safety distance. Observe Q204.
The sign of the cycle parameter for depth determines the working direction. If depth = 0, the cycle will not be executed.
At the end of the cycle, the tool is moved back to the start position.
At the end of a roughing operation, the tool is moved to the pocket center in rapid traverse. The tool is one safety clearance above the current infeed depth. Enter the safety distance such that the tool cannot jam with removed chips during traverse.
When plunging with a helix, WinNC issues an error message if the calculated helix has a diameter that is less than two diameters of the tool.
WinNC reduces the infeed depth to the LCUTS cutting length defined in the tool table, for the case that the cutting length is shorter than the Q202 infeed depth set for the cycle.

emco

Cycle description

Cycle 252 has the following machining alternatives:

- Complete machining: Roughing, depth finishing, side finishing
- Roughing only
- Side and depth finishing only
- Depth finishing only
- Side finishing only

Roughing

1. WinNC moves the tool in rapid traverse to the safety distance above the workpiece.
2. The tool plunges in the pocket centre by the infeed depth value. The plunging strategy is determined using parameter Q366.
3. WinNC clears the pocket from the inside out, taking into account overlap factor Q370 and finishing allowances Q368 and Q369.
4. At the end of the roughing operation, WinNC moves the tool in the machining plane tangentially a safety distance away from the pocket wall, raises the tool by the safety distance in rapid traverse, and moves it from there back to the pocket centre in rapid traverse.
5. Steps 2-4 are repeated until the programmed pocket depth is achieved. The Q369 finishing allowance is thus observed.
6. If only roughing is programmed (Q215=1), the tool moves tangentially away from the pocket wall maintaining the safety distance, rapidly rises in the tool axis to the 2nd safety distance, and rapidly traverses back to the pocket centre.

Finishing

1 If finishing allowances are defined, WinNC finishes the pocket walls first; in several infeeds, if specified.

2 WinNC moves the tool in the tool axis to the position distant from the pocket wall by the finishing allowance and the safety distance.

3 WinNC clears the pocket from the inside out to diameter Q223.

4 Then, WinNC sets the tool in position in the tool axis, ensuring that it is a finishing allowance and safety distance away from the pocket wall, and carries out the side wall finishing operation again at the new depth.

5 WinNC repeats the operation until the programmed diameter is achieved.

6 Once the diameter Q223 has been machined, the tool returns tangentially by the finishing allowance plus the safety distance in the machining plane and moves at rapid traverse in the tool axis to the safety distance and then to the centre of the pocket.

7 Finally, the tool moves in the tool axis to depth Q201, and finishes the floor of the pocket from the inside out. The pocket floor is approached tangentially.

8 WinNC repeats this sequence until depth Q201 plus finishing allowance Q369 are achieved.

9 Finally, the tool moves tangentially a safety distance away from the pocket wall, rapidly rises in the tool axis to achieve the safety distance, and then rapidly traverses back to the pocket centre.
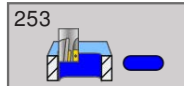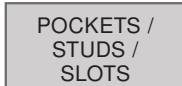
CYCL DEF	POCKETS / STUDS / SLOTS	253	**SLOT MILLING (cycle 253)**

Parameter	Description	Unit
Q215	Scope of machining: • 0 = Roughing and finishing • 1 = Roughing only • 2 = Finishing only Side and depth finishing are only carried out if the corresponding finishing allowance is defined (see Q368, Q369)	
Q218	Slot length: specify longer side of the slot. Value parallel to the main axis of the machining plane.	mm
Q219	Slot width: If the value of the slot width is equal to the tool diameter, only roughing will be performed (oblong slot milling). The maximum slot width when roughing is equal to two tool diameters. Value parallel to the minor axis of the machining plane.	mm
Q368	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q374	Turning position (absolute): Angle, about which the whole slot is turned. The centre of rotation is in the position, in which the tool is when the cycle is initiated. Value: -360 to +360	°
Q367	Reference for slot position: Position of the slot relative to the position of the tool when the cycle is initiated: • 0 = Slot centre • 1 = Left end of slot • 2 = Centre of left slot circle • 3 = Centre of right slot circle • 4 = Right end of slot	

Parameter	Description	Unit
Q207	Milling feed: Tool traverse speed during milling. Alternatively FAUTO, FU, FZ	mm/min
Q351	Type of milling: Milling machining at M3: • +1 = Down-cut milling • -1 = Up-cut milling	
Q201	Depth (incremental): Distance work piece surface - slot bottom.	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed. Enter positive value.	mm
Q369	Finishing allowance for depth: (incremental)	mm
Q206	Feed for depth infeed. Traverse speed during movements into the depth. Alternatively: F AUTO, FU, FZ	mm/min
Q338	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
Q200	Safety distance (incremental): between tip of the tool and the work-piece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2. Safety distance (incremental): Coordinate of the spindle axis, in which no collision between the tool and the work piece can occur.	mm
Q366	Plunging strategy: • 0 = Plunge vertically. The ANGLE plunging angle is not assessed in the tool table • 1,2 = Plunge in a reciprocating motion. The plunging angle for the active tool must not be equal to 0 in the tool table.	
Q385	Finishing feed: Traverse speed for side and depth finishing. Alternatively: F AUTO, FU, FZ	mm/min
Q439	Feed reference: determines, what the programmed feed refers to: • 0 = Feed refers to the midpoint path of the tool. • 1 = Feed refers to the tool cutting edge only for side finishing, otherwise, it refers to the midpoint path. • 2 = Feed refers to the tool cutting edge only for side finishing and depth finishing, otherwise, it refers to the midpoint path. • 3 = Feed always refers to the tool cutting edge.	

Note:
If the tool table is inactive, plunging is always vertical, because no plunging angle can be defined.
Pre-position the tool at the starting position in the machining position with R0.
Observe Q367 (position).
WinNC automatically positions the tool before the 2nd safety distance.
At the end of the cycle, WinNC moves the tool in the machining plane back to the slot centre; on the other axis of the machining plane, the controller does not carry out any positioning. If the slot position is set to a value not equal to 0, WinNC positions the tool in the tool axis, at the 2nd safety distance. Before a new cycle call, move the tool back to the starting position or always program absolute movements after the cycle call.
The sign of the cycle parameter for depth determines the working direction. If depth = 0, the cycle will not be executed.
If the slot width is more than two tool diameters, then WinNC clears the slot from the inside out. Thus, even small tools can be used to mill any slots. WinNC reduces the infeed depth to the LCUTS cutting length defined in the tool table, for the case that the cutting length is shorter than the infeed depth set for the cycle.
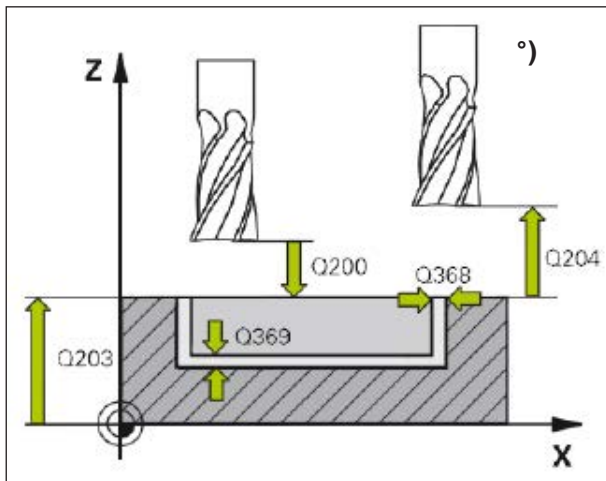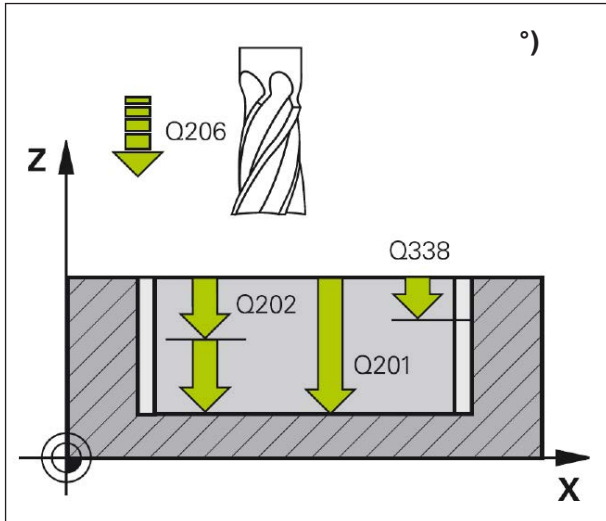
Cycle description

Cycle 253 has the following machining alternatives:

- Complete machining: Roughing, depth finishing, side finishing
- Roughing only
- Side and depth finishing only
- Depth finishing only
- Side finishing only

Roughing

1 The tool reciprocates to the first infeed depth from the left slot circle centre point at the plunging angle defined in the tool table. The plunging strategy is determined using Q366.

2 WinNC clears the slot from the inside out, taking into account finishing allowances Q368 and Q369.

3 WinNC moves the tool back by the safety distance Q200. If the slot width matches the mill diameter, WinNC positions the tool out of the slot after every infeed.

4 The operation is repeated until the programmed slot depth is achieved.

Finishing

5 If the finishing allowances are defined, WinNC finishes the slot walls first; in several infeeds, if specified. In doing so, the slot wall is approached tangentially in the left slot circle.

6 Finally, WinNC finishes the slot floor from the inside out.

CYCL DEF	POCKETS / STUDS / SLOTS	254	CIRCULAR SLOT (cycle 254)

Parameter	Description	Unit
Q215	Scope of machining: • 0 = Roughing and finishing • 1 = Roughing only • 2 = Finishing only Side and depth finishing are only carried out if the corresponding finishing allowance is defined (see Q368, Q369)	
Q219	Slot width: If the value of the slot width is equal to the tool diameter, only roughing will be performed (oblong slot milling). The maximum slot width when roughing is equal to two tool diameters. Value parallel to the minor axis of the machining plane.	mm
Q368	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q375	Pitch circle diameter	mm
Q367	Reference for slot position: Position of the slot relative to the position of the tool when the cycle is initiated: • 0 = Tool position, the slot position results from the entered pitch circle centre and start angle • 1 = Tool position = left slot circle centre. Start angle Q376 refers to this position. The entered pitch circle centre is not taken into account • 2 = Tool position = middle axis centre. Start angle Q376 refers to this position. The entered pitch circle centre is not taken into account • 3 = Tool position = right slot circle centre. Start angle Q376 refers to this position. The entered pitch circle centre is not taken into account	

Parameter	Description	Unit
Q216	Centre 1st Axis (absolute): Centre of the pitch circle in the main axis of the machining plane, only in effect if Q376=0	mm
Q217	Centre 2nd Axis (absolute): Centre of the pitch circle in the minor axis of the machining plane, only in effect if Q376=0	mm
Q376	Start angle (absolute): Polar angle of the starting point. Value -360 to +360	°
Q248	Opening angle (incremental) of the slot. Value 0 to 360	°
Q378	Angle step (incremental): Angle, about which the slot is rotated. The centre of rotation is in the centre of the pitch circle. Value -360 to +360	°
Q377	Number of machining operations on the pitch circle	
Q207	Milling feed: Tool traverse speed during milling. Alternatively FAUTO, FU, FZ	mm/min
Q351	Type of milling: Milling machining at M3: • +1 = Down-cut milling • -1 = Up-cut milling	
Q201	Depth (incremental): Distance work piece surface - slot bottom.	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed. Enter positive value.	mm
Q369	Finishing allowance for depth: (incremental)	mm
Q206	Feed for depth infeed. Traverse speed during movements into the depth. Alternatively: F AUTO, FU, FZ	mm/min
Q338	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
Q200	Safety distance (incremental): between tip of the tool and the work-piece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2. Safety distance (incremental): Coordinate of the spindle axis, in which no collision between the tool and the work piece can occur.	mm
Q366	Plunging strategy: • 0 = Plunge vertically. The ANGLE plunging angle is not assessed in the tool table • 1,2 = Plunge in a reciprocating motion. The plunging angle for the active tool must not be equal to 0 in the tool table.	
Q385	Finishing feed: Traverse speed for side and depth finishing. Alternatively: F AUTO, FU, FZ	mm/min
Q439	Feed reference: determines, what the programmed feed refers to: • 0 = Feed refers to the midpoint path of the tool. • 1 = Feed refers to the tool cutting edge only for side finishing, otherwise, it refers to the midpoint path. • 2 = Feed refers to the tool cutting edge only for side finishing and depth finishing, otherwise, it refers to the midpoint path. • 3 = Feed always refers to the tool cutting edge.	

Note:
If the tool table is inactive, plunging is always vertical, because no plunging angle can be defined.
Pre-position the tool at the starting position in the machining position with R0.
Observe Q367 (position).
WinNC automatically positions the tool before the 2nd safety distance.
At the end of the cycle, WinNC moves the tool in the machining plane back to the starting point (pitch circle centre). Exception: If the slot position is set to a value not equal to 0, WinNC positions the tool only in the tool axis, at the 2nd safety distance. In such cases, only absolute traverse movements may be programmed after the cycle is initiated.
The sign of the cycle parameter for depth determines the working direction. If depth = 0, the cycle will not be executed.
If the slot width is more than two tool diameters, then WinNC clears the slot from the inside out. Thus, even small tools can be used to mill any slots.
If cycle 254 (round slot) is used in combination with cycle 221, slot position 0 is not allowed.
WinNC reduces the infeed depth to the LCUTS cutting length defined in the tool table, for the case that the cutting length is shorter than the infeed depth set for the cycle.
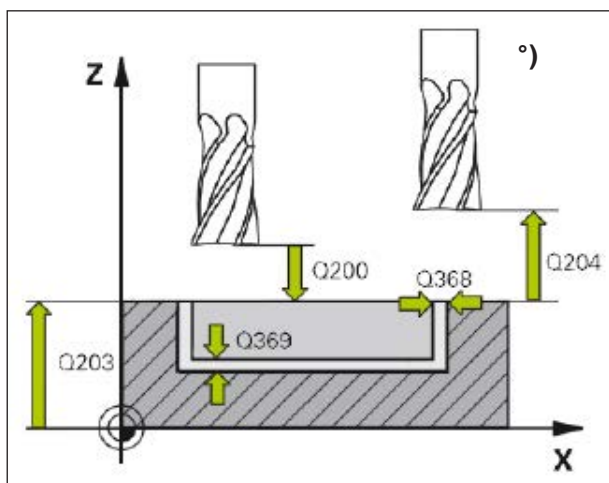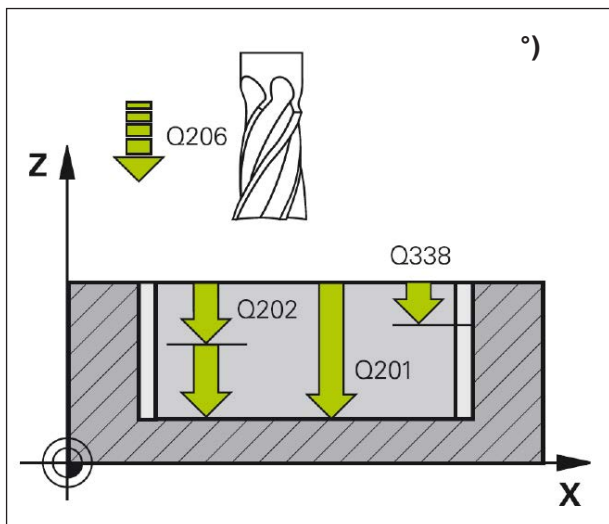
Cycle description

Cycle 254 has the following machining alternatives:

- Complete machining: Roughing, depth finishing, side finishing
- Roughing only
- Side and depth finishing only
- Depth finishing only
- Side finishing only

Roughing

1 The tool reciprocates to the first infeed depth in the slot centre, at the plunging angle defined in the tool table. The plunging strategy is determined using Q366.

2 WinNC clears the slot from the inside out, taking into account finishing allowances Q368 and Q369.

3 WinNC moves the tool back by the safety distance Q200. If the slot width matches the mill diameter, WinNC positions the tool out of the slot after every infeed.

4 The operation is repeated until the programmed slot depth is achieved.

Finishing

5 If the finishing allowances are defined, WinNC finishes the slot walls first; in several infeeds, if specified. In doing so, the slot wall is approached tangentially in the left slot circle.

6 Finally, WinNC finishes the slot floor from the inside out.

CYCL DEF | POCKETS / STUDS / SLOTS | 256

RECTANGULAR STUD (cycle 256)

Parameter	Description	Unit
Q218	1. Side length: Length of stud, parallel to the main axis of the machining plane	mm
Q424	Raw part dimension side length 1: Length of the stud workpiece, parallel to the main axis of the machining plane. Raw part dimension side length 1 greater than 1. Enter side length. WinNC carries out multiple side infeeds, if the difference between raw part dimension 1 and finished dimension 1 is greater than the allowed side infeed (tool radius x path overlap Q370). WinNC always calculates a constant side infeed	mm
Q219	2. Side length: Length of stud, parallel to the minor axis of the machining plane. Raw part dimension side length 2 greater than 2. Enter side length. WinNC carries out multiple side infeeds, if the difference between raw part dimension 2 and finished dimension 2 is greater than the allowed side infeed (tool radius x path overlap Q370). WinNC always calculates a constant side infeed	mm
Q425	Raw part dimension side length 2: Length of the stud workpiece, parallel to the minor axis of the machining plane.	mm
Q220	Corner radius: Radius of the stud corner.	mm
Q368	Side finishing allowance (incremental): Finishing allowance in the machining plane that WinNC leaves there during the milling.	mm
Q224	Turning position (absolute): Angle, by which the whole machining is turned. The centre of rotation is in the position, in which the tool is when the cycle is initiated.	°

Parameter	Description	Unit
Q367	Stud length: Position of the stud relative to the position of the tool when the cycle is initiated: • 0 = Tool position = stud centre • 1 = Tool position = lower left corner • 2 = Tool position = lower right corner • 3 = Tool position = upper right corner • 4 = Tool position = upper left corner	
Q207	Milling feed: Tool traverse speed during milling. Alternatively FAUTO, FU, FZ	mm/min
Q351	Type of milling: Milling machining at M3: • +1 = Down-cut milling • -1 = Up-cut milling	
Q201	Depth (incremental): Distance work piece surface - stud bottom	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed. Enter positive value.	mm
Q206	Feed for depth infeed: Tool traverse speed during milling. Alternatively FAUTO, FU, FZ	mm/min
Q200	Safety distance (incremental): Distance between the tip of the tool and the workpiece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm
Q204	2. Safety distance (incremental): Coordinate of the spindle axis, in which no collision between the tool and the work piece can occur.	mm
Q370	Path overlap factor: the tool radius results in side infeed k	mm
Q437	Approach position: Tool approach strategy • 0 = To the right from the stud (basic position) • 1 = Lower left corner • 2 = Lower right corner • 3 = Upper right corner • 4 = Upper left corner Should approaching with the 0 setting result in approach marks on the stud surface, a different approach position should be chosen.	

Note:
Preposition the tool at the start position in the machining plane using R0, observe parameter Q367.

WinNC automatically positions the tool in the tool axis before the 2nd safety distance.

The sign of the cycle parameter for depth determines the working direction. If the depth = 0, then the cycle is not executed.

WinNC reduces the infeed depth to the LCUTS cutting length defined in the tool table, for the case that the cutting length is shorter than the infeed depth set for the cycle.
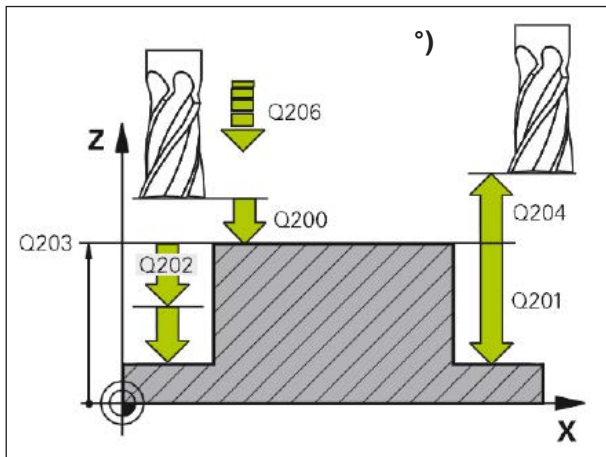
Cycle description

Cycle 256 enables the machining of rectangular studs. If a raw part dimension is greater than the maximum possible side infeed, WinNC executes multiple side infeeds, until the finished dimension is achieved.

1 The tool moves from the cycle starting position (=stud centre) to the starting position for the stud milling. The starting position is determined using parameter Q437. Standard setting Q437=0 is 2 mm to the right of the stud blank.

2 If the tool is at the 2nd safety distance, WinNC moves the tool to the safety distance in rapid traverse, and moves it from there with depth infeed to the first infeed depth.

3 Subsequently, the tool approaches the stud contour tangentially and cuts one revolution.

4 If the finished dimension cannot be achieved in one revolution, WinNC feeds the tool laterally to the current infeed depth, and continues milling for another revolution. The controller accounts for the raw part dimension, the finished dimension and the permissible side infeed. This operation is repeated until the programmed finished dimension is achieved. If the starting point is on a corner (Q437 not equal to 0), WinNC mills in a spiral motion from the starting point inwards, until the finished dimension is achieved.

5 If any additional infeeds are required, the tool tangentially retracts from the contour back to the starting point of the stud machining.

6 Subsequently, WinNC moves the tool to the next infeed depth and machines the stud at that depth.

7 This operation is repeated until the programmed stud depth is achieved.

8 At the end of the cycle, WinNC positions the tool in the tool axis to the safe elevation defined in the cycle. This results in the end position not being equal to the starting position.
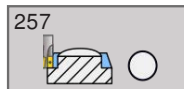
| CYCL DEF | POCKETS / STUDS / SLOTS | 257 | **CIRCULAR STUD (cycle 257)** |

Parameter	Description	Unit
Q223	Finished part diameter: Diameter of the finished stud.	mm
Q222	Raw part diameter: define the raw part diameter bigger than the finished part diameter. WinNC carries out multiple side infeeds if the difference between raw part diameter and finished part diameter is greater than the allowed side infeed (tool radius x path overlap Q370). WinNC always calculates a constant side infeed	mm
Q368	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q207	Milling feed: Tool traverse speed during milling. Alternatively FAUTO, FU, FZ	mm/min
Q351	Type of milling: Milling machining at M3: • +1 = Down-cut milling • –1 = Up-cut milling	
Q201	Depth (incremental): Distance work piece surface - stud bottom	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed. Enter positive value.	mm
Q206	Feed for depth infeed: Tool traverse speed during milling. Alternatively FAUTO, FU, FZ	mm/min
Q200	Safety distance (incremental): Distance between the tip of the tool and the workpiece surface.	mm
Q203	Work piece surface coordinates (absolute)	mm

Parameter	Description	Unit
Q204	2. Safety distance (incremental): Coordinate of the spindle axis, in which no collision between the tool and the work piece can occur.	mm
Q370	Path overlap factor: the tool radius gives the side infeed k. Input 0.1 to 1.414	mm
Q376	Start angle: polar angle, relative to the stud centre point, from which the tool approaches the stud	°

Note:
Pre-position the tool at the start position in the machining plane (stud centre) using R0. WinNC automatically positions the tool in the tool axis before the 2nd Safety clearance, observe Q204.

The sign of the cycle parameter for depth determines the working direction. If the depth = 0, then the cycle is not executed.

WinNC reduces the infeed depth to the LCUTS cutting length defined in the tool table, for the case that the cutting length is shorter than the infeed depth set for the cycle (Q202).

Cycle description

Cycle 257 enables the machining of circular studs. WinNC produces the circular stud in a spiral infeed, based on the diameter of the raw part.

1. If the tool is at a distance that is less than the 2nd safety distance, WinNC moves the tool back to the 2nd safety distance.
2. The tool moves from the stud centre to the starting position for the stud milling. The starting position is defined by the polar angle relative to the stud center with parameter Q376.
3. WinNC moves the tool to the safety distance in rapid traverse, and moves it from there with depth infeed to the first infeed depth.
4. Then, WinNC produces the circular stud in a spiral infeed observing the Q370 overlap factor.
5. WinNC moves the tool along a tangential path by 2 mm away from the contour.
6. If multiple depth infeeds are necessary, the new depth infeed is carried out at the point closest to the exit movement.
7. This operation is repeated until the programmed stud depth is achieved.
8. At the end of the cycle, the tool rises - after the tangential exit movement - in the tool axis, to reach the 2nd safety distance defined in the cycle.

emco

| CYCL DEF | POCKETS / STUDS / SLOTS | 233 | FACE MILLING (cycle 233) |

Parameter	Description	Unit
Q215	Scope of machining: • 0 = Roughing and finishing • 1 = Roughing only • 2 = Finishing only Side and depth finishing are only carried out if the corresponding finishing allowance is defined (parameters Q368, Q369)	
Q389	Milling strategy: determines, how the controller must machine the surface • 0 = Meander machining: side infeed in positioning feed outside the surface to be machined • 1 = Meander machining: side infeed in milling feed on the edge of the surface to be machined • 2 = Line-by-line machining: retraction and side infeed in positioning feed outside the surface to be machined • 3 = Line-by-line machining: retraction and side infeed in positioning feed on the edge of the surface to be machined • 4 = Spiral machining: even infeed from outside inwards	
Q350	Milling direction: axis of the machining plane, to which the machining must be aligned: • -1 = Main axis = machining direction • -2 = Minor axis = machining direction	
Q218	1. Side length (incremental): Length of the surface to be machined by multipass milling in the main axis of the machining plane, relative to the starting point 1st Axis	mm
Q219	2. Side length (incremental): Length of the surface to be machined by multipass milling in the minor axis of the machining plane. The sign determines the direction of the first cross infeed relative to the starting point on the 2nd axis.	mm

Parameter	Description	Unit
Q227	Starting point 3rd Axis (absolute): Coordinates of the work piece surface, from which the infeeds are calculated.	mm
Q386	End point 3. Axis (absolute): Coordinates of the spindle axis, on which the surface must be face-milled.	mm
Q369	Finishing allowance for depth (incremental): Value, at which the last infeed must traverse.	mm
Q202	Infeed depth (incremental): Dimension by which the tool is fed in each time. Enter a positive value.	mm
Q370	Path overlap factor: Maximum side infeed rate k. The controller calculates the actual side infeed based on the 2nd side length and the tool radius such that the machining is carried our at a constant side infeed rate. Input: 0.1 to 1.9999	mm/min
Q207	Milling feed: Tool traverse speed during milling. Alternatively: FAUTO, FU, FZ	mm/min
Q385	Finishing feed: Tool traverse speed during milling of the last infeed. Alternatively: FAUTO, FU, FZ	mm/min
Q253	Preliminary positioning feed: Traverse speed of the tool approaching the start position and during movement to the next line. If cross movements take place on the material (Q389=1), then the controller moves the cross infeed with milling feed Q207 Alternatively: FMAX, FAUTO	mm/min
Q357	Side safety distance (incremental): Side distance between the tool and the workpiece when approaching the first infeed depth and the distance, by which the side infeed is moved with machining strategies Q389=0 and Q389=2.	mm
Q200	Safety distance (incremental): Distance between tool point and work piece surface.	mm
Q204	2. Safety distance (incremental): coordinate of the spindle axis, in which no collision between the tool and the workpiece (clamping device) can occur.	mm
Q347	1. Limitation: Select the workpiece side, on which the face is limited by a side wall - not possible with spiral machining. Depending on the position of the side wall, the controller limits the machining of the face to the corresponding starting point coordinates or side length • 0 = No limitation • -1 = Limitation in negative main axis • -+1 = Limitation in positive main axis • -2 = Limitation in negative minor axis • -+2 = Limitation in positive minor axis	

emco

Parameter	Description	Unit
Q348	2. Limitation: see Q347	
Q349	3. Limitation: see Q347	
Q220	Corner radius: Radius for the corner with limitations Q347-Q349	
Q368	Finishing allowance, side: Finishing allowance in the machining plane	mm
Q338	Infeed for finishing: Dimension by which the tool is fed in on the spindle axis when finishing. Q338=0: Finishing in one infeed	mm

Note:

Pre-position the tool at the start position in the machining plane (stud centre) using R0. Observe the machining direction.

WinNC automatically positions the tool in the tool axis before the 2nd Safety clearance, observe Q204.

2. The Q204 safety distance must be set in such a way that prevents any collision with the work piece or clamping devices.

If the starting point of the 3rd Axis Q227 and the end point of the 3rd Axis Q386 are set as equal, the controller does not execute the cycle (programmed depth value = 0).
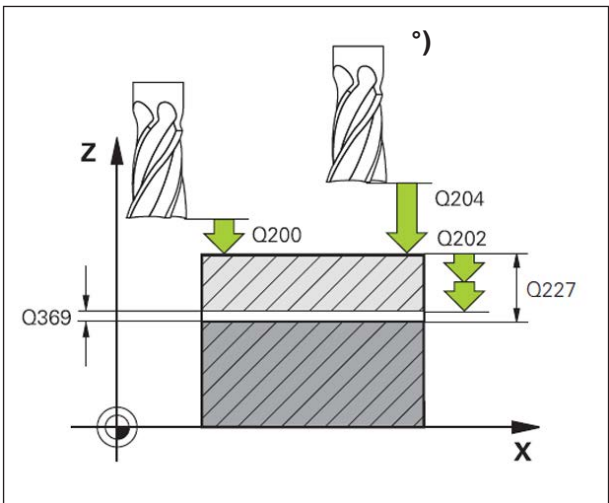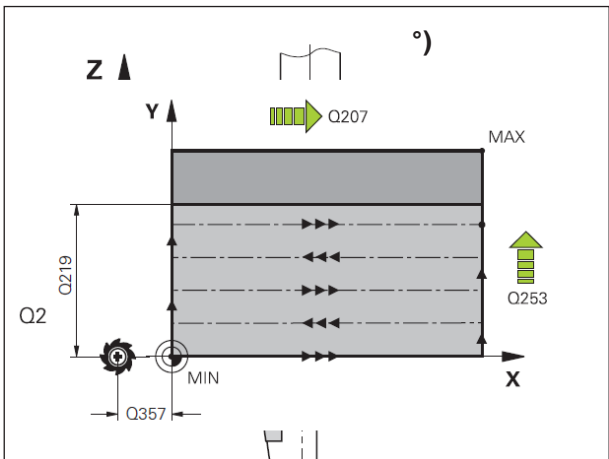
Cycle description

Cycle 233 enables face milling of a surface in multiple infeeds, accounting for a finishing allowance. Side walls that must be taken into account as part of machining of the face surface may be defined in the cycle for this purpose. The following machining strategies are available:

- Strategy 389=0: Meander machining, side infeed outside the surface to be machined.
- Strategy 389=1: Meander machining, side infeed on the edge of the surface to be machined.
- Strategy 389=2: Line-by-line machining with overrun, side infeed, rapid traverse during retraction.
- Strategy 389=3: Line-by-line machining without overrun, side infeed, rapid traverse during retraction.
- Strategy 389=4: Spiral machining from outside inwards.

1 WinNC positions the tool at rapid traverse from the current position in the machining plane on the starting point **1:** The starting point in the working plane is offset from the workpiece by the tool radius and the side safety distance.

2 The controller then positions the tool at rapid traverse in the spindle axis to the safety distance.

3 Then, the tool moves with milling feed Q207 in the spindle axis to the first infeed depth calculated by WinNC.

°)
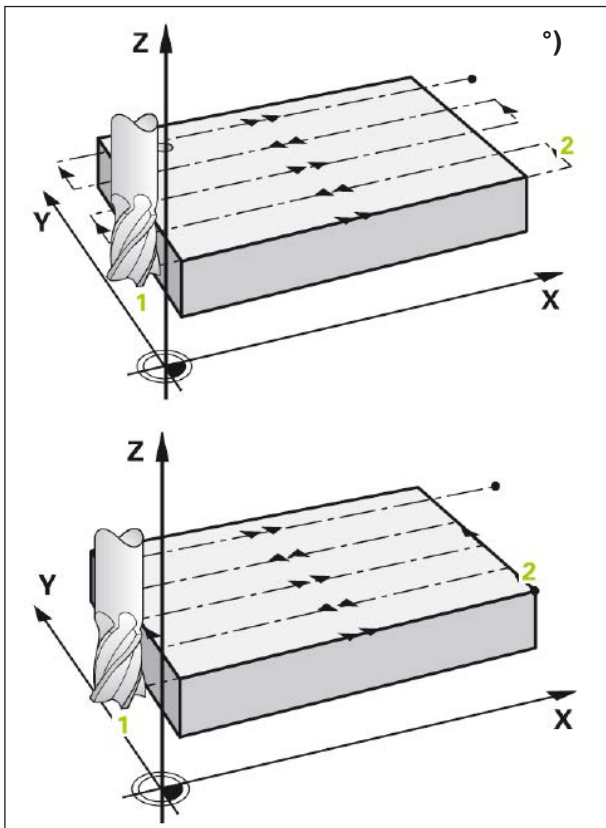
Cycle description:
Strategies Q389=0 and Q389=1

Strategies Q389=0 and Q389=1 differ in the overrun during face milling. With Q389=0, the end point is outside the surface, and with Q389=1 it is at the edge of the surface. The controller calculates end point **2** based on the side length and the side safety distance. With strategy Q389=0, WinNC additionally moves the tool by the tool radius beyond the face surface.

4 WinNC moves the tool with the programmed milling feed to the end point **2**.

5 The controller then offsets the tool with the preliminary positioning feed in cross direction to the starting point of the next line. WinNC calculates the offset based on the programmed width, tool radius, maximum path overlap factor and side safety distance.

6 Then, WinNC moves the tool back in the opposite direction with the milling feed.

7 The procedure is repeated, until the specified surface is completely machined.

8 After that, WinNC positions the tool back to start point **1** in rapid traverse.

9 If multiple infeeds are necessary, WinNC moves the tool in the spindle axis to the next infeed depth, with positioning feed.

10 The procedure is repeated, until all infeeds are executed. With the last infeed, only the entered finishing allowance is milled in the finishing feed.

11 At the end, WinNC returns the tool at rapid traverse back to the 2nd safety distance.
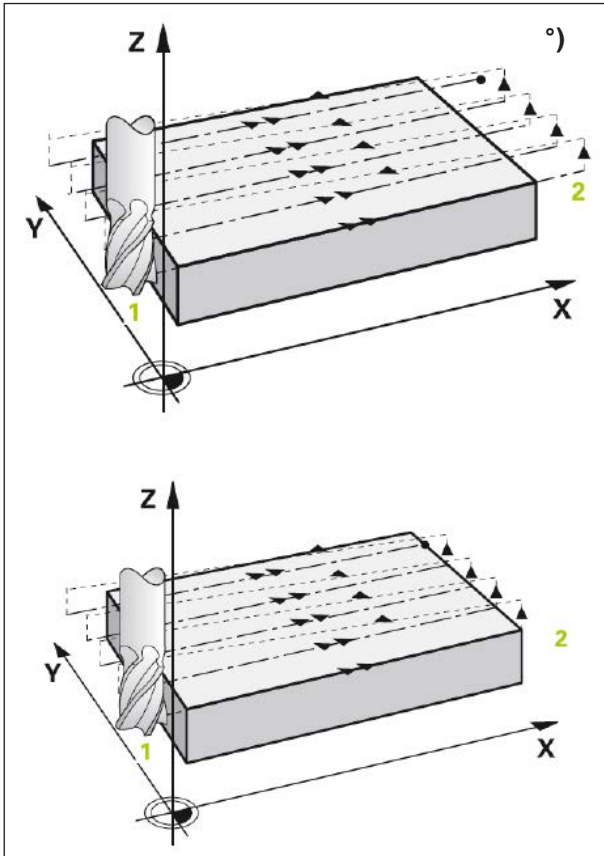
Cycle description:
Strategies Q389=2 and Q389=3

Strategies Q389=2 and Q389=3 differ in the overrun during face milling. With Q389=2, the end point is outside the surface, and with Q389=3 it is at the edge of the surface. The controller calculates end point **2** based on the side length and the side safety distance. With strategy Q389=2, WinNC additionally moves the tool by the tool radius beyond the face surface.

4 WinNC moves the tool with the programmed milling feed to the end point **2**.

5 The controller moves the tool in the spindle axis to the safety distance over the current infeed depth and returns at rapid traverse directly to the starting point of the next line. WinNC calculates the offset based on the programmed width, tool radius, maximum path overlap factor and side safety distance.

6 Subsequently, the tool moves back to the current infeed depth and then back in the direction of the end point **2**.

7 The multipass milling procedure is repeated, until the specified surface is completely machined. At the end of the last path, the controller positions the tool back to start point **1** in rapid traverse.

8 If multiple infeeds are necessary, WinNC moves the tool with positioning feed in the spindle axis to the next infeed depth.

9 The procedure is repeated, until all infeeds are executed. With the last infeed, only the entered finishing allowance is milled in the finishing feed.

10 At the end, WinNC returns the tool at rapid traverse back to the 2nd safety distance.
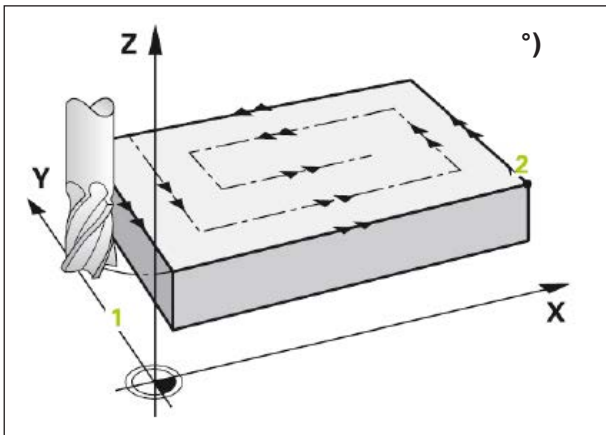
Cycle description:
Strategy Q389=4

4 The tool moves to the starting point of the milling path with the programmed milling feed in a tangential approaching movement.

5 The controller machines the face with milling feed from the outside inwards, with increasingly shorter milling paths. Due to the constant side infeed, the tool remains continuously engaged.

6 The procedure is repeated, until the specified surface is completely machined. At the end of the last path, the controller positions the tool back to start point **1** in rapid traverse.

7 If multiple infeeds are necessary, WinNC moves the tool with positioning feed in the spindle axis to the next infeed depth.

8 The procedure is repeated, until all infeeds are executed. With the last infeed, only the entered finishing allowance is milled in the finishing feed.

9 At the end, WinNC returns the tool at rapid traverse back to the 2nd safety distance.

Limitation:

Limitations can be used to restrict the machining of the face, in order to take the side walls or offsets into account during the machining, for example. A side wall defined through a limitation is machined based on the dimension based on the starting point and/or the side length of the face. In roughing operations, the WinNC takes the allowance for the side into account; in the finishing operation, the allowance is used to pre-position the tool.

Converting the coordinates

COORD.
TRANSF.

- 7 Zero offset
- 8 Mirroring
- 10 Rotation
- 11 Scaling
- 19 Working plane
- 247 Datum setting

emco
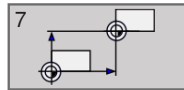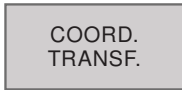
| CYCL DEF | COORD. TRANSF. | 7 | | ZERO OFFSET (cycle 7) |
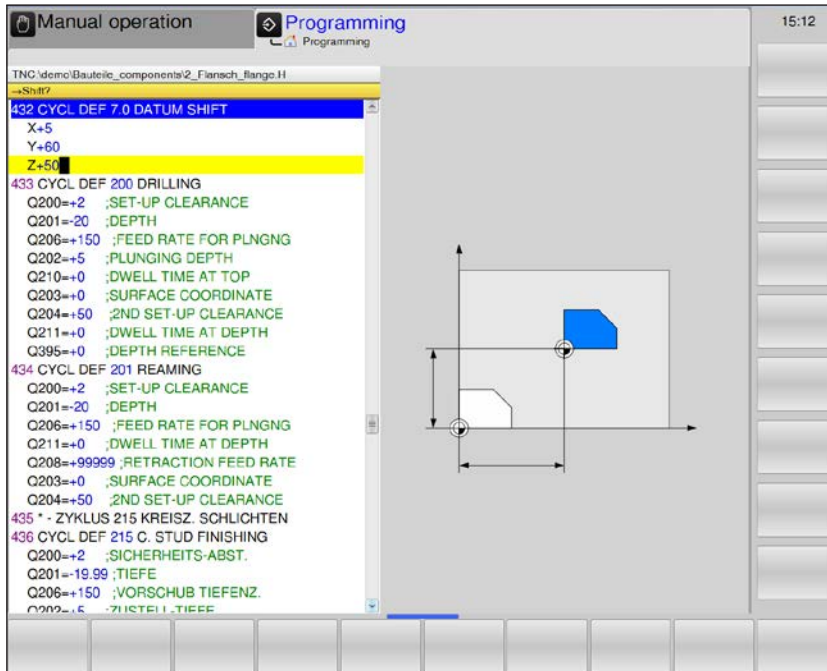
Cycle description

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.

CYCL DEF | COORD. TRANSF. | 8
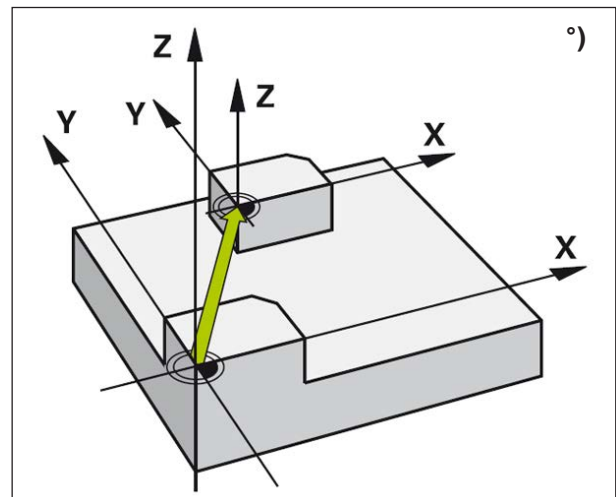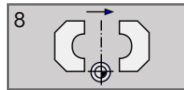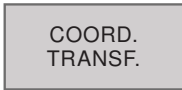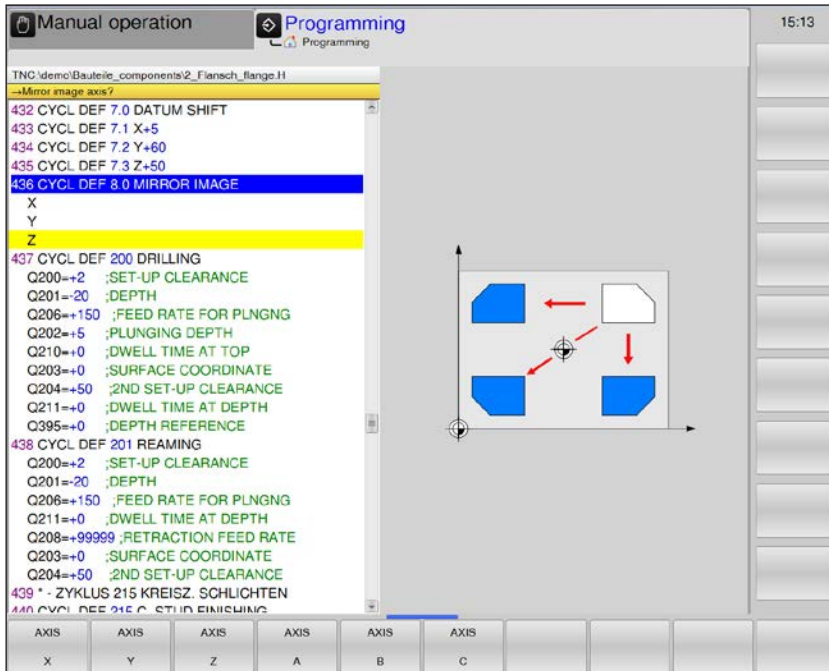
MIRRORING (cycle 8)

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.

emco

CYCL DEF	COORD. TRANSF.	10
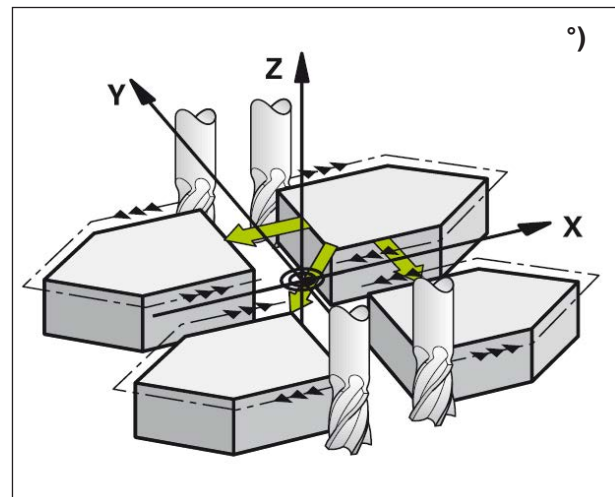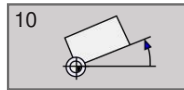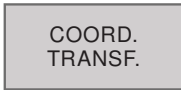
ROTATION (cycle 10)
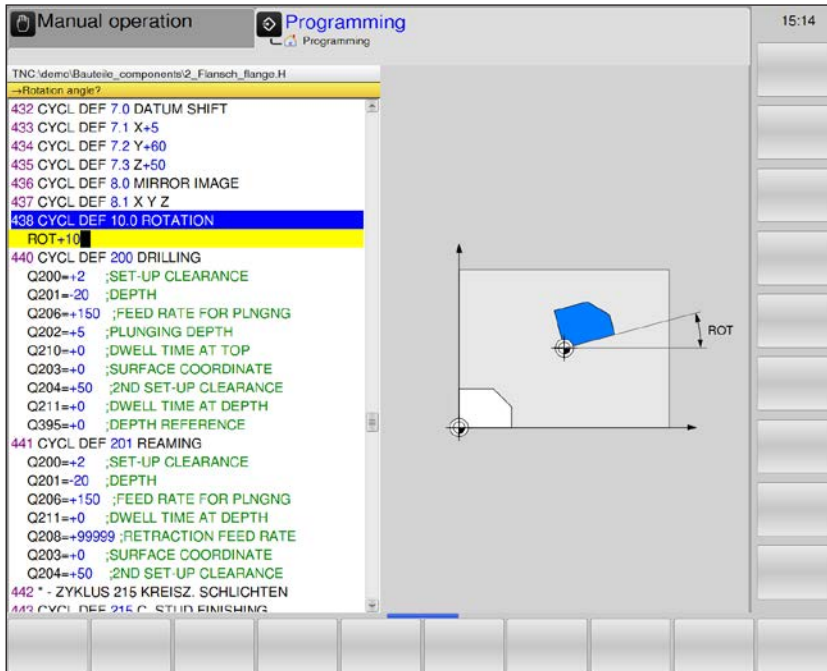
Cycle description

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°.

CYCL DEF	COORD. TRANSF.	11	

SCALING (cycle 11)

Parameters	Description	Unit
SCL	Factor: The WinNC multiplies the coordinates and radii by the SCL factor (as described under "Effect" above	

Effect

The WinNC can enlarge or reduce contours within a programme. This means that shrinkage and allowance factors can be taken into account.

The scaling factor is effective from the definition in the program. It is also effective in the Positioning with Manual Data Input mode. The WinNC shows the active scaling factor in the additional status display.

The scaling factor has an effect on

- all 3 coordinate axes at the same time
- all dimensions in cycles

Prerequisite

It is advisable to set the datum to an edge or a corner of the contour before enlarging or reducing the contour.

Resetting

Program the SCALING cycle once again with a scaling factor of 1.

CYCL DEF	COORD. TRANSF.	19
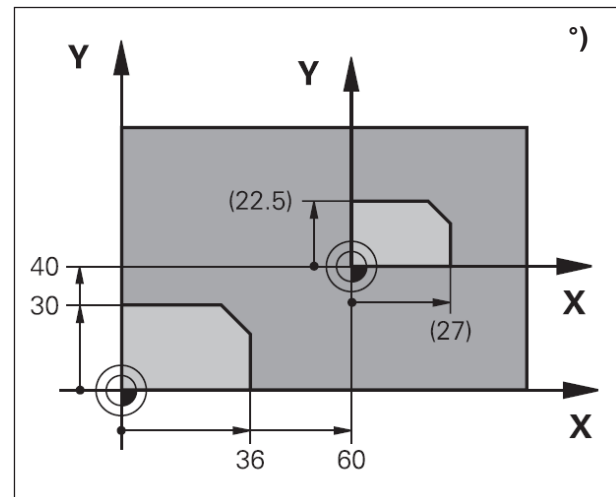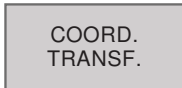
MACHINING PLANE (Cycle 19)
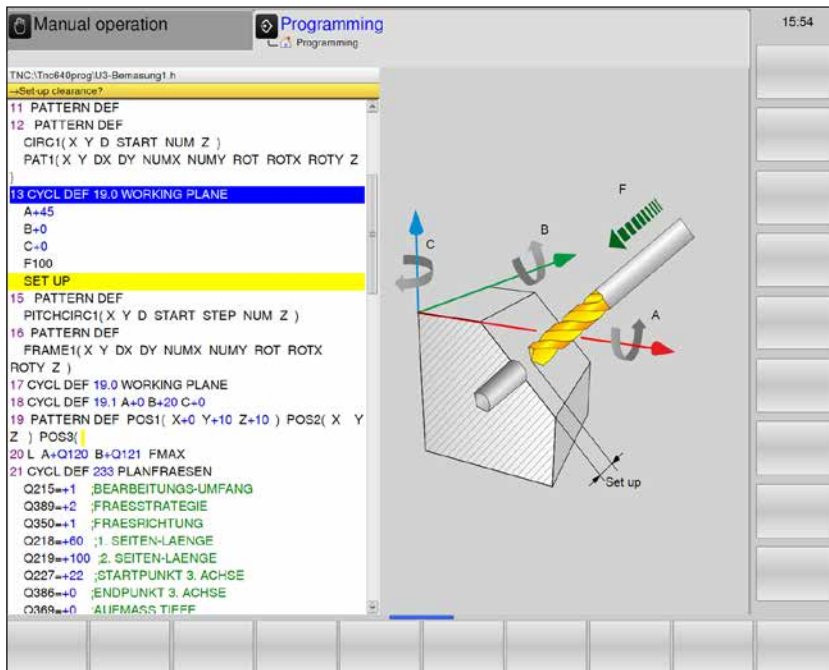
Parameters	Description	Unit
A,B,C	Enter rotation axis with associated angle of rotation	°
F	Feed rate related to traversing on DIST	mm/min
DIST	Traversing movement in Z incrementally upwards. Maximum value: Tool change point	

Effect

The position of the machining plane is defined in Cycle 19.

• Describe the position of the machining plane by up to three rotations (spatial angle) of the machine-fixed coordinate system. The spatial angles to be entered are obtained by making a section perpendicular to the tilted machining plane and viewing the section from the axis around which it is tilted. With two spatial angles, any tool position is already clearly defined in space.

Resetting

To reset the tilt angles, redefine the MACHINING PLANE cycle and enter 0° for all rotation axes. Then define the MACHINING PLANE cycle once again and confirm the entry with NO ENT. This sets the function to inactive.

Note:
The PLANE SPATIAL cycle (tilting the machining plane) is preferred, see page D39 et seq.!
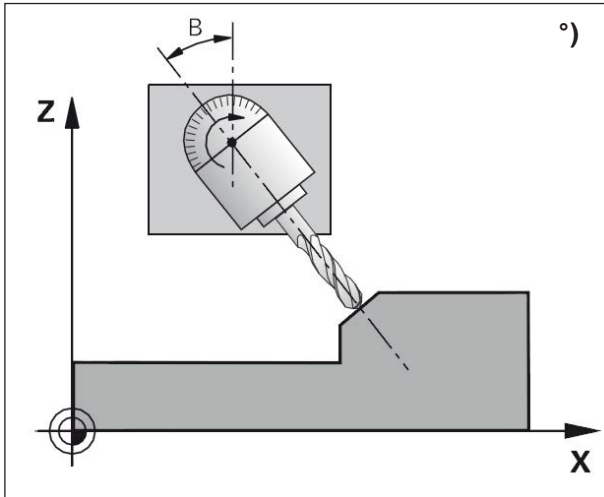
Position rotation axes manually

Since the rotation axes are not positioned automatically in Cycle 19, the rotation axes must be positioned in a separate L block after the cycle definition.

If you work with axis angles, you can define the axis values directly in the L block. If you are working with spatial angles, use the Q parameters Q120 (A axis value), Q121 (B axis value) and Q122 (C axis value) described by Cycle 19.

CYCL DEF	COORD. TRANSF.	247
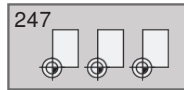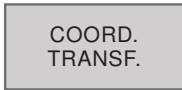
DATUM SETTING (cycle 247)
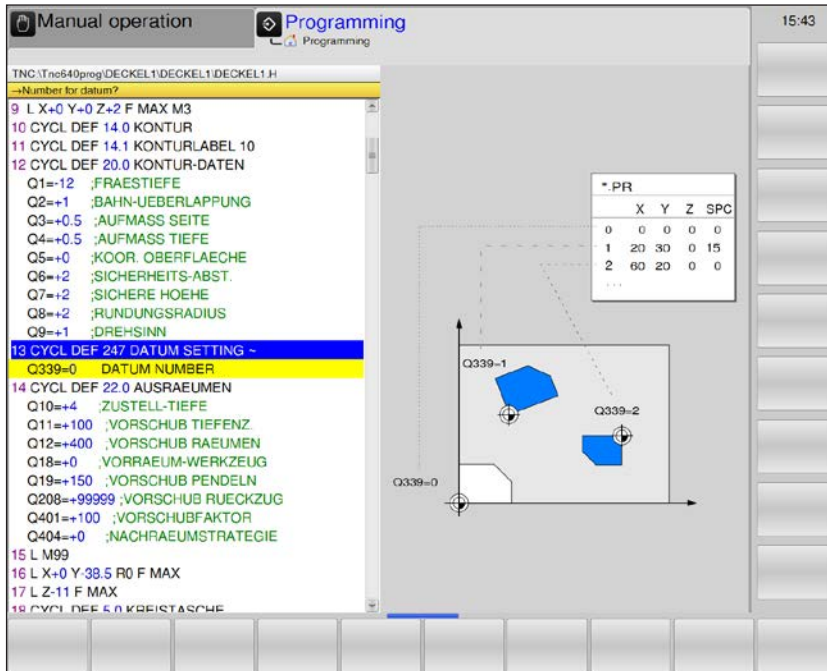
Cycle description

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
IIn 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.
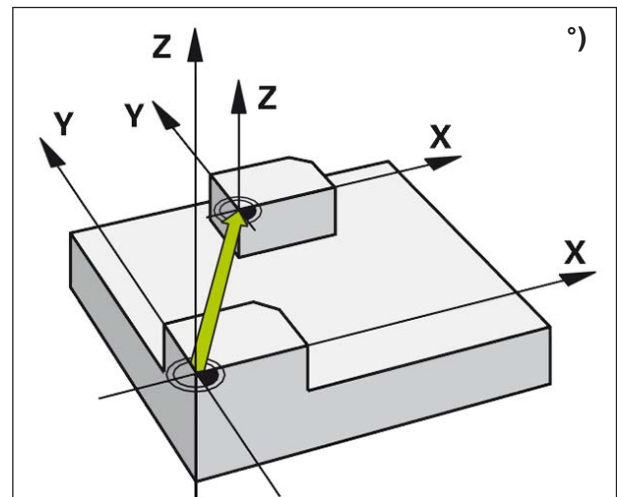
SL Cycles

SL
CYCLES

- 14 Contour geometry
- 20 Contour data
- 21 Pilot drilling
- 22 Rough-out
- 23 Floor finishing
- 24 Side finishing
- 25 Contour train
- 27 Cylinder surface

emco

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 GEOMETRY (cycle 14)
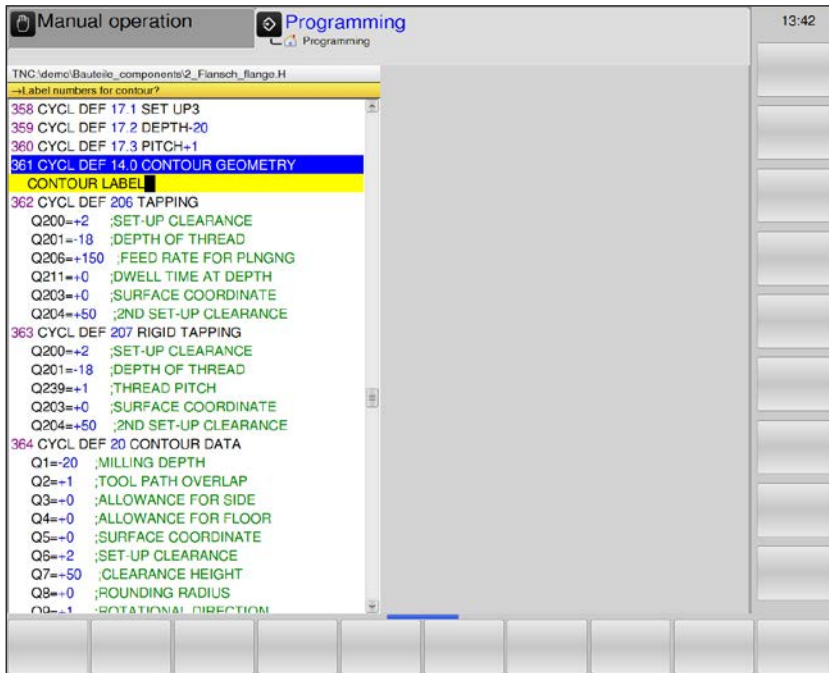
14
LBL 1..N

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

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

ENT

END

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
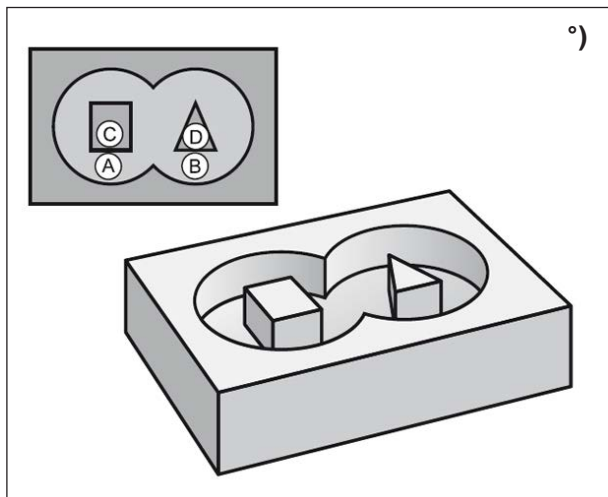
Note:
Prior to programming, please take note:
Cycle 14 is DEF-active, which means that it is effective in the program as soon as it is defined.
In cycle 14, a maximum of 12 subprograms (subcontours) can be listed.

emco

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.

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

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
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

°)
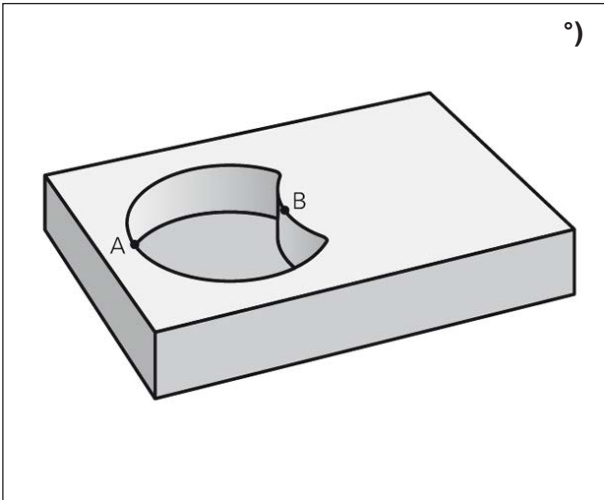
"Difference" surface

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

°)
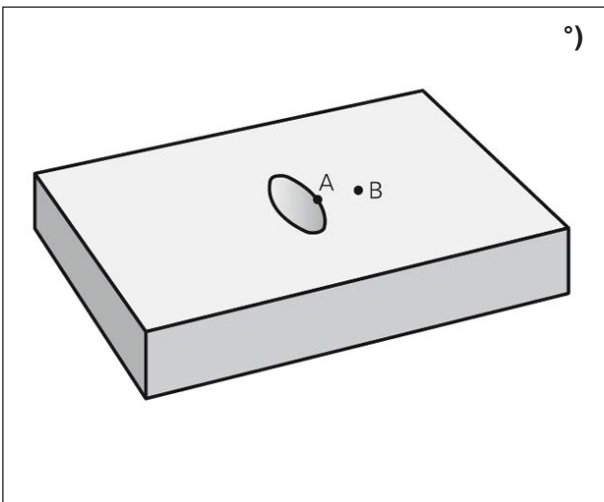
"Overlap" surface

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

emco

| 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. **Note:** 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 • -1 = up-cut for pocket and island • +1 = down-cut for pocket and island	

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

°)
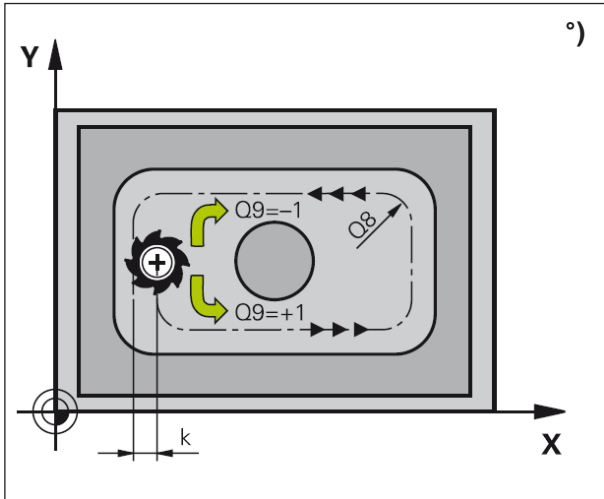
Cycle description

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

°)
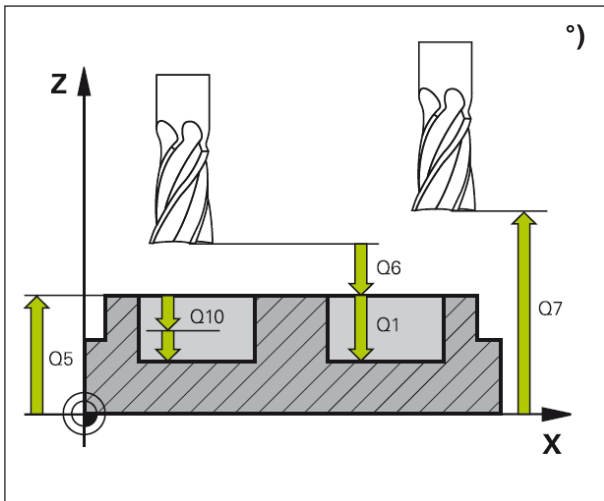
Note:

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.

emco
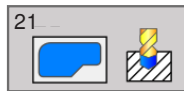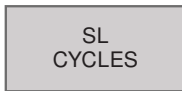
| CYCL DEF | SL CYCLES | 21 | PILOT-DRILLING (cycle 21) |

Parameter	Description	Unit
Q10	Infeed depth (incremental): Dimension by which the tool is fed in. "-" sign for negative alignment	mm
Q11	Feed for depth infeed: Tool traverse speed during plunging	mm/min
Q13	Clearing tool number: Number and name of the clearing tool. Input range: 0 - 32767.9 for number entry. At value 0, the tool data of the tool that is currently in the spindle are used.	

°)
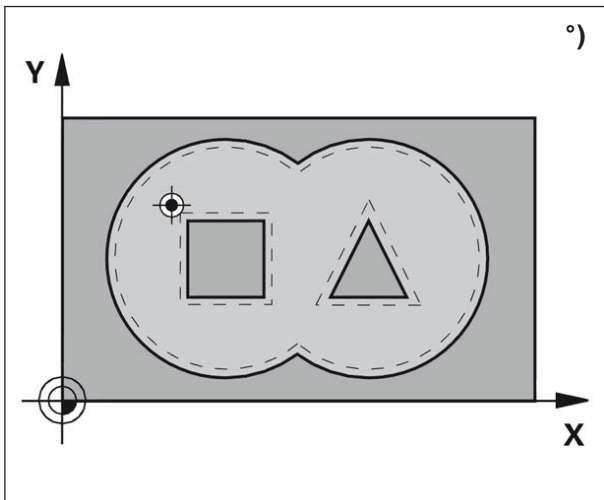
Note:
Prior to programming, please take note:
The DR delta value programmed in a TOOL CALL set is not taken into account for calculating the infeed points.
At narrow points, WinNC may probably not be able to pre-drill with a tool that is larger than the roughing tool.
If Q13=0 is set, the tool data of the tool that is in the spindle are used.

Cycle description
Cycle 21 is used when after the cycle, a tool that does not have a centre-cut end mill (DIN 844) is used for clearing out the contour.
The cycle produces a hole in the area, to be cleared later, e.g. as part of Cycle 22. Cycle 21 takes into account the depth finishing allowance and the radius of the clearing tool for the infeed points. The infeed points simultaneously serve as starting points for the clearing.

Before cycle 21 can be called, two further cycles need to be programmed:

- Cycle 14 CONTOUR: it is required by Cycle 21 PRE-DRILLING, in order to determine the drilling position on the plane.
- Cycle 20 CONTOUR DATA: it is required by Cycle 21 PRE-DRILLING, in order to determine the drilling depth and safety distance.

1 WinNC first positions the tool on the plane: the position is based on the contour previously defined with Cycle 14 and on the information about the clearing tool.
2 The tool moves back to the safety distance in rapid traverse FMAX. The safety distance data are entered in Cycle 20 CONTOUR DATA.
3 The tool drills at the entered feed F from the current position to the first infeed depth.
4 After that, WinNC moves the tool back at FMAX rapid traverse to the first infeed depth reduced by the advance stop distance t.
5 WinNC independently determines the advance stop distance:

- Drilling depth up to 30 mm: t = 0.6 mm
- Drilling depth over 30 mm: Drilling depth / 50
- maximum advance distance: 7 mm
6 Subsequently, the tool drills at the programmed feed F by a further infeed depth.
7 WinNC repeats steps 1 to 4, until the specified drilling depth is achieved, taking into account the depth finishing allowance.
8 Finally, the tool moves in the tool axis back to the safe height or to that position which was programmed last before the cycle.
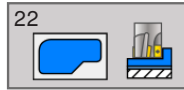
| CYCL DEF | SL CYCLES | 22 | | **ROUGH-OUT (cycle 22)** |

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	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. • Q404 = 0: WinNC moves the tool between the fine roughing areas at the current depth along the contour. • 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.	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.

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
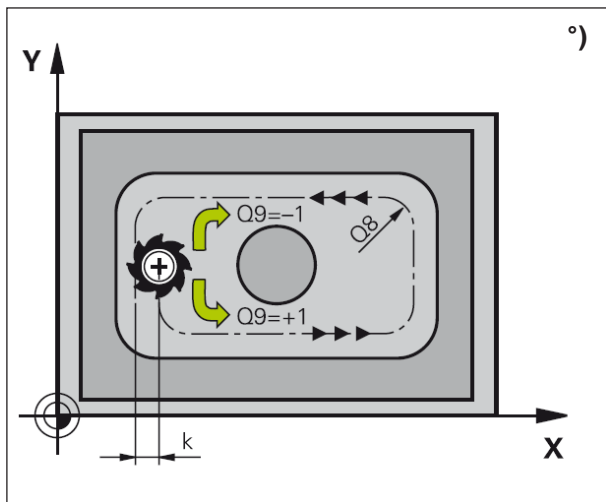
Cycle description
1. WinNC positions the tool above the cutter infeed point in consideration of the side finishing allowance.
2. In the first infeed depth, the tool cuts the contour from the inside to the outside at milling feed Q12.
3. In doing so, the island contours (here: C/D) with an approach to 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.

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.

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

| CYCL DEF | SL CYCLES | 23 | **FLOOR FINISHING (cycle 23)** |
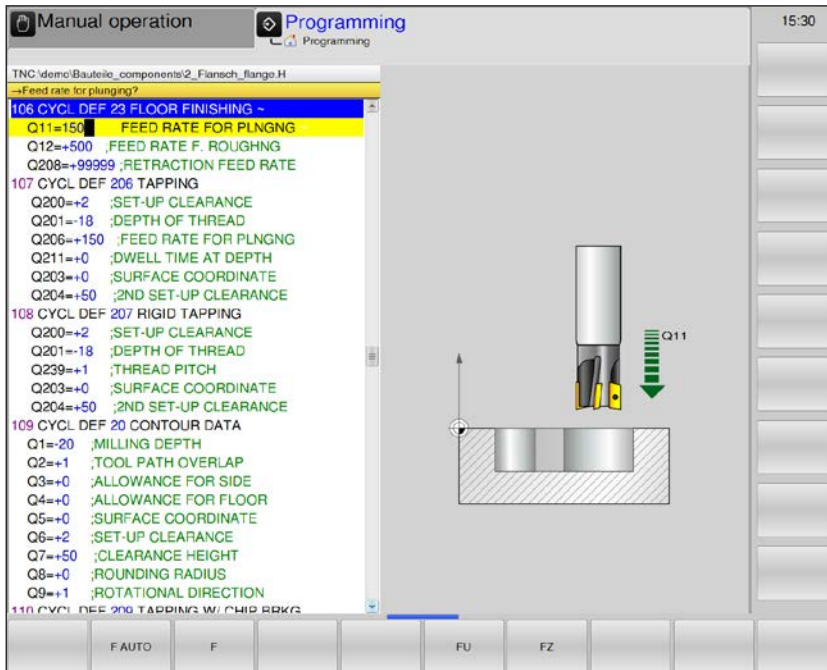
Parameter	Description	Unit
Q11	Feed for depth infeed: Tool traverse speed during plunging.	mm/min
Q12	Milling feed: Feed during movements in the machining plane	mm/min
Q208	Retraction feed: Traverse speed of the tool during retraction after machining. With input =0: The tool retracts with feed Q12	mm/min

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.

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
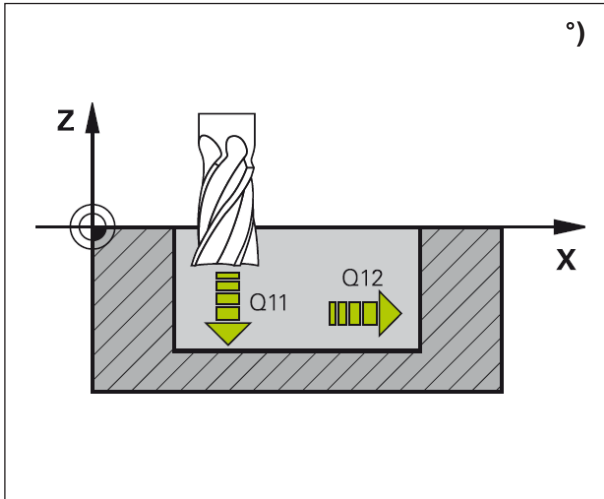
Cycle description
1. WinNC positions the tool in rapid traverse at the safe height.
2. A movement in the tool axis at feed Q11 follows.
3. The control unit moves the tool smoothly (vertical tangential circle) to the surface that is to be machined if there is enough space. In narrow conditions, WinNC 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.

Instructions fro programming

1. The control unit independently determines the starting point for finishing depth. The starting point depends on the spacial 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.

emco

| CYCL DEF | SL CYCLES | 24 | | **SIDE FINISHING (cycle 24)** |
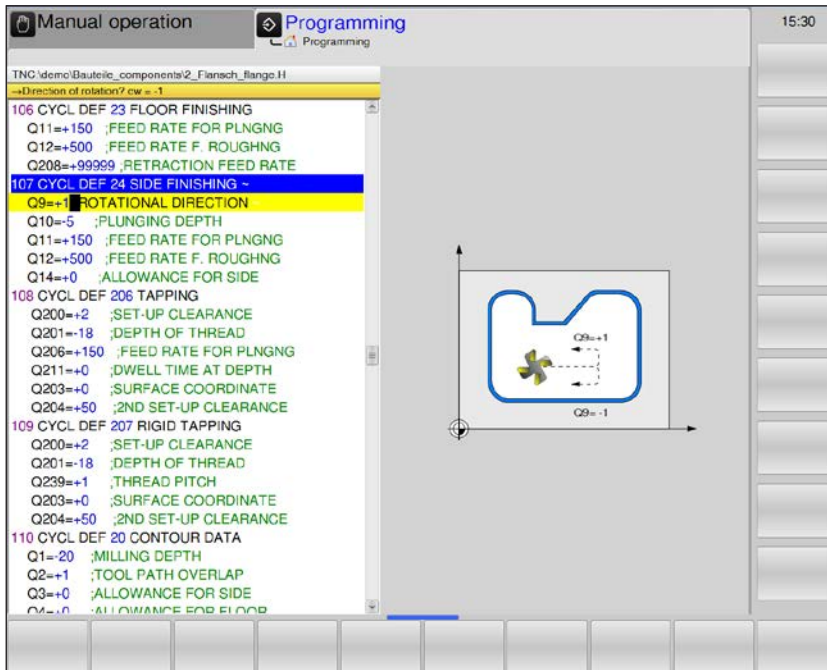
Parameter	Description	Unit
Q9	Direction of rotation: Machining direction • +1: Rotation counterclockwise. • -1: Rotation clockwise	
Q10	Infeed depth (incremental): Dimension by which the tool is fed in respectively.	mm
Q11	Feed for depth infeed: Tool traverse speed during plunging.	mm/min
Q12	Milling 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
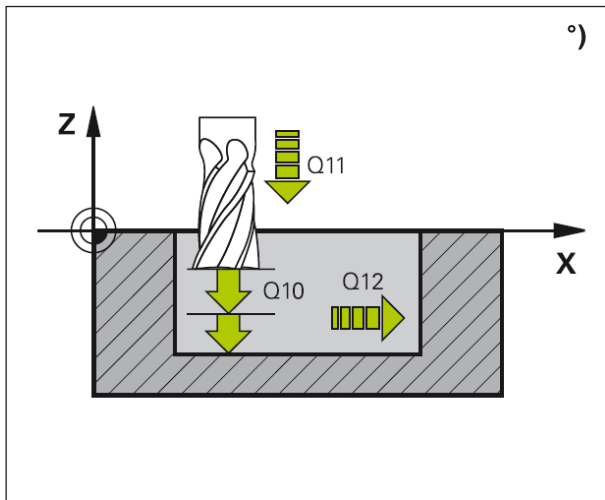
Cycle description

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.

| CYCL DEF | SL CYCLES | 25 |
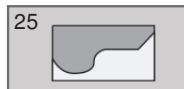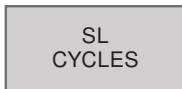
CONTOUR TRAIN (cycle 25)
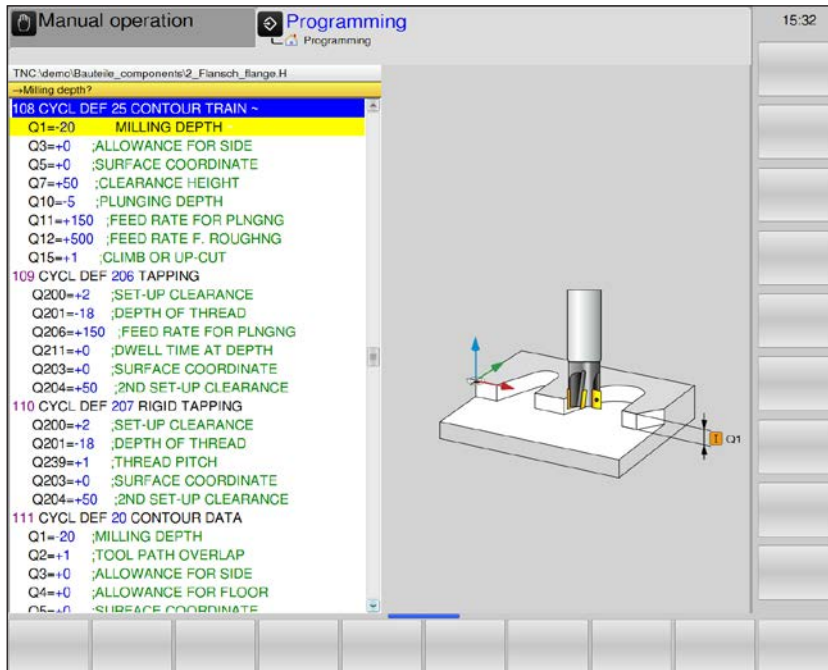
Parameter	Description	Unit
Q1	Cutting depth (incremental): Distance between work piece surface and contour bottom:	mm
Q3	Side finishing allowance (incremental): Finishing allowance in the machining plane.	mm
Q5	Work piece surface coordinate (absolute)	mm
Q7	Safe height (absolute): Absolute height, where 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.	

°)
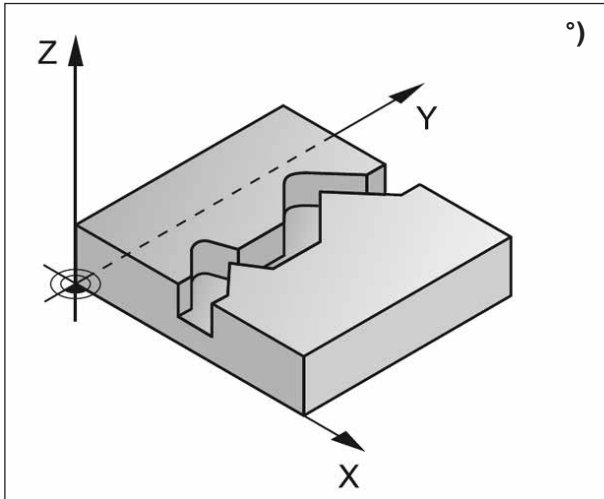
Cycle description

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.

emco
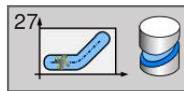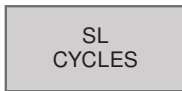
| CYCL DEF | SL CYCLES | 27 | CYLINDER SURFACE (Cycle 27) |
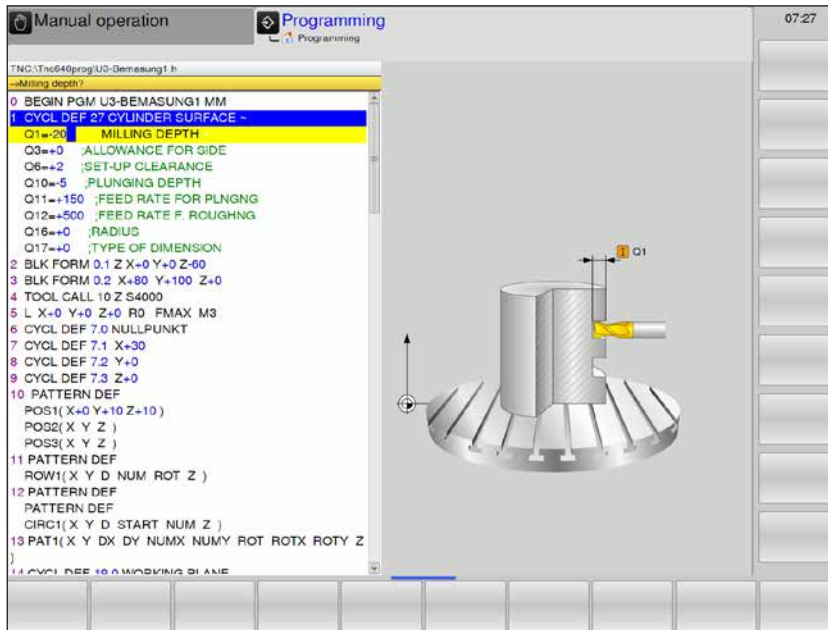
Parameters	Description	Unit
Q1	Cutting depth (incremental): Distance between cylinder jacket and contour bottom.	mm
Q3	Side finishing allowance (incremental): Finishing allowance in the plane of the unrolled cylinder jacket. The allowance acts in the direction of the radius correction.	mm
Q6	Safety distance: Distance between tool face and cylinder jacket surface.	mm
Q10	In-feed depth: Dimension, by which the tool is fed in respectively.	
Q11	Feed for depth infeed: Feed during movements in the spindle axis. Alternatively FAUTO, FU, FZ	mm/min
Q12	Milling feed: Feed during movements in the machining plane. Alternatively FAUTO, FU, FZ	mm/min
Q16	Cylinder radius: Radius of the cylinder, on which the contour is to be machined.	mm
Q17	Dimension type: • 0: Degrees • 1: MM/INCH Program the coordinates of the rotation axis in the subprogram in degrees or mm (inch).	mm or inches
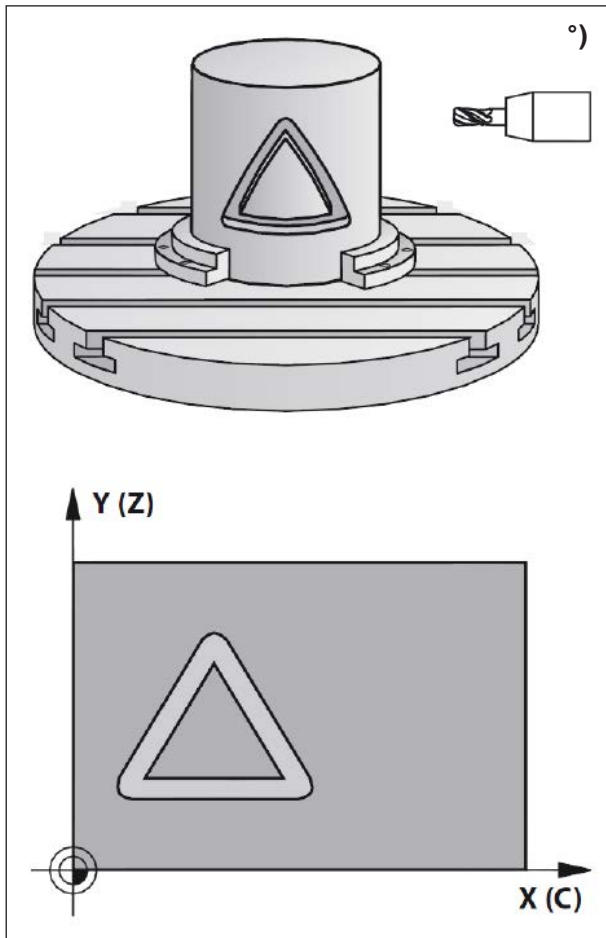
Cycle description
With this cycle, a contour defined on the unrolled jacket can be transferred to the jacket of a cylinder. Cycle 28 is used to mill guide grooves on the cylinder. You can describe the contour in a subprogram that is defined in Cycle 14 (CONTOUR). In the subprogram, you always describe the contour with the coordinates X and Y, regardless of which rotation axes are present on the machine. The contour description is therefore independent of your machine configuration. L, CHF, CR, RND and CT are available as path functions.
The data for the angular axis (X coordinates) can be entered either in degrees or in mm (inches) (specify in the cycle definition via Q17).

- WinNC positions the tool over the infeed point, taking into account the finishing allowance for the side.
- In the first in-feed depth, the tool cuts using milling feed Q12 along the programmed contour.
- At the end of the contour, WinNC moves the tool to the safety distance and then back to the infeed point.
- Repeat steps 1 to 3 until the programmed end cutting depth Q1 is reached.
- The tool then moves to the safety distance.

Note:
Prior to programming, please note:
Both cylinder jacket coordinates must always be programmed in the first NC block of the contour subprogram.
The memory for one SL cycle is limited as follows:
A maximum of 16384 contour elements can be programmed in one SL cycle.
The sign of the cycle parameter for depth determines the working direction.
If depth = 0 is programmed, the cycle will not be executed.
Use a centre-cut end mill (DIN 844).
The cylinder must be clamped centrally on the rotary table. The reference point must be in the centre of the rotary table.

The spindle axis must be perpendicular to the rotary table axis when the cycle is activated. If this is not the case, WinNC produces an error message.
It may be necessary to switch the kinematics.
This cycle can also be executed with the machining plane tilted.
The safety distance must be greater than the tool radius.
The machining time may increase if the contour consists of many non-tangential contour elements.
If the local Q parameter QL is used in a contour subprogram, it must also be assigned or calculated within the contour subprogram.

Dot patterns

PATTERN

- 220 Pattern on circle
- 221 Pattern on line

emco
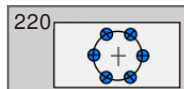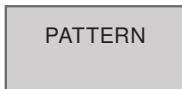
| CYCL DEF | PATTERN | 220 | | PATTERN ON CIRCLE (cycle 220) |

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: • 0: Move to the safety distance between the machining operations • 1: Move between the machining operations to 2nd safety distance	
Q365	Type of movement? Straight line or circle: Determine, according to which path function the tool should move between the machining operations: • 0: Move on a straight line between the machining operations. • 1: Move circularly on the pitch circle diameter between the machining operations.	

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.
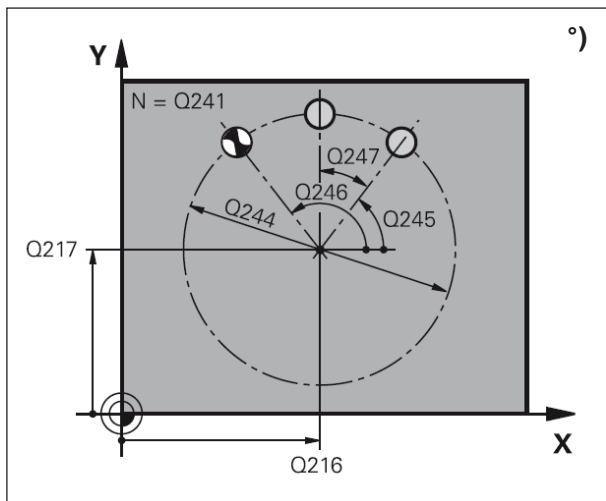
Cycle description
1 WinNC positions the tool in rapid traverse from the current position to the starting point of the first machining operation.
Order:
• 2. Approach safety distance (spindle axis).
• Approach starting point in the machining plane.
• Move to the safety distance above work piece surface (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.
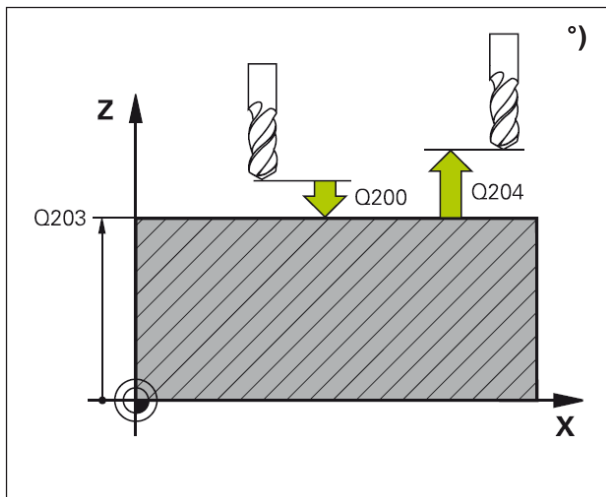
Note:
Prior to programming, please take note:
Cycle 220 is DEF-active, which means that cycle 220 automatically calls the last defined machining cycle. If you combine one of the machining cycles 200 to 209 and 251 to 267 with cycle 220, the safety distance, the work piece surface and the 2nd safety distance from cycle 220 are effective.

emco
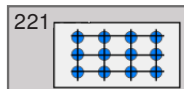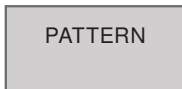
| CYCL DEF | PATTERN | 221 | **PATTERN ON LINES (cycle 221)** |

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: • 0: Move to the safety distance between the machining operations • 1: Move between the machining operations to 2nd safety distance	

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
- 13 Orientation
- 225 Engraving

emco

| CYCL DEF | SPECIAL CYCLES | 9 🕐 |
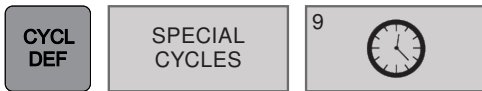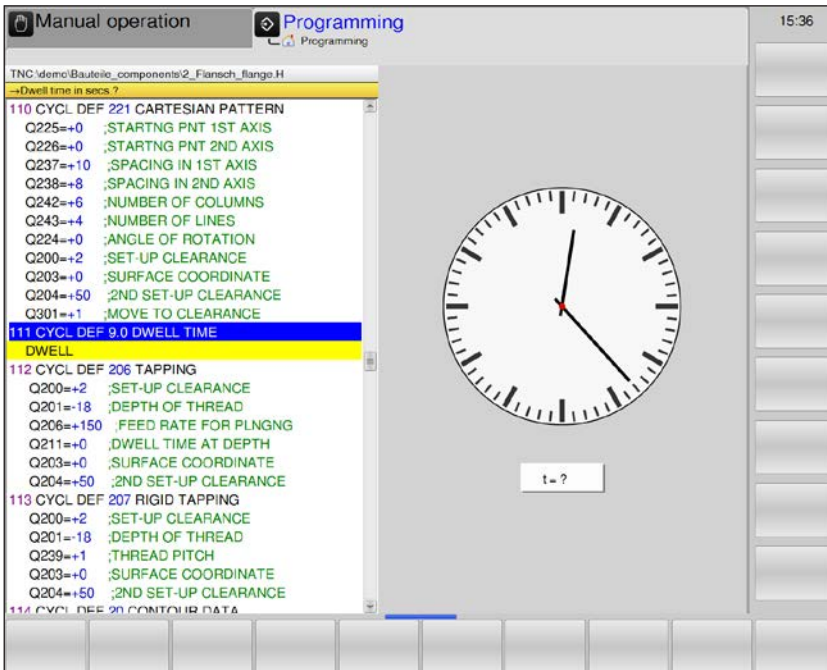
DWELL TIME (cycle 9)

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)
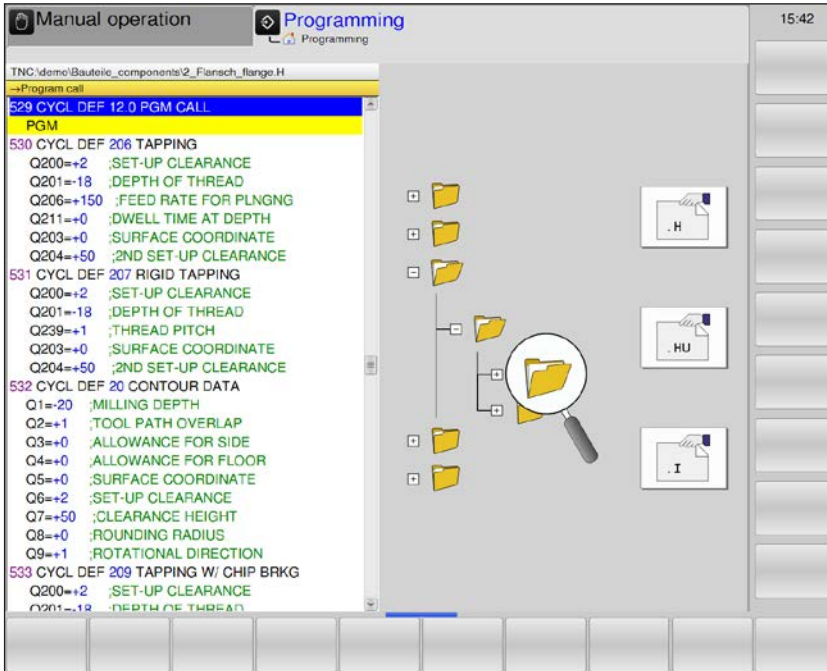
Parameter	Description	Unit
	Enter program name and extension	

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:\... \ ...

emco

| CYCL DEF | SPECIAL CYCLES | 13 | **SPINDLE ORIENTATION (Cycle 13)** |
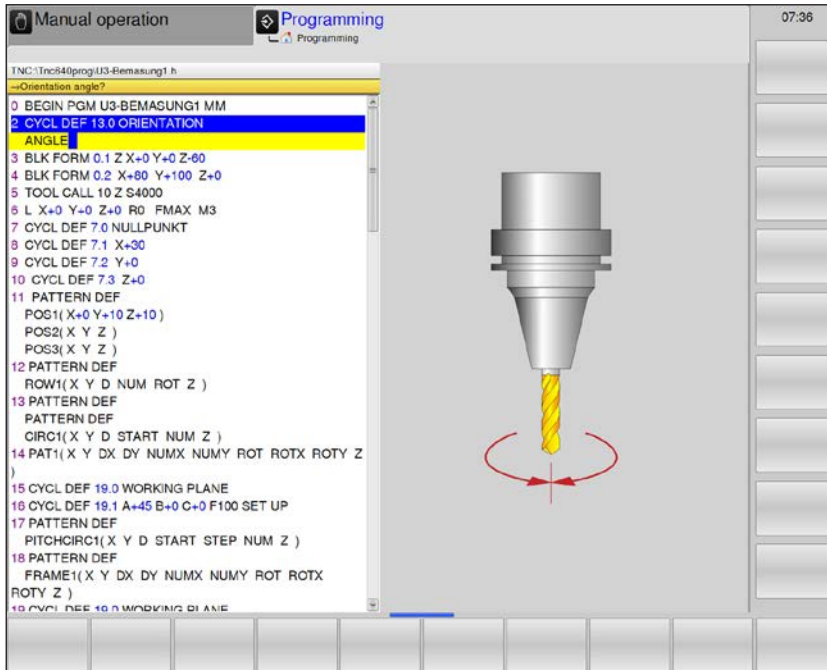
Parameters	Description	Unit
Orientation angle	Enter the angle with respect to the angle reference axis of the work plane	Degrees

Cycle description

WinNC can control the main spindle of the machine tool and turn it to a position determined by an angle.

The spindle orientation is required, for example, for tool change systems with a specific change position for the tool

The angular position defined in the cycle is positioned by WinNC by programming M19 or M20.

When Cycle 13 becomes active, there is no spindle movement.
M19 always positions to 0 degrees
M20 positions to the value programmed in Cycle 13.
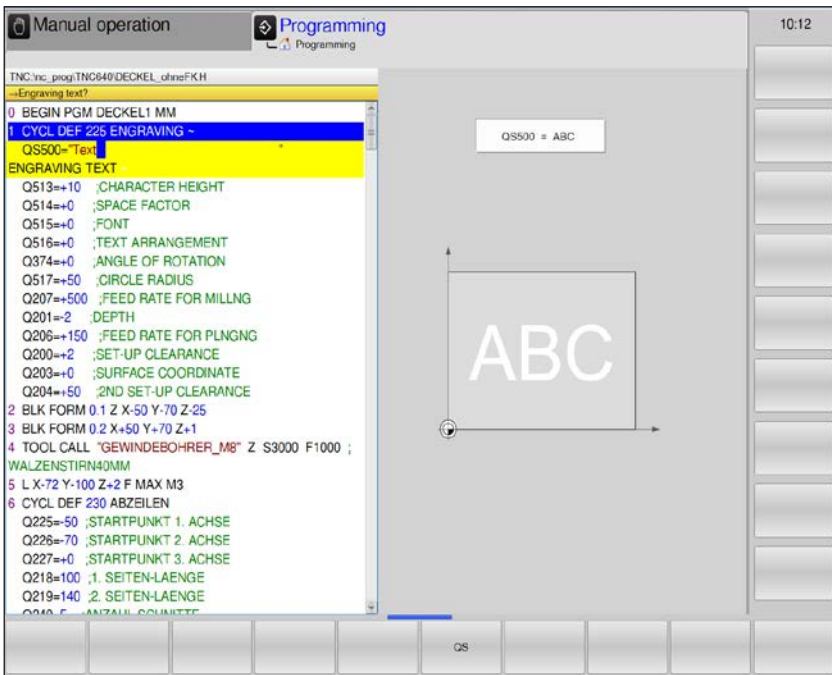
Parameters	Description	Unit
QS500	Engraving text: Text to be engraved inside quotation marks. Assignment of a string variable through the Q key of the numerical keypad.	
Q513	Character height: Height of the characters to be engraved in mm.	mm
Q514	Space factor : The font used is a proportional font. Each character has its own width, which is engraved correspondingly by the WinNC if you program Q514 = 0. If Q514 is not equal to 0, the WinNC scales the space between the characters.	
Q515	• 0: without function	
Q516	Text in a straight line/on arc: • 0: Engrave text in a straight line. • 1: Engrave text on an arc	
Q374	Angle of rotation: Center angle if the text is to be arranged on an arc. Engraving angle if the text is arranged in a straight line.	
Q517	Radius of text on an arc (absolute): Radius of the arc in mm on which the TNC is to arrange the text.	mm
Q207	Feed rate for milling: Traversing speed of the tool in while milling.	mm/min
Q201	Depth (incremental): Distance between workpiece surface and engraving floor.	mm
Q206	Feed rate for plunging: Traversing speed of the tool when moving into the workpiece.	mm/min
Q200	Set-up clearance (incremental): Distance between tool tip and workpiece surface.	mm

Q203	Coordinate of workpiece surface (absolute): Coordinate of the workpiece surface.	
Q204	2nd set-up clearance (incremental): Coordinate in the spindle axis at which no collision between tool and workpiece (fixtures) can occur.	

$a = x \cdot Q514$

°)

x

a

A B C
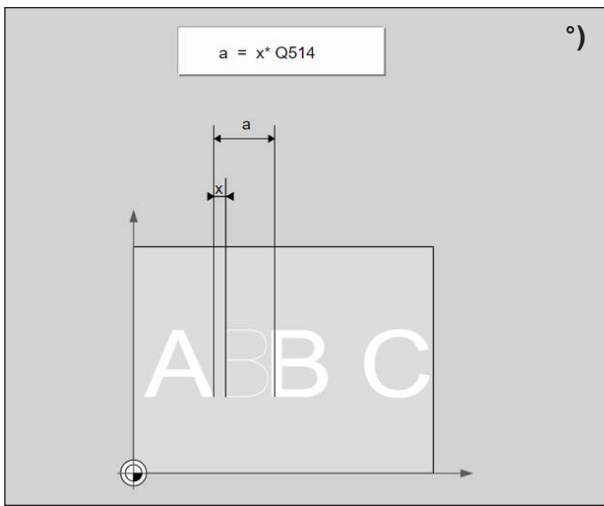
Cycle description

This cycle is used to engrave texts on a flat surface of the workpiece. The texts can be arranged in a straight line or along an arc.

1 The WinNC positions the tool in the working plane to the starting point of the first character.
2 The tool plunges perpendicularly to the engraving floor and mills the character. The WinNC retracts the tool to the set-up clearance between the characters when required.
After machining the character, the tool is at the set-up clearance above the workpiece surface.
3 This process is repeated for all characters to be engraved.
4 Finally, the WinNC retracts the tool to the 2nd set-up clearance.

> **Note:**
> The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the cycle will not be executed.
> If you engrave the text in a straight line (Q516=0), the starting point of the first character is determined by the tool position at the time the cycle is called.
> If you engrave the text along an arc (Q516=1), the arc's center is determined by the tool position at the time the cycle is called.
> The text to be engraved can also be transferred with a string variable (QS).

emco

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

emco

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 1 | **DEEP DRILLING (cycle 1)** |

Parameter	Description	Unit
DIST	Safety distance **1** (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Drilling depth **2** (incremental): Distance between work piece surface - hole bottom (= drill taper tip).	mm
INF.DT	Infeed depth **3** (incremental): Dimension by which the tool is fed in respectively. The control unit moves in one work step to the drilling depth, if: • infeed depth and depth are identical • the infeed depth is bigger than the drilling depth	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
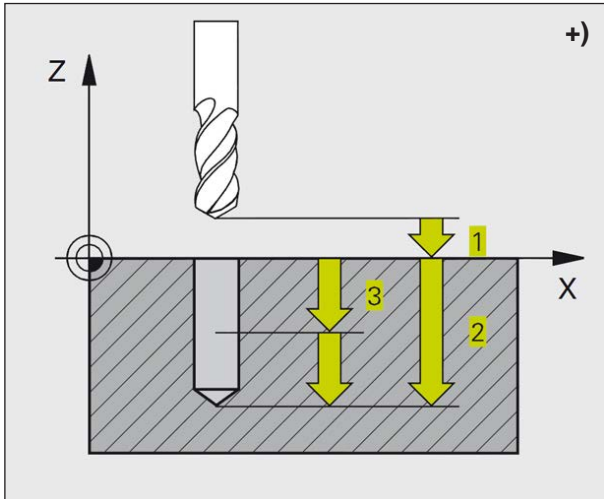
Cycle description

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 mm
- Drilling depth over 30 mm: t = 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.

emco

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 2 | **THREAD TAPPING (cycle 2)** |
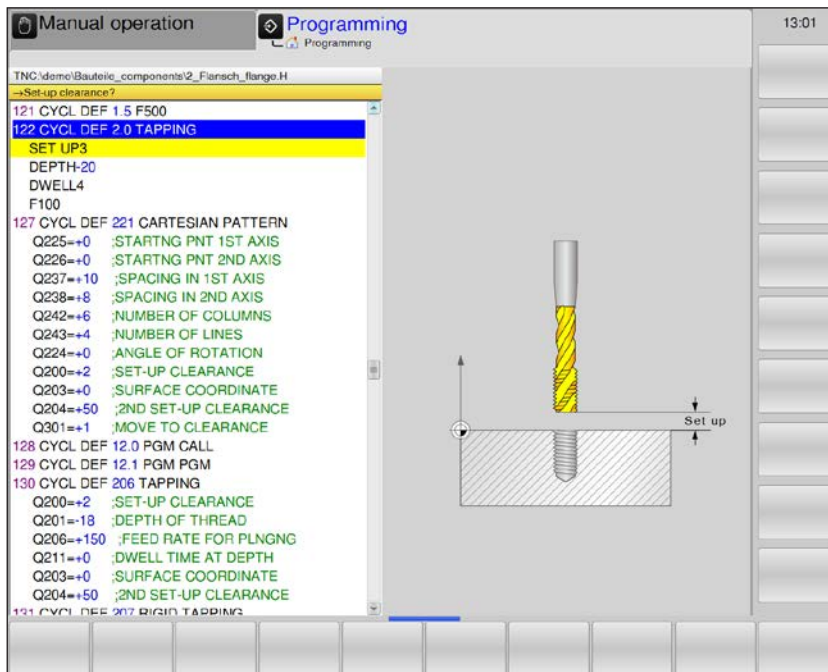
Parameter	Description	Unit
DIST	Safety distance **1** (incremental): Distance between tool point (= starting position) and work piece surface. Guideline value: 4x thread pitch	mm
DEPTH	Drilling depth **2** (thread length, incremental): Distance work piece surface - thread end.	mm
D.TIME	Dwell time in 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 x p F Feed (mm/min) S: Spindle speed (rpm) p: Thread pitch (mm)	mm/min
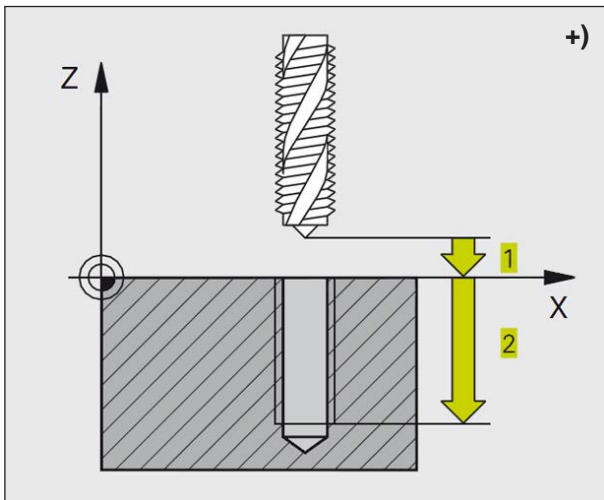
Cycle description

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.

emco

CYCL DEF	SPECIAL CYCLES	OLD CYCLS	17 RT

RIGID TAPPING (cycle 17)
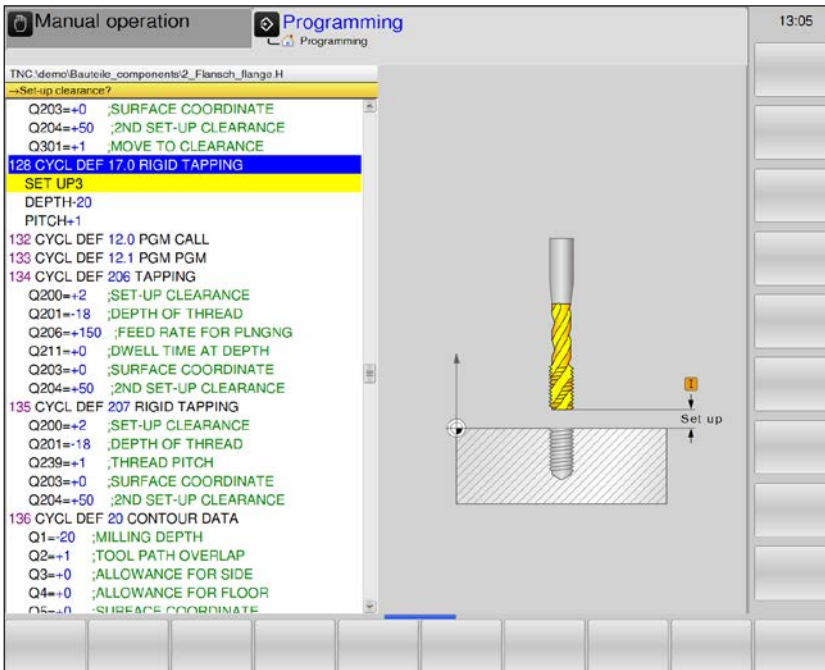
Parameter	Description	Unit
DIST	Safety distance **1** (incremental): Distance between tool point (= starting position) and work piece surface. Guideline value: 4x thread pitch	mm
DEPTH	Thread depth **2** (incremental): Distance work piece surface (thread start) - thread end.	mm
PITCH	Thread pitch **3**: Sign determines right-hand or left-hand thread: • + = Right-hand thread • - = Left-hand thread	mm
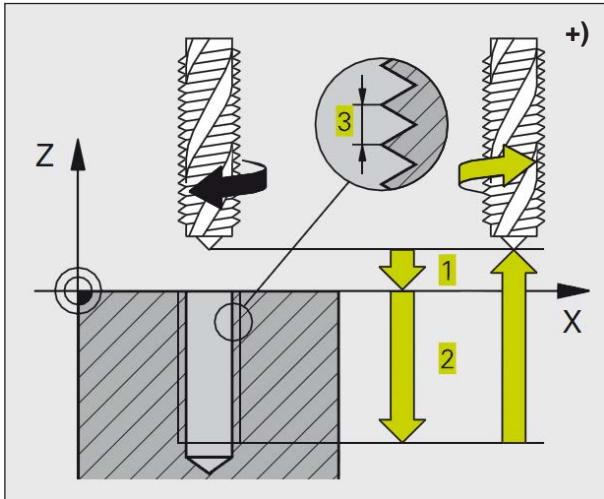
Cycle description

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.

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 3 | **SLOT MILLING (cycle 3)** |

Parameter	Description	Unit
DIST	Safety distance **1** (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Cutting depth **2** (incremental): Distance work piece surface - pocket bottom.	mm
INF.DT	Infeed depth **3**: Dimension by which the tool is fed in respectively. The control unit moves in one work step to the depth, if: • Infeed depth = depth • Infeed depth > depth	mm
F	Feed for depth infeed: Tool traverse speed during plunging	mm/min
X	1. Side length **4**: Length of the slot. Determine the cutting direction via the algebraic sign.	mm
Y	2nd Side length **5**: Width of the slot.	mm
F	Feed: Traverse speed of the tool in the machining plane.	mm/min
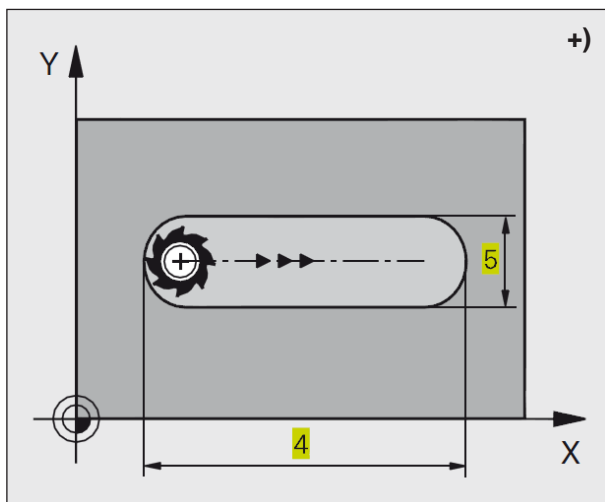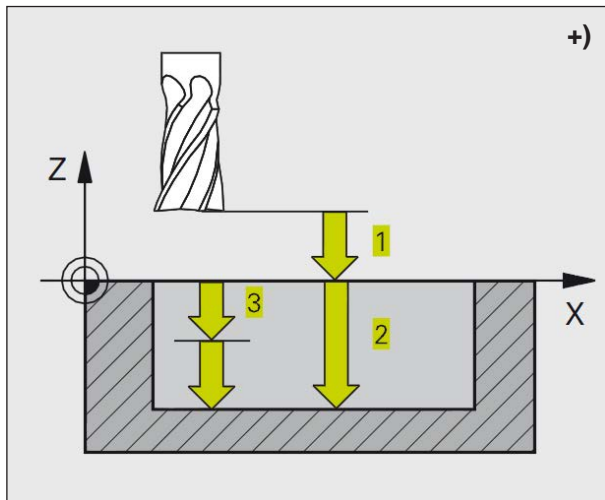
Cycle description
Roughing
1 WinNC shifts the tool by the finishing allowance (half the difference between slot width and tool diameter) toward the inside. From there, the tool cuts into the work piece and mills in longitudinal direction of the slot.
2 At the end of the slot, a depth infeed is executed, 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.

emco

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 4 | **POCKET MILLING (cycle 4)** |

Parameter	Description	Unit
DIST	Safety distance **1** (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Cutting depth **2** (incremental): Distance work piece surface - pocket bottom.	mm
INF.DT	Infeed depth **3**: Dimension by which the tool is fed in respectively. The control unit moves in one work step to the depth, if: • Infeed depth = depth • Infeed depth > depth	mm
F	Feed for depth infeed: Tool traverse speed during plunging:	mm/min
X	1. Side length **4**: Length of the pocket. Parallel to the main axis of the machining plane.	mm
Y	2nd Side length **5**: Width of the pocket.	mm
F	Feed: Traverse speed of the tool in the machining plane.	mm/min
DR	Rotation clockwise: • DR + : Down-cut milling at M3 • DR - : Up-cut milling at M3	
RADIUS	Rounding radius: Radius for the pocket corners. Radius =0: Rounding radius equals the tool radius.	
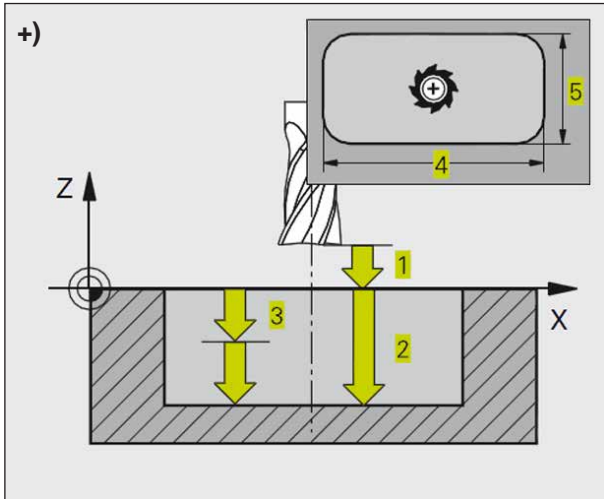
Cycle description
Roughing
1 The tool cuts into the work piece at the starting position (pocket centre) and moves to the first infeed depth.
2 Subsequently, the tool initially moves in the positive direction of the longer side, in case of square pockets in positive Y-direction, and then clears the pocket 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 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.
For the 2nd side length, the following applies:
2nd side length larger than (2x rounding radius + side infeed k)

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 5 | | **CIRCULAR POCKET (cycle 5)** |

Parameter	Description	Unit
DIST	Safety distance **1** (incremental): Distance between tool point (= starting position) and work piece surface.	mm
DEPTH	Cutting depth **2** (incremental): Distance work piece surface - pocket bottom.	mm
INF.DT	Infeed depth **3**: Dimension by which the tool is fed in respectively. The control unit moves in one work step to the depth, if: • Infeed depth = depth • Infeed depth > depth	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: • DR + : Down-cut milling at M3 • DR - : Up-cut milling at M3	

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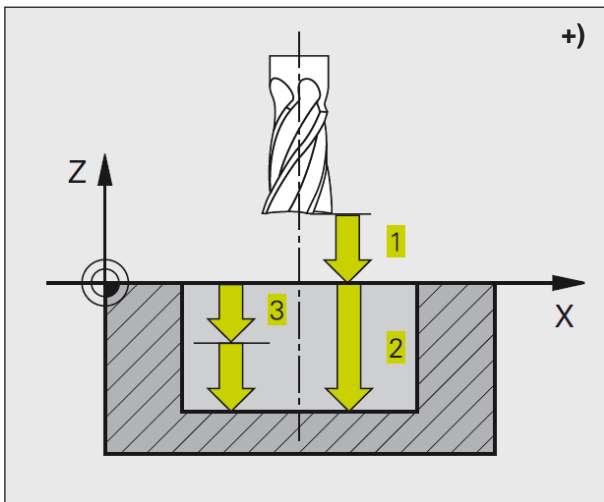
Note:
Prior to programming, please take note:
Use a centre-cut end mill (DIN 844) or pre-drilling at the starting point.
Program the positioning block on the starting point (= pocket centre) of the machining plane with radius compensation R0.
Program the positioning block on the starting point in the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

emco
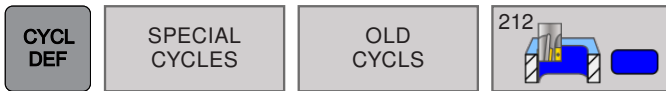
| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 212 | **POCKET FINISHING (cycle 212)** |

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
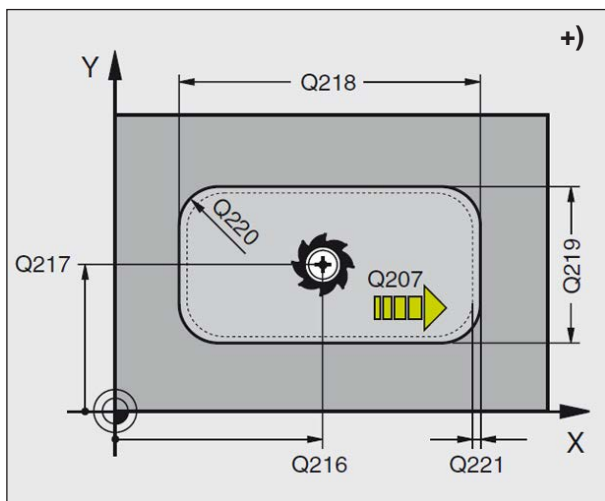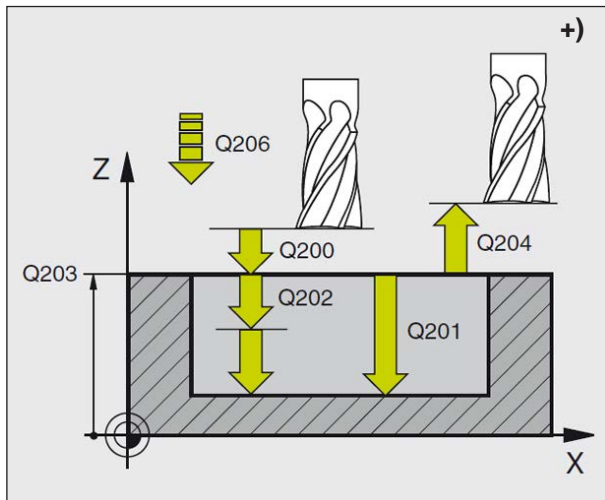
Cycle description

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.

emco

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 213 | **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.

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 214 | CIRCULAR POCKET FINISHING (cycle 214) |

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
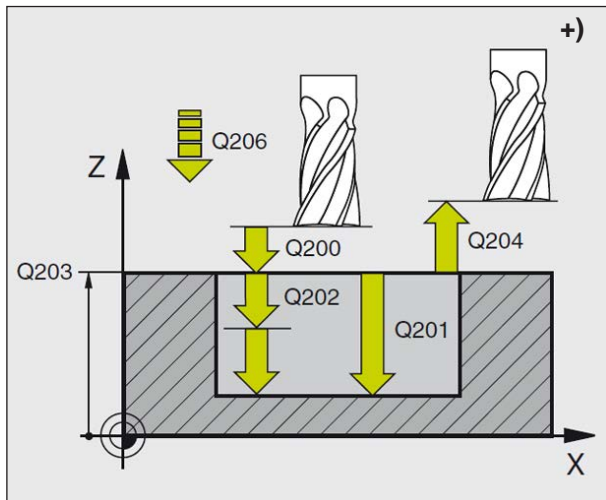
Cycle description

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.

---

emco

CYCL DEF	SPECIAL CYCLES	OLD CYCLS	215	**CIRCULAR STUD FINISHING (cycle 215)**

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. 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
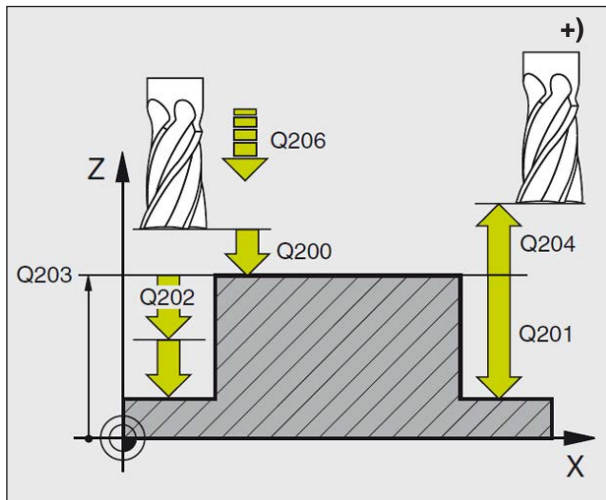
Cycle description

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.

emco

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 210 | **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.
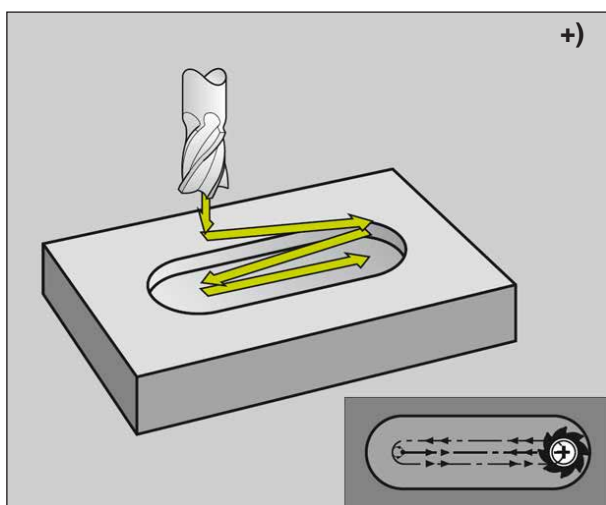
Finishing

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.

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 211 | **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: • 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
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.

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 230 | **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
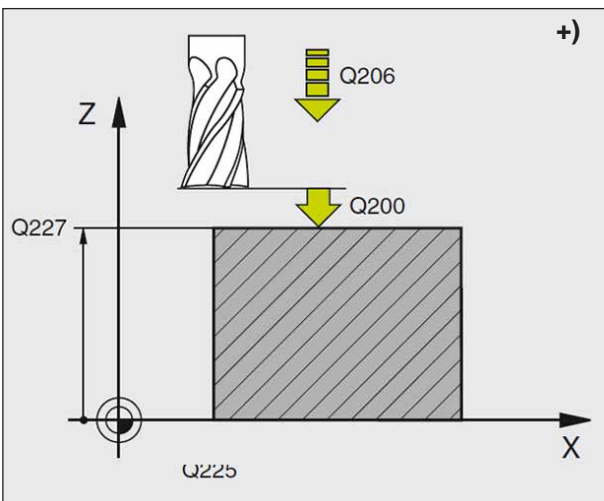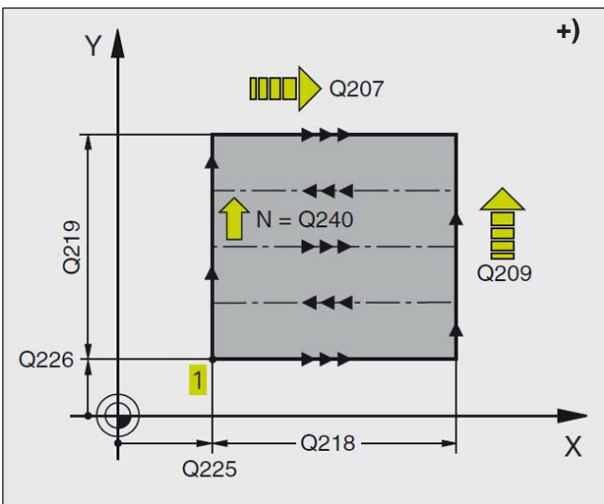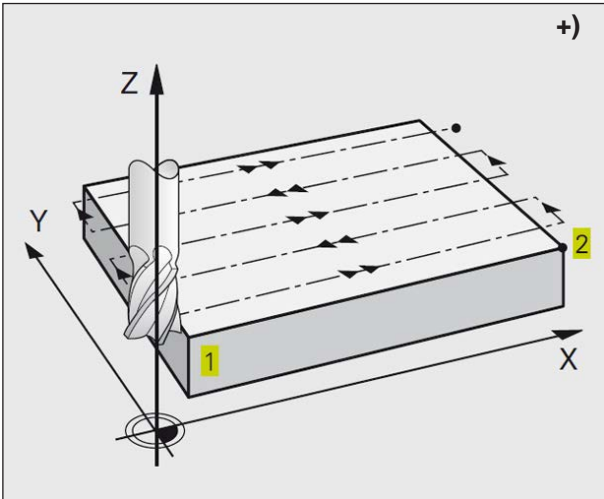
Cycle description

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.

| CYCL DEF | SPECIAL CYCLES | OLD CYCLS | 231 | **RULED SURFACE (cycle 231)** |
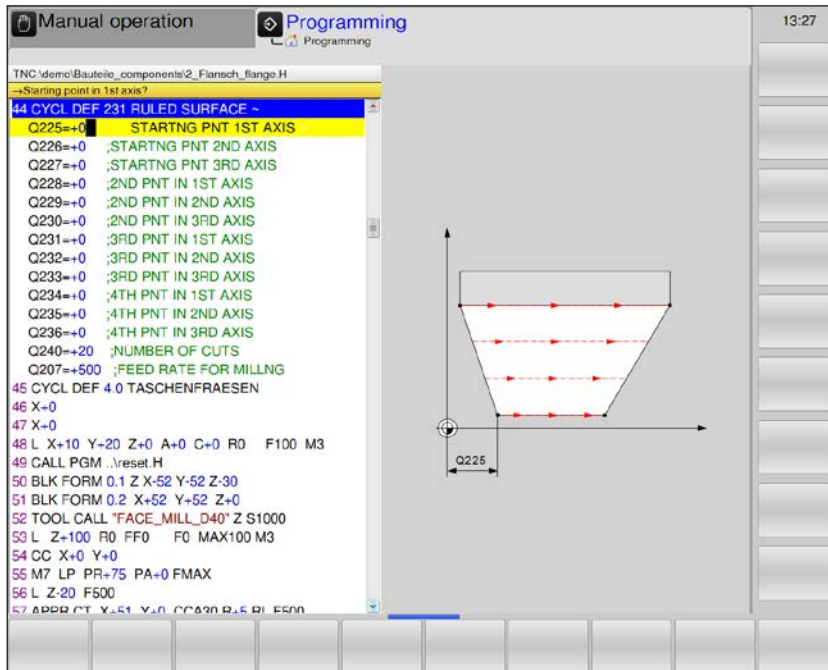
Parameter	Description	Unit
Q225	Starting point 1st axis (absolute): Starting point coordinate of the surface to be milled in the main axis of the machining plane.	mm
Q226	Starting point 2nd axis (absolute): Starting point coordinate of the surface to be milled in the minor axis of the machining plane.	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 **1** and **4** and/or between point **2** and **3**.	mm
Q207	Milling feed: Tool traverse speed during milling. WinNC executes the first step with half of the programmed value.	mm/min
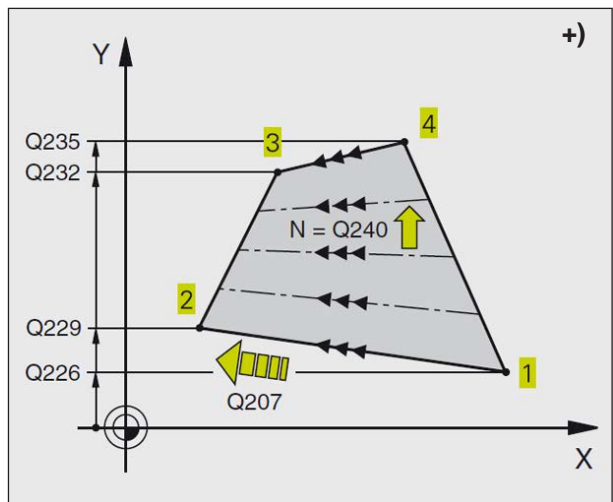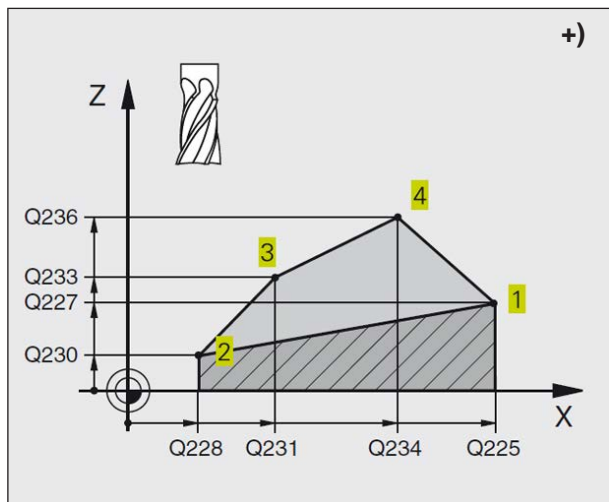
Note:

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 compensation R0 between the entered positions.

Possibly, use a centre-cut end mill (DIN 844).
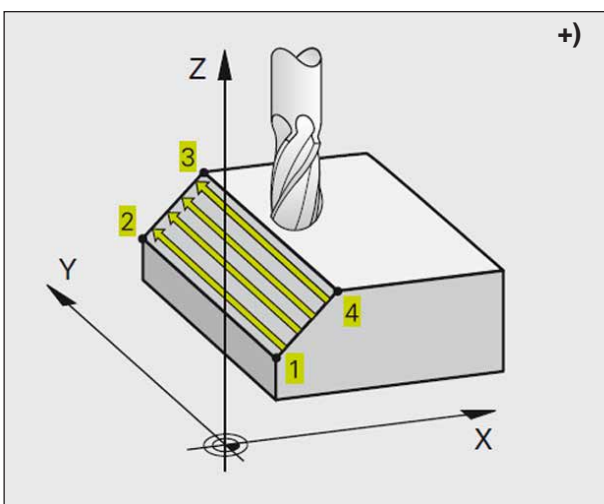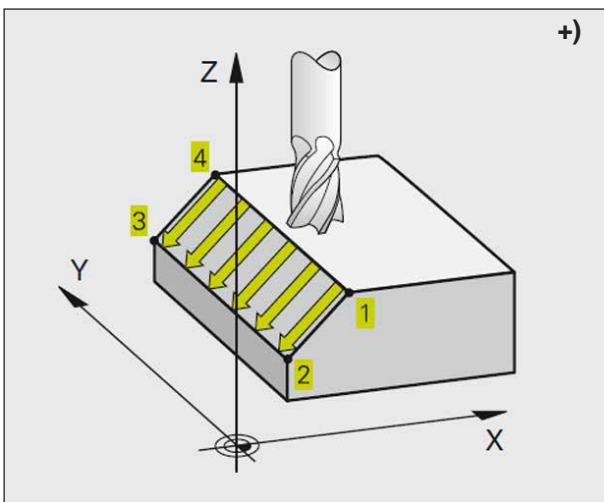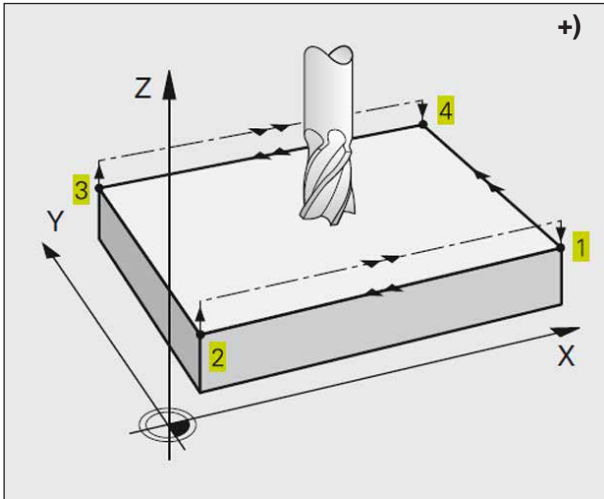
Cycle description

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.

```
0   BEGIN PGM ...                         *)
        ①
    CALL LBL1
        ②
    L Z+100 M2
    LBL1
        ③
    LBL0
    END PGM ...
```
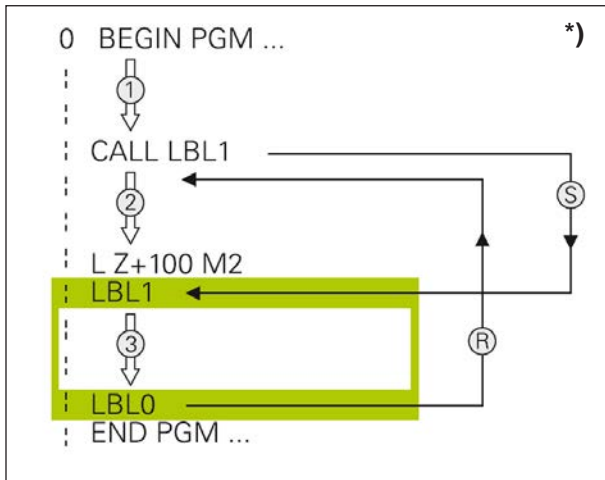
Subprograms *)

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.

LBL
SET

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.

LBL
CALL

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

0   BEGIN PGM ...   *)
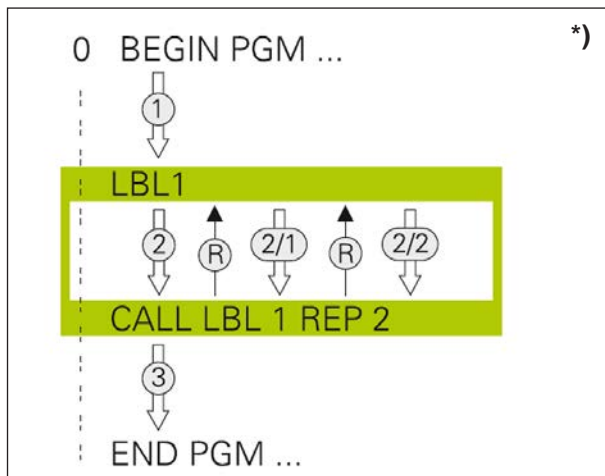
Program section repeats *)

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.

emco

*)
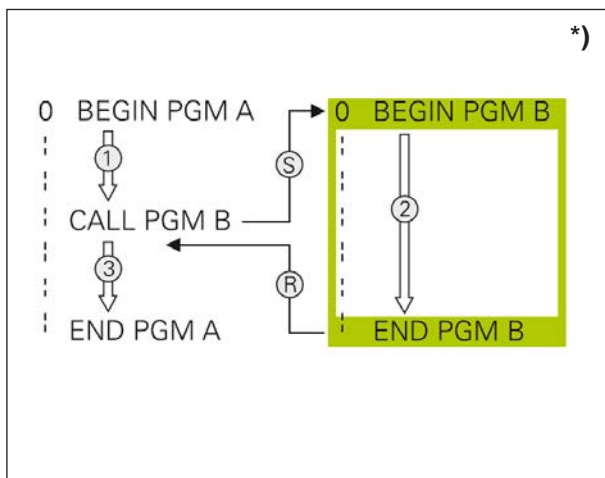
Any desired program as subprogram *)

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.

PGM
CALL

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
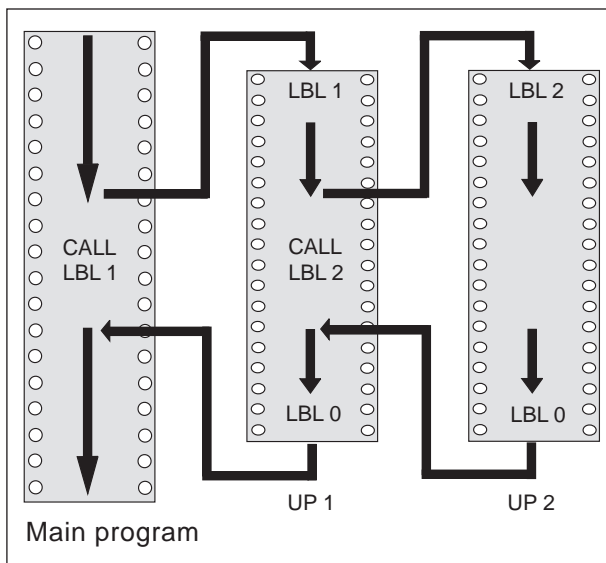
Program execution

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

emco

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	
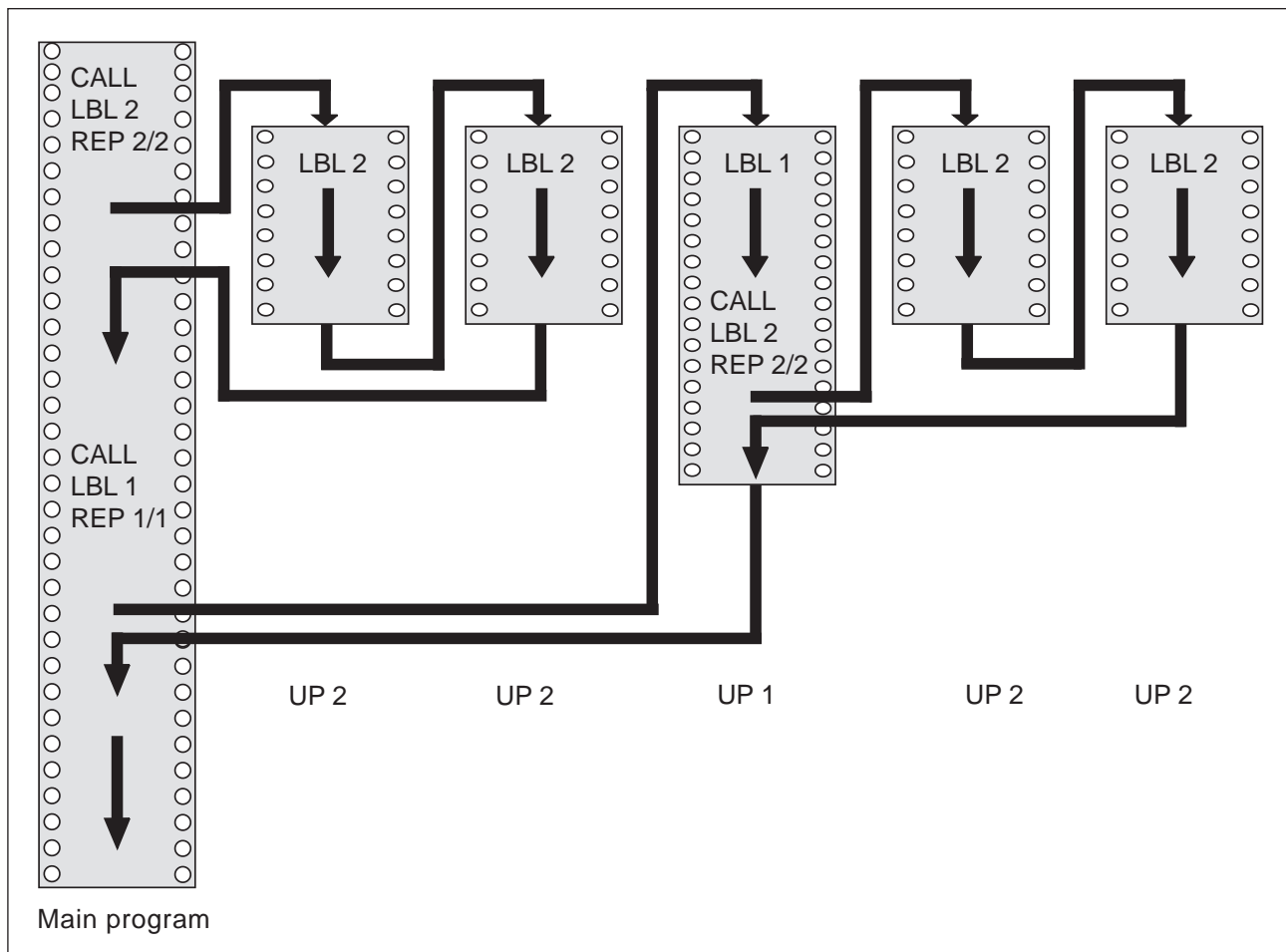
Program execution

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
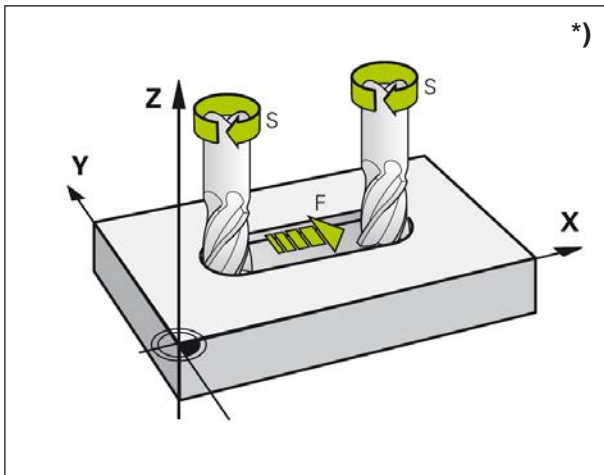
Feed and spindle speed

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.

emco

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.
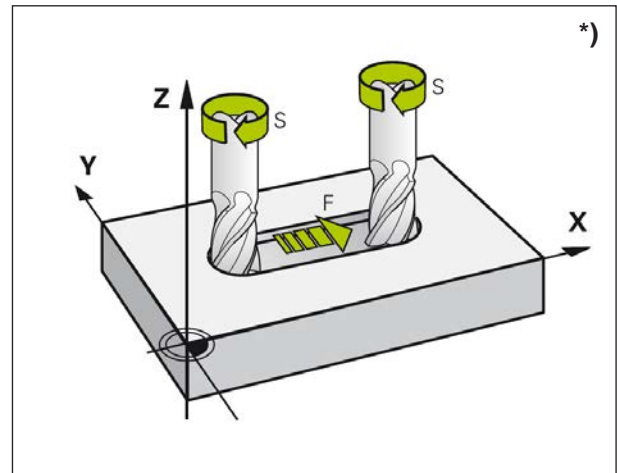
Feed and spindle speed

Tool data

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 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

emco
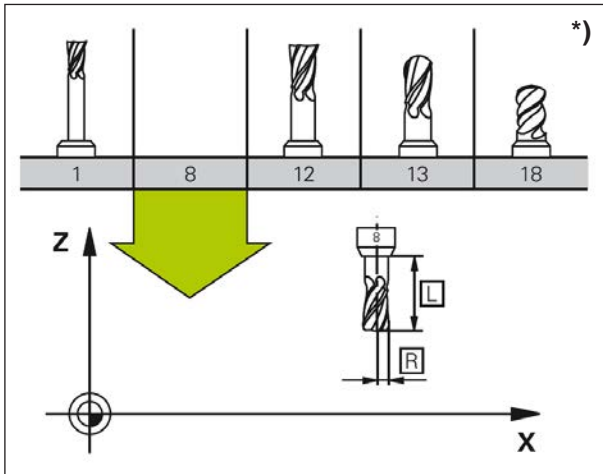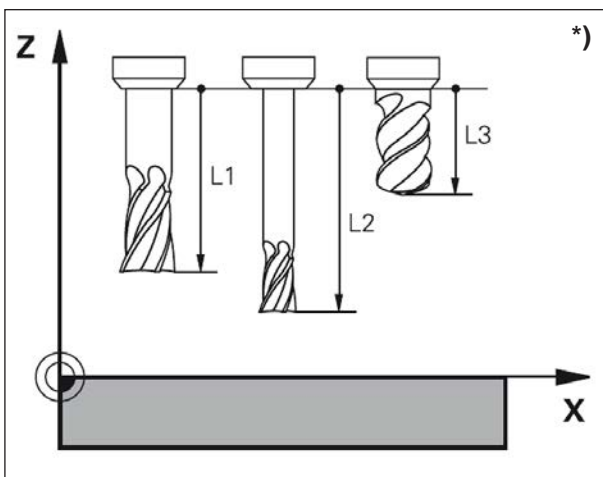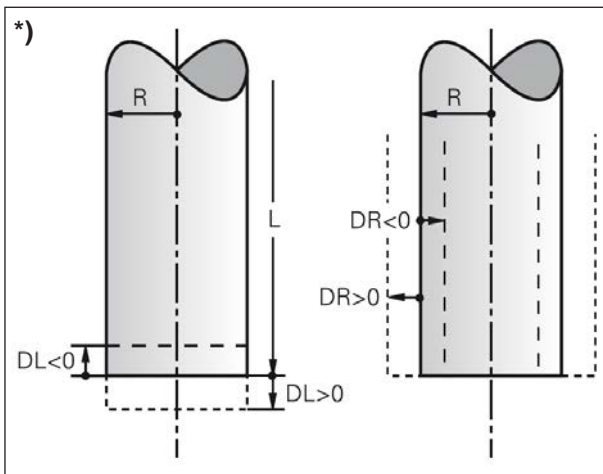
Tool radius

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 ± 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?**

emco

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".
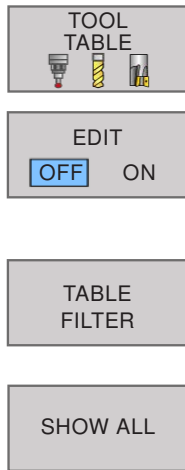
Display only certain tool types:
1 Switch the softkey bar until TABLE FILTER appears.
2 Select desired tool type via softkey.
3 Cancel filter: Press softkey SHOW ALL

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.
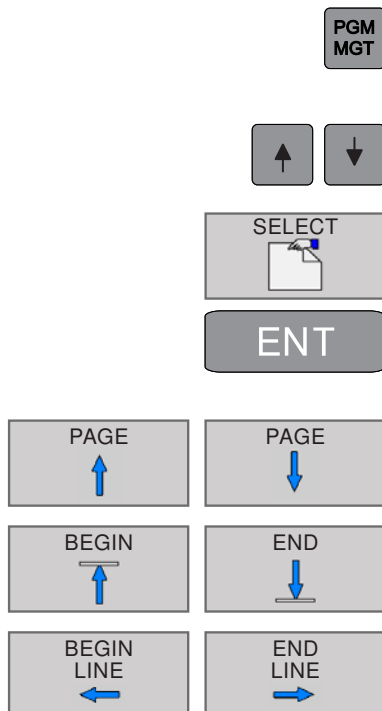
If you open a tool table for editing, you can move the illuminated field with the arrow keys or softkeys to any desired position. You can overwrite the saved values or enter new values on any desired position.
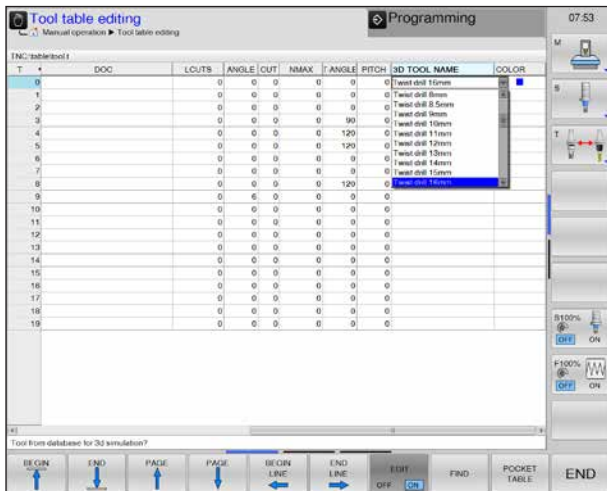
3D tool-name

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.
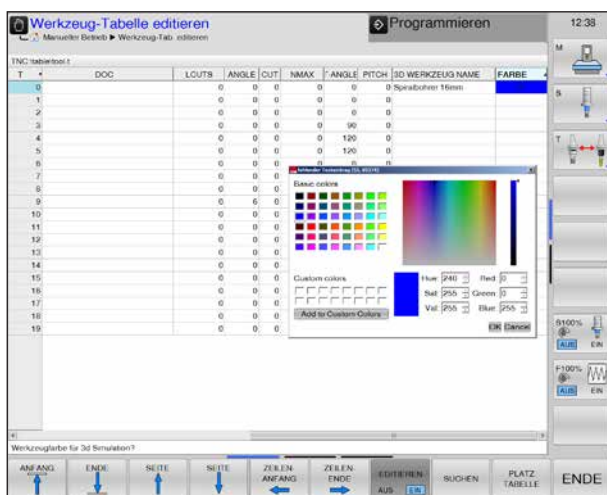
3D tool-color

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 an click to select. Alternatively the values for R,G,B can be entered manually.
• By clicking "Add to Custom Colors" a new color is added.

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 seperate table **TOOL.3d**.

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

emco

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.

| TOOL TABLE |

3 Press softkey to open POCKET TABLE.

| POCKET TABLE |

4 Set softkey EDIT to „ON".

| EDIT OFF ON |

Refer to the tool table to see functions for edit options of the pocket table.
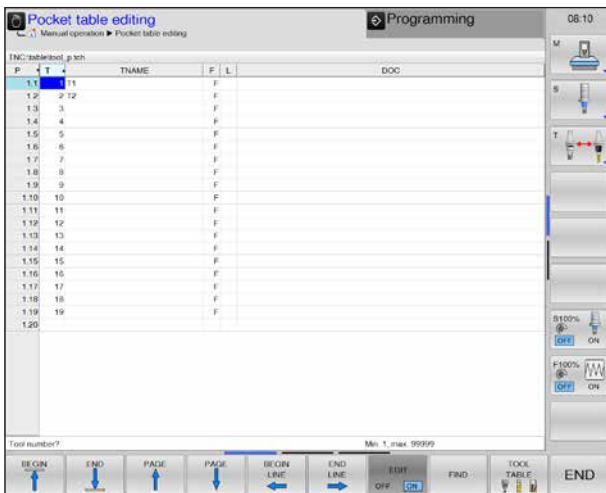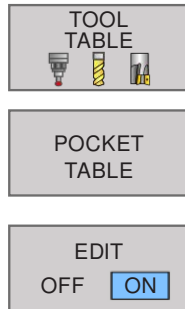
For automatic tool changing you need the pocket table.

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 transfered.

Tool change via softkey

In the vertical softkey bar it is possibile to change the tool via softkey.
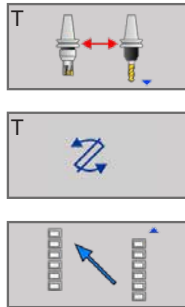
1 Press softkey for tool change.

2 Swivel in a new tool.

3 Return to the pocket table.

More functions in the horizontal soft-key bar
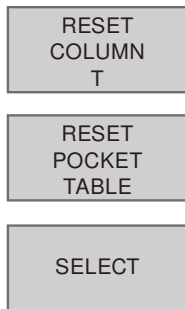
RESET COLUMN T

1 Position the cursor to column T and press softkey to reset the column.

RESET POCKET TABLE

2 Reset the entire table.

SELECT

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.
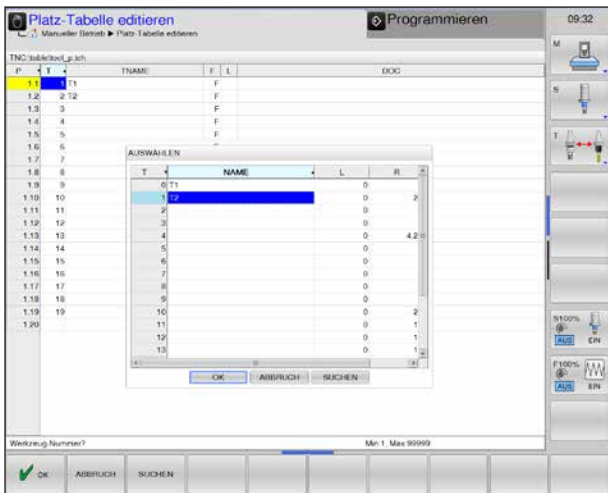
OK   CANCEL

4 Complete input by pressing OK or CANCEL your input.
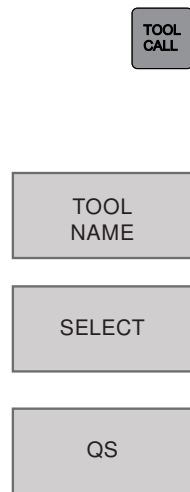
Call tool data *)

You have to program a **TOOL CALL** in the machining program by entering the following information:

TOOL
CALL

TOOL
NAME

SELECT

QS

- 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 correction

Tool compensation

Introduction *)

WinNC corrects the tool path by the compensation value for the tool length in the spindle axis and by the tool radius in the machining plane.

If you create the machining program directly in the WinNC, the tool radius compensation is only effective in the machining plane. In doing so, WinNC takes up to five axes, incl. the rotation axes, into account.
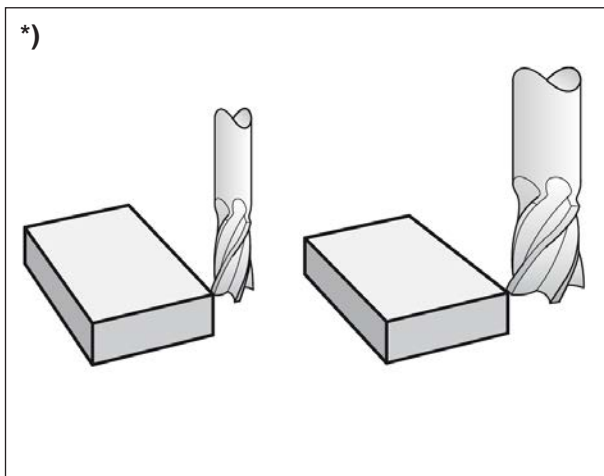
Tool length compensation

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

$DL_{TOOL\ CALL}$:   Allowance **DL** for length from **TOOL CALL** block

DL_{TAB} :   Allowance **DL** for length from the tool table

emco
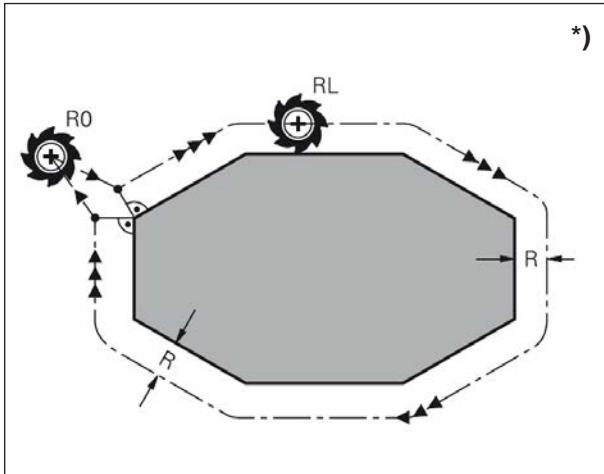
Tool radius compensation

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**.
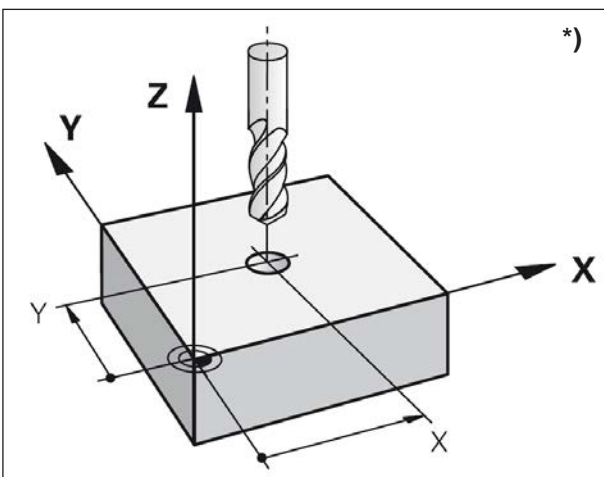
Tool radius compensation

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

$DR_{TOOL\ CALL}$:   Allowance **DR** for radius from the **TOOL CALL** block

DR_{TAB}:   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.
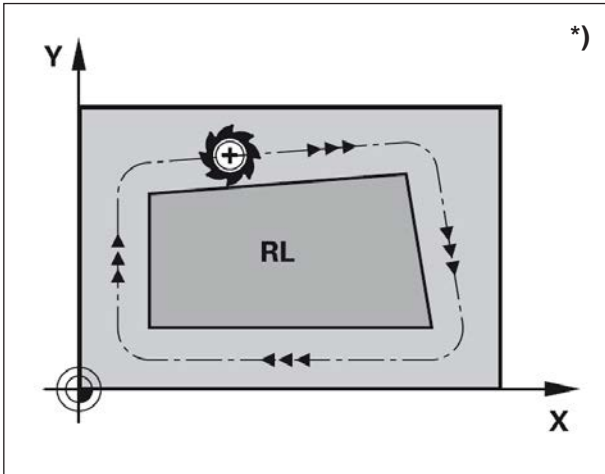
Path movements
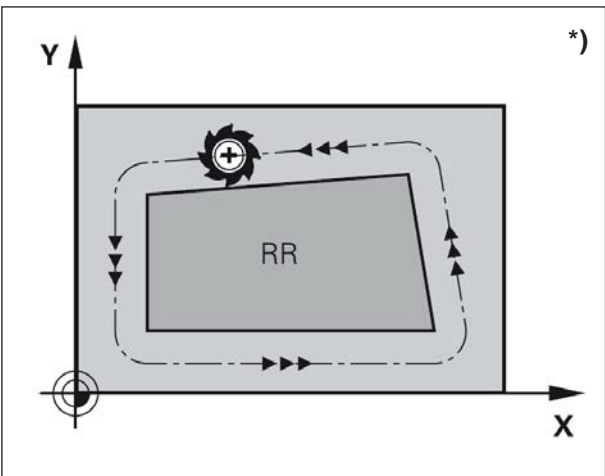
Path movements

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 ist 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

The radius compensation is entered via an **L** block. Confirm and finish entry of the target point coordinates with ENT.

Radius com.: RL/RR/No comp.?
Tool movement on left side of the programmed contour: Press RL softkey or

Tool movement on right side of the programmed contour: Press RR softkey or

Tool movement without radius compensation and/or cancel radius compensation: Press ENT key or softkey R0.

Finish block: Press END key.

E13
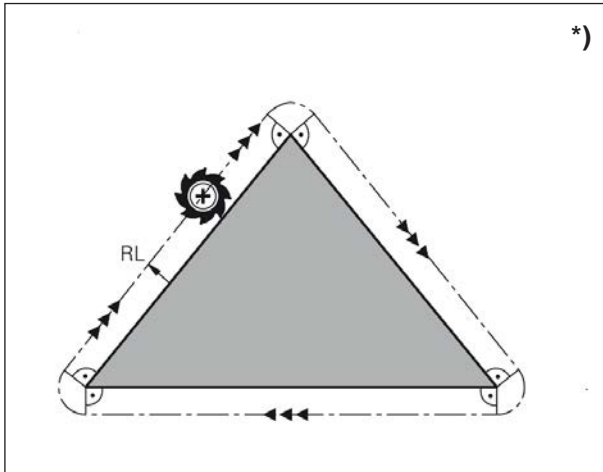
Machining corners

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.

emco

Program start, program stop
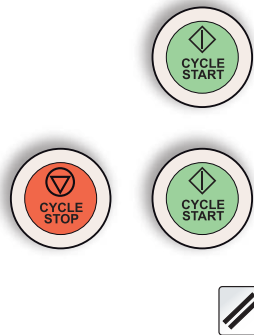
Press the "Cycle Start" button.

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.
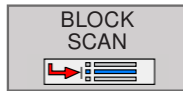
BLOCK
SCAN

• 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.
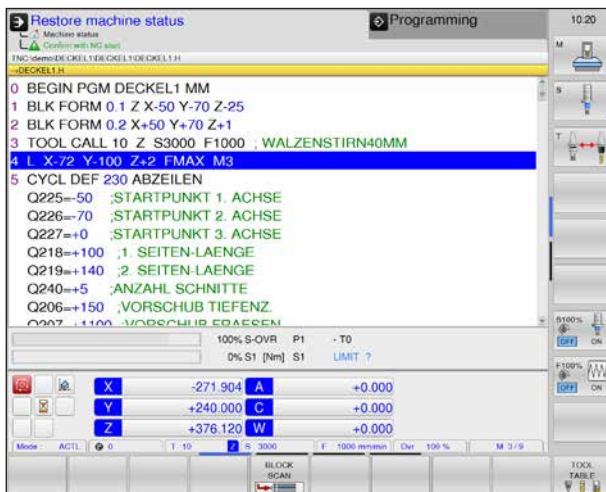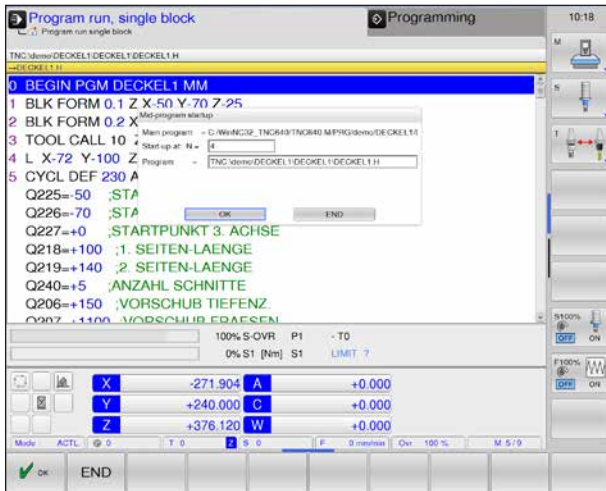
After executing block scan the following message appears: "Restore machine status".

Press NC-start to confirm restore machine status.

emco

F MAX   *)

• Switch the softkey bar until the function for F MAX appears.

F MAX

• 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.
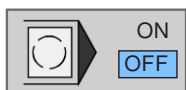
Do not interrupt program run or test run at blocks containing M1: Set soft key to OFF

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 fro 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

emco

Calling up Q parameters

Q

Press the Q key during the input of a machining program.

The following functions are available:

1 Basic mathematical functions

BASIC ARITHM.

2 Trigonometric functions

TRIGO-NOMETRY

3 If / then decisions, jumps

JUMP

4 Function string formula
Before string variables are used, they must be defined:
example: QS10="WERKSTÜCK"

STRING FORMULA

Basic mathematical functions

The following functions are available:

FN0
X=Y

1 FN 0: Assignment
Directly assign a value
e.g.: FN 0: Q5= +60

FN1
X+Y

2 FN 1: Addition
Create the sum of two values and assign it
e.g.: FN 1: Q1 = -Q2= +5

FN2
X-Y

3 FN 2: Subtraction
Create the difference of two values and assign it
e.g.: FN 2: Q1 = +10 - +5

FN3
X*Y

4 FN 3: Multiplication
Create the product of two values and assign it
e.g.: FN 3: Q2 = +3 * +3

FN4
X/Y

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!

FN5
SQRT

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)**

1 FN 6: SINE
Determine the sine of an angle in degrees ° and assign it
e.g.: FN 6: Q20= SIN-Q5

**FN7
COS(X)**

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: ....$\tan \alpha = a / b = \sin \alpha / \cos \alpha$
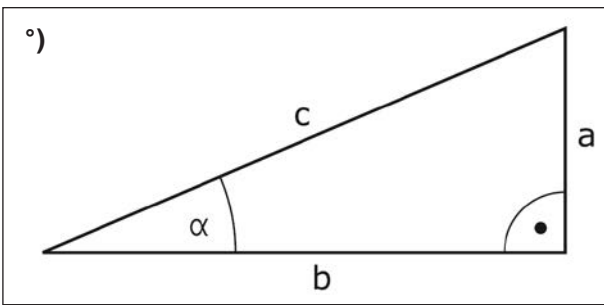
In this case:

- **c** is the side opposite the right angle
- **a** is the side opposite angle α
- **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 mm
b = 50 mm
$\alpha = \arctan (a / b) = \arctan 0.5 = 26.57°$

Additionally, the following applies:
$a^2 + b^2 = c^2$ (with $a^2 = a \times a$)

°)

emco

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:

JUMP

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

FN9 IF X EQU Y GOTO

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

FN10 IF X NE Y GOTO

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

FN11 IF X GT Y GOTO

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

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

emco

6022: CLAMPING DEVICE BOARD DEFECTIVE
The signal "clamping device clamped" is constantly released, although no command has been given.
Replace the board.

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

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

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

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

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

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

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

6047: TOOL DISK UNLOCKED
Tool drum turned out of locked position, inductive proximity switch defective or disadjusted, fuse defective, hardware defective.
A running CNC program will be interrupted.
Contact EMCO service.
When the tool drum is turned out of locked position (no defect), act as following:

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

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

6049: INTERLOCKING TIME EXCEEDED
see alarm 6048

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

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

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

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

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

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

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

6074: DIVIDING TIME EXCEEDED
Cause: dividing device stucks mechanically
          locking switch defective
          cabling defective
          fuses defective
          insufficient compressed-air supply.
A running program will be aborted.
The auxilliary drives will be switched off.
Remedy: Check for collision, check the compressed-
          air supply or contact the EMCO service.

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

6110: 5TH AXIS NOT CONNECTED
Cause: 4th/5th axis was selected in EMConfig, but
          not connected electrically.
Remedy: Connect 4th/5th axis or deselect in 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 depart-
          ment.

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.

emco

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

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

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

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

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

7051: DIVIDING HEAD NOT LOCKED!
Either the dividing head is in an undefined position after the machine has been switched on, or the locking signal after a dividing process is missing.
Initiate the dividing process, check, respectively adjust the proximity switch for locking.

7054: VICE OPEN
Cause: the workpiece is not clamped
When switching on the main spindle with M3/M4 alarm 6072 (vice not ready) will be released.
Remedy: Clamp

7055: OPEN TOOL CLAMPING SYSTEM
A tool is clamped in the main spindle and the control does not recognize the corresponding T number.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7056: SETTING DATA INCORRECT
An invalid tool number is stored in the setting data.
Delete the setting data in the machine directory xxxxx.pls.

7057: TOOLHOLDER OCCUPIED
The clamped tool cannot be positioned in the tool turret since the position is occupied.
Eject the tool from the main spindle when the door is open by means of the PC keys "Strg" and " 1 ".

7058: RETRACTING THE AXES
The position of the tool turret arm cannot be clearly defined during the tool change.
Open the machine door, push the tool turret magazine backwards to the stop. Move the milling head in the JOG mode upwards to the Z reference switch and then traverse the reference point.

7087: MOTOR PROTECTION HYDRAULIC CLAMPING RELEASED!
Hydraulic motor is defective, stiff, circuit breaker is set incorrectly.
Replace motor or check circuit breaker and replace if necessary.

7090: ELECTRICAL CABINET OVERRIDE SWITCH ACTIVE
The cabinet door can only be opened when the key switch is switched on without raising an alarm.
Switch off key switch.

7107: OTOR PROTECTION RELEASED
A motor protection switch has triggered. A possibly running NC program will be completed. A new NC start will be prevented.
Check the device connected to the motor protection switch and switch on again. If recurring, contact EMCO service department.

7270: OFFSET COMPENSATION ACTIVE !
Only with PC-MILL 105
Offset compensation activated by the following operation sequence.
- Reference point not active
- Machine in reference mode
- Key switch in manual operation
- Press STRG (or CTRL) and simultaneously 4
This must be carried out if prior to the tool change procedure spindle positioning is not completed (tolerance window too large)

7271: COMPENSATION FINISHED, DATA SAVED !
see 7270

PC TURN 50 / 55 / 105 / 120 / 125 / 155
Concept TURN 55 / 60 / 105 / 155 / 250 / 260 / 460
Concept MILL 250
EMCOMAT E160
EMCOMAT E200
EMCOMILL C40
EMCOMAT FB-450 / FB-600

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

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

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

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

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

6005: K2 OR K3 NOT DE-ENERGIZED
Turn machine on/off. Defective security board.

6006 EMERGENCY-OFF RELAY K1 NOT DE-ENERGIZED
Turn machine on/off. Defective security board.

6007 SAFETY CIRCUIT FAULT

6008: MISSING CAN SUBSCRIBER
The PLC-CAN board is not identified by the control.
Check the interface cable and the power supply of the CAN board.

6009: SAFETY CIRCUIT FAULT

6010: DRIVE X-AXIS NOT READY
The step motor board is defective or too hot, a fuse is defective, over- or undervoltage from mains.

A running program will be stopped, the auxiliary drives will be switched off, the reference position will be lost.
Check fuses or contact EMCO service.

6011: DRIVE Z-AXIS NOT READY
see 6010.

6012: DRIVE Z-AXIS NOT READY
see 6010.

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

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

6015: NO DRIVEN TOOL SPINDLE SPEED
see 6014.

6016: AUTOMATIC TOOL TURRET SIGNAL COUPLED MISSING

6017: AUTOMATIC TOOL TURRET SIGNAL UNCOUPLED MISSING
In the tool turret that can be coupled, the position of the coupling and uncoupling magnet is monitored by means of two proximity switches. It has to be made sure that the coupling is in the rear stop position so that the tool turret can get to the next tool position. Equally, during operation with driven tools the coupling has to be safe in the front stop position.
Check and adjust the cables, the magnet and the stop position proximity switches.

6018: AS SIGNALS, K4 OR K5 NOT DE-ENERGIZED
Turn machine on/off. Defective security board.

6019: POWER SUPPLY MODULE NOT READY
Turn machine on/off. Power supply module, defective axis controller 6020 AWZ drive failure turn machine on/off, defective axis controller.

6020: DRIVEN TOOL DRIVE MALFUNCTION
The driven tool power supply defective or the driven tool drive is too hot, fuse defective, mains overvoltage or undervoltage. A running CNC program will be aborted, the auxiliary drives will be stopped. Check the fuses or contact EMCO service.

6021: COLLET TIME OUT
During closing of the clamping device the pressure switch has not reacted within one second.

6022: CLAMPING DEVICE BOARD DEFECTIVE
The signal "clamping device clamped" is constantly released, even though no command has been given. Replace the board.

6023: COLLET PRESSURE MONITORING
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

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

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

6026: MOTOR PROTECTION COOLANT PUMP RELEASED

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

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

6029: TAILSTOCK QUILL TIME EXCEED
The tailstock quill does not reach a final position within 10 seconds.
Adjust the control and the stop position proximity switches, or the tailstock quill is stuck.

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

6031: QUILL FAILURE

6032: TOOL CHANGE TIMEOUT
see alarm 6041.

6033: TOOL TURRET SYNC ERROR
Hardware defective.
Contact EMCO service.

6037: CHUCK TIMEOUT
The pressure switch does not react within one second when the clamping device is closed.

6039: CHUCK PRESSURE FAILURE
The pressure switch turns off when the clamping device is closed (compressed air failure for more than 500ms).

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

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

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

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

6044: BRAKING RESISTANCE - MAIN DRIVE OVERLOADED
Reduce number of speed changes in the program.

6045: TOOL TURRET SYNC MISSING
Hardware defective.
Contact EMCO service.

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

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

6049: COLLET NOT READY
see 6048

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

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

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

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

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

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

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

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

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

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

6067: NO COMPRESSED AIR
Turn the compressed air on, check the setting of the pressure switch.

6068: MAINDRIVE OVERTEMPERATURE

6070: LIMIT SWITCH TAILSTOCK SLEEVE ACTIVE
Cause: The axis arrived in the tailstock sleeve.
Remedy: Drive the travel off the tailstock sleeve.

6071: LIMIT SWITCH X AXIS ACTIVE
Cause: The axis arrived to the end switch.
Remedy: Drive the axis off the end switch again.

6072: LIMIT SWITCH Z AXIS ACTIVE
see 6071

6073: CHUCK GUARD OPEN
Cause: The chuck guard is open.
Remedy: Close the chuck guard.

6074: NO FEEDBACK FROM USB-PLC
Turn machine on/off. Check cabling, defective USB board.

6075: AXIS LIMIT SWITCH TRIGGERED
see 6071

6076: DRIVE Y-AXIS NOT READY
see 6010

6077 VICE NOT READY
Cause: Loss of pressure in clamping system.
Remedy: Check pressurised air and air ducts.

**6078 MOTOR PROTECTION TOOL MAGA-
ZINE RELEASED**
Cause: Swing intervals are too short.
Remedy: Raise swing intervals.

**6079 MOTOR PROTECTION TOOL CHANG-
ER RELEASED**
see 6068

**6080 PRESSURE SWITCH FOR TANI MISS-
ING**
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 sev-
        eral 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 sev-
        eral 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 sev-
        eral 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 ma-
        chine 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.

emco

6097 EMERGENCY-OFF TEST REQUIRED

Cause: Functional test of Emergency off.
Remedy: Press EMERGENCY-OFF button on the control panel and unlock again. Press Rest-button to acknowledge the emergency stop state.

6098 FLOAT SWITCH HYDRAULIC MISSING

Effect: Auxiliary units are switched off.
Meaning: The hydraulic float switch has released.
Remedy: Refill hydraulic oil.

6099 PROX. SWITCH SPINDLE-BRAKE 1 MISSING

Effect: Feed stop, Read in lock
Meaning: M10 Spindle brake on ➙ Inductive proximity switch stays 0.
M11 Spindle brake off ➙ Inductive proximity switch stays 1.
Remedy: Check inductive proximity switch, check magnetiv valve spindle brake.

6100 LOW PRESSURE QUILL

Effect: Auxiliary units are switched off.
Meaning: At the moment of a spindle start command the tailstock pressure was not built up or the pressure dropped while spindle run.
Remedy: Check clamping device pressure and pressure switch.
Check program.

6101 QUILL -B3 OR -B4 MISSING

Effect: Feed stop, Read in lock
Meaning: The magnetic valve for the movement of the quill was activated, but the switches –B3 and –B4 does not alter its status.
Remedy: Check switches, magnetic valves.

6102 QUILL POSITION ALARM (PART MOVED?)

Effect: Feed stop, Read in lock
Meaning: The tailstock target position was overtravelled in AUTOMATIC mode.
Remedy: Check tailstock target position, check technology (clamping device pressure higher, tailstock pressure lower)

6103 QUILL NO BACKPOSITION

Effect: Feed stop, Read in lock
Meaning: The magnetic valve for the tailstock was activated, but the switch for "tailstock back" stays 0.
Remedy: Check magnetic valve, check switch.

6104 LOW PRESSURE CLAMPING EQUIPMENT 1

Effect: Auxiliary units are switched off.
Meaning: At the moment of a spindle start command the clamping pressure was not built up or the clamping pressure dropped while spindle run.
Remedy: Check clamping device pressure and pressure switch.
Check program.

6105 CLAMPING EQUIPMENT 1 NOT OPEN

Effect: Feed stop, Read in lock
Meaning: The analogous proximity switch for clamping device 1 was not actuated.
Remedy: Adjust inductive proximity switch (see "Machine Description - chapter "C Programming and operation").

6106 CLAMPING EQUIPMENT 1 NOT CLOSED

Effect: Feed stop, Read in lock
Meaning: The pressure switch for "clamping device closed" does not switch.
Remedy: Check pressure switch

6107 LIMIT SWITCH CLAMPING EQUIPMENT 1

Effect: Auxiliary units are switched off.
Remedy: Adjust clamping device (do not clamp in end position of the clamping system - see "Machine Description, chapter C Programming and Operation").

6108 PARTS CATCHER FORWARD MISSING

Effect: Feed stop, Read in lock
Meaning: The magnetic valve for "collection tray forward/backward" was activated, but the switch for "collection tray forward/backward" does not alter its status.
Remedy: Check switches, magnetic valves.

6109 PARTS CATCHER ROTATE IN MISSING

Effect: Feed stop, Read in lock
Meaning: The magnetic valve for "collection tray in/out" was activated, but the switch for "collection tray in/out" does not alter its status.
Remedy: Check switch, magnetic valve.

6900 USBPLC not available

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

6901 Error emergency-off relay USBPLC
Cause: USBPLC EMERGENCY-OFF relay error.
Remedy: Switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6902 Error standstill monitoring X
Cause: Unauthorized movement of the X axis in the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6903 Error standstill monitoring Z
Cause: Unauthorized movement of the Z axis in the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6904 Error alive circuit PLC
Cause: Error in the connection (Watchdog) of the safety board with the PLC.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6906 Error overspeed spindle
Cause: The main spindle speed exceeds the maximum permissible value for the current operating condition.

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

6907 Error enable pulses ER-module
Cause: ACC-PLC did not shutdown the input/negative feeder-module.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6908 Error standstill monitoring main drive
Cause: Unexpeced warm up of the main spindle in the operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6909 Error main drive enable without spindle start
Cause: The release of the control unit of the main spindle was given by the ACC-PLC without the spindle-start key being pressed.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6910 Error standstill monitoring Y
Cause: Unauthorized movement of the Y axis in the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6911 Error standstill axes
Cause: Unauthorized movement of the axis in the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6912 Error overspeed axis
Cause: The feed of the axes exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6913 Error overspeed X
Cause: The feed of the X axis exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6914 Error overspeed Y
Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

emco

6915 Error overspeed Z
Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6916 ERROR: X-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by X axis Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6917 ERROR: Y-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by Y axis Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6918 ERROR: Z-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by Z axis Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6919 ERROR: SPINDLE-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by main spindle Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6920 INVERSION OF X-DIRECTION TOO LONG "1"
Cause: The change in direction of X axis was being sent to USBSPS for more than three seconds.
Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6921 INVERSION OF Y-DIRECTION TOO LONG "1"
Cause: The change in direction oY axis was being sent to USBSPS for more than three seconds.
Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6922 INVERSION OF Z-DIRECTION TOO LONG "1"
Cause: The change in direction of Z axis was being sent to USBSPS for more than three seconds.
Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6923 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC
Cause: ACC-PLC and USBSPS receive different door status reports.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6924 ERROR ENABLE PULSES MAIN DRIVE
Cause: The pulse release on the main spindle control element was interrupted by the USBSPS, as the PLC did not shut it down in a timely fashion.
Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

6925 GRID PROTECTION ERROR!
Cause: Grid protection does not drop out in current operating state, or does not engage.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6926 MOTOR PROTECTION ERROR!
Cause: Motor protection drops out in current operating state.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6927 EMERGENCY OFF ACTIVE ERROR!
Cause: Emergency off button was pressed.
Remedy: Restart the machine.

6928 TOOL CHANGER SHUTDOWN MONITORING ERROR
Cause: Unauthorised tool changer movement in the current operating state.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6929 MACHINE DOOR CLOSING/LOCKING ERROR
Cause: State of the door lock not plausible or door closure unserviceable.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6930 BEROS MAIN SPINDLE PLAUSIBILITY ERROR
Cause: Beros main spindle signal different.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6931 MAIN DRIVE QUICK STOP FUNCTION PLAUSIBILITY ERROR
Cause: Main drive actuator does not confirm the quick stop function in the current operating state.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6999 USB-EXTENSION FOR ROBOTIK NOT AVAILABLE
Cause: The USB extension for robotics cannot be addressed by ACC.
Remedy: Contact EMCO.

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

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

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

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

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

7019: PNEUMATIC LUBRICATION MONITORING!
Refill pneumatic oil

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

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

7022: COLLECTION DEVICE MONITORING
Time exceed of the swivelling movement.
Check the pneumatics, respectively whether the mechanical system is jammed (possibly a workpiece is jammed).

7023: ADJUST PRESSURE SWITCH !
During opening and closing of the clamping device the pressure switch has to turn off and on once.
Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !
When the clamping device is open and the position stop control is active, the respective proximity switch has to feed back that the clamping device is "Open".
Check and adjust the clamping device proximity switch, check the cables.

7025 WAITING TIME MAIN DRIVE !
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

7026 PROTECTION MAIN MOTOR FAN RELEASED!

EMCO

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 opera-
tion 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 tur-
ret 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. Trav-
erse the tailstock back to the back end position
to continue.

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

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

7060 RETRACT SLEEVE LIMIT SWITCH !
The axis arrived in the tailstock sleeve. Drive the
travel off the tailstock sleeve.

7061 RETRACT X AXIS LIMIT SWITCH !
The axis arrived to the end switch. Drive the axis
off the end switch again.

7062 RETRACT Z AXIS LIMIT SWITCH !
see 7061

7063 OIL LEVEL CENTRAL LUBRICATION !
Low oil level in central lubrication. Refill oil as per
maintenance instructions to the machine.

7064 CHUCK GUARD OPEN !
The chuck guard is open. Close the chuck guard.

**7065 MOTOR PROTECTION COOLANT
      PUMP RELEASED !**
Overheated coolant pump. Check the coolant
pump for ease of motion and presence of dirt.
Ensure sufficient amount of coolant fluid in the
coolant facility.

7066 CONFIRM TOOL !
To confirm the tool change, press T after the
change has been completed.

7067 MANUAL OPERATING MODE
The Special Operation key switch is in the Set
position (manual).

7068 X AXIS HANDWHEEL ACTIVE
The safety wheel is locked for manual travel
movement. The safety wheel locking is monitored
by contactless switches. With the manual wheel
locked, the axis feed cannot be switched on. For
automatic processing of a program, the manual
wheel must be released again.

7069 Y AXIS HANDWHEEL ACTIVE
see 7068

7070   Z AXIS HANDWHEEL ACTIVE
see 7068

7071   VERTICAL TOOL CHANGE
The sheath for manual clamping of the tool holder is monitored by a switch. The switch reports a unaccepted socket wrench or a sheath which was left open. Remove the socket wrench after clamping the tool and close the sheath.

7072   HORIZONTAL TOOL CHANGE
The turning knob for manual tool clamping on the horizontal spindle is monitored by a switch. The switch reports a tightened turning knob. The spindle gets locked. Release the turning knob after clamping the tool.

7073   RETRACT Y AXIS LIMIT SWITCH !
see 7061

7074   CHANGE TOOL
Clamp programmed tool.

7076:   SWIVEL UNIT VOR MILLING HEAD UNLOCKED
The milling head is not fully swung. Fix the milling head mechanically (the end switch must be pushed).

7077:   ADJUST TOOL TURRET
No valid machine data for tool change are available. Contact EMCO.

7078:   POCKET NOT IN HOME POSITION
Cancel during tool change. Swing back tool recessed in setup operation.

7079:   TOOL ARM NOT IN HOME POSITION
see 7079

7080:   INCORRECT TOOL CLAMPED !
The tool cone lies beyond tolerance. The clamped tool is twisted by 180°. Bero tool clamping is displaced. Check the tool and clamp it again. If this problem occurs with more tools, contact EMCO.

7082:   MOTOR PROTECTION CHIP CONVEYOR RELEASED
The scrap belt is overloaded. Check the conveyor belt for ease of motion and remove jammed scrap.

7083:   MAGAZINE IS ACTIVE !
A tool has been removed from the non-chaotic tool administration from the main spindle. Fill the tool drum.

7084:   VICE OPEN !
The vice is not clamped. Clamp the vice.

7085   ROUNDAXIS A MOVE TO 0 DEGRE !
Cause: The MOC only shuts down if the A Round axis is at 0°.
When 4.5. is present, a round axis must be made each time before the machine is switched off.
Remedy: Move round axis to 0°.

7088   SWITCHGEAR CABINET OVERHEATING
Cause: Temperature monitoring responded.
Remedy: Check switchgear cabinet filter and fan, raise trigger temperature.

7089   SWITCHGEAR CABINET DOOR OPEN
Cause: Switchgear cabinet door open.
Remedy: Close switchgear cabinet door.

7091   WAIT FOR USB-I2C SPS
Cause: Communication with the USB-I2C PLC could not be established.
Remedy: If the message does not self-extinguish, switch off and on the machine. Please contact EMCO service when the message occurs permanently even after power off.

7092   TEST STOP ACTIVE
Cause: Safety test for monitoring the safety functions is active.
Remedy: Wait until the safety test is finished.

7093   SET REFERENCE POINT
Cause: The reference point acknowledge mode was activated by the operator.

7094   SET X-REFERENCE POINT
Cause: The reference value of the X axis was adopted to the acc.esd file.

7095   SET Y-REFERENCE POINT
Cause: The reference value of the Y axis was adopted to the acc.esd file.

7096   SET Z-REFERENCE POINT
Cause: The reference value of the Z axis was adopted to the acc.esd file.

7097   FEED OVERRIDE 0
Cause: The override switch (feed control) was set by the operator on 0%

emco

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 EQUIP-MENT 1

Effect: Locking of NC start and main drive start, spindle stop S1
Meaning: The pick-up for the analouge value detects the clamping position as end position.
Remedy: Change clamping range of the clamping device (see "Machine Description - chapter C Programming and Operation").

7104 QUILL IN INTERMEDIATE POSITION

Effect: Feed hold / read in stop.

7105 SET AWZ REFERENCE POINT

Effect: The reference value of the AWZ motor was transferred to the acc.msd file.

7900   INITIALIZE EMERGENCY OFF!

Cause: The emergency off button must be initialized.
Remedy: Press and then release emergency off button.

7901   INITIALIZE MACHINE DOORS!

Cause: The machine doors must be initialized.
Remedy: Open the machine doors and close them again.

7906   SET A-REFERENCE POINT

Cause: The reference value of the A axis was adopted to the acc.esd file.

Inputunit alarms 1700 - 1899

These alarms and messages are raised by the control keyboard.

1701   Error in RS232

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

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

1703   Ext. keyboard not available

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

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

1704   Ext. keyboard: checksum error

Cause: Error in the transmission.

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

1705   Ext. keyboard: general error

Cause: The attached keyboard reported an error.

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

1706   General USB error

Cause: Error in the USB communication.

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

1707   Ext. Keyboard: no LEDs

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

Remedy: EMCO-Service kontaktieren.

1708   Ext. Keyboard: unknown command

Cause: Unknown command was sent to the keyboard.

Remedy: Contact EMCO Customer Service

1710   Installation of Easy2control is damaged!

Cause: Incorrect installation of Easy2control

Remedy: Reinstall software and/or contact EMCO Customer Service

1711   Initialization of Easy2Control failed!

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

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

1712   License for Easy2control could not be found!

Cause: USB-Dongle or a valid license key for Easy2control is not connected. Easy2control is displayed but can not be operated.

Remedy: Connect USB-Dongle for Easy2control.

1801   Keytable not found!

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

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

1802   Connection to keyboard lost

Cause: Connection to the serial keyboard was interrupted.

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

EMCO

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.

emco

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The combination of linear and circular axes does not match.
Remedy: Program correction.

8140 Maschine (ACIF) not responding
Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

8141 Internal PC-COM error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8142 ACIF Program error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

8145 Fatal init data error (acif.hex)
see 8142.

8146 Multiple request for axis
see 8142.

8147 Invalid PC-COM state (DPRAM)
see 8142.

8148 Invalid PC-COM command (CNo)
see 8142.

8149 Invalid PC-COM command (Len)
see 8142.

8150 Fatal ACIF error
see 8142.

8151 AC Init Error (missing RPG file)
see 8142.

8152 AC Init Error (RPG file format)
see 8142.

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

8155 Invalid FPGA packet acknowledge
see 8142 or hardware error on ACIF board (contact EMCO Service).

8156 Sync within 1.5 revol. not found
see 8142 or Bero hardware error (contact EMCO Service).

8157 Data record done
see 8142.

8158 Bero width too large (referencing)
see 8142 or Bero hardware error (contact EMCO Service).

8159 Function not implemented
Bedeutung: In normal operation this function can not be executed

8160 Axis synchronization lost axis 3..7
Cause: Axis spins or slide is locked, axis synchronisation was lost
Remedy: Approach reference point

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:
- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm) or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost
see 8161

8163 Z-Axis synchronization lost
see 8161

8164 Software limit switch max axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8168 Software limit overtravel axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.
Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running
Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

8174 INC not allowed
Cause: At the moment the axis is in motion.
Remedy: Wait until the axis stops and then traverse the axis.

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

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

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message
see 8175.

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

8192 Invalid axis in command
see 8175.

8193 Fatal PLC error
see 8175.

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

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

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

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

8198 Internal error (to manny threads)
see 8175.

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

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

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

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

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8212 Rotation axis not allowed
see 8199.

8213 Circle and rotation axis can't be interpolated

8214 Thread and rotation axis cant't be interpolated

8215 Invalid state
see 8199.

8216 No rotation axis for rotation axis switch
see 8199.

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

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

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

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

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

emco

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 configured for touch probes. A traversing movement with rotary axis is not allowed during touch probe operating mode.
Remedy: Remove the rotary axis movement from the traversing movement.

8226 Rotary axis switch not valid (MSD data)!
Cause: The indicated spindle does not have a rotary axis.

8228 Rotary axis switch not allowed while axis move!
Cause: The rotary axis has moved during switching over to the spindle operating mode.
Remedy: Stop the rotary axis before switching.

8229 Spindle on not allowed while rotary axis is active!

8230 Program start not allowed due to active spindle rotation axis!

8231 Axis configuration (MSD) for TRANSMIT not valid!
Cause: Transmit is not possible at this machine.

8232 Axis configuration (MSD) for TRACYL not valid!
Cause: Tracyl is not possible at this machine.

8233 Axis not available while TRANSMIT/TRACYL is active!
Cause: Programming of the rotary axis is not allowed during Transmit/ Tracyl.

8234 Axis control grant removed by PLC while axis interpolates!
Cause: Internal error
Remedy: Delete error with reset and inform EMCO.

8235 Interpolation invalid while axis control grant is off by PLC!
see 8234.

8236 TRANSMIT/TRACYL activated while axis or spindle moves!
see 8234.

8237 Motion through pole in TRANSMIT!
Cause: It is not allowed to move through the coordinates X0 Y0 inTransmit.
Remedy: Alter the traversing movement.

8238 Speed limit in TRANSMIT exceeded!
Cause: The traversing movement gets too close to the coordinates X0 Y0. In order to observe the programmed feed rate, the maximum speed of the rotary axis would have to be exceeded.
Remedy: Reduce the feed rate. Set the value of the C-axis feed limitation in WinConfig, machine data settings / general machine data/ to 0.2. Thus, the feed rate will be automatically reduced near the coordinates X0 Y0.
The distance to the center is calculated with the following formula:
for CT155/CT325/CT450:
F[mm/min] * 0.0016 = distance [mm]
for CT250:
F[mm/min] * 0.00016 = distance [mm]
This applies for rapid traverse in transmit:
CT155/250/325: 4200 mm/min
CT450: 3,500 mm/min

8239 DAU exceeded 10V limit!
Cause: Internal error
Remedy: Start the software again or install it anew. Report the error to EMCO.

8240 Function not valid during active transformation (TRANSMIT/TRACYL)!
Cause: The Jog and INC operating mode are not possible during Transmit in X/C and during Tracyl in the rotary axis.

8241 TRANSMIT not enabled (MSD)!
Cause: Transmit is not possible at this machine.

8242 TRACYL not enabled (MSD)!
Cause: Tracyl is not possible at this machine.

8243 Round axis invalid during active transformation!
Cause: It is not allowed to program the rotary axis during Transmit/Tracyl.

8245 TRACYL radius = 0!
Cause: When selecting Tracyl, a radius of 0 was used.
Remedy: Correct the radius.

8246 Offset alignment not valid for this state!
see 8239.

8247 Offset alignment: MSD file write protected!

8248 Cyclic supervision failed!
Cause: The communication with the machine keyboard is interrupted.
Remedy: Start the software again or install it anew. Report the error to EMCO.

8249 Axis motion check alarm!
see 8239

8250 Spindle must be rotation axis !
see 8239

8251 Lead for G331/G332 missing !
Cause: The threading pitch is missing or the starting coordinates are identical to the target coordinates.
Remedy: Program the threading pitch.
Correct the target coordinates.

8252 Multiple or no linear axis programmed for G331/G332 !
Remedy: Program exactly one linear axis.

8253 Speed value for G331/G332 and G96 missing !
Cause: No cutting speed has been programmed.
Remedy: Program the cutting speed.

8254 Value for thread starting point offset not valid!
Cause: The thread starting point offset is not within the range of 0 to 360°.
Remedy: Correct the thread starting point offset.

8255 Reference point not in valid software limits!
Cause: The reference point has been defined outside the software limit switches.
Remedy: Correct the reference points in WinConfig.

8256 Spindle speed too low while executing G331/G332!
Cause: During tapping the spindle speed has decreased. Perhaps the incorrect threading pitch was used or the core drilling is not correct.
Remedy: Correct the threading pitch. Adapt the diameter to the core drilling.

8257 Real Time Module not active or PCI card not found!
Cause: ACC could not be started correctly or the PCI card in the ACC was not recognized.
Remedy: Report the error to EMCO.

8258 Error allocating Linux data!
see 8257.

8259 Current thread in sequence not valid!
Cause: One block of a thread in sequence has been programmed without thread G33.
Remedy: Correct the program.

8260 Change of leading axis within the thread sequence !
Cause: Occurs if the thread run-out of the longitudinal thread is set so that it's not possible to reach the target point. with the required stopping distance
Remedy: The thread run-out should be at least as large as the pitch. this error also occurs. if the pitch of a thread chain is too large when changing the leading axis.

8261 Missing thread in sequence !
Cause: A successive thread has not been programmed for a thread in sequence, the number has to be in accordance with the SETTHREADCOUNT () that has been defined before.
Remedy: Correct the number of threads in the thread in sequence and add a thread.

8262 Reference marks are not close enough !
Cause: The settings of the linear scale have been changed or the linear scale is defective.
Remedy: Correct the settings. Contact EMCO.

8263 Reference marks are too close together!
see 8262.

8265 No or wrong axis in axis switch command!
Cause: Internal error.
Remedy: Please contact the EMCO after-sales service.

8266 Invalid tool
Cause: Programmed tool is not set in magazine.
Remedy: Correct tool number and/or load tool in magazine.

8267 Speed difference to high
Cause: Die Soll- und Istgeschwindigkeit der Achse weichen zu stark voneinander ab.
Remedy: Run the program again with reduced feed. If this does not remedy the problem, contact EMCO.

8269 USBSPS and ACC speed values or override are different
Cause: USBSPS and ACC have diferent rotations saved.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

8270 Reference switch defective
Cause: The reference switch did not switch within the specified range.
Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

8271 Tool load in locked place not possible
Cause: There was an attempt to swing a tool into a locked place in the magazine.
Remedy: Choose a free, unlocked place in the magazine and then swing the tool into the magazine.

8272 Old PLC version, update necessary
Cause: The PLC version is too old to fully support randomised tool management.
Remedy: Update the PLC.

emco

8273 Spindle overload
Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).
Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

8274 Define tool before loading
Cause: The tool must be defined in the tool list before it is possible to transfer the tool into the spindle.
Remedy: Create the tool in the tool list, then load.

8275 Could not read position from absolute encoder
Cause: The position of an absolute encoder could not be read.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times

8276 Physical axis left the valid travel range
Cause: An axis with absolute encoder is outside the valid range for travel.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times.

8277 Sinamics drive package error active
Cause: Error in Sinamics drive.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times.

8278 Control not compatible with ACpn
Cause: The WinNC control is incompatible with the ACpn machine.
Remedy: Install a WinNC control which is compatible with the Acpn.

8279 Connection to drive got lost
Cause: the connection between Acpn and CU320 was lost.
Remedy: Turn off and on the machine. Contact EMCO service it the error occurs multiple times

8280 Reference point in setting data not equivalent with machine data, please check!
Cause: The reference point saved in the AC setting data does not match the reference point in the machine data (ACC_MSD).
Remedy: Measure the reference point of all axes anew and enter into EMConfig.

8704 Feed override absent, REPOS is not executed
Cause: The REPOS command is not executed because the feed override is set to 0%.
Remedy: Change the feed override and restart REPOS.

8705 Tool sorting active
Cause: The tools will be re-sorted with random tool management to facilitate non-random operation (tool 1 at place 1, tool 2 at place 2, etc.).
Remedy: Wait until sorting has finished. The controller will delete the report independently.

8706 Check new controller - tool table
Cause: The controller was changed with random tool management active.
Remedy: Check the tool or place table to clear the alarm.

8707 Ending with auxiliary drives switched on not possible
Cause: An attempt was made to end the controller, although the auxiliary drives are still switched on.
Remedy: Switch off the auxiliary drives and then end the controller.

8710 Initiating communication with drives
Cause: Acpn connects to the Sinamics drives
Remedy: Wait until the connection was established.

8712 JOG in X and C deactivated during Transmit
Cause: JOG in X and C ist not possible at active Transmit.

22000 Gear change not allowed
Cause: Gear step change when the spindle is active.
Remedy: Stop the spindle and carry out a gear step change.

22270 Feed too high (thread)
Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed
Remedy: Program correction, lower pitch or lower spindle speed for thread

200000 to 300000 are specific to the drives and show up only in combination with the alarm # "8277 Sinamics error".

201699 - „(F) SI P1 (CU): Test of the shutdown path is necessary"
Cause: A test of the shutdownpath is necessary. The machine remains operational.
Remedy: The test is performed automatically during the next restart of the WinNC.

235014 TM54F: Teststop necessary
Cause: A teststop is necessary.
Remedy: Shutdown and restart the WinNC. The test is performed automatically during restart.

Axis Controller Messages

8700 Execute REPOS in all axes before program start
Cause: After the program was stopped, the axes were manipulated with the hand wheel or with the jog keys and then a restart of the program was attempted.
Remedy: Before 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 diamter is too large for the intended bore.
Remedy: Enlarge the nominal diameter and use a smaller milling cutter.

4009 length too small
Cause: Width and length must be larger than the double tool radius.

4010 diameter less equal zero
Cause: The pocket diameter, the stud diameter, etc. must not be zero.

4011 blank diameter too big
Cause: The diameter of the machined pocket must be larger than the diameter of the premachined pocket.

4012 blank diameter too small
Cause: The diameter of the machined stud must be smaller than the diameter of the premachined stud.

emco

4013 start angle equal to end angle
Cause: Start angle and end angle for hole pattern are identical.

4014 tool radius 0 not permitted
Cause: Tool radius zero is not permitted.
Remedy: Select a valid tool.

4015 no outer contour defined
Cause: The contour file indicated in the cycle was not found.

4017 tool radius too big
Cause: For the programmed machining, a tool being too large was selected. Therefore, machining is not possible.

4018 allowance must not be 0
Cause: There were programmed finishing operations without finishing offset.

4019 too many iterations
Cause: The contour definitions are too complex for the roughing-out cycle.
Remedy: Simplify the contour.

4020 illegal radian correction
Cause: An error has occured during the programming of the radius compensation.
Remedy: Check the cycle parameters.

4021 can't calculate parallel contour
Cause: The control was not able to calculate the tool radius compensation.
Remedy: Check the programmed contour for plausibility. Maybe contact EMCO.

4022 illegal contour definition
Cause: The programmed contour is not suited to the selected machining.
Remedy: Check the programmed contour.

4024 no contour definition
Cause: The contour file being defined in the cyle has not been found.

4025 internal calculation error
Cause: An unexpected error has occured during calculation of the cycle movements.
Remedy: Please inform the EMCO after-sales service.

4026 allowance too big
Cause: A part of the finishing offset (for several finishing passes) is larger than the total finishing offset.
Remedy: Correc the finishing offsets.

4028 pitch 0 not permitted
Cause: The thread was programmed with pitch zero.

4029 undefinded working mode
Cause: Internal error (invalid machining type for the thread).

4030 function not yet supported
Cause: Roughing out with pockets is not implemented yet.
Remedy: Please inform the EMCO after-sales service.

4031 value not permitted
Cause: An invalid retracting direction was programmed during inside turning.

4032 plunging must be defined
Cause: For the programmed cycle no cutting depth has been programmed.

4033 radius/chamfer too big
Cause: The radius, respectively the chamfer, cannot be inserted in the programmed contour.
Remedy: Reduce the radius, respectively the chamfer.

4034 diameter too big
Cause: The programmed starting point and the machining diameter are contradictory.

4035 diameter too small
Cause: The programmed starting point and the machining diameter are contradictory.

4036 unknown working direction
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4037 unknown working type
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4038 unknown sub cycle
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4039 rounding not possible
Cause: The programmed radius contradicts the rest of the cycle parameters.

4042 illegal tool width
Cause: The tool width for the cutting-off cycle must be defined.

4043 groove width too small
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4044 distance not defined
Cause: The distance for the multiple grooving cycle must not be zero.

4045 illegal allowance type
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4046 invalid speed
Cause: The spindle speed must be nonzero.

4047 invalid end point
Cause: The programmed end point contradicts the rest of the cycle definition.

4048 tool cut width too small
Cause: The cutting edge is too small for the programmed cutting depth.

4050 invalid distance
Cause: The hole patterns do not tally with the selected distance.

4052 working pattern not possible
Cause: Error in the definition of the hole pattern. The number of bores is contradictory.

4053 invalid start point
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4055 illegal working direction
Cause: The machining direction is contradictory to the rest of the cycle definition.

4057 plunging angle less equal zero
Cause: The plunging angle must be between 0 and 90 degree.

4058 chamfer too large
Cause: The programmed chamfer is too large for the pocket cycle.

4062 radius/chamfer too small
Cause: The radius, respectively the chamfer, cannot be machined with the current tool radius.

4066 invalid mill step
Cause: The mill step must be greater than zero.

4069 invalid angle
Cause: An angle of zero degree is not permitted.

4072 plunging too small
Cause: For the cycle, a cutting depth has been selected that leads to extra-long machining time.

4073 invalid clearance angle
Cause: The clearance angle indicated for the tool cannot be machined.
Remedy: Correct the clearance angle for the tool.

4074 contour-file not found
Cause: The contour file indicated in the cycle has not been found.
Remedy: Please select the contour file for the cycle.

4075 not machinable with selected tool
Cause: The tool is too wide for the programmed groove.

4076 reciprocating plunge cut impossible (initial move too short)
Cause: The first movement of the contour is shorter than the double tool radius and cannot be therefore used for the swinging delivery.
Remedy: Extend the first movement of the contour.

4077 wrong tool type in grooving or cut-off cylce
Cause: The wrong tool type was used in the cutting cycle.
Remedy: Use only grooving and punch tools in the cutting cycles.

4078 radius of helix too small
Cause: The pitch of the helix is $\leq$ 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..

emco

4080 radius of helix or tool too large
Cause: The helical approach cannot be executed with the selected data for the helix and the current tool radius without a contour breach.
Remedy: Use a tool with a smaller radius or reduce the radius of the helix.

4200 leaving movement is missing
Cause: No movement after the tool radius compensation was deactivated in the current plane.
Remedy: Insert the departing movement in the current plane after having deactivated the tool radius compensation.

4201 TPC off missing
Cause: The tool radius compensation has not been deactivated.
Remedy: Deactivate the tool radius compensation.

4202 TPC requires at least three movements
Cause: The tool radius compensation requires at least 3 movements in the current plane in order to calculate the tool radius compensation.

4203 approaching movement not possible
Cause: It was not possible to calculate the approaching movement.

4205 leaving movement not possible
Cause: It was not possible to calculate the departing movement.

4208 TPC curve could not be calculated
Cause: It was not possible to calculate the tool radius compensation for the programmed contour.

4209 switching the plane is not allowed when TPC is switched on
Cause: The programmed plane must not be changed during the tool radius compensation.
Remedy: Remove the change of planes during the tool radius compensation.

4210 tool path compensation already activated
Cause: G41 is active and G42 was programmed or G42 is active and G41 was programmed.
Remedy: Switch tool radius compensation off with G40 before programming the radius compensation again.

4211 Bottleneck detected
Cause: In the radius correction calculation some parts of the contour were omitted, as too large a milling cutter tool was used.
Remedy: Use a smaller milling cutting tool to process the contour completely.

4212 Infeed has been programmed twice during approach
Cause: After the approach movement a second infeed has been programmed, without previously moving to the work plane.
Remedy: First program a movement to the work plane before programming a second infeed.

5000 drill manually now

5001 contour has been adjusted to the programmed clearance angle
Cause: The programmed contour was adapted to the programmed clearance angle. Maybe there will remain rest material that cannot be machined with this tool.

5500 3D simulation: Internal error
Cause: Internal error inside the 3D simulation.
Remedy: Restart the software or, if necessary, report the error to EMCO customer service.

5502 3D simulation: Tool place invalid
Cause: Tool place unavailable on the machine used.
Remedy: Correct tool call-up.

5503 3D simulation: Chuck invalid owing to the unmachined part definition
Cause: The distance from the front of the unmachined part is > the unmachined part length.
Remedy: Change the distance.

5505 3D simulation: Unmachined part definition invalid
Cause: Implausibility in the unmachined part geometry (e.g. expansion in one axis ≤ 0, inside diameter > outside diameter, unmachined part contour not closed, etc.).
Remedy: Correct unmachined part geometry.

5506 3D simulation: STL chuck file has auto-overcuts
Cause: Error in the chuck description.
Remedy: Correct file.

5507 3D simulation: Pole transit on TRANS-MIT!
Cause: Travel comes too close to the X0 Y0 co-
               ordinates.
Remedy: Change travel.

emco

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.

1500   Collision with sensor during approach to measuring detected
Explanation: The probe collided with an obstacle during positioning.
Remedy:
- Check spigot diameter (possibly too small)
- Check measuring path (possibly too large)

15040 Preset not found
Explanation: Preset with the specified number was not found.

15056 No tool pocket available
Explanation: There is no suitable tool location for the tool in the pocket table.
Remedy: Provide a suitable tool location.

22504 Pocket table incorrect
Explanation: The pocket table contains more spindle pockets than are given in the machine parameter CfgAxes.spindleIndices.
Remedy: Delete the invalid pockets from the pocket table.

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.

38013 Error when opening the file '%1'
Explanation: An error occurred while a file was being opened.
Remedy: Ensure that the file exists, that the given path is correct, and that the file has a readable format.

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).

38031 Wrong axis programmed
Explanation: An axis programmed in the NC block is not configured.
Remedy: Edit the NC program.

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.

38050   Arithmetical error
Explanation: Incorrect Q parameter calculation: Division by 0, square root of a negative number, or similar error.
Remedy: Check the input values.

38060 CYCL DEF incomplete
Explanation: You programmed an incomplete cycle definition or inserted other NC blocks between cycle blocks.
Remedy: Edit the NC program.

38073 Touch point inaccessible
Explanation: No signal came from the touch probe within the measuring distance.
Remedy: Pre-position as appropriate and repeat the probing process.

38100   Table access failed
Explanation: The syntax of the programmed SQL statement is incorrect
- The given table could not be opened
- The given table is incorrectly defined
- The symbolic name of the table is not defined
- The table does not contain the given columns
- A data record could not be read because it is locked

emco

Remedy: Check the table definition and edit the NC program.

38170 Excessive program nesting
Explanation: Program nesting through CALL LBL or CALL PGM is too deep, presumably because of a recursive call.
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.

38220 Wrong rpm
Explanation: The programmed spindle speed does not lie in the existing pattern of spindle speed stages.
Remedy: Enter the correct rotational speed.

38221 Check the depth sign
Explanation: The cycle can only run in negative direction (positive in Cycle 204) because the configuration datum "displayDepthErr" is set to on.
Remedy: - Enter the depth as a negative value (positive in Cycle 204) to run the cycle
- To run the cycle in the positive direction (negative in Cycle 204,), set the configuration datum "displayDepthErr" to off.

38224 FZ was programmed in conjunction with the number of teeth 0
Explanation: You used FZ to define a feed per tooth, although no tooth number is defined in the tool table.
Remedy: Add to the CUT column in the tool table for the active tool.

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.

38227 Spindle speed S is greater than NMAX from the tool table
Explanation: You entered a spindle speed S greater than the maximum speed defined for this tool in the tool table.

Remedy: - Enter a lower spindle speed S.
- Change the maximum spindle speed in the NMAX column of the tool table.

38237 Point pattern call or definition faulty
Explanation: A CYCL CALL PAT or PATTERN DEF block could not be executed because:
- The machining plane programmed with TOOL CALL is not supported or
- The point table or a pattern definition with PATTERN DEF is faulty.
Remedy: - Check the NC program or point table and correct if necessary.

38238 No point pattern definedt
Explanation: CYCL CALL PAT can only work if beforehand a point pattern was selected with SEL PATTERN or PATTERN DEF.
Remedy: -Edit the NC program.

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.

38284 Touch probe not defined
Ursache: - Sie haben ein Tastsystem aufgerufen, das in der Tastsystem-Tabelle nicht definiert ist.
- Tastsystem-Tabelle ist nicht vorhanden oder schreibgeschützt.
Fehlerbehebung: - Ergänzen Sie die Tastsystem-Tabelle um das fehlende Tastsystem.
- Tastsystem-Tabelle erstellen oder Schreibschutz aufheben.

42023 Rounding radius too large
Explanation: - In the definition of a contour, a contour pocket or a contour train, you programmed a rounding arc (RND, ISO: G25) with so large a radius that it does not fit between the adjoining elements.
- In a fixed cycle (rectangular pocket/rectangular stud), you defined a rounding arc that cannot be inserted.
Remedy: - Define a smaller rounding radius in the contour subprogram
- Check the cycle definition and correct the input values.

42072 Tolerance exceeded

Explanation- The limits entered in the tool table TOOL.T in the LTOL or RTOL column were exceeded.

Remedy: Check the limit values for the active calibration tool.

42078 Q303 in meas. cycle undefined!

Explanation- In one of the measuring cycles 410 to 418, you did not define the parameter Q303 (measured value transfer)(current value = -1). For reasons of security, however, selection of the measured value transfer is required to write the results of measurement in a table (datum table or preset table).

Remedy: Change parameter Q303 (measured value transfer) in the measuring cycle that is causing the error:
- Q303=0: Write the measured values with respect to the active workpiece coordinate system into the active datum table (activate in the program with Cycle 7!).
- Q303=1: Write the measured values with respect to the active machine-based coordinate system (REF values) into the preset table (activate in the program with Cycle 247!).
- Q303=-1: Measured value transfer is undefined. This value is automatically generated by the TNC when you download a program that was written on a TNC 4xx or with an old software level of the iTNC 530, or if during the cycle definition you skipped the
prompt for the measured value transfer with the END key.

42081   Contradictory measuring points

Explanation - In one of the probing cycles 400, 403 or 420, you defined a contradictory combination of measuring points and measuring axes.

- The selection of measuring points in Cycle 430 results in division by 0.

Remedy: - For measuring axis = reference axis (Q272=1), the parameters Q264 and Q266 are defined at different values.
- For measuring axis = minor axis (Q272=2), the parameters Q263 and Q265 are defined at different values.
- For measuring axis = probe axis (Q272=3), the parameters Q263 and Q265 or Q264 and Q266 are defined at different values.
- Select the measuring points so that they always have different coordinates in all axes.

42083   Contradictory plunge type!

Explanation The plunging strategy defined in Cycles 251 to 254 contradicts the defined plunging angle of the active tool.

Remedy: Change parameter Q366 in one of the Cycles 251 to 254 or the plunging angle ANGLE of

the tool in the tool table. Permissible combinations of parameter Q366 and the plunging ANGLE are:
For perpendicular plunging: Q366 = 0 and ANGLE = 90
For helical plunging: Q366 = 1 and ANGLE > 0
For inactive tool table, define Q366 with 0 (only perpendicular plunging allowed).

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.

42089 Slot position 0 not allowed!

Explanation: You tried to run Cycle 254 with the slot position 0 (Q367=0) in conjunction with the point pattern Cycle 221.

Remedy:
Use slot position Q367 = 1, 2 or 3 if you want to run Cycle 254 with the point pattern cycle 221.

42090 Enter an infeed not equal to 0

Explanation: You defined a fixed cycle with the depth 0.

Remedy:

Enter a depth unequal to 0.

42098   Contradictory workpc. blank dim.

Explanation: The workpiece blank dimensions you have defined in a fixed cycle are smaller than the dimensions of the finished part.

Remedy:
 Check the cycle definition and correct the input values.

42121   Too many limits

Explanation: Too many limits were selected for face milling.

Remedy:
Set no more than 3 limits for face milling.

42122   Machining strategy with limits not possible

Explanation: Spiral machining strategy: face milling with limits is not possible

Remedy:
Deactivate the limits or select another machining strategy.

emco

42131 Line does not exist in table
Explanation: The programmed action cannot be executed because the given table line does not exist.
Remedy:
Check your NC program.
Create the given line in your preset or datum table.

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.

42531 Calibrate touch probe
Explanation: You tried to automatically measure a tool although the tool touch probe is not yet calibrated.
Remedy:
Calibrate the TT tool touch probe with the cycle TCH PROBE 30.

42544 Tool broken
Explanation: Automatic tool measurement: The breakage tolerance (LBREAK or RBREAK) from the tool table was exceeded.
Remedy:
Check the tool and, if necessary, replace it.

43076 Tangent at start point of element not defined
Explanation: A geometrical element with tangential transition (e.g. CT) was programmed. However, the tangent at the start point is undefined, e.g. because of an immediately preceding verticaldeparture. or because it is the 1st block of the NC program..
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.

43133 Circle incorrectly programmed
Ursache: The end point of a CT block lies in the extension of the tangent of the previously programmed contour element.
Remedy: Edit the coordinates of the end point

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 <= ANGLE <= 90.0
LCUTS : Tooth length in mm, 0.1 <= LCUTS <= 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
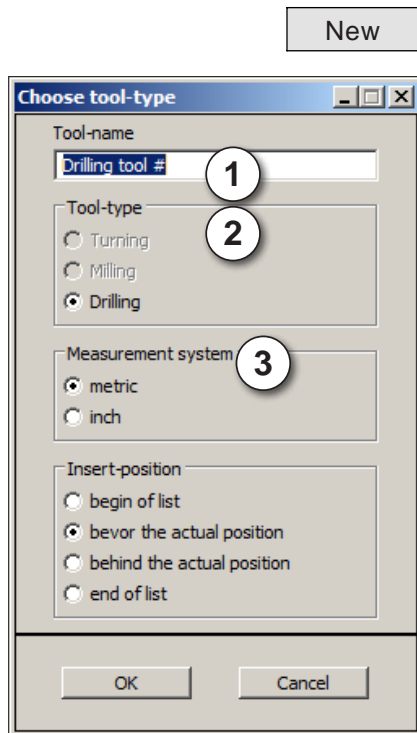
emco
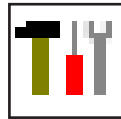
Tool modelling with the 3D-Tool Generator

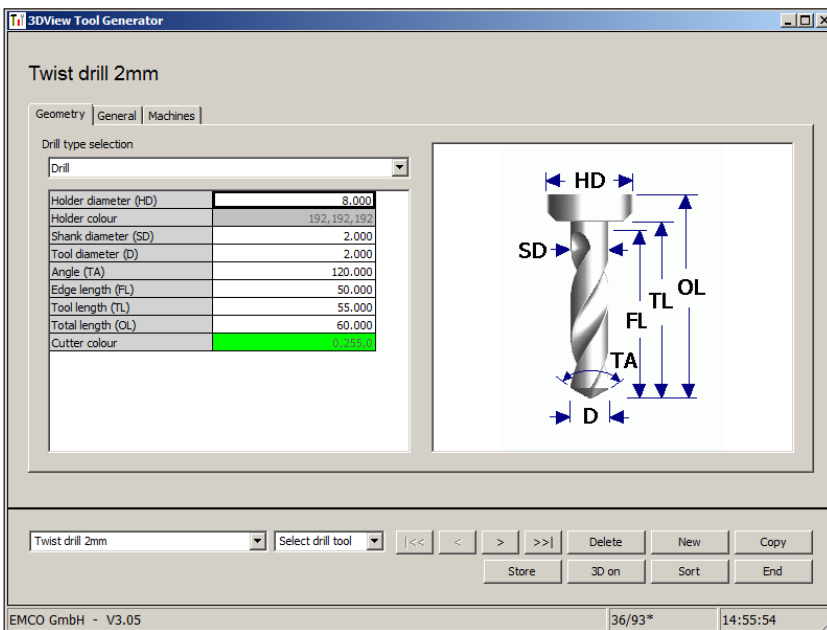With the 3D-ToolGenerator you can modify existing tools and create new tools.
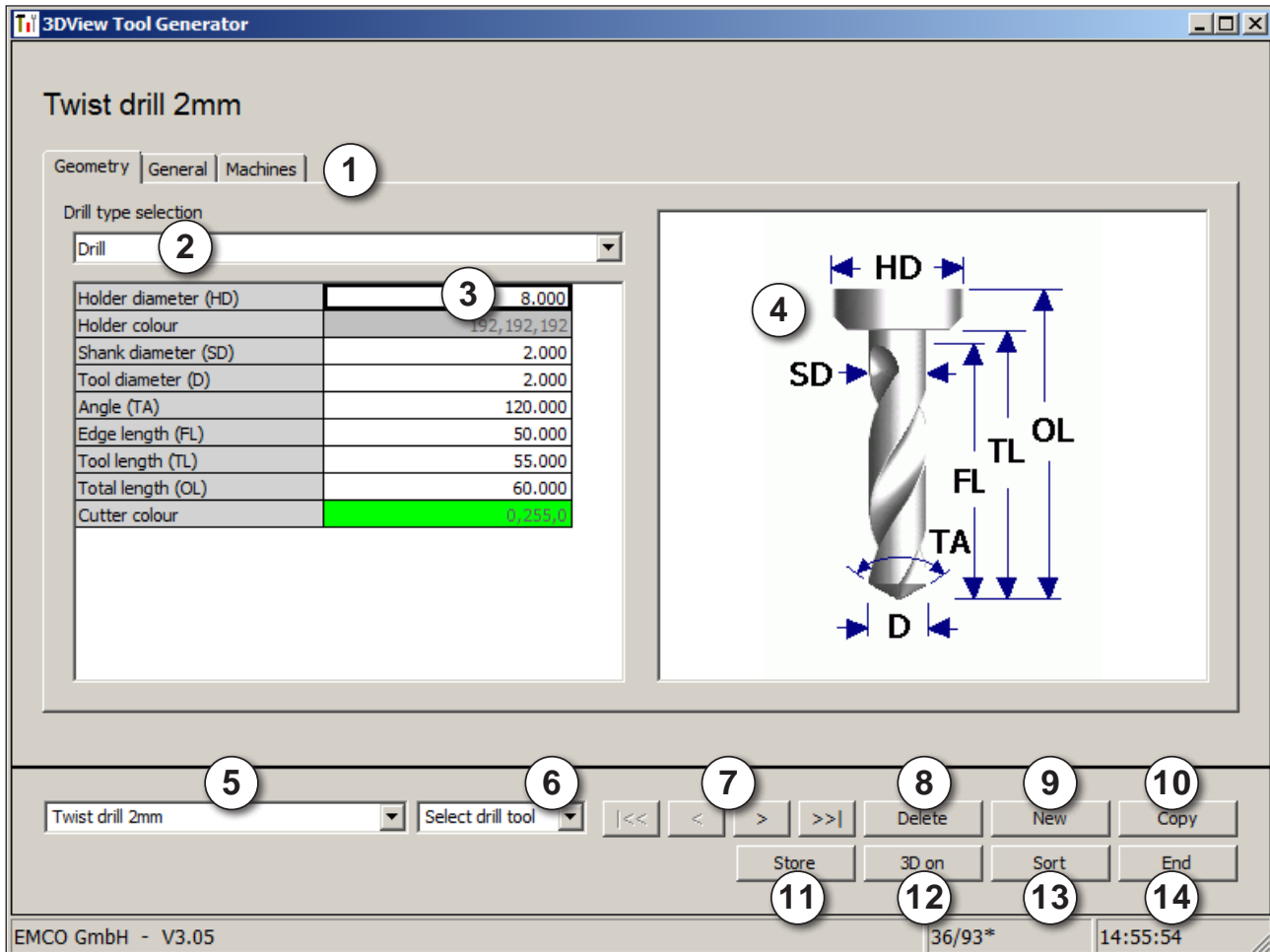
1 Register cards for "Geometry", "General" and "Machines" for drilling and milling tools and "Tip", "Holder", "General" and "Machines" for turning tools.
2 Selection of tool types
3 This window enables the input of tool dimensions.
4 Graphical support for the tool dimensioning
5 Choice of tools for the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

New

- 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).

Choose tool-type

Tool-name

Drilling tool # **(1)**

Tool-type **(2)**
- Turning
- Milling
- ● Drilling

Measurement system **(3)**
- ● metric
- inch

Insert-position
- begin of list
- ● bevor the actual position
- behind the actual position
- end of list

OK   Cancel

OK

- Confirm the entries with "OK".

3DView Tool Generator

Twist drill 2mm

Geometry | General | Machines

Drill type selection

Drill

Holder diameter (HD)	8.000
Holder colour	192,192,192
Shank diameter (SD)	2.000
Tool diameter (D)	2.000
Angle (TA)	120.000
Edge length (FL)	50.000
Tool length (TL)	55.000
Total length (OL)	60.000
Cutter colour	0,255,0

Twist drill 2mm   Select drill tool |<< < > >>| Delete New Copy
Store 3D on Sort End

EMCO GmbH - V3.05   36/93*   14:55:54

- Define all tool dimensions.
- Define all tool colours (see "select tool colour").

Store

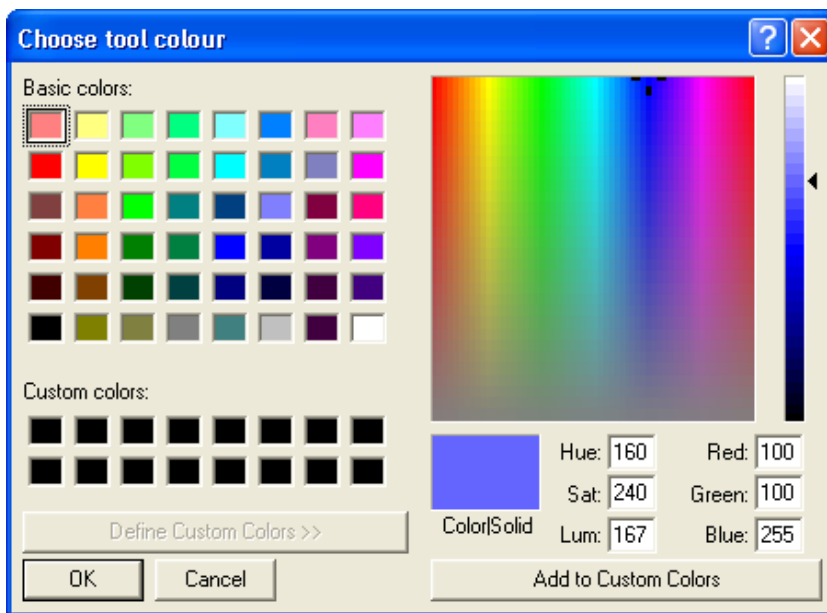- Confirm the entries with "Store".

emco

Copying a tool

Copy

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

Changing an existing tool

Store

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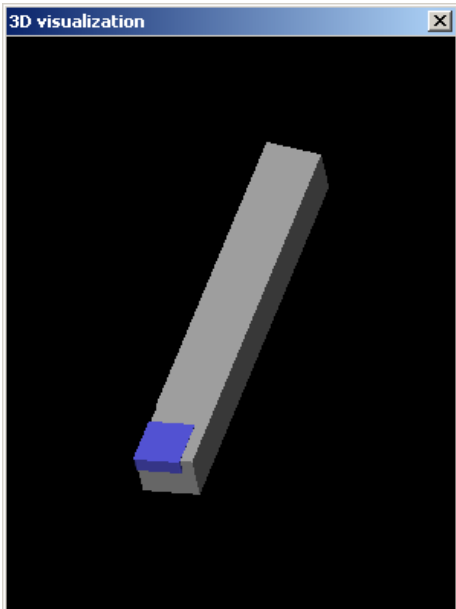- Change the values.
- Confirm the entries with "Store".

Selecting a tool colour

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

OK

- Confirm the entries with "OK".

Visualizing a tool

3D on

- Press the button for 3D visualization.

Rotating image

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

Zooming

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

Shifting

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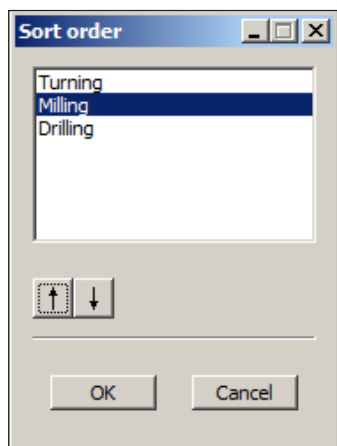
Sorting function

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

Sort

- Press the button for sorting.

- Define new sorting sequence.

OK

- Confirm the entries with "OK".

emco

DNC interface

The DNC interface (Distributed Numerical Control) enables the control system (WinNC) to be controlled remotely via a software protocol.

The DNC interface is activated with EMConfig, by indicating TCP/IP or a serial interface for the DNC.

During the installation of the control software, the DNC interface is enabled and configured, and can be reconfigured with EMConfig later on.

The DNC interface creates a connection between a higher-level computer (production control computer, FMS computer, DNS host computer, etc.) and the control computer of an NC machine. After activation of the DNC drive the DNC computer (Master) takes over control of the NC machine (Client). The DNC computer takes over complete control of the manufacturing. The automation fittings such as door, chuck (collet), quill, coolant, etc. can be controlled from the DNC computer. The current status of the NC machine is visible on the DNC computer.

The following data can be transferred or loaded via the DNC interface:
- NC Start
- NC Stop
- NC programs *)
- Zero point shifts *)
- Tool data *)
- RESET
- Approach reference point
- Periphery control
- Override data

The DNC interface can be operated with the following CNC control types:
- SINUMERIK Operate T and M
- FANUC 31i T and M

Further details of the functions and the DNC protocol can be found in the accompanying product documentation.

If the DNC interface is operated with TCP/IP, it will wait for incoming connections on port 5557.

*) not for SINUMERIK Operate and FANUC 31i

X: EMConfig

Note:
The settings which are available in EMConfig are depending on the machine and the control that is used.

General

EMConfig is a configuration software for WinNC. EMConfig helps you to alter the settings of WinNC.

The most important settings are:
*   Control language
*   System of measurement mm - inch
*   Activate accessories
*   Selection of interface for control keyboard

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

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

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

EMLaunch configuration
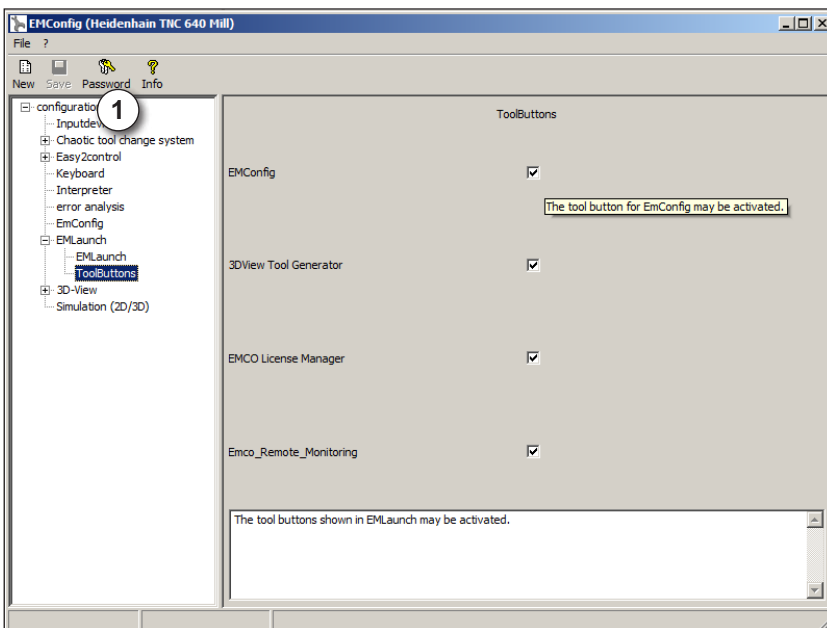
The following ToolButtons for EMLauch can be activated or deactivated, e.g.:
*   EMConfig
*   3DView Tool Generator
*   EMCO License Manager
*   Emco_Remote_Monitoring

emco

Icon for EMConfig

How to start EMConfig

Open EMConfig.

Selection box for control type
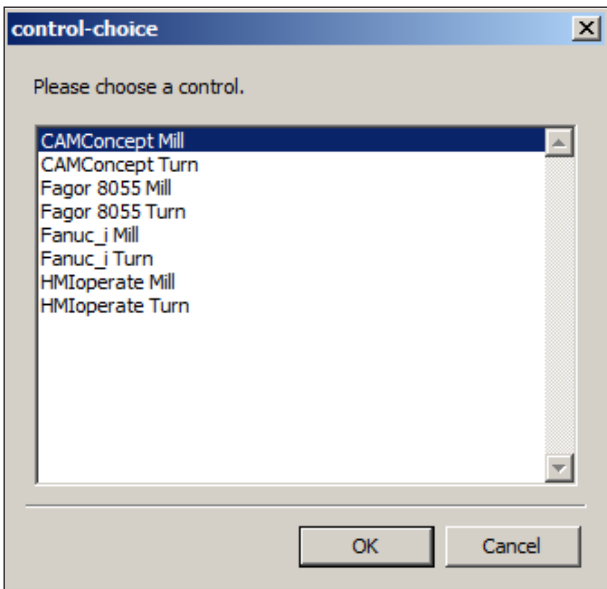
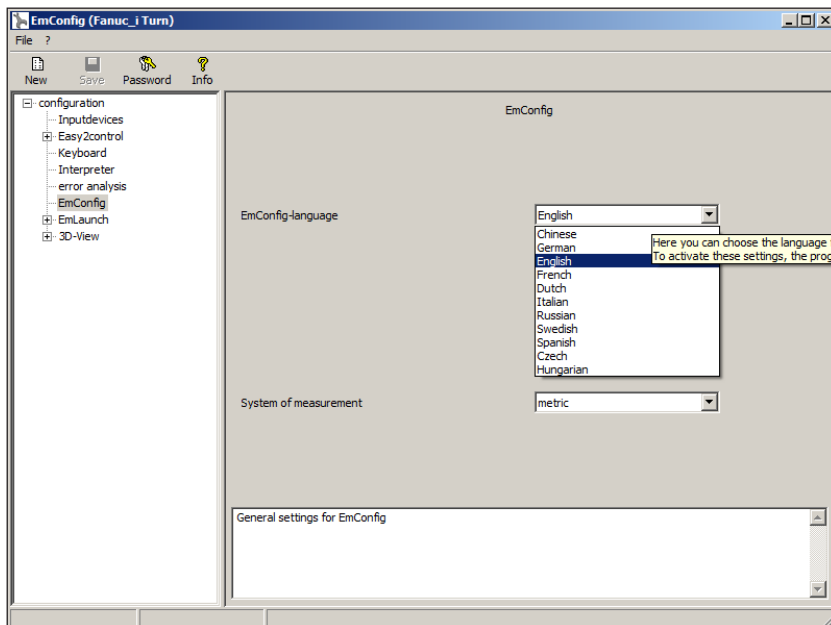In case several control types are installed, a selection box will appear on the screen.

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

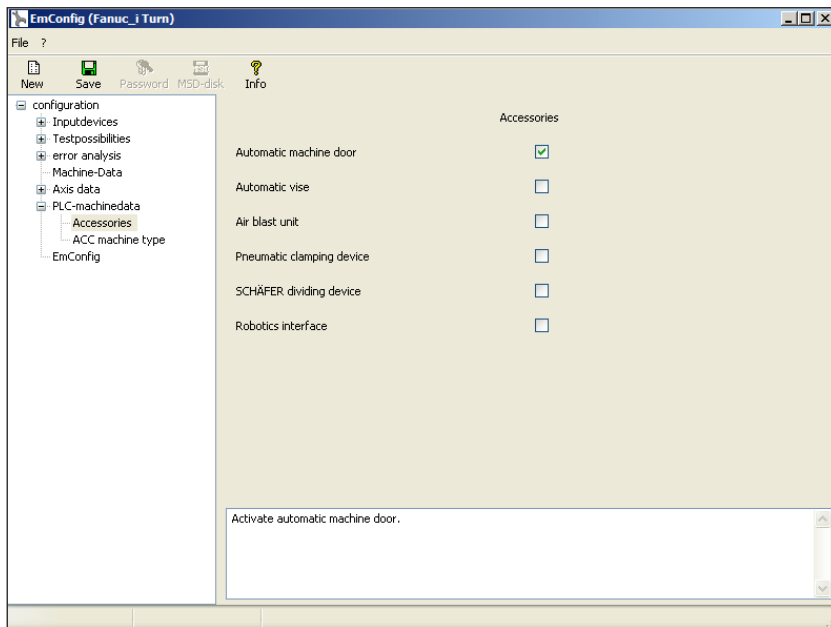Here you can change the language of EMConfig. In order to activate the settings, restart the program.

Note:
Select the desired menu item. The appropriate function is explained in the text box.
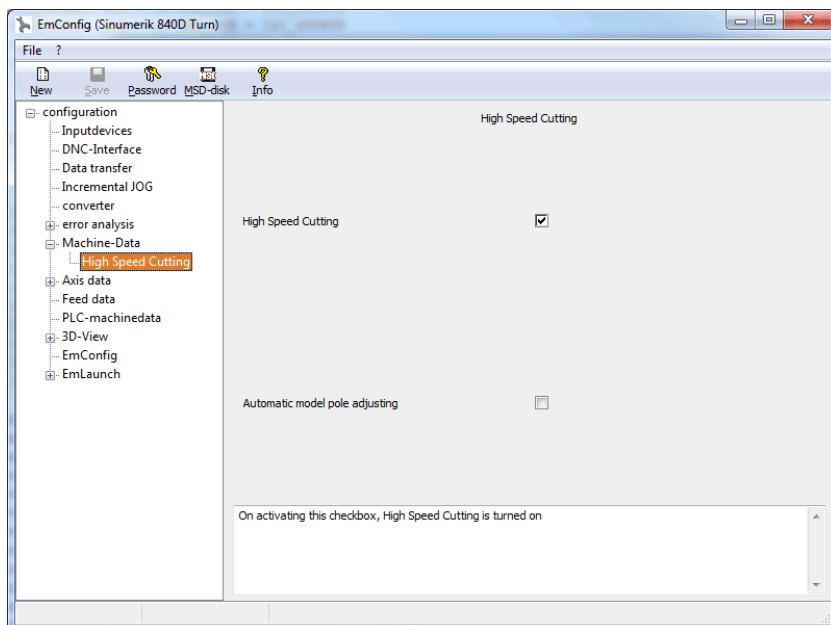
How to activate accessories

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.

emco

Easy2control on screen operation

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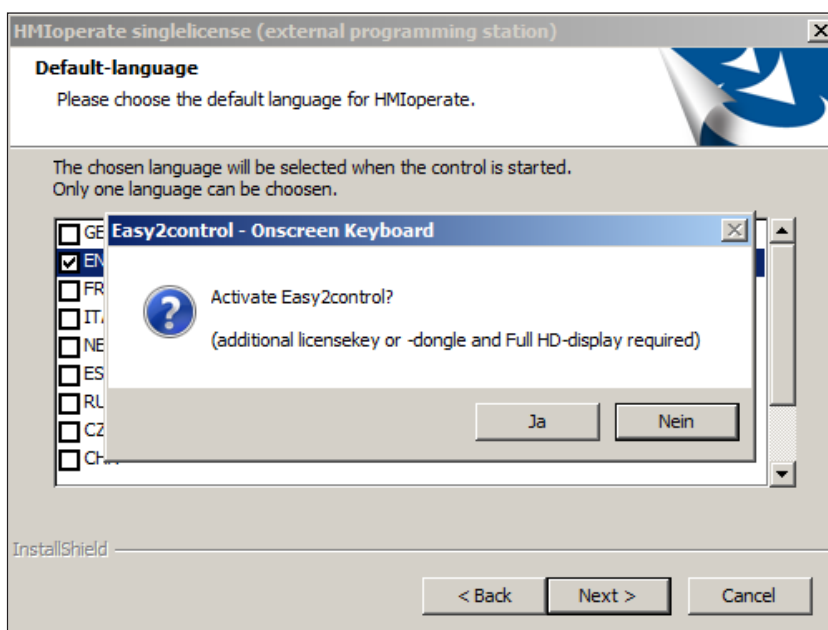
Activating Easy2control

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

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

Using Easy2operate on a programming station requires either
• a license dongle, or
• a license key, or
• a connected Easy2Operate keyboard.

Easy2control settings

This mask allows you to enable or disable Easy-2control and make settings.

Easy2control settings
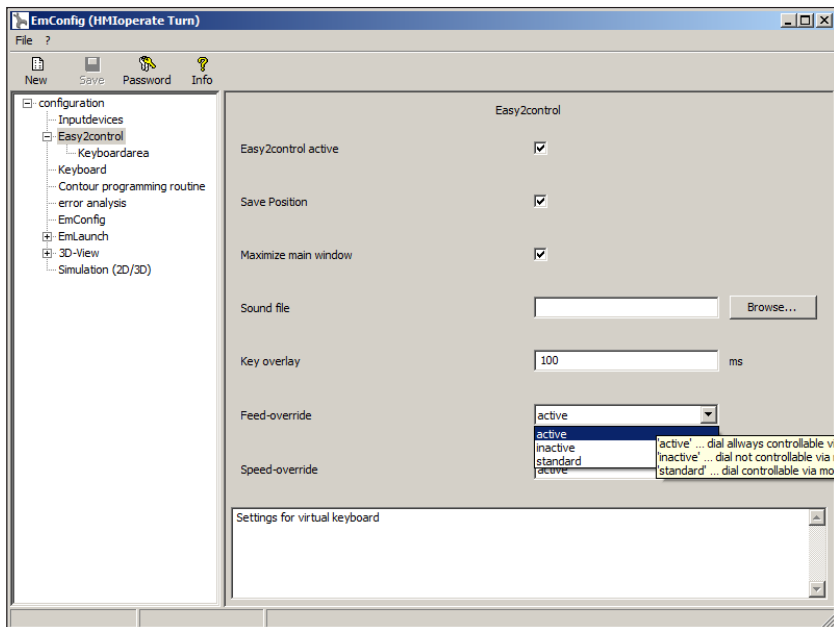
**Dial feed-override and
dial speed-override:**

- **Aktive**: dial always controllable via mouse/touch-screen (even with available hardware- version).

- **Inaktive**: dial not controllable via mouse/touchscreen.

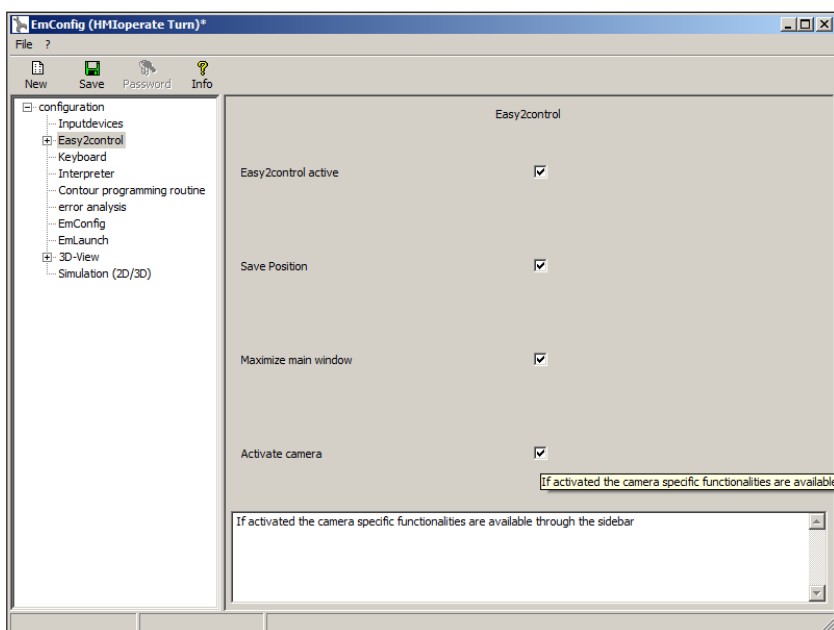- **Standard**: dial controllable via mouse/touchscreen when no hardware-version is available.

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.

Please refer to chapter Y "external input devices" for more information about the installation guide for the the camera.
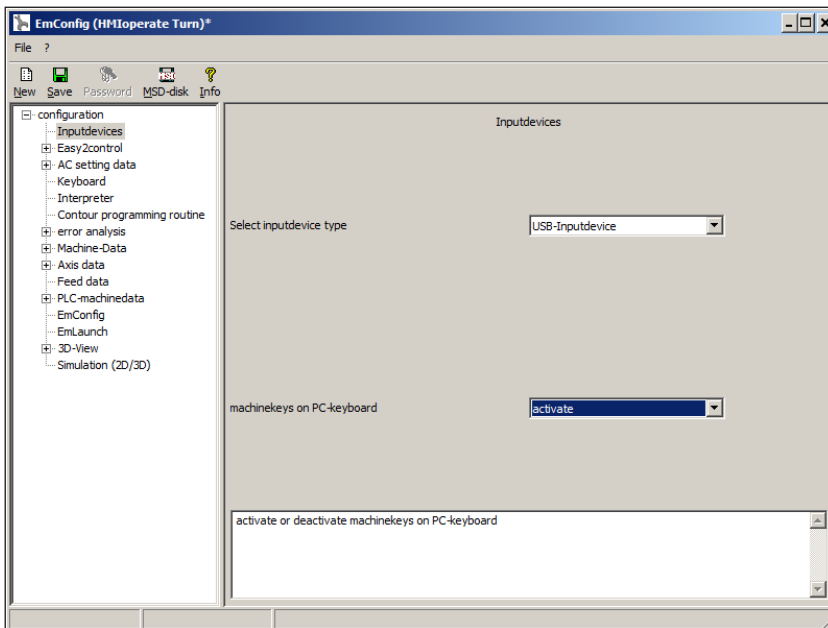

Activate camera

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.

emco

Machinekeys on PC keyboard

Setting for the machinekeys

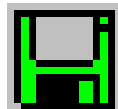You can activate or deactivate the machinekeys on the PC keyboard.

The setting is available for the following controls:

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

How to save changes

After the settings, the changes must be saved.

Select "Save" or click on the icon.

Note:
Input fields highlighted in red indicate inadmissible values. Inadmissible values are not saved in EMConfig.

After saving the changes, create a machine data floppy disk (MSD) or a machine data USB flash drive.

How to create machine data floppy disk or machine data USB flash drive

After having changed the machine data, the machine data floppy disk or the machine data USB flash drive must be in the appropriate drive.
Otherwise your changes cannot be saved and get lost.
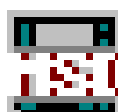
Y: External Input Devices

Easy2control On Screen operation

Easy2control adds a range of attractive applications to the successful interchangeable control system used in EMCO training machines. Suitable for use in machines and simulation workplaces alike, it displays additional control elements directly on the screen and, when used together with a touchscreen monitor, provides the ideal input interface.

Scope of supply

The software for Easy2control is part of the control software.

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

Technical data for the screen:

16:9 Full-HD Monitor (1920x1080) at the minimum

Currently available controls (T and M):

- Sinumerik Operate
- Fanuc 31i
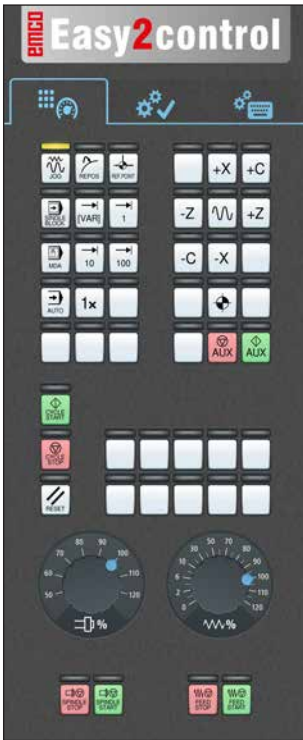- Emco WinNC for Heidenhain TNC 640 (M only)

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

Note:
If a Full HD monitor is used without touchscreen function, the control is operated just with mouse and keyboard.

emco

Operating areas

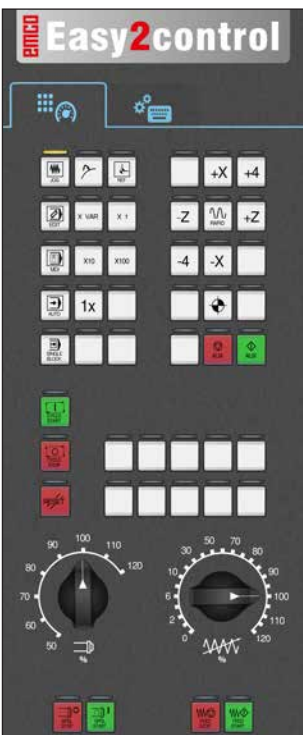## Sinumerik Operate

Machine control panel

Controller-specific operation

Control operation complete

Fanuc 31i

Machine control panel

Control operation complete

Emco WinNC for Heidenhain TNC 640
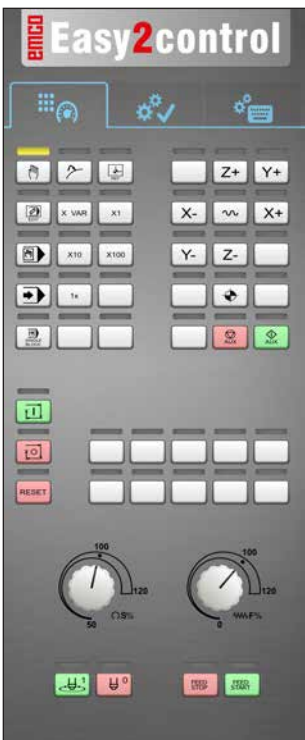

Machine control panel

Controller-specific operation

Control operation complete

Heidenhain TNC 426
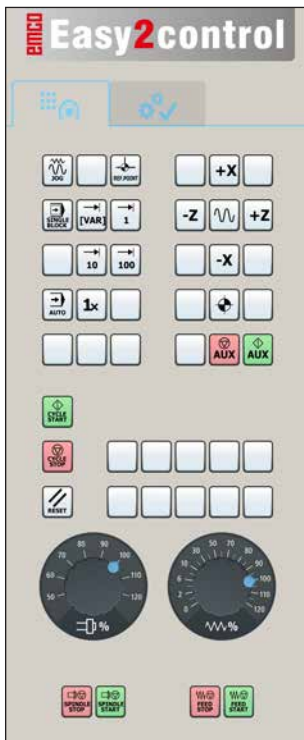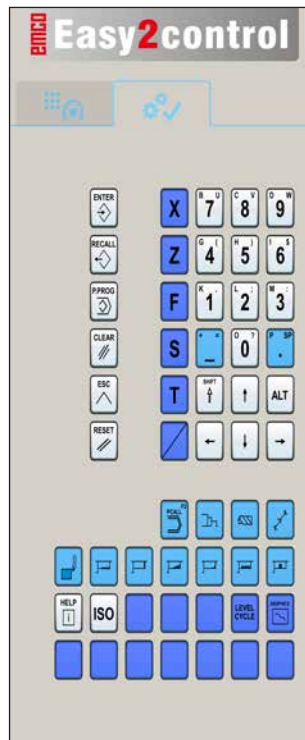

Machine control panel

Controller-specific operation

Control operation complete

emco

Fagor 8055

Machine control panel

Control operation complete

Refer to the chapter "Key Description" of the respective control description for operation and key function.
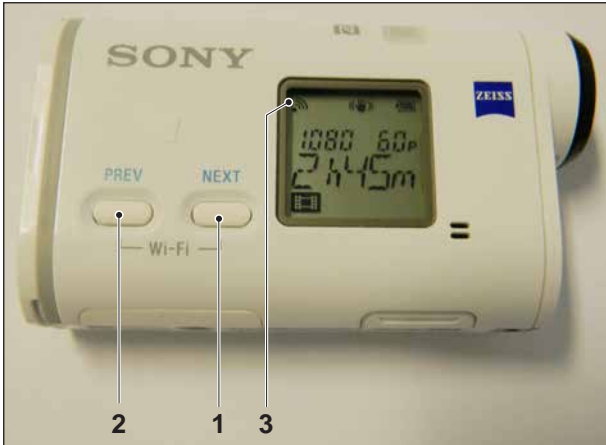
> **Note:**
> The screen display, based on customer-specific configurations, may look different.

Machine room camera

This accessory is no longer available!

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.

emco

Operating the camera

• Click the Easy2control logo to open the sidebar.

Functions of the sidebar
• Click the camera symbol to open the preview window (2).

• Open the operating manual of the control.

• Option for the second screen:
  - duplicate screen
  - screen extention to two monitors

• Creates a screenshot of the control in *.png format.
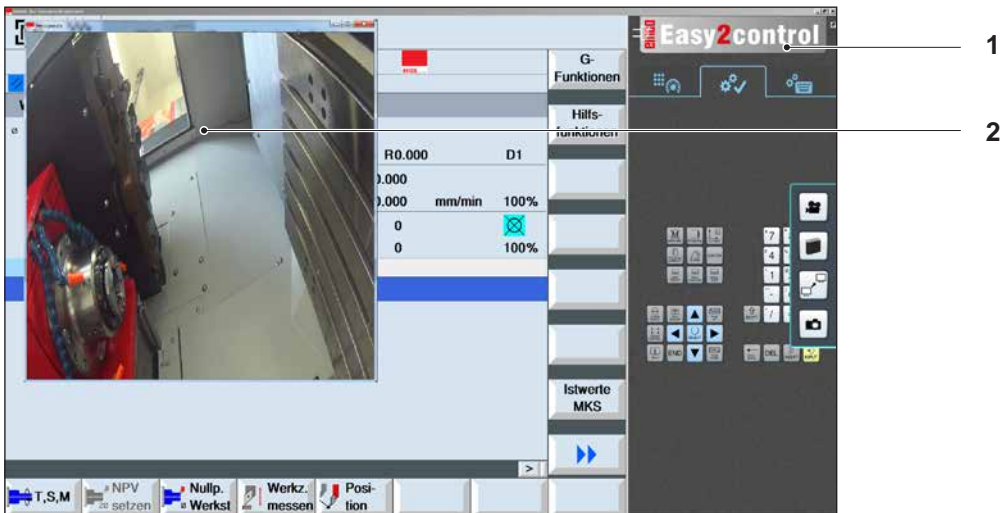
Operating the machine room camera

Note:
The option for the second screen is only available for machines CT/CM 260 and 460.

Attention:
The camera may not be operated without the supplied waterproof case.

If the camera is operated without the waterproof case it may be damaged by coolant and chips.

Z: Software Installation Windows

System prerequisites

Machine with integrated control PC
- All Concept machines
- Machines that were converted to ACC
- MOC with Windows 7 or higher (32 / 64 Bit)

Machines with included control PC and programming stations
- Windows 7 or higher (32 / 64 Bit)
- free hard drive space 400 MB
- Programming station: 1*USB, machine version: 2*USB
- TCP/IP-capable network card for machine version)

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

Software installation

- Start Windows
- Start the installation application on the USB stick or your download file.
- Follow the instructions from the installation guide.

For more informations regarding software installation and / or software update please refer to the documentation "short description for WinNC update installation".

Variants of WinNC

You can install EMCO WinNC for the following CNC control types:

- WinNC for SINUMERIK Operate T and M
- WinNC for FANUC 31i T and M
- HEIDENHAIN TNC 426
- Emco WinNC for HEIDENHAIN TNC 640
- FAGOR 8055 TC and MC
- CAMConcept T and M

In case there are several control types installed, a menu appears when starting EM Launch from which you can select the desired type.
The following versions can be installed from the WinNC variants:

- Demo licence:
  The demo licence is valid for 30 days after the first use. 5 days before the demo licence expires, you can enter another valid licence key (see licence manager)

- Programming station:
  Programming and operation of the appropriate CNC control type is simulated by WinNC on your PC.
  - Single user licence:
    Authorizes to external programming of CNC-controlled machine tools on one PC workstation.
  - Multi-user licence:
    Authorizes to external programming of CNC-controlled machine tools. The multi-user licence can be installed on an unlimited number of PC workstations or in a network within the institute registered by the licensor.
  - Educational licence version:
    Is a time-limited multi-licence especially for schools and educational institutes.

- Machine licence:
  This licence allows to directly operate a PC-controlled machine (PC TURN, Concept TURN, PC MILL, Concept MILL) of WinNC as if it was operated by an ordinary CNC control.

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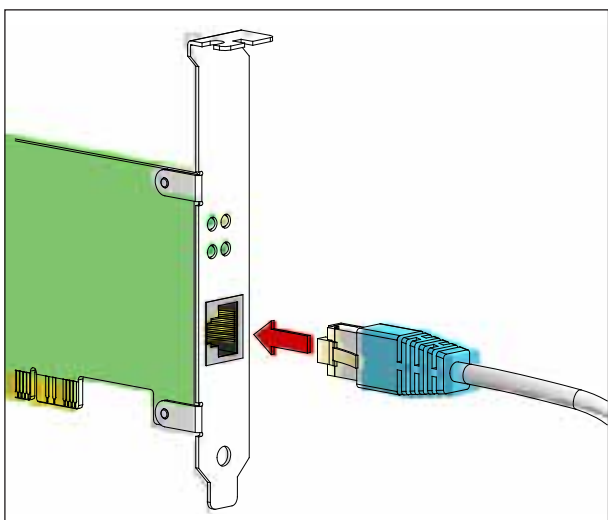
EMCO

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, the installed folder, and start WinNC Launch.

6 The start image will be shown on the screen. The licence holder is registered in the start screen.

7 If you have only installed one CNC control type, it starts immediately.

8 If you have installed several CNC control types, the selection menu appears.

9 Select the desired CNC control type (use cursor buttons or mouse) and press ENTER to start it.

10 If you use the control keyboard, you can select the desired CNC control type with the cursor buttons or mouse and start with the "NC-Start" button.

11 Tool Buttons (1) can be configured in EMConfig.

Selection menu EMLaunch

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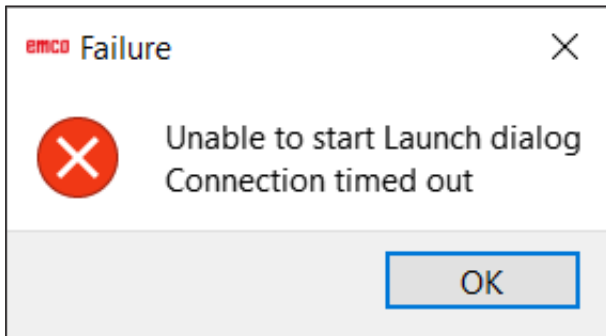
Terminating WinNC

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.
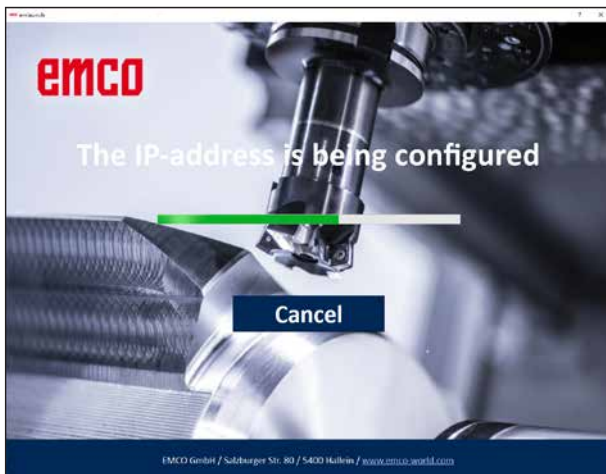This corresponds to Alt+F4 on the PC keyboard.

Checks by EmLaunch

Connection to the machine failed

An incorrect IP address was entered, the connection to the machine cannot be established.

IP-address configuration

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

Network connection is not possible

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.
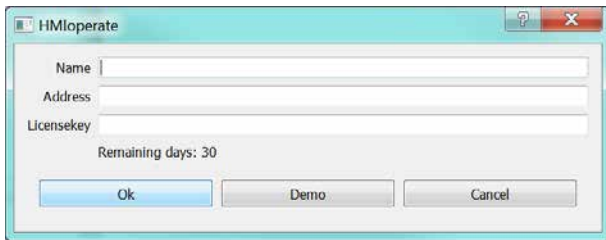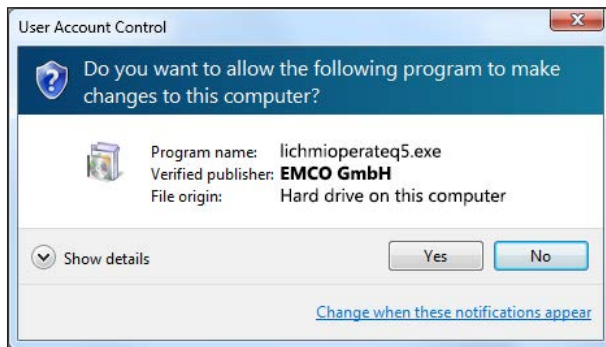
Conncection to the machine is OK

The connection to the machine is completed and the corresponding control can be started.
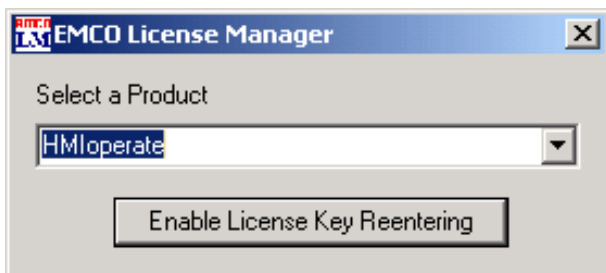
Input window licence key enquiry

Run EMCO licence manager upon licence key input

EMCO Licence Manager

Licence input

After the installation of an EMCO software product, an input window appears during initial operation and asks for name, address and licence key. If an Emco USB-Stick is connected, this data is taken from the USB-Stick.

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

This input window appears for every software product that is installed. In case a demo licence is desired (see page Z1), please select "DEMO". Then the input window reappears only 5 days before the expiry date of the demo licence. A subsequent input of a licence key is also possible via the licence manager (see licence manager below).

Licence manager

For the release of additional function groups of existing EMCO software products it is necessary to enter a new licence key (exception: demo licence).

The EMCO Licence Manager enables the input of further new licence keys. For this purpose select the new product in the selection window and confirm the input.

The next time you start your control software an input window appears and asks you to enter name, address and licence key.

Please note that the licence key is asked for each software product individually. The picture on the left shows e.g. the input prompt for the licence key for the software product "HMIoperate".