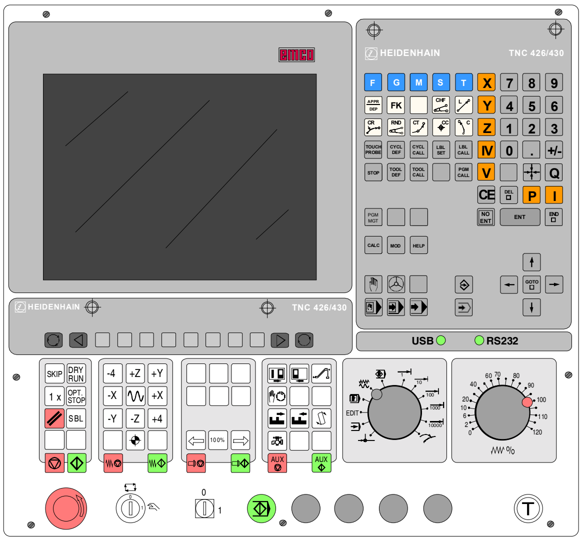
EMCO WinNC Heidenhain TNC 426 Conversational Software Description Software version since 2.08



Software Description
EMCO WinNC
Heidenhain TNC 426

Ref.-Nr. EN 1816 Edition E2013-12

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

Original operating instructions

EMCO Maier Ges.m.b.H.

P.O. Box 131

A-5400 Hallein-Taxach/Austria

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

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

Internet: www.emco-world.com

E-Mail: service@emco.at

Designed for your profit

Notice

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

Preface

The EMCO WinNC Heidenhain TNC 426 Milling Software is part of the EMCO training concept on PC-basis.

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

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

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

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

This manual does not include the whole functionality of the control software Heidenhain TNC 426 Milling, however emphasis was laid on the simple and clear illustration of the most important functions so as to achieve a most comprehensive learning success.

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

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

emco group
Designed for your profit

blank page

Contents

A: Fundamentals ... A1
- Reference points of EMCO milling machines ... A1
- Datum shift ... A2
- Reference system for milling ... A3
- machines ... A3
- Polar coordinates ... A4
- Absolute and incremental workpiece positions ... A5
  - Incremental workpiece positions ... A5
  - Absolute and incremental polar coordinates ... A5
- Tool data ... A6

B: Key Description ... B1
- Address keyboard and ... B2
- numerical keyboard ... B2
- Key functions ... B3
- German PC keyboard ... B6
  - Description of keys for German PC keyboard ... B7
- English PC keyboard ... B8
  - Descripton of keys for English PC keyboard ... B9

Machine control panel ... B10

Key Description ... B10
- Skip (block mask) ... B10
- Dryrun (test-run feed) ... B10
- Individual piece mode ... B11
- Optional stop ... B11
- Reset key (Reset) ... B11
- Single block ... B11
- NC Stop ... B11
- NC Start ... B11
- Arrow keys ... B12
- Reference point ... B12
- Rapid Traverse ... B12
- Feed Stop ... B12
- Feed Start ... B12
- Spindle speed correction ... B12
- Spindle Stop ... B12
- Spindle Start ... B12
- Automatic machine doors ... B13
- Chip conveyor (Option) ... B13
- Swing tool drum ... B13
- Manual tool change ... B13
- Clamping device ... B13
- Coolant ... B14
- Auxiliary OFF ... B14
- Auxiliary ON ... B14
- Types of operation ... B14
- Override switch (feed rate override) ... B16
- EMERGENCY STOP ... B16
- Key Switch Special Operations Mode ... B16
- Multifunction switch for operating modes ... B17
- Key switch ... B20
- Additional NC start button ... B20
- USB connection (USB 2.0) ... B20
- Enable button ... B20

C: Operation ... C1
- Switch-off ... C1
- Modes of operation ... C1
  - Calling operating modes ... C1
  - Navigation in the menu window ... C1
- Machine operation ... C3
  - Positioning with Manual Data Input (MDI) ... C6
  - Target file = BOREHOLE ... C7
- Fundamentals of file management ... C9
  - Files ... C9
- Standard file management ... C10
- Advanced file management ... C14
  - DEMO ... C17
- Block ... C21
  - 10 L X+10 Y+5 R0 F100 M3 ... C21
- Creating and writing programs ... C21
  - Programming tool movements in conversational format ... C23
- MOD-functions ... C25

D: Programming ... D1
- Pocket Calculator ... D4
- Programming graphics ... D5
- Tool movements ... D7
- Fundamentals of path functions ... D8
  - Overview: Types of paths for contour approach and departure ... D11
  - Important positions for approach and departure ... D12
  - Approaching on a straight line with tangential connection: APPR LT ... D13
  - Approaching on a straight line perpen-dicular to the first contour point: APPR LN ... D13
  - Approaching on a circular path with tangential connection: APPR CT ... D14
  - Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT ... D14
  - Departing on a straight line with tangential connection: DEP LT ... D15
  - Departing on a straight line perpendicular to the last contour point: DEP LN ... D15
  - Departure on a circular arc with tangential connection: DEP CT ... D16
  - Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT ... D16
  - Overview of path functions ... D17
  - Straight line L ... D18
  - Inserting a chamfer CHF between two straight lines ... D19
  - Corner Rounding RND ... D20
  - Circle center CC ... D21
  - Circular path C around circle center CC ... D22
  - Circular path CR with defined radius ... D23
  - Circular path CT with tangential connection ... D24
  - example: square ... D25
  - example: nooks round / chamfer 1 ... D26
  - example: nooks round / chamfer 2 ... D27
  - example: circular motions ... D28
  - example: circular arc with CC, C ... D29
  - example: milling with multi-infeed ... D30
  - Overview ... D31

Polar coordinate origin: Pole CC D31

Straight line LP P D32

Circular path CP around pole CC P D32

Circular path CTP with tangential connection P D33

Helix P D33

Fundamentals...D35
Graphics during FK programming...............................D36
Initiating the FK dialogue...D37
Free programming of straight lines D37
Free programming of circular arcs...............................D38
Input possibilites...D39
Converting FK programs..D40
example: FK telephone D41

Cycles D43
Working with cycles...D43
Defining a cycle using soft keys..................................D43
Defining a cycle using the GOTO function...................D43
Calling a cycle D44
Point Tables...D46
PECKING (Cycle 1) D50
DRILLING (Cycle 200) D51
REAMING (Cycle 201) D53
BORING (Cycle 202)...D54
UNIVERSAL DRILLING (Cycle 203)..............................D56
BACK BORING (CYCLE 204) D58
UNIVERSAL PECKING (Cycle 205)...............................D60
BORE MILLING (Cycle 208) D62
TAPPING with a floating tap holder (Cycle 2)..............D64
TAPPING NEW with floating tap holder (Cycle 206)....D65
RIGID TAPPING (Cycle 17) D66
RIGID TAPPING without a floating tap holder
TAPPING (Cycle 207) D67
THREAD CUTTING (CYCLE 18) D68
TAPPING WITH CHIP BREAKING (Cycle 209) D69
THREAD MILLING (Cycle 262) D72
THREAD MILLING/COUNTERSINKING (Cycle 263).. D74
THREAD DRILLING/MILLING...
(Cycle 264) D76
HELICAL THREAD DRILLING/MILLING (Cycle 265)..D78
OUTSIDE THREAD MILLING (Cycle 267)...................D80
POCKET MILLING (Cycle 4)...D84
POCKET FINISHING (Cycle212)..................................D85
STUD FINISHING (Cycle 213)......................................D87
CIRCULAR POCKET MILLING (Cycle 5).....................D89
CIRCULAR POCKET FINISHING (Cycle 214).............D90
CIRCULAR STUD FINISHING (Cycle 215).................D92
SLOT MILLING (Cycle 3) D94
SLOT (oblong hole) with reciprocating plungecut (Cycle 210)...D96
CIRCULAR SLOT (oblong hole) with reciprocating plunge-cut (Cycle 211) D98
Cycles for Machining Hole Patterns D101
Overview...D101
CIRCULAR PATTERN (Cycle 220)...............................D102
LINEAR PATTERN (Cycle 221) D104
Fundamentals..D107
Overview of SL-Cycles...D108
SL-cycles, program flowchart.......................................D109
M2...D109
CONTOUR GEOMETRY (Cycle 14) D110
Overlapping contours...D111
CONTOUR DATA (Cycle 20)..D113
REAMING (Cycle 21) D114
ROUGH-OUT (Cycle 22)..D115
FLOOR FINISHING (Cycle 23) D116
SIDE FINISHING (Cycle 24) D117
CONTOUR TRAIN (Cycle 25).......................................D118
CYLINDER SURFACE (Cycle 27).................................D119
Cycles for multipass milling D121
Overview...D121
MULTIPASS MILLING (Cycle 230) D122
RULED SURFACE (Cycle 231) D124
Coordinate Transformation Cycles D127
Overview...D127
Effect of coordinate transformation.............................D127
DATUM SHIFT (Cycle 7)..D128
DATUM SHIFT with datum tables (Cycle 7) D129
MIRROR IMAGE (Cycle 8)...D132
ROTATION (CYCLE 10) D133
SCALING FACTOR (Cycle 11).....................................D134
Special Cycles..D135
Overview...D135
DWELL TIME (Cycle 9)..D135
PROGRAM CALL (Cycle 12) D136
ORIENTED SPINDLE STOP (Cycle 13).....................D137
Subprograms..D139
Labeling subprograms and program section repeats.D139
Labels D139
Subprograms...D140
Program section repeats...D141
Separate any program as subprogram D142
Nesting D143

E: Tool programming...E1
Entering tool-related data E1
Feed rate F E1
Spindle speed S E1
Tool Data E2
Requirements for tool compensation...........................E2
Tool number, tool name..E2
Tool length L...E2
Load the tools into the magazine.....................................E6
(random tool system)..E6
Decide tool and magazine E7
place...E7
Loading the tools in the magazine...................................E8
with a random tool system E8
Place the tool in the magazine......................................E9
Pre-positioning of the tool (only random tool system) .. E10
Calling tool data...E12
Tool Compensation E13
Introduction..E13
Tool length compensation E13
Tool radius compensation E14

F: Program run F1
Requirements...F1
Program start, Program stop F2

G: Flexible NC programming G1
Q parameters G1
Calling Q parameter functions G1
Calculating with Q parameters......................................G2
Trigonometric functions...G3
If-Then decisions with Q parameters G4
Entering formulas directly G5

H: Alarms and Messages

Machine Alarms 6000 - 7999 .... H1
PC MILL 50 / 55 / 100 / 105 / 125 / 155 .... H1
Concept MILL 55 / 105 / 155 .... H1
PC TURN 50 / 55 / 105 / 120 / 125 / 155 .... H6
Concept TURN 55 / 60 / 105 / 155 / 250 / 260 .... H6
Concept MILL 250 .... H6
EMCOMAT E160 .... H6
EMCOMAT E200 .... H6
EMCOMILL C40 .... H6
EMCOMAT FB-450 / FB-600 .... H6
Inputunit alarms 1700 - 1899 .... H15
Axis Controller Alarms H17
8000 - 9000, 22000 - 23000, 200000 - 300000 .... H17
Axis Controller Messages H24
Control alarms 2000 - 5999 .... H25
Fagor 8055 TC/MC H25
Heidenhain TNC 426 .... H25
CAMConcept H25
EASY CYCLE H25
Sinumerik for OPERATE H25
Fanuc 31i H25

W: Accessory Functions

Activating accessory functions W1
Robotic Interface W1
Automatic doors W1
Win3D View W1
DNC interface W2

X: EMConfig

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

Y: External Input Devices

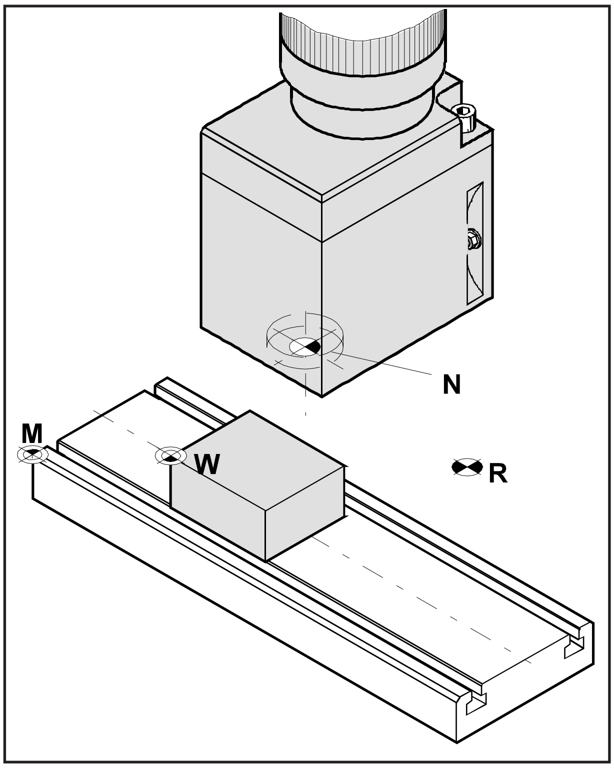
EMCO Control Keyboard USB Y1
Scope of supply Y1
Assembling Y2
Connection to the PC Y4
Settings at the PC software Y4
Setting during new installation of the PC software Y4
Setting in case of PC software already installed Y4
Easy2control On Screen operation Y5
Scope of supply Y5
Operating areas Y6

Z: Software Installation Windows

System prerequisites Z1
Software installation Z1
Variants of WinNC Z1
Network card (ACC) Z2
Starting WinNC Z3
Terminating WinNC Z3
Checks by EmLaunch Z4
Licence input Z6
Licence manager Z6

blank page

A: Fundamentals



Reference points in the working space

Reference points of EMCO milling machines

M = Machine datum

"M" is an unchangeable reference point, fixed by the manufacturer.
From this point the whole maschine will be measured.
At the same time "M" is the basis of the coordinate system.

R = Reference point

"R" is an exactly defined position in the working space of the machine.
When the slides move to "R", the control is informed of the slide positions. This is necessary after every interruption of circuit.

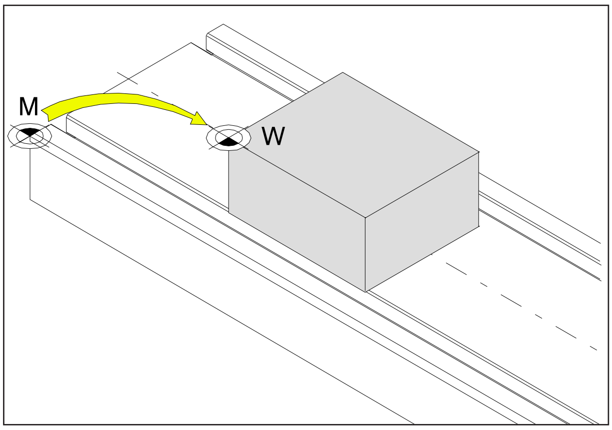
N = Tool-holding-fixture reference point

"N" is the starting point for measuring the machine tools. "N" is positioned at a suitable place of the tool holding system and is set by the machine tool manufacturer.

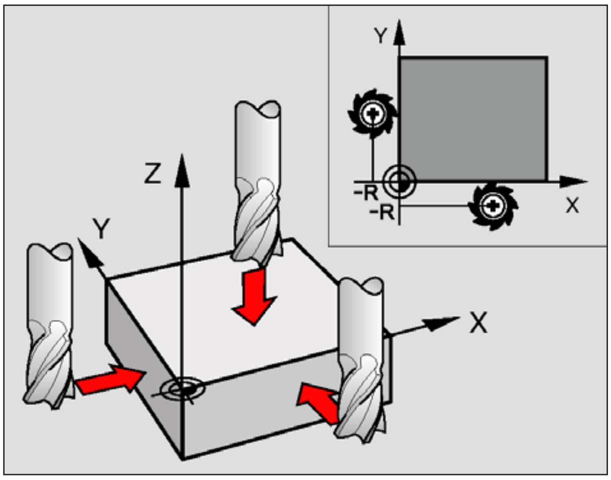
W = Workpiece datum

"W" is the starting point for the unit of measurement in the part program.
It can be freely set by the programmer and may be shifted as often as desired within the part program.

Note:
When HSC mode is active, the feedrate during contour machining must be reduced to 2500 mm/min (at 100% OVR).



Datum shift from machine datum M to workpiece datum W



Datum shift

EMCO milling machines have their machine datum "M" located on the left front edge of the machine table. This location is not suited as starting point for programming.

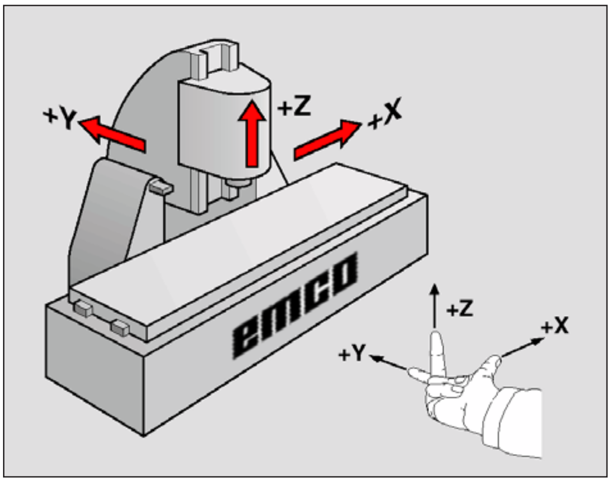
Heidenhain TNC 426 knows 2 methods, which can be combined, in order to set a datum shift:

1.) reference point setting (see below)
2.) Cycle 7- Datum shift. Absolute or incremental coordinates are usable. (see chapter D, Coordinate Transformation Cycles)

Datum/ reference point setting

- Select **Manual operation mode**  .
- Move the tool slowly until it touches the workpiece surface +X, -X, +Y, -Y, +Z, -Z, +4, -4.
- Select an axis (all axes can also be selected via the ASCII-keyboard).
- ENT zero tool, spindle axis: set the display to a known position on the workpiece (e.g. 0). In the tool axis: offset the tool radius.
- Repeat the process for the remaining axes.

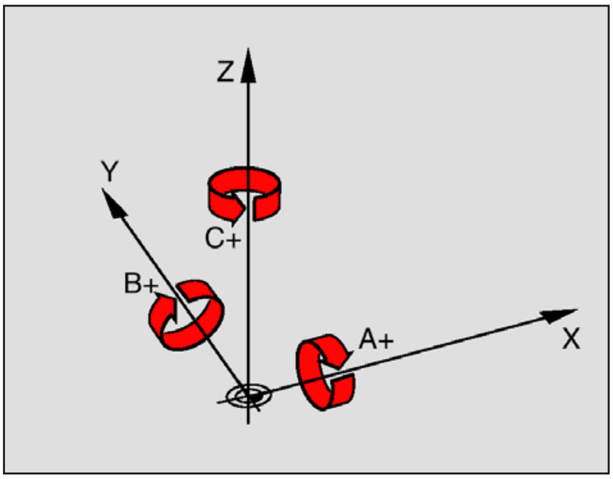
If you use a preset tool, set the display of the tool axis to the length L of the tool.



Reference system for milling machines

A reference system is required to define positions in a plane or in space. The position data are always referenced to a predetermined point and are described through coordinates.
The Cartesian coordinate system (a rectangular coordinate system) is based on three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate indicates the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

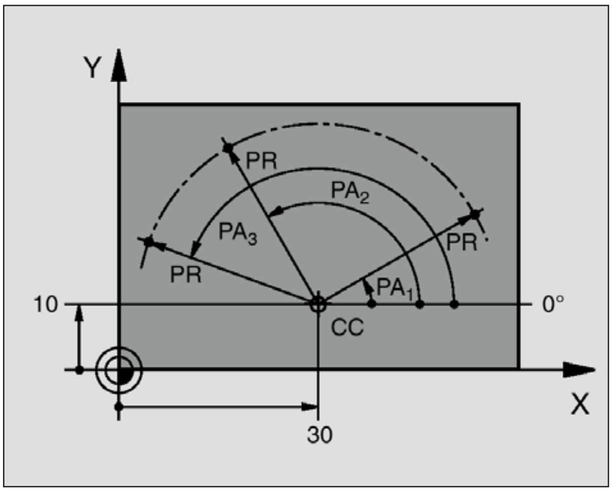
Coordinates that are referenced to the datum are called **absolute coordinates**. Relative coordinates are referenced to any other known position (reference point) within the coordinate system. Relative coordinate values are also referred to as **incremental coordinate** values.

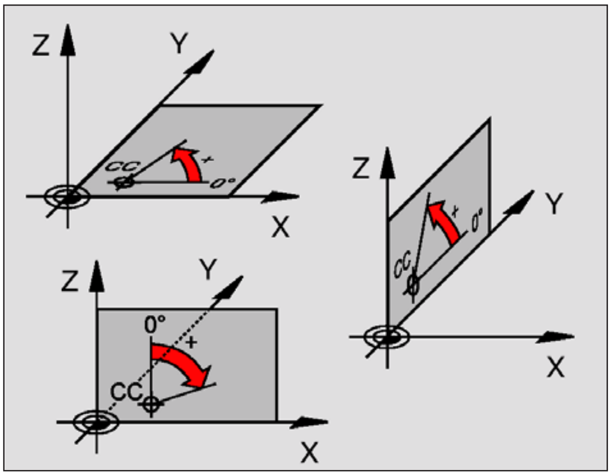


When working a workpiece on a milling machine you generally orient tool movement to the Cartesian coordinate system. The illustration on the left shows how the Cartesian coordinate system describes the machine axes. The "right-hand-rule" helps to remember the three axes directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece towards the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.

The TNC 426 is able to control up to 5 axes. The axes U, V and W are secondary linear axes parallel to the main axes X, Y and Z. Rotary axes are designated as A, B and C. The illustration on the lower left shows the assignment of secondary axes, respectively rotary axes to the main axes.

Note:
EMCO PC-machines do not endue at secondary axes.





Polar coordinates

If the production drawing is dimensioned in Cartesian coordinates, you also write the part program using rectangular coordinates. For parts with circular arcs or angles it is often simpler to fix the positions in polar coordinates.

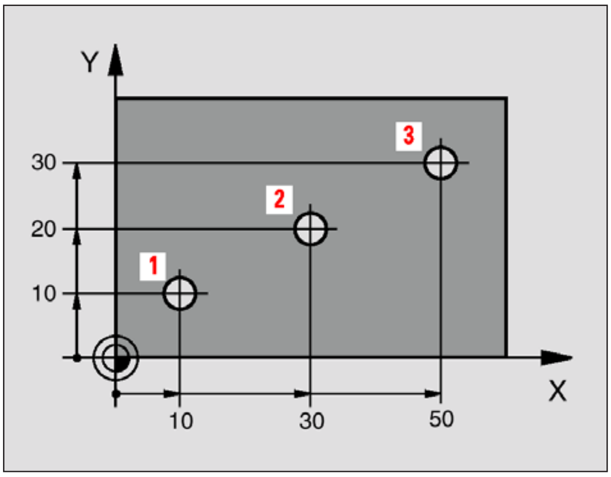
In contrast to the Cartesian coordinates X, Y and Z, which are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum in the circle center (CC) or pole.

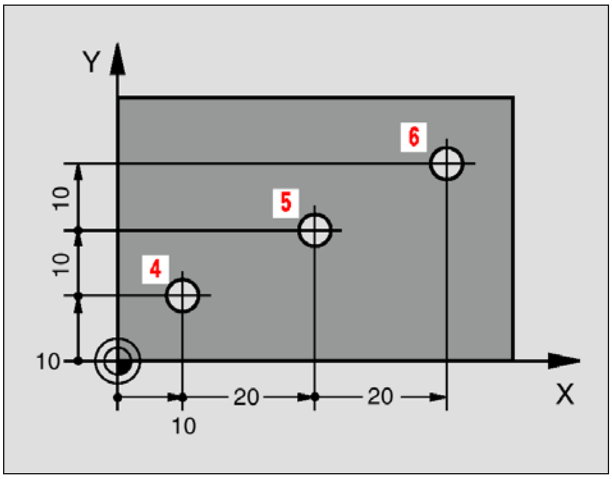
Therefore, a position in a plane is clearly defined by:

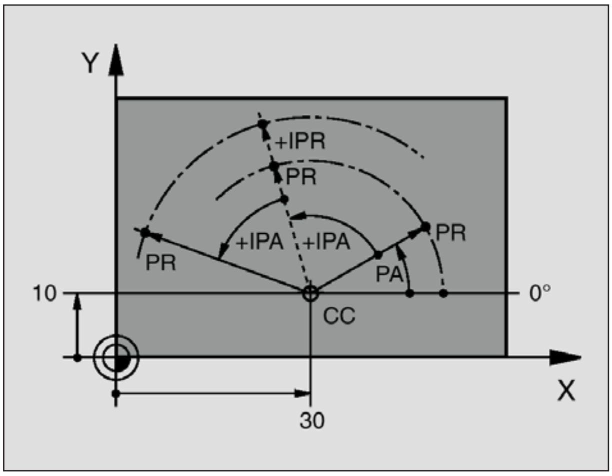
- Polar radius: the distance from the circle center CC to the position
- Polar angle: the size of the angle between the reference axis and the line that connects the circle center CC with the position (see figure on upper left).

Definition of pole and angle reference axis
The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the angle reference axis for the polar angle PA.

Coordinates of the pole (plane)	Reference axis of the angle
X/Y	+X
Y/Z	+Y
Z/X	+Z







Absolute and incremental workpiece positions

Absolute workpiece positions
Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is clearly defined by its absolute coordinates.

Example 1: Holes dimensioned in absolute coordinates

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

Incremental workpiece positions

Incremental coordinates are referenced to the last programmed nominal position of the tool, which serves as the relative (imaginary) datum. When a part program is written in incremental coordinates , the tool is programmed to move by the distance between the previous and the subsequent nominal positions. That is why incremental coordinates are also referred to as chain dimensions.

To program a position in incremental coordinates, enter the prefix "I" before the axis.

Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole 4
IX = 10 mm
IY = 10 mm

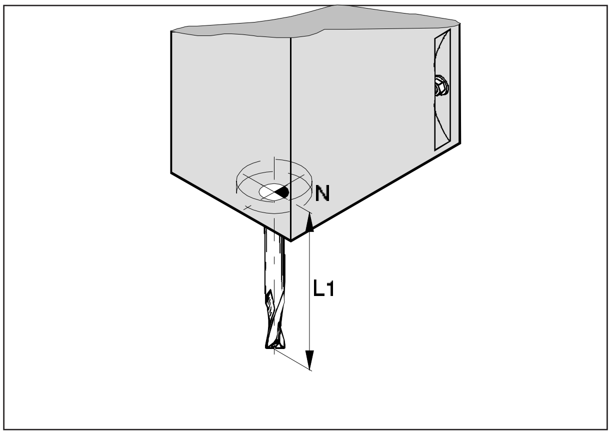
Hole 5, referenced to 4
IX = 20 mm
IY = 10 mm

Hole 6, referenced to 5
IX = 20 mm
IY = 10 mm

Absolute and incremental polar coordinates

Absolute coordinates always refer to the pole and the angle reference axis. Incremental coordinates always refer to the last programmed position of the tool.

Tool data



Correction of length

The aim of the tool data recording is that the software uses the tool tip, respectively the tool center for positioning and not the reference point for the toolholding fixture.

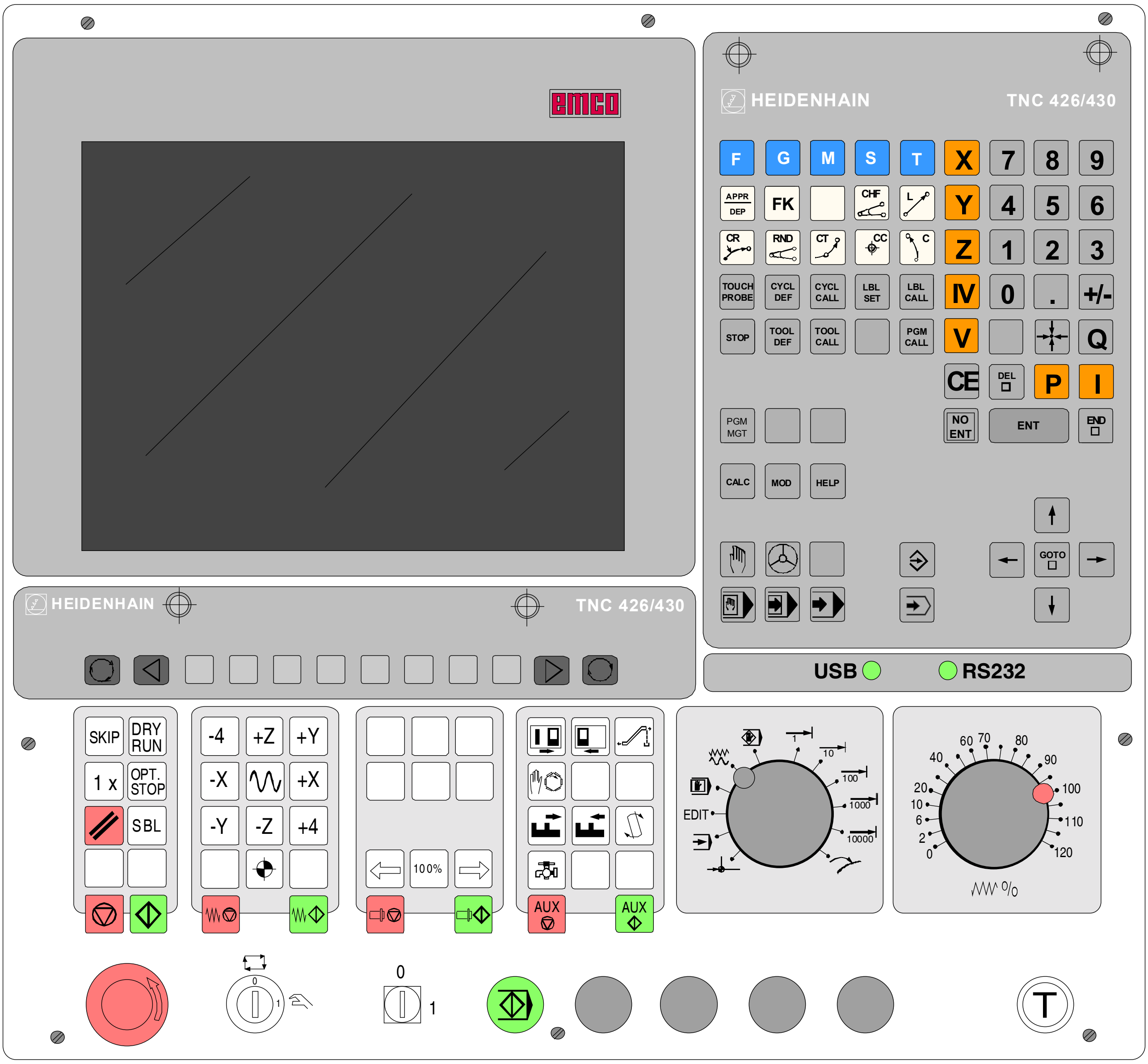
Every tool that is used in the working process must be measured. Therefore, it is necessary to calculate the distance between the tool tip and the toolholding-fixture reference point "N".

In the so-called tool data store the measured length compensation values and the mill radius can be stored.

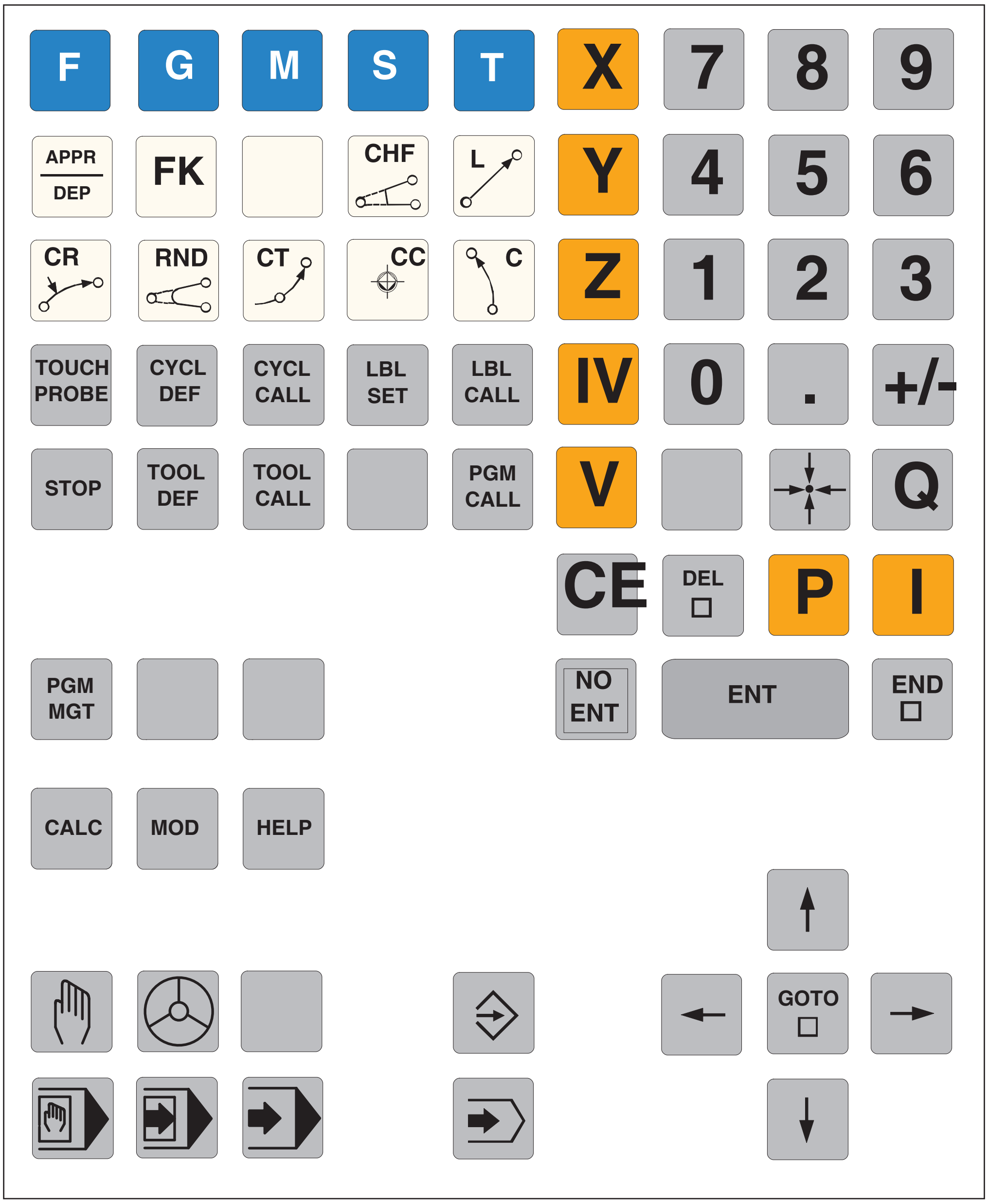
The specification of the mill radius is **only** necessary if a **mill radius compensation** or a milling cycle is selected for the corresponding tool!
(see chapter E tool programming)

B: Key Description

Control keyboard, Digitizer overlay



Address keyboard and numerical keyboard



Key functions

Programming path movements

APPR DEP	Approach/ depart contour
FK	FK free contour programming
L	Straight line
CC	Circle center/ pole for polar coordinates
C	Circle with center
CR	Circle with radius
CT	Circular arc with tangential connection
CHF	Chamfer
RND	Corner rounding

Entering letters and symbols

F G M S T	Entering letters and symbols (DIN/ISO-programming)

Cycles, subprograms and program section repeats

CYCL DEF, CYCL CALL	Define and call cycles
LBL SET, LBL CALL	Enter and call subroutines and program section repeats
STOP	Enter program stop in a program
TOUCH PROBE	Enter touch probe functions in a program

Tool functions

TOOL DEF	Enter tool length and tool radius
TOOL CALL	Call tool length and tool radius

Program/ file management, TNC functions

PGM MGT	Select or delete programs and files External data transfer
PGM CALL	Enter program call in a program
MOD	Select MOD functions
HELP	Displaying help texts for NC error messages
CALC	Pocket calculator

Selecting machine operating modes

Key	Function
	Manual operation
	Electronic handwheel
	Positioning with manual data input
	Program run, single block
	Program run, full sequence

Selecting programming operation modes

Key	Function
	Programming and editing
	Test run

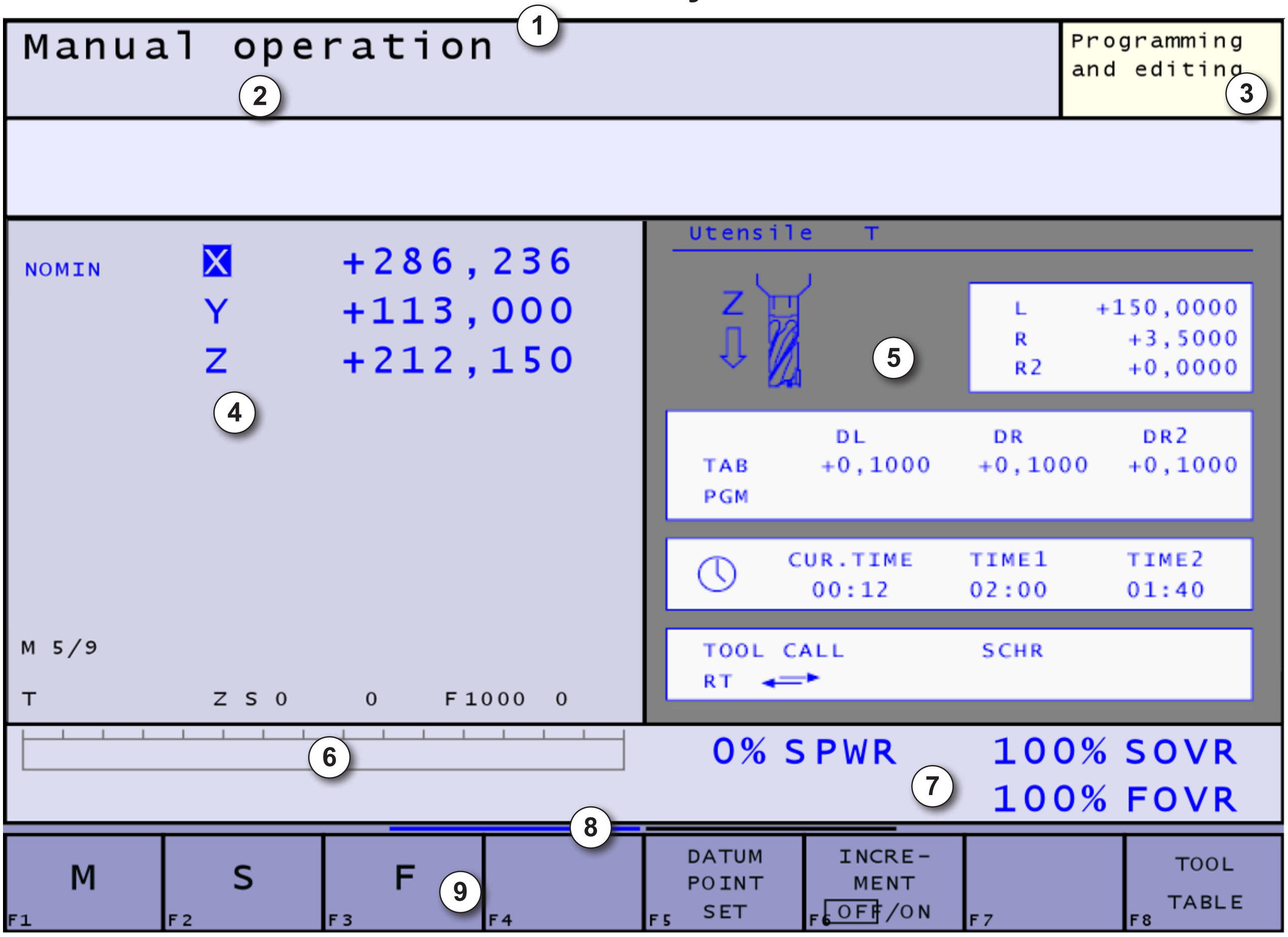
Moving the highlight, going directly to blocks, cycles and parameter functions

Key	Function
↑ ↓ ← →	Move highlight
GOTO □	Go directly to blocks, cycles and parameter functions

Entering and editing coordinate axes and numbers

Key	Function
Y Z IV V	Select coordinate axes or enter them into the programn
7 8 9 4 5 6 1 2 3 0	Numbers
.	Decimal point
+/-	Change arithmetic sign
P	Enter polar coordinates
I	Incremental dimensions
Q	Q parameters
	Actual position capture
NO ENT	Skip dialgue questions and delete words
ENT	Confirm entry and resume dialogue
END □	End block
CE	Clear numerical entry or TNC error message
DEL □	Abort dialogue, delete program section

Screen layout



1 Display of machine`s operating mode, dialogue line

2 Alarm and message line

3 Display of the programming operation mode

4 Working window, NC display

5 Additional status displays contain detailed information on the program run. They can be called in all operating modes except for the programming and editing mode of operation.

6 Power display

7 The general status display informs on the actual condition of the machine. It appears automatically.

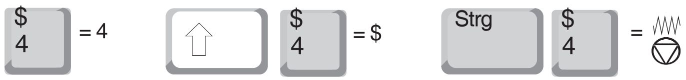
SPWR...... Power of main spindle
SOVR ...... Correction of spindle
FOVR ...... Correction of feed rate

8 The menu row shows the number of soft-key rows that can be chosen by using the or F9 or Shift F9 keys.

9 Softkey row

You can choose the screen layout in the corresponding menus by means of the or keys.

German PC keyboard



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

Several alarms are confirmed with the ESC key.

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

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

Description of keys for German PC keyboard

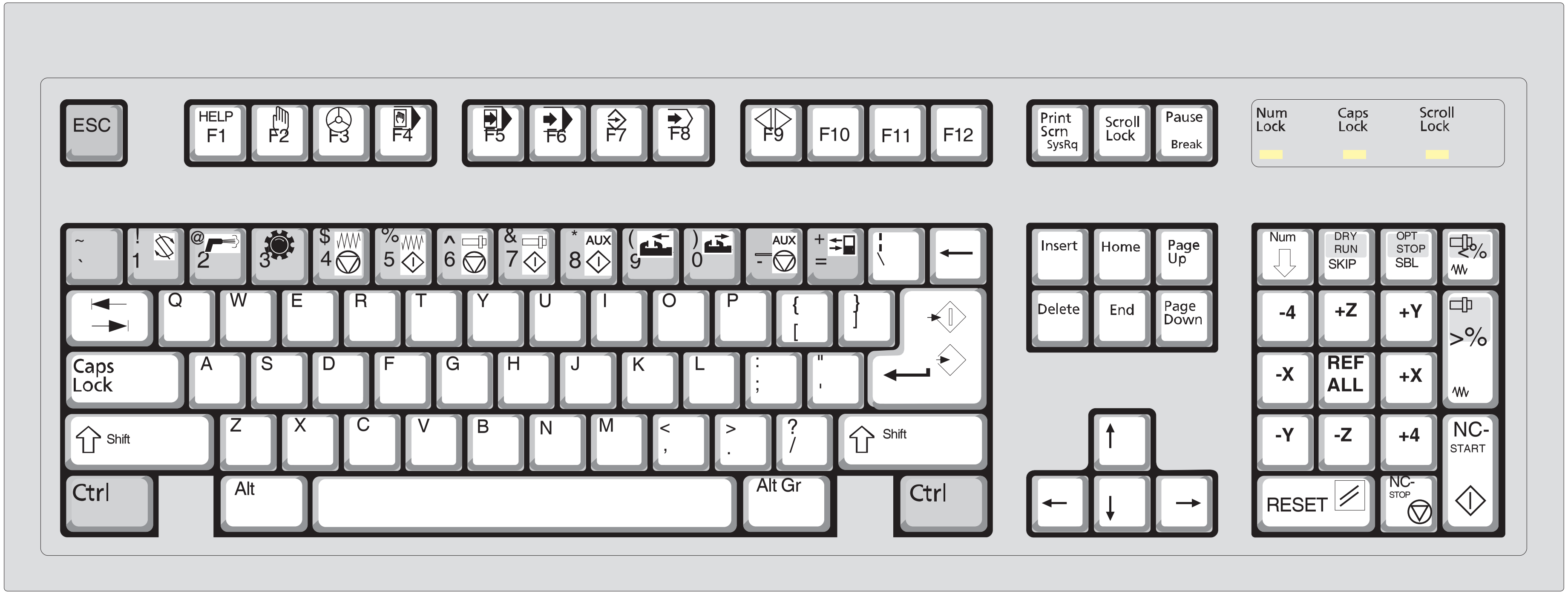
Keys	Function
F11 – F8	Soft keys
F9	Shift softkey rows (forward)
⇆	Select screen display
Strg A	APPR/DEP
Strg B	Actual position capture
Strg C	CC (Circle Center)
Strg D	TOOL DEF
Strg E	LBL
Strg F	FK
Strg G	LBL CALL
Strg H	CHF
Strg I	C (circle)
Strg K	I (incrementall)
Strg L	L (line)
Strg M	+/- key
Strg N	RND
Strg O	TOOL CALL
Strg P	P (polar)
Strg Q @	PROG CALL
Strg R	CR (circle with radius)
Strg S	STOP
Strg T	CT (circle tangential)
Strg U	Q parameter

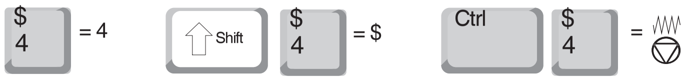
Keys	Function
Strg V	NO ENT
Strg W	CALC
Strg X	CE
Strg Y	CYCLE DEF
Strg Z	CYCLE CALL
Strg F11	MOD
Strg ↓	GO to
⇧ HELP F1	HELP
⇧ F2	Manual operation
⇧ F3	Electronic handwheel
⇧ F4	Positioning with manual data input
⇧ F5	Program run, single block
⇧ F6	Program run, full sequence
⇧ F7	Programming/ Editing
⇧ F8	Test run
⇧ F9	Shift soft-key rows (back)
⇧ F10	Shift machining/ programming operating mode
⇧ F11	PGM MGT

Note:
Selecting machine keys via the PC keyboard:

1.) Press and hold Strg key.
2.) Press machine key and then release it.
3.) Release Strg key.

English PC keyboard





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

Several alarms are confirmed with the ESC key.

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

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

Descripton of keys for English PC keyboard

Key	Function
F11 – F8	Soft keys
F9	Shift softkey rows (forward)
	Select screen display
Ctrl A	APPR/DEP
Ctrl B	Actual position capture
Ctrl C	CC (circle center)
Ctrl D	TOOL DEF
Ctrl E	LBL
Ctrl F	FK
Ctrl G	LBL CALL
Ctrl H	CHF
Ctrl I	C (circle)
Ctrl K	I (incremental)
Ctrl L	L (line)
Ctrl M	+/- key
Ctrl N	RND
Ctrl O	TOOL CALL
Ctrl P	P (polar)
Ctrl Q @	PROG CALL
Ctrl R	CR (circle with radius)
Ctrl S	STOP
Ctrl T	CT (circle tangential)
Ctrl U	Q parameter

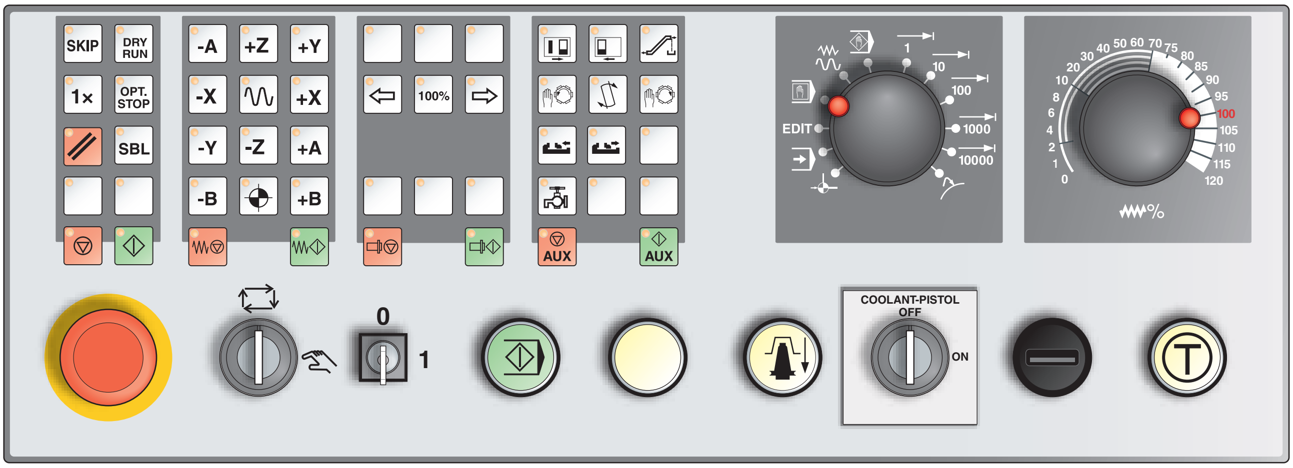
Key	Function
Ctrl V	NO ENT
Ctrl W	CALC
Ctrl X	CE
Ctrl Y	CYCLE DEF
Ctrl Z	CYCLE CALL
Ctrl F11	MOD
Ctrl ↓	GO TO

Key	Function
⇧ HELP F1	HELP
⇧ F2	Manual operation
⇧ F3	Electronic handwheel
⇧ F4	Positioning with manual data input
⇧ F5	Program run, single block
⇧ F6	Program run, full sequence
⇧ F7	Programming/ Editing
⇧ F8	Test run
⇧ F9	Shfit soft-key rows (back)
⇧ F10	Shift machining/ programming operating mode
⇧ F11	PGM MGT

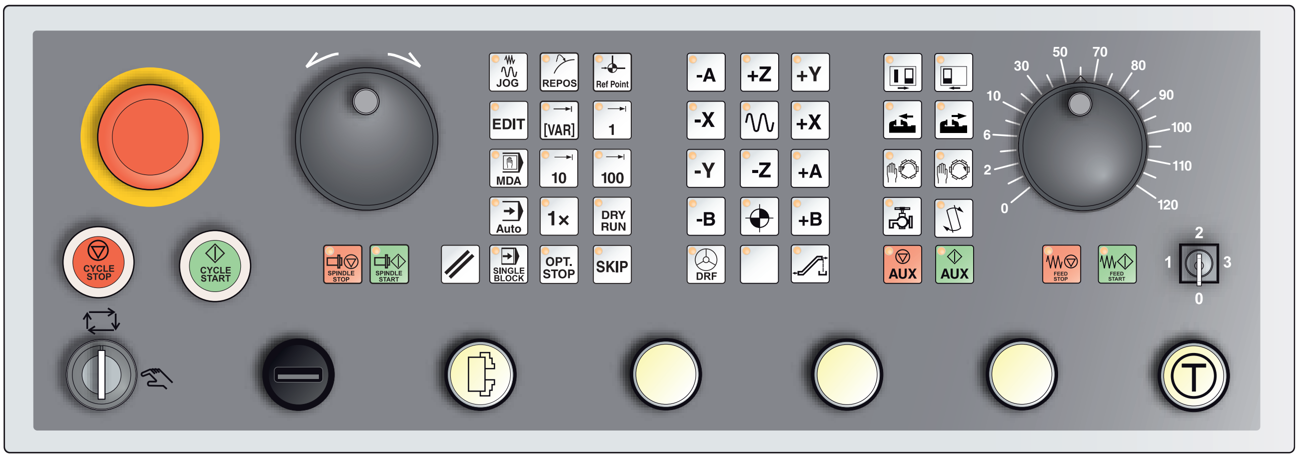
Note:
Selecting machine keys via PC keyboard:

1.) Press and hold Ctrl key.
2.) Press machine key and then release it.
3.) Release Ctrl key.

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

Instructions:
The buttons for the Concept Mill 250 machine are explained below. For other machines always take note of Chapter D EMCO-specific Programming and Operation in the operating manual.

Skip (block mask)

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

Dryrun (test-run feed)

In Dry-run mode, positioning movements will be carried out with the dry-run feed.
The dry-run feed works instead of the programmed movement commands.
On starting the NC program, the main spindle will not be activated and the slides will move with dry-run feed speed.
Only perform the test run without a workpiece to avoid the danger of collisions.
If the test run is engaged, the test "DRY" appears in the simulation window.

Individual piece mode

This button makes individual piece mode or constant operation in conjunction with automatic loading equipment available for selection.
Individual state is the default state when switched on.

Optional stop

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

Reset key (Reset)

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

Single block

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

When single block processing is active:
- SBL" (=SingleBlock) is shown on the screen.
- the current block of the part program is only processed when you press the NC Start button.
- processing stops after a block is executed.
- the following block is executed by pressing the NC Start key again.

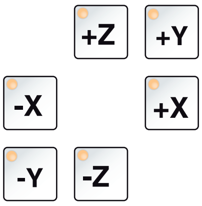
The function can be deselected by pressing the Single Block key again.

NC Stop

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

NC Start

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



Arrow keys

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

Reference point

Pressing this button causes the reference points to be approached in all axes.

Rapid Traverse

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

Feed Stop

In "AUTOMATIC" operation mode, this function cancels a slide movement.

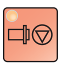
Feed Start

This function resumes a programmed slide movement which has been interrupted.
If the main spindle motion was also broken off, it must be switched on first.

Spindle speed correction

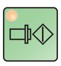
The set spindle speed value S will be shown on the screen as an absolute value and as a percentage.
Effective for the milling spindle.

Adjustment range:	50 - 120% of the programmed spindle speed
Increment:	5% per button press
100% spindle speed:	100% button



Spindle Stop

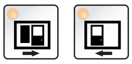
This button interrupts the motion of the milling spindle. If it happens during a feed movement, that has to be stopped first.



Spindle Start

This function resumes the programmed spindle motion.

Automatic machine doors



To open and close the machine doors.

Chip conveyor (Option)

Switch on chip conveyor:
Forwards: Press button for less than 1 second.
Backwards: Press button 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

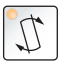
Pressing this button causes the tool drum to swivel by one position:

Cycle in the clockwise direction (one position further)

Cycle in the counter-clockwise direction (one position back)

Preconditions:
- Machine doors closed
- "JOG" operating mode
- Key switch in "Hand" position

Manual tool change

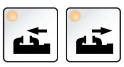


Pressing this button starts a manual tool change.
The tool clamped in the milling spindle will be removed and replaced with the tool from the currently swivelled-in tool drum.

Preconditions:
- Machine doors closed
- "JOG" operating mode
- Key switch in "Hand" position

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

Clamping device



These functions activate the clamping device.

Coolant

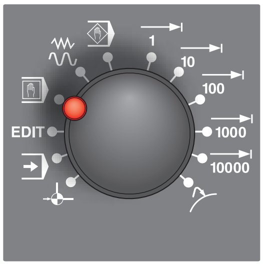
This function switches the coolant equipment on or off.

Auxiliary OFF

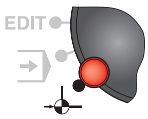
This function switches off the machine's auxiliary unit. Only effective if spindle and program are off.

Auxiliary ON

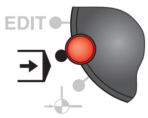
This function makes the machine's auxiliary unit ready for operation (e.g.: hydraulics, feed drives, spindle drives, lubrication, chip conveyors, coolant).
The button must be pressed for around 1 second.
Briefly pressing the AUX ON button is a quit function and causes the central lubrication system to perform a lubrication impulse.



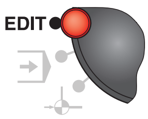
Types of operation



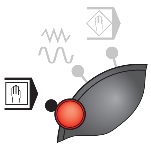
REF - Reference mode
Approaching the reference point (Ref) in the JOG operating mode.



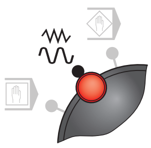
AUTO - Automatic mode
Control the machine by automatically executing programs.
Here part programs are selected, started, adjusted, deliberately influenced (e.g. individual block) and executed.



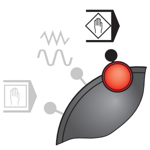
EDIT
no function



MDA - Semi-automatic mode
Control the machine by executing a set or a sequence of sets. Block input is performed via the operating panel.



JOG - Jogging
Standard movement of the machine by continuous movement of the axes via the directional buttons or by incremental movement of the axes via the directional buttons or the handwheel.
JOG is used in manual mode as well as for set-up of the machine.



TEACH IN
no function

1

Inc 1 - Incremental Feed
Move step by step a predefined distance of 1 increment.
Metrical measurement system: Inc 1 corresponds to 1µm
Imperial (inch-based) measurement system: Inc 1 corresponds to 0,1 µinch

10

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

100

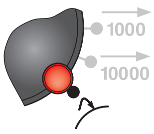
Inc 100 - Incremental Feed
Move step by step a predefined distance of 100 increments.
Metrical measurement system: Inc 100 corresponds to 100µm
Imperial (inch-based) measurement system: Inc 100 corresponds to 10 µinchh

1000

Inc 1000 - Incremental Feed
Move step by step a predefined distance of 1000 increments.
Metrical measurement system: Inc 1000 corresponds to 1000µm
Imperial (inch-based) measurement system: Inc 1000 corresponds to 100 µinch

10000

Inc 10000 - Incremental Feed
Move step by step a predefined distance of 10000 increments.
Metrical measurement system: Inc 10000 corresponds to 10000µm
Imperial (inch-based) measurement system: Inc 10000 corresponds to 1000 µinch



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

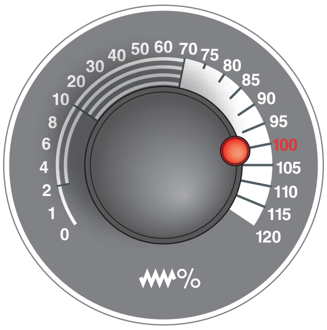
Instructions:

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

Note:
The allocation from metric to the imperial system is as follows:

feed:
millimeter to inch:
mm/min => inch/min
mm/U => inch/U

constant cutting speed:
meter to feet:
m/min => feet/min

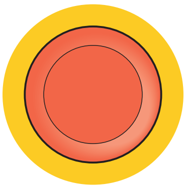


Override switch (feed rate override)

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



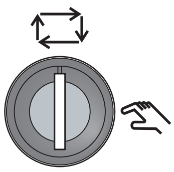
EMERGENCY STOP

Press the red button in emergency situations only.

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

Unlock: Twist button

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

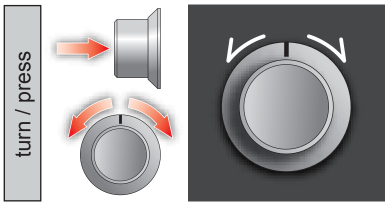


Key Switch Special Operations Mode

The key switch can be set to "AUTOMATIC" or "READY" (hand) mode. 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 authorised persons.
After work is carried out in Special Operations mode, always withdraw key (accident danger).
Observe country-specific safety instructions (e.g.: SUVA, BG, UVV ....).



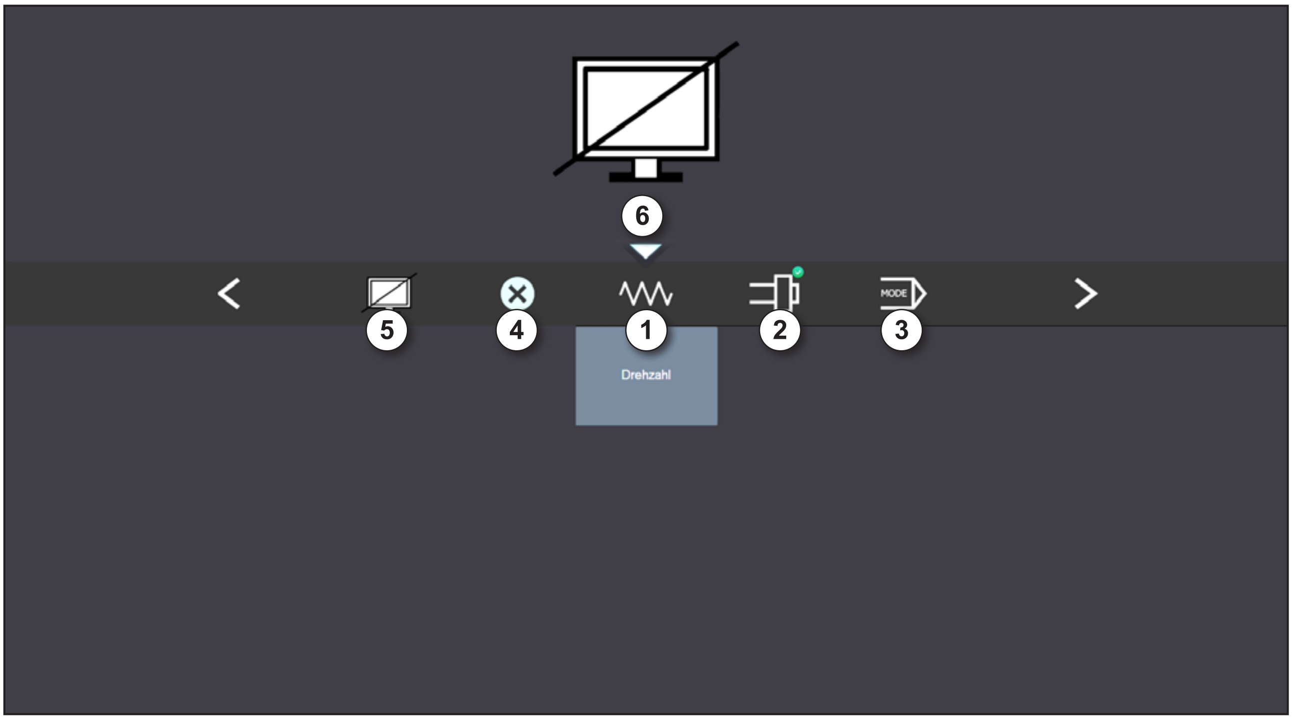
Multifunction switch for operating modes

The multi-function switch is designed as a rotary switch with a press feature.

Populated function

- The user interface is opened by pressing the multifunction operation. The active function is indicated by a green check-box.
- Turning the switch allows you to switch between the functions. The black bar with the symbols moves to the left or to the right.
- Activating a function or a change to a sub-menu is executed by pressing the button.

The interface offers the following functions:



Overwiew

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

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

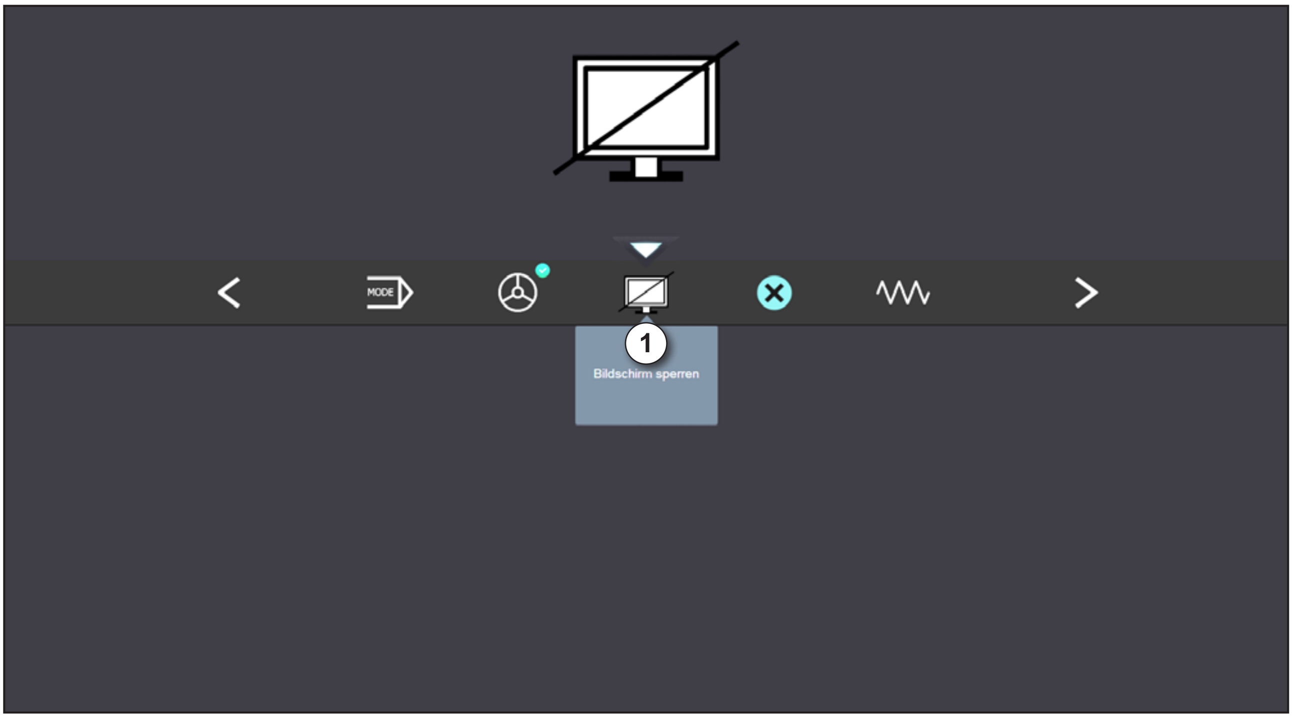
3 Modes: allows you to select the operating mode using the multifunction operation

4 Close: The user interface is closed. The menu disappears, return to the control surface

5 Settings: opens another level with settings

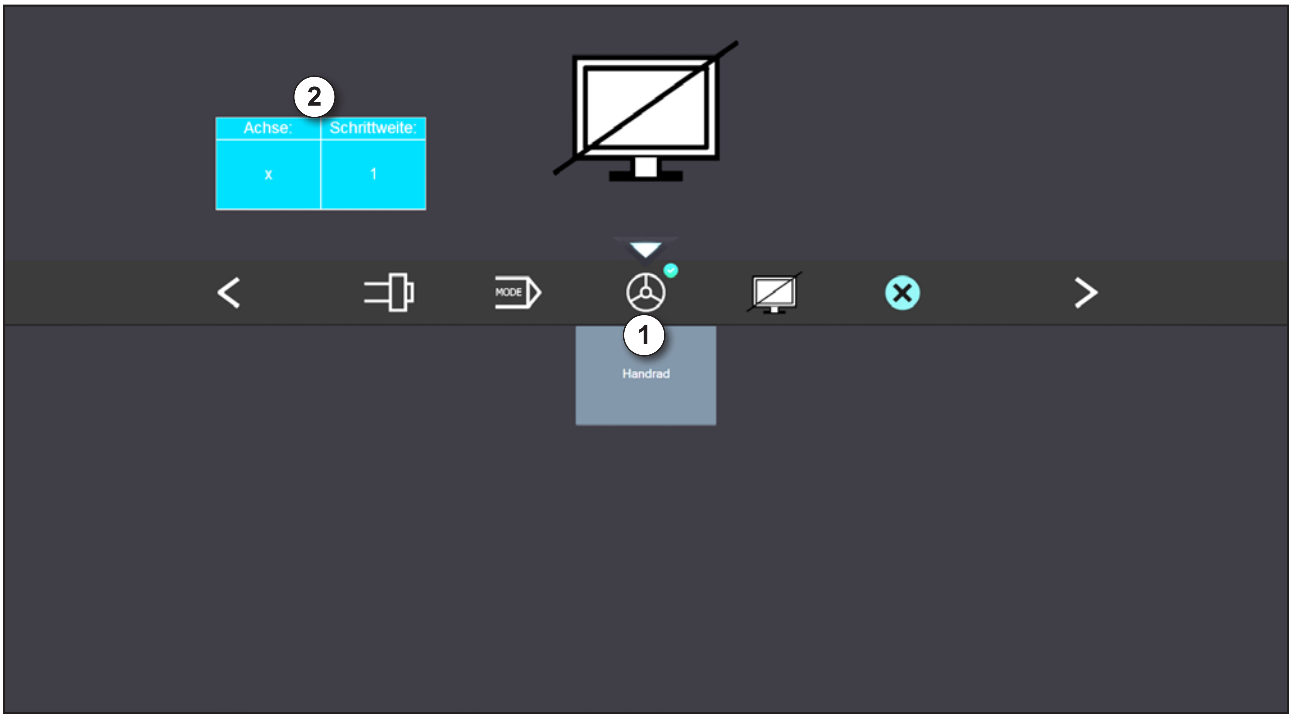
6 Cursor: shows the actual position in the menu

Note:
The functionality of the multifunction operation is depending on the installed software version.



Lock screen

1 Lock screen: pressing again unlocks the screen and closes the unser interface.



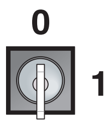
Handwheel function

The handwheel (1) activates the handwheel mode. The parameters for axis and step width (2) are set with the axis- and operating -mode buttons on the machine keyboard.

Operation

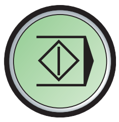
- The electronic handwheel is used to traverse the slides at a defined step width.
- The step width depends on the selected Inc mode: Inc 1, Inc 10, Inc 100.
- There must be one pre-selected Inc mode and an axis defined by a direction key.
- Also refer to "types of operation" und "arrow keys" in chapter B.

Note:
In the mode "Inc 1000" the slides cannot be moved with the hand-wheel. "Inc 1000" operates with "Inc 100".



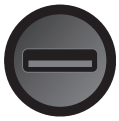
Key switch

The function of the key switch is machinespecific



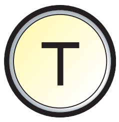
Additional NC start button

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



USB connection (USB 2.0)

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



Enable button

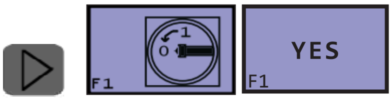
When the door is open, axis movements via direction buttons and tool changer movements are authorized by pressing the enable button (precondition: key switch in SET-UP position).
In machines with automated doors (option) pressing the enable switch opens the machine doors.

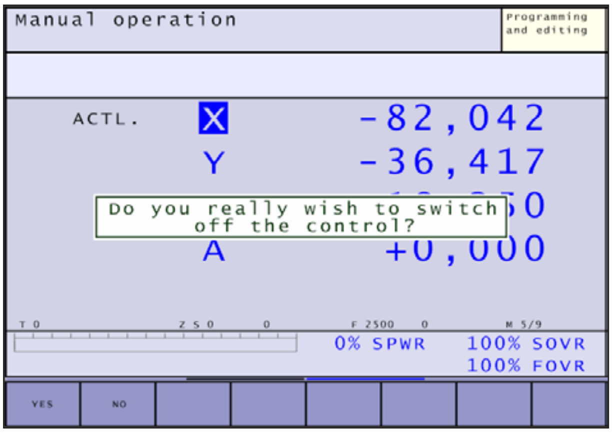
C: Operation

Switch-off

To prevent data from being lost at switch-off, you need to run down the operating system of the WinNC as follows:

- Select the manual operation mode .
- Press button AUX OFF.
- Select the funktion for run-down, confirm again with the YES soft key.





Now you may cut off the power supply to the WinNC.

Inappropriately switching the WinNC off can lead to data loss.

> **Note:**
> The operating mode "Electronic handwheel" behaves as "Manual operation" in EMCO simulation. In order to proceed with the handwheel, switch to one of the INC modes (1 - 100) on the machine control panel, and then select the appropriate axis (see machine description manual). .

Modes of operation

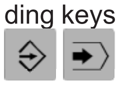
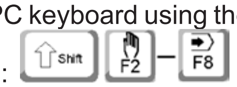
The modes of operation of the WinNC Heidenhain TNC 426 are divided into five machining modes and two programming modes:

Machining modes:
- Manual operation
- Electronic handwheel
- Positioning with manual data input
- Program run, single block
- Program run, full sequence

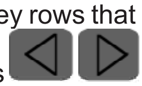
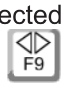
Programming modes:
- Programming and editing
- Test run

The header shows the machining modes on the left and the programming modes on the right side. The current operating mode is displayed in the larger box of the header, where dialogue prompts and messages also appear.

Calling operating modes

Operating modes are called either via the corresponding keys on the keyboard   or via the PC keyboard using the following combination of keys:  or via the mode select switch.

Navigation in the menu window

In the footer the WinNC indicates additional functions in a soft-key row. The lines directly above the soft-key row indicate the number of soft-key rows that can be selected with the black arrow keys  or with the  key. The active soft-key row is highlighted.

Manual operation and electronic handwheel

The manual operation mode is required for setting up the machine tool. In this operating mode you can position the machine axes manually or by increments and set the reference points.
The operation mode electronic handwheel is currently not available.

Positioning with manual data input

This mode of operation enables the programming of simple traversing movements, e.g. face milling or pre-positioning.

Program run, full sequence and program run, single block

In the program run, full sequence, the WinNC executes a program to its end or to a manual or programmed stop. You can resume the program run after an interruption.
In the program run, single block, you start each block separately by pressing the external START button.

Programming and Editing

In this operating mode you can write your part programs. The Free-Contour- Programming feature , the various cycles and the Q parameter functions support you in the programming process and provide additional information. If desired, the programming graphics show the individual steps, or you can use another screen window to draw up your program structure.

Test run

In the Test run operating mode the WinNC simulates programs and program sections, so as to check them for geometrical incompatibilities, missing or incorrect data within the program or violations of the working space. This simulation is supported graphically by different display modes.

Machine operation

The machine operation includes all functions and influencing variables that lead to machine actions or that control the condition of the machine.

Four operating modes are distinguished:

- Manual operation
  Rapid traverse is used for manual operation and for adjusting the machine.
  The following functions are available for adjusting the machine:

  Traverse the reference point (Ref)

  Move in incremental steps 1 ... 10000

- Positioning with manual data input (MDI)
  Semi-automatic operation, positioning with manual data input
  Here part programs can be created and worked off blockwise.

- Program run, single block
  Here part programs are selected, started, corrected, directly inflcuenced and worked off.

- Program run, full sequence
  Part programs are worked off in a fully automatic way.

You select the modes of operation via the softkeys (PC keyboard or Heidenhain TNC426-keyboard) or by using the operating mode selection key.

Traversing the reference point

By traversing the reference point the control is synchronized with the machine.

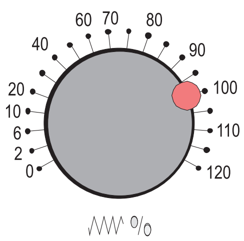
- The operation mode is selected automatically.
- Use the directions keys -X or +X in order to traverse the reference point in the corresponding axis, analogue for all other axes.
- By using the REF ALL or key the reference point is automatically traversed first in the Z- and then in the X- and Y-axis.

When the reference point is reached, its position is shown as actual position on the screen. Now the control is synchronized with the machine.

Traversing the slide manually

You can traverse the machine axes manually by using the direction keys.

- Change to the manual operation mode .
- By using the +X, -X, +Y, -Y, +Z, -Z, +4, -4 ....keys the axes are moved in the chosen direction as long as the key is pressed and held.
- By using the +X, -X, +Y, -Y, +Z, -Z, +4, -4 ... keys and simultaneously pressing the key the axes are traversed continuously until key is pressed (not available for CM 300).



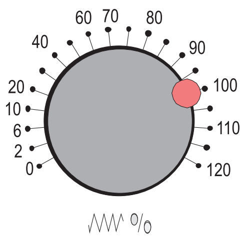
- The feed rate is set with the override switch.
- If the the key is pressed simultaneously, the slides traverse in rapid traverse (only for CM 250, CM 300 and CM 450 available).

Traversing the slide incrementally

With incremental jog positioning the WinNC traverses the machine axis by a distance you preset.
You can move the machine axes incrementally by using the axis direction buttons.

INC 1	1/1000 mm	per depression of key
INC 10	1/100 mm	per depression of key
INC 100	1/10 mm	per depression of key
INC 1000	1 mm	per depression of key
INC VAR	variable distance	

- Set the switch for the selection of the operating mode to INC (1 ... 10000 or Alt+0 ... Alt+4 on PC or with the softkey INCRE-MENT OFF/ON F6 for individually incremental traversing).



- With every depression of the keys +X, -X, +Y, -Y, +Z, -Z, +4, -4, and so on, the axes are moved in the corresponding direction by the preset distance.
- The feed rate is set by the override switch.
- If the the key is pressed simultaneously, the slides traverse in rapid traverse (only for PC MILL 300, CM 250 and CM 450 available).

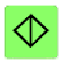
Positioning with Manual Data Input (MDI)

Programming and Executing Simple Machining Operations

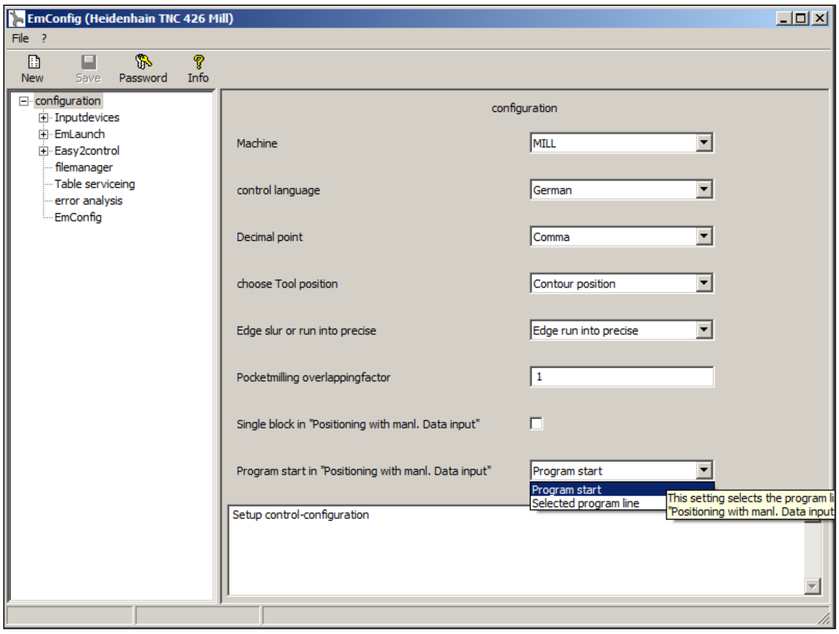
The operating mode Positioning with Manual Data Input is particularly convenient for simple machining operations or pre-positioning of the tool. It enables you to write a short program in conversational programming and execute it immediately. You can also call WinNC cycles. The programm is stored in the file $MDI. In the operating mode Positioning with MDI, the additional status displays can also be activated. see chapter B - screen layout

Positioning with Manual Data Input (MDI)

Select the Positioning with MDI mode of operation. Program the file $MDI as you wish.



To start program run, press the machine START button.



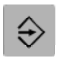
Program start with "positioning with manual data input"

With EMConfig it can be selected if the program starts at program start or with the actual selected program line. If the program starts at the actual selected program line, all other previously programmed records will be ignored.

Note:
Positioning with Manual Data Input is only in the Conversational programming available. Positionieren mit Handeingabe ist nur im Klartext-Dialog möglich. FK free contour programming, programming graphics and program run graphics cannot be used. The $MDI file must not contain a program call (**PGM CALL**).

Protecting programs in $MDI

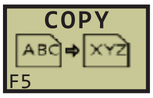
The $MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:



Select the Programming and Editing mode of operation

To call the file manager, press the PGM MGT key (programm-management).

Move the highlight to the $MDI file.

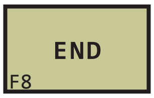


To select the file copying function, press the COPY softkey.

Target file = BOREHOLE

Enter the name under which you want to save the current contents of the $MDI file.

Copy the file.



To close the file manager, press the END softkey.

further informations: see "Copying a single file" chapter C - "Advanced file management".

Program run, single block / full sequence

In the operating mode Program run, single block / full sequence part programs can be run in a fully automatic way.

Preconditions for working off part programs:

- The reference point was traversed.
- The part program was called into the control.
- The necessary correction values were checked, respectively entered (e.g datum shift, tool corrections).
- The safety lockings are activated (e.g. chip protection door closed).

Possibilities in the program run, single block / full sequence operation mode:

- Block search run
- Influencing the program

(see chapter F - program run)

Fundamentals of file management

Note:
You may select between the standard and advanced file management by using the MOD function PGM MGT. If the WinNC is connected to a network you use the file management with additional functions (= advanced) - (see Selection of MOD functions).

Files

Files in the TNC	Type
Programs	
in HEIDENHAIN format	.H
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D
Points (digitizing range of measuring touch probe)	.PNT
Texts as	
ASCII-files	.A

The WinNC has a special file management window so that you can easiliy find and manage your files. Here you can call, copy, rename and delete files.

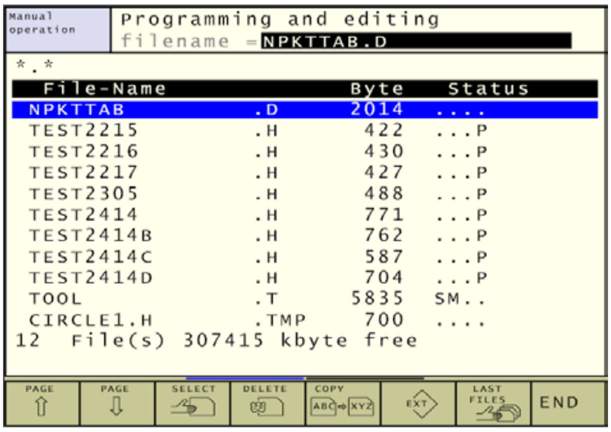
You can manage any number of files with the WinNC, however their total size must not exceed your hard disk capacity.

File names
When you store programs, tables and texts as files you have to add an extension to the file name, separated by a point. This extension indicates the file type.

PROG20	.H
File name	File type

Standard file management

Note:
The standard file management is perfectly suited when you wish to save all files in one directory, or if you are acquainted with the file management of old TNC controls. To use it please set the MOD function **PGM MGT** to **Standard** (see Selection of MOD functions).



Calling the file manager

Press the PGM MGT key: The WinNC displays the file management window.

The window shows all files that are stored in the WinNC. Various information is provided for every file.

Display	Meaning
File name	Name with up to 16 characters and file type
Byte	File size in bytes
Status	Properties of the file:
E	Program is selected in the Programming and Editing mode of operation
S	Program is selected in the Test Run mode of operation
M	Program is selected in the Program Run operating mode
P	File is protected against editing and deletion
Date	Date the file was last changed
Time	Time the file was last changed

Tag a single file

Tag all files

Untag a single file

Copy all tagged files

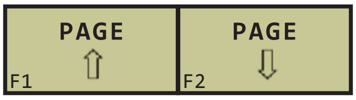
Untag all files

Selecting a file

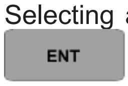
Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to select:

Moves the highlight up and down in the window **file by file**.



Moves the highlight up and down in the window **page by page**.

Selecting a file: Press the SELECT softkey or the 

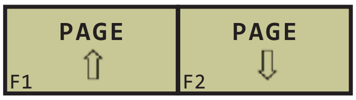
key.

Deleting a file

Call the file manager.

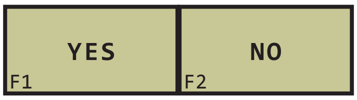
Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to delete:

Moves the highlight up and down in the window **file by file**.



Moves the highlight up and down in the window **page by page**.

Deleting a file: Press the DELETE soft key and confirm with the YES soft key
or



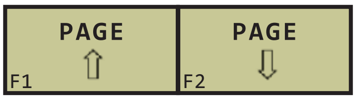
abort with the NO soft key.

Copying a file

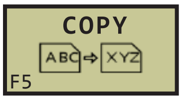
Call the file manager.

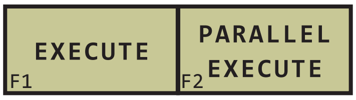
Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to copy:

Moves the highlight up and down in the window **file by file**.



Moves the highlight up and down in the window **page by page**.





Copying a file: Press the COPY soft key
Enter the new file name and confirm by pressing the EXECUTION soft key or the ENT key. The WinNC displays a status window that keeps you informed about the copying progress. As long as the WinNC is copying you cannot work. If you wish to copy very long programs, enter a new file name and confirm with the PARALLEL EXECUTE soft key. The WinNC will then copy the file in the background, so you can continue to work after the copying process has started.

Selecting one of the last 10 files selected

Call the file manager.

Selecting the last 10 files selected: Press the LAST FILES soft key.

Use the arrow keys to move the highlight to the file you wish to select:
Moves the highlight up and down in the window.

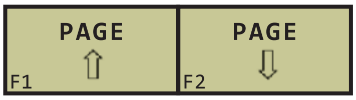
Selecting a file: Press the SELECT soft key or the ENT key.

Renaming a file

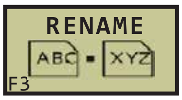
Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to rename:

Moves the highlight up and down in the window **file by file**.



Moves the highlight up and down in the window **page by page**.



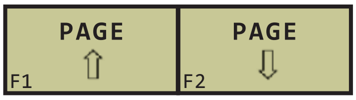
Renaming a file: Press the RENAME soft key, enter the name of the new file and confirm with the EXECUTE or ENT key.

Converting an FK program into conversational format

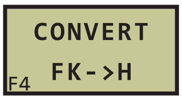
Call the file manager.

Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to convert:

Moves the highlight up and down in the window **file by file**.



Moves the highlight up and down in the window **page by page**.

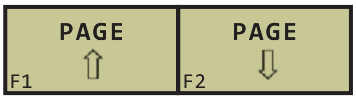


Converting the file: Press the CONVERT FK-> H. Enter the name of the new file and confirm with the EXECUTE soft key or the ENT key.

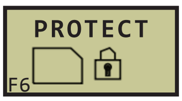
Protecing a file / Cancelling a file protection

Call the file manager.
Use the arrow keys or the arrow soft keys to move the highlight to the file you wish to protect or whose protection you wish to cancel:

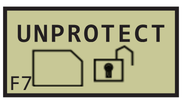
Moves the highlight up and down in the window **file by file**.



Moves the highlight up and down in the window **page by page**.



Protecting a file: Press the PROTECT soft key. The file now has the status P, or



Cancelling a file protection: Press the UNPROTECT soft key. The P status is canceled.

Advanced file management

Note:
Use the advanced file manager if you wish to save your files in different directories.
To use it, set the MOD function **PGM MGT** to **Enhanced** (see selection of MOD functions).

Directories

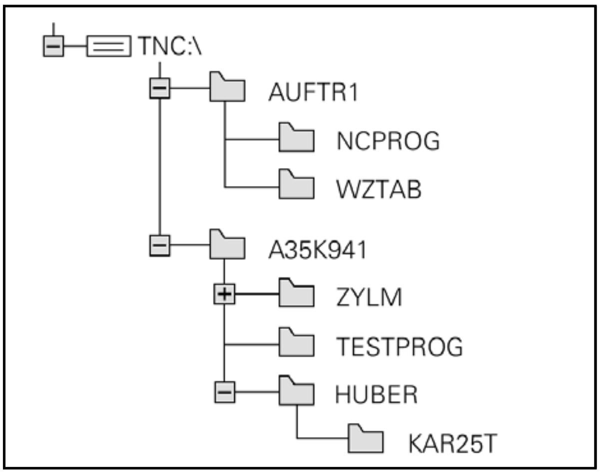
To ensure that you can easily find your files, we recommend that you organize your hard disk into directories (files). You can divide these directories up into further directories, so-called subdirectories.

Note:
The WinNC can manage up to 6 directory levels!
If you save more than 512 files in one directory the WinNC no longer sorts them alphabetically!

Directory names

The name of a directory can contain up to 8 characters and does not have an extension. If you enter more than 8 characters for the directory name the WinNC will display an error message.

Paths



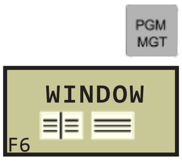
A path indicates the drive and all directories, resp. subdirectories, under which a file is saved. The individual details are separated by the symbol „\“.

Example:
On drive **TNC:** the directory AUFTR1 was created. Then, in the directory **AUFTR1** the subdirectory NCPROG was created and the part program PROG1.H was copied into it. The part program now has the following path:
TNC:\AUFTR1\NCPROG\PROG1.H

The graphic chart on the left shows an example of a directory display with different paths.

Overview: Functions of the expanded file management

Function	Soft key
Copy (and convert) individual files	COPY ABC XYZ F5
Display a specific file type	TYP F5 SELECT
Display the last 10 files selected	LAST FILES F7
Delete a file or directory	DELETE F3
Tag a file	TAG F4
Rename a file	RENAME ABC XYZ F3
Convert an FK program into conversational format	CONVERT F4 FK->H
Protect a file against editing and deletion	PROTECT F6
Cancel file protection	UNPROTECT F7
Network drive management	NET F6



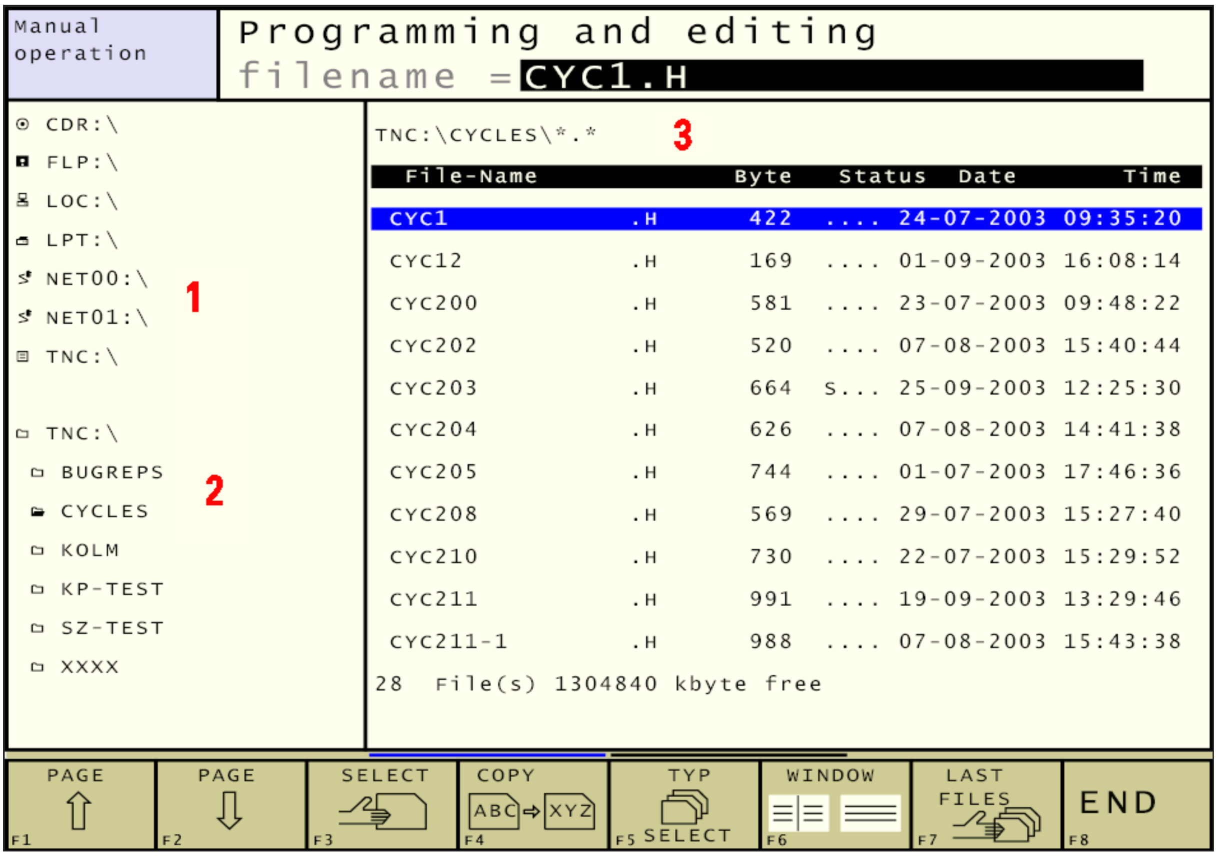
Display	Meaning
File name	Name with up to 16 characters and file type
Byte	File size in bytes
Status	Properties of the file:
E	Program is selected in the Programming and Editing mode of operation
S	Program is selected in the Test Run mode of operation
M	Program is selected in the Program Run operating mode
P	File is protected against editing and deletion
Date	Date the file was last changed
Time	Time the file was last changed

Calling the file manager

Press the PGM MGT key: the WinNC displays the file management window (The picture on the left shows the fundamental setting. If the WinNC displays a different screen layout, press the WINDOW soft key).

The narrow window on the left shows here seven drives 1. Drives are devices by means of which data are stored or transferred. One drive is here the hard disk of the WinNC, other drives are the CD-Rom drive (CDR:\), the floppy drive (FLP:\), one local drive (LOC:\), two network drives (NET00:\ and NET01:\) and a printer (LPT:\). The selected (active) drive is shown in a different colour.
Shown drives can be select in **WinConfig** (see Starting-Information capture X "Change Ini Data of WinNC"). You can activate the required drives in the menu item for activating drives of the Heidenhain TNC426 file manager.
You can choose:

- floppy disk drive (FLP:\)
- CD-Rom drive (CDR:\)
- local drives (LOC:\)
- network drives (NET:\)
- printer (LPT:\) (see "printing a file")



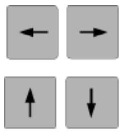
In the lower part of the narrow window the WinNC shows all directories **2** of the selected drive. A directory is always identified by a file symbol (left) and the directory name (right). Subdirectories are indented to the right.
The selected (active) directory is shown in a different colour.

The wide window on the right shows all files **3** that are stored in the selected directory. Each file is shown with additional information which is illustrated in detail by the table on the left.

Selecting drives, directories and files

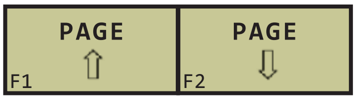
Call the file manager.

Use the arrow keys or the soft keys to move the highlight to the desired position on the screen:



Moves the highlight from the right to the left window and vice versa.

Moves the highlight up and down within a window.

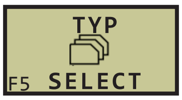


Moves the highlight one page up and down within a window.

1. Step: Select a drive
Mark the drive in the left window:
To select a drive, press the SELECT soft key or the ENT key.

2. Step: Select a directory
Mark the directory in the left window: The right window automatically shows all files stored in the marked (highlighted) directory.

3.Step : Select a file



Press the SELECT TYPE soft key.

Press the soft key for the desired file type, or

Press the SHOW ALL soft key to display all files.

Mark the file in the right window:
The selected file is activated in the operating mode from which you have called the file manager:

Press the SELECT soft key or the ENT key.

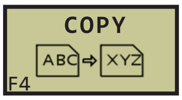
Creating a new directory (only possible on TNC:\ drive)

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

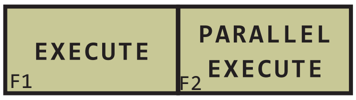
DEMO

Enter the new directory name and confirm with ENT .

Copying a single file



- Move the highlight to the file you wish to copy.
- Press the COPY soft key to select the copying function.
- Enter the name of the target file and confirm your entry with the ENT key or the EXECUTE soft key: The WinNC copies the file into the active directory. The original file is retained, or



- Press the PARALLEL EXECUTE soft key to copy the file in the background. You should use this function when you copy large files, since it enables you to continue working after the copying process has been started.

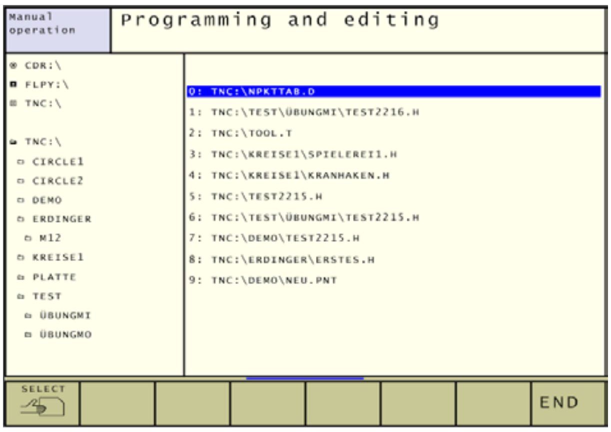
Choosing one of the last 10 files selected

Call the file manager.

Display the last 10 files selected: Press the LAST FILES soft key.

Use the arrow keys to move the highlight to the file you wish to select:
Moves the highlight up and down within the window.

Select a file: Press the SELECT soft key or the ENT key.



Choosing one of the last 10 files selected

Deleting a file

- Move the highlight to the file you want to delete.
- Press the DELETE soft key to select the erasing function. The WinNC inquires whether the file should really be deleted.
- To confirm the deletion, press the YES soft key or
- To abort the deletion, press the NO soft key.

Deleting a directory

- Delete all files and subdirectories from the directory you wish to erase.
- Move the highlight to the directory you want to delete.
- Press the DELET soft key to select the erasing function. The WinNC inquires whether the directory should really be deleted.
- To confirm the deletion, press the YES soft key or
- To abort the deletion, press the NO soft key.

Renaming a file

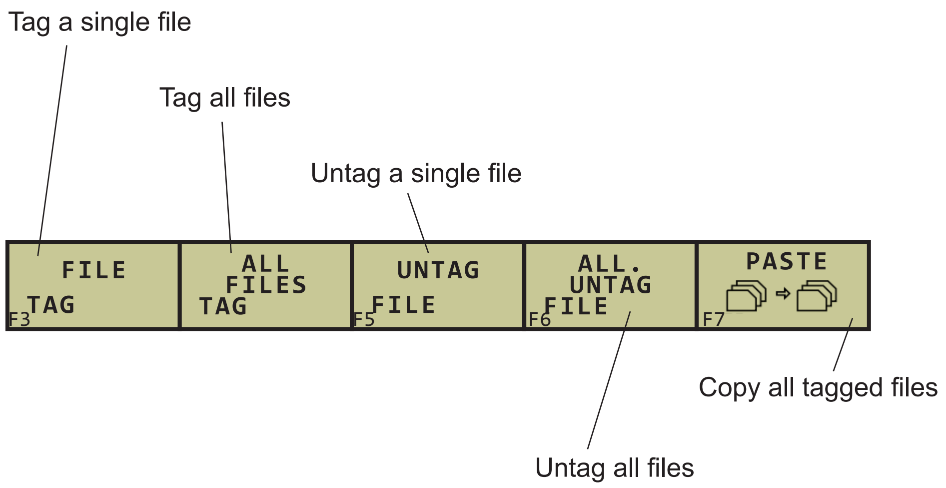
- Move the highlight to the file you wish to rename.
- Press the RENAME soft key to select the renaming function.
- Enter the new file name: the file type cannot be changed.
- To exectue the renaming process, press the  key.

Printing files

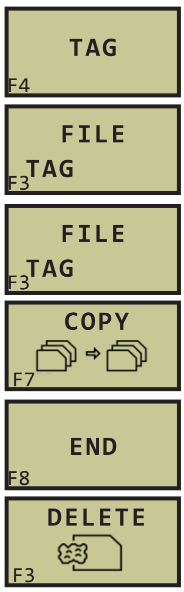
- Move the highlight to the file you wish to print.
- copy the file to the printer. target-file: LPT:\
- press Soft key Execute

Note:
Printing is only in the operation mode Programming and Editing available.

Tagging files



You can use functions like copying or deleting of files not only for single files but also for several files simultaneously. To tag several files, proceed as follows:



Move the highlight to the first file.

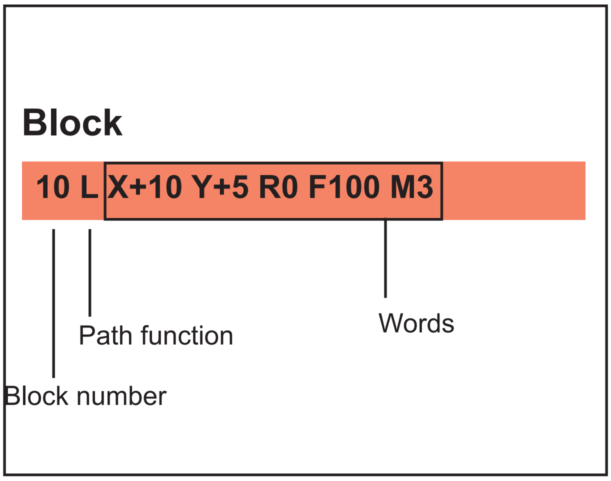
To show the tagging functions, press the TAG soft key.

Tag a file by pressing the TAG FILE soft key.

Move the highlight to the next file you wish to tag: You can tag further files, if you wish to: Press TAG FILE soft key, and so on.
To copy the tagged files, press the COPY TAG soft key, or

Delete the tagged files by pressing the END soft key in order to leave the marking function and

then press the DELETE soft key to delete the tagged files.



Creating and writing programs

Organization of an NC program in HEIDENHEIN conversational format

A part program consists of a series of program blocks. The left illustration shows the elements of a block. The WinNC numbers the blocks of a part program in ascending sequence.

The first block of a program is identified by **BEGIN PGM**, the program name and the active unit of dimension.

The subsequent blocks contain information on:

- the blank workpiece
- tool definitions and tool calls
- feed rates and spindle speeds
- path contours, cyles and other functions

The last block of a program is identified by **END PGM**, the program name and the active unit of dimension.

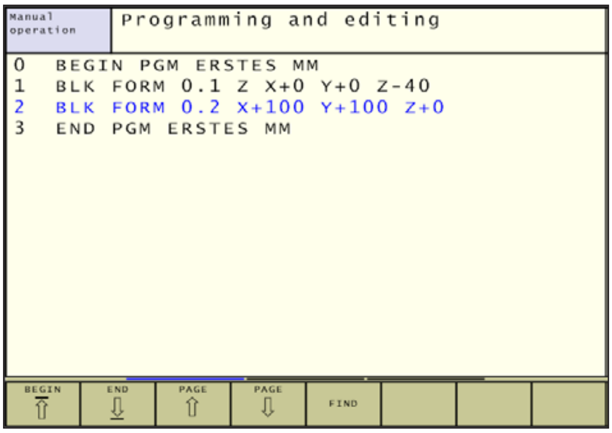
Defining the blank workpiece: BLK FORM

Immediately after initiating a new program you define a cuboid, blank workpiece. If you wish to define the blank workpiece at a later stage, press the BLK FORM soft key. The WinNC needs this definition for the graphic simulations. The sides of the blank workpiece are parallel to the X, Y and Z axes and must not exceed a length of 100 000 mm.
The blank form is defined by two of its corner points:

- MIN-point: smallest X, Y and Z coordinate of the blank form, entered as absolute values.
- MAX-point: largest X, Y and Z coordinate of the blank form, entered as absolute or incremental values.

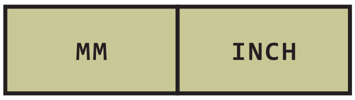
Note:
The definition of the blank form is necessary if you want to run a graphic test for the program!

Creating a new part program



You always enter a part program in the **Programming and Editing** mode of operation.

Example for a program creation:
Select the **Programming and Editing** mode of operation.
To call the file manager, press the PGM MGT key.
Select the directory in which you want to store the new program:
Enter the new program name and confirm your entry with the ENT key.



To select the unit of dimension, press the MM or INCH soft key. The WinNC switches to the program window and opens the dialogue for the definition of the **BLK-FORM** (blank workpiece).

Spindle axis parallel to X/Y/Z?
Enter the spindle axis.

0 ENT
0 ENT
-40 ENT

Def BLK-FORM: Min-point?
Enter the X, Y and Z coordinates of the MIN point in sequence.

100 ENT
100 ENT
0 ENT

Def BLK-FORM: Max-point?
Enter the X, Y and Z coordinates of the MAX point in sequence.

Display of the BLK Form in the NC program	
0 BEGIN PGM NEU MM	Program begin, name, unit of dimension
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Spindle axis, MIN point coordinates
2 BLK FORM 0.2 X+100 Y+100 Z+0	MAX point coordinates
3 END PGM NEU MM	Program end, name, unit of dimension

The WinNC automatically generates the block numbers as well as the **BEGIN** and **END** blocks.

Programming tool movements in conversational format

To program a block, start by pressing the dialogue key. In the screen headline, the WinNC then asks you for all the information necessary to program.

Example of a dialogue:

L

Initiate the dialogue.

Coordinates

X 10

Enter the target coordinate for the X axis.

Y 20

Enter the target coordinate for the Y axis and go to the next question by pressing ENT.

Radius compensation: RL/ RR/ no compensation?
Enter "No radius compensation" and got to the next question by pressing ENT.

Feed rate F=? / F MAX = ENT

100

Enter a feed rate of 100 mm/min for this path contour; go to the next question by pressing ENT.

Miscellaneous function M?

3

Enter the miscellaneous function **M3** "spindle ON clockwise"; pressing the ENT key will terminate this dialogue.

The program window displays the following line:
3 L X+10 Y+5 R0 F100 M3

Functions for setting the feed rate

F MAX

Rapid traverse

Function	Key
Ignore the dialogue question	NO ENT
End the dialogue immediately	END
Abort the dialogue and delete	DEL

emco

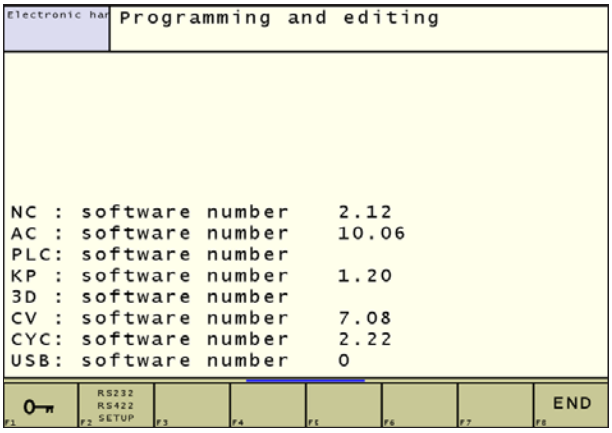
Editing a program

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys:

Function	Soft keys/ keys
Go to previous page	PAGE ⇧ F3
Go to next page	PAGE ⇩ F4
Go to beginning of program	BEGIN ⇧ F1
Go to end of program	END ⇩ F2
Move from one block to the next	↑ ↓
Select individual words in a block	← →
Set the selected word to zero	CE
Delete an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	NO ENT
Delete the selected block	DEL □
Erase cycles and program sections: Select the last block of the cylcle or program section to be deleted, then erase with the DEL key	DEL □

MOD-functions

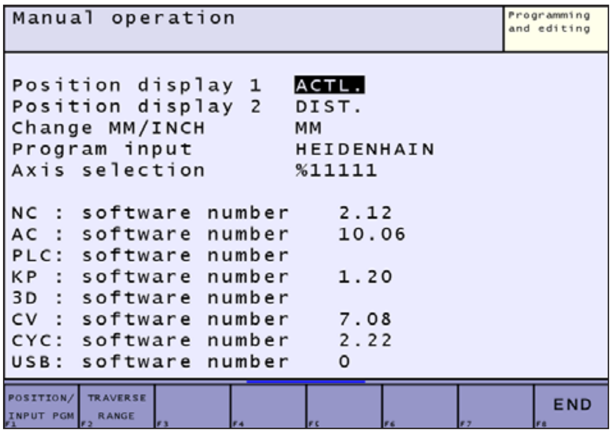
The MOD functions provide additional displays and input possibilities. The available MOD functions depend on the selected operating mode.



Selecting MOD functions

Select the operating mode in which you wish to change the MOD function.

Press the MOD key. The illustrations on the left show the typical screen menus in test run (see picture on the upper left) and in a machine operating mode (see picture on the lower left).



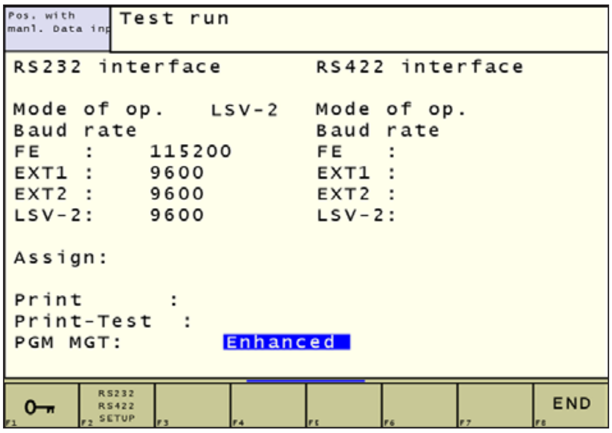
Changing the settings

There are three possibilites to change a setting, depending on the function selected:

- Enter a numerical value directly, e.g. when determining the traverse range limit.
- Change a setting by pressing the ENT key, e.g. when setting the program input.
- Change a setting via a selection window. If there are more setting possibilities available, you can superimpose a window, that lists all given possibilities, by pressing the GOTO key. Directly select the desired setting by pressing the arrow key and then confirming with the ENT key. If you don`t want to change the setting, close the window with the END key.

Exiting the MOD-Funktionen

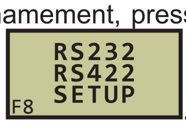
Press the END F8 soft key or the END key.



Overview of MOD functions

Depending on the selected mode of operation, you can make the following changes:

- Display different software numbers
- Select position display
- Set unit of measurement (MM/inches)
- Set the axis traverse limits
- Display the datums

Example: In order to select the standard or advanced file manamement, press the Programming/ Editing soft key 

.

Select the desired file managment by pressing the 

key in the **PGM MGT** line.

The standard or simplified file management does not have a directory display.
The advanced file management has enhanced functions and a directory display.

D: Programming

Notice:

This programming manual describes all functions that can be carried through by means of the Heidenhain TNC 426.
However, it depends on the machine, which you use the WinNC for, which functions are available.
Example:
The milling machine Concept Mill 55 does not have a position-controlled main spindle, which means that no spindle positions can be programmed.

Overview M- commands

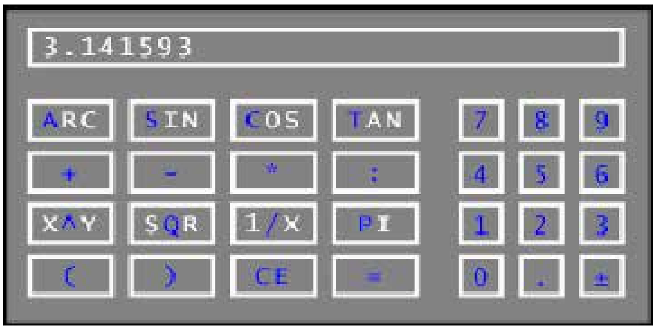
COMMAND	EFFECT
M0	Programmed stop
M1	Optional stop (program stop only with opt. stop)
M2	Program end
M3	Spindle ON clockwise
M4	Spindle ON counterclockwise
M5	Spindle OFF
M6	Tool change
M7	Minimal lubrication ON
M8	Coolant ON
M9	Coolant OFF
M10	Dividing head, clamping ON
M11	Dividing head, clamping OFF
M17	End of subprogram
M25	OPEN clamp/ machine vice
M26	CLOSE clamp/ machine vice
M27	Swivel dividing head
M30	Main program end
M70	Position controlled spindel
M71	Puff blowing device ON
M72	Puff blowing device OFF
M91	In the positioning block: Coordinates are referenced to the machine zero point
M99	Cycle call

Overview Cycles

Cycle	Softkey
DRILLING/THREAD Cycles for Drilling, Tapping and Thread Milling	DRILLING/ THREAD F1
POCKETS/STUDS/SLOTS Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS F2
PATTERN Cycles for Machining Hole Patterns	PATTERN F5
SL-CYCLES Cycles for complex contours	SL- CYCLES F4
MULTIPAGES Cycles for multipass milling	MULTIPAGES F6
COORDINATE TRANSFORMATION Cycles for Coordinate Transformation	COORD. TRANSF. F3
SPECIAL CYCLES Dwell Time, Program Call, Oriented Spindle Stop	SPECIAL- CYCLES F7

Calculating operators

COMMAND	DESIGNATION	MEANING
+, -, *, :		Fundamental operations
S	SIN	Sine function
C	COS	Cosine function
T	TAN	Tangent function
AS	ARCSIN	Arc sine function
AC	ARCCOS	Arc cosine function
AT	ARCTAN	Arc tangent function
^		Powers
Q	SQR	Square root
/	1/x	Inversion
( )		Parenthetic calculations
P	PI	Circular graduation number pi (3.14159265359)
=		Display result
ENTER		Display result

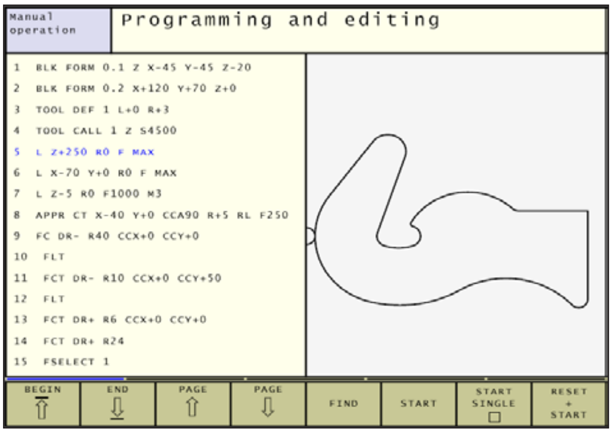


Pocket Calculator

Operation

The WinNC features an integrated pocket calculator with the most important mathematical functions. You can open and close the window for the pocket calculator with the CALC key.
You can move the window to any desired location on the screen by using the arrow keys. You select the calculating function via short commands on the keyboard. The short commands are shown in a special colour in the calculator window.
When you are writing a program and the programming dialogue is active, you can use the actual-position-capture key to copy the result of the calculator window directly into the highlighted position in the current block.

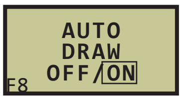
Programming graphics



Generating/ not generating programming graphics during programming

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

- Press the SPLIT SCREEN key and the PGM+GRAPHICS soft key to switch the screen layout to show the program on the left and the graphics on the right.
- Set the AUTO DRAW soft key to ON. While you are entering the program lines, the WinNC shows every programmed path contour in the graphics window on the right screen half.



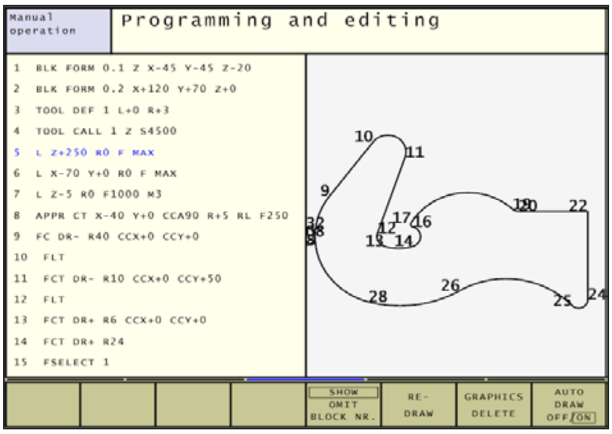
If you do not wish to have graphics generated during programming, set the AUTO DRAW to OFF.

Note:
If AUTO DRAW is ON, graphics are not generated for program section repeats or subprograms.
Coordinates that are relative to the machine zero point with M91, are shown incorrectly in the graphical view.

Generating a graphic for an existing program

- To generate graphics, press the RESET + START soft key.

Additional functions:



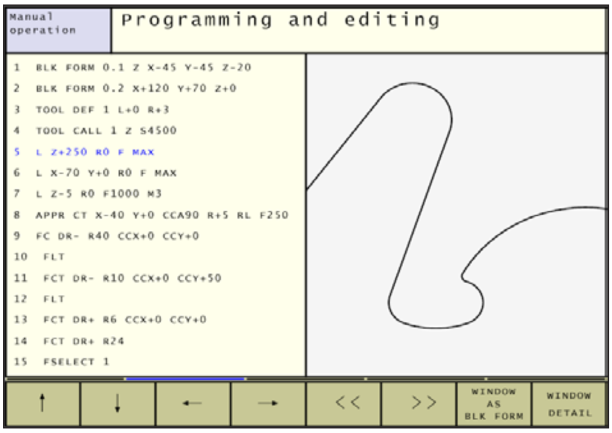
Block number display ON/ OFF

- Shift the soft-key row: see illustration on upper left
- To show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW.
- To omit block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT.

Deleting the graphic

- Shift the soft-key row: see illustration on upper left.
- To delete the graphic, press the CLEAR GRAPHIC soft key

Magnifying or reducing a detail



You can determine the display for a graphic by yourself. You select a detail for mangification or reduction by using a frame.

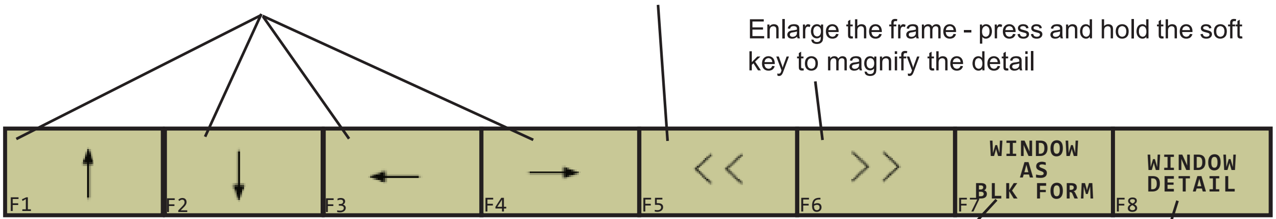
- Select the soft-key-row for detail magnification/ reduction (second row, see illustration at the left center).

The following functions are available:

Show and move the frame. Press and hold the desired soft key to move the frame.

Reduce the frame - press and hold the soft key to reduce the detail

Enlarge the frame - press and hold the soft key to magnify the detail



Confirm the selected area with the WINDOW DETAIL soft key.

With the WINDOW BLK FORM soft key you can restore the original section.

Tool movements

Path functions

A workpiece contour is composed of several contour elements such as straight lines and circular arcs. With the path functions you can program the tool movements for straight lines and circular arcs.

FK Free Contour programming

If a production drawing is not dimensioned for NC and the dimensions given are not sufficient for the NC program, you can program the workpiece contour with the FK free contour programming. The WinNC will calculate the missing data.

Miscellaneous functions M

With the miscellaneous functions of the WinNC you can control

- the program run, e.g. a program interruption
- the machine functions such as switching the spindle rotation and coolant supply on and off
- the contouring behaviour of the tool

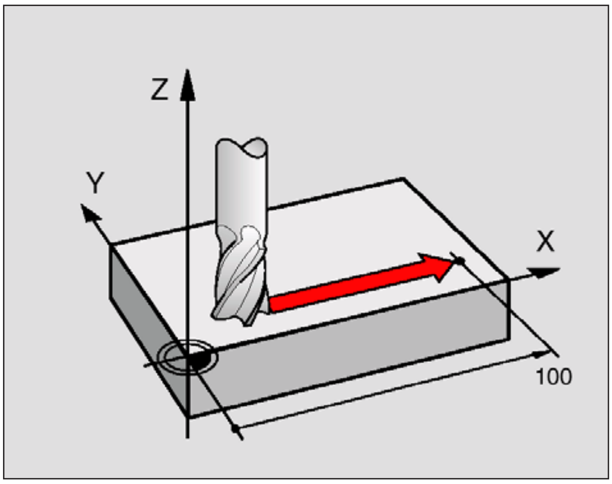
Subprograms and Program section repeats

If a machine sequence occurs several times in a program, you enter the sequence once and define it as subprogram or program section repetition. If you wish to execute a specific program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a further part program for execution.

Programming with Q parameters

In a part program Q parameters stand for numerical values. You assign the values to the Q parameters separately with the Q parameter functions. You can use Q parameters for programming mathematical functions that control program execution or describe a contour.

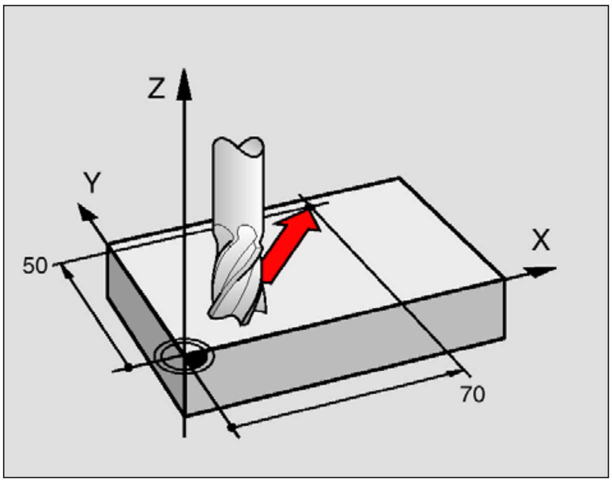
Fundamentals of path functions



Programming tool movements for workpiece machining

You create a part program by programming the path functions for the single elements of the workpiece contour in sequence. You do this by entering **the coordinates of the end points of the contour elements,** given in the production drawing. The WinNC calculates the actual path of the tool from these coordinates and from the tool data and radius compensation.

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

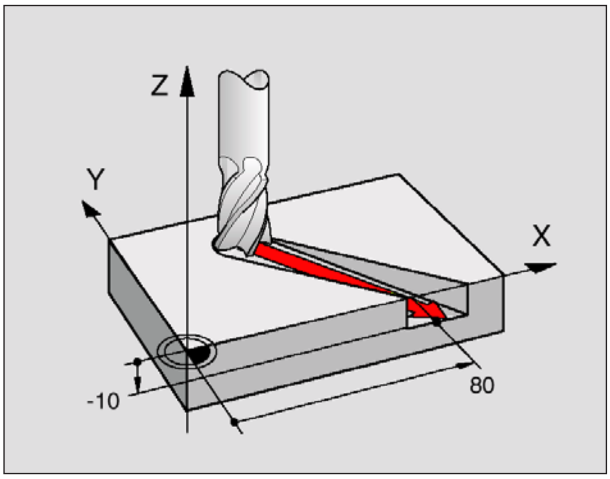


Movements parallel to the machine axes

The program block contains one coordinate: The WinNC moves the tool parallely to the programmed machine axis.

The part program is executed by movement of the machine table on which the workpiece is clamped. You always program path contours as if the tool moves.

Examplel:
L X+100
L .... Path function for „Straight line“
X+100 ... Coordinate of the end point
The tool retains the Y and Z coordinates and moves to the position X=100 (see illustration on upper left).



Movements in the main planes

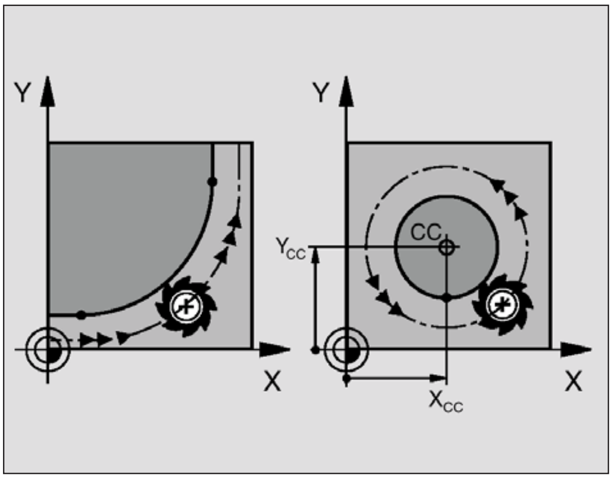
The program block contains two coordinates: The WinNC moves the tool in the programmed plane.

Example:
L X+70 Y+50
The tool retains the Z coordinate and moves in the XY plane to the position X=70, Y=50 (see illustration at the left center).

Three-dimensional movement

The program block cotnains three coordinates. The WinNC moves the tool in space to the programmed position.

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



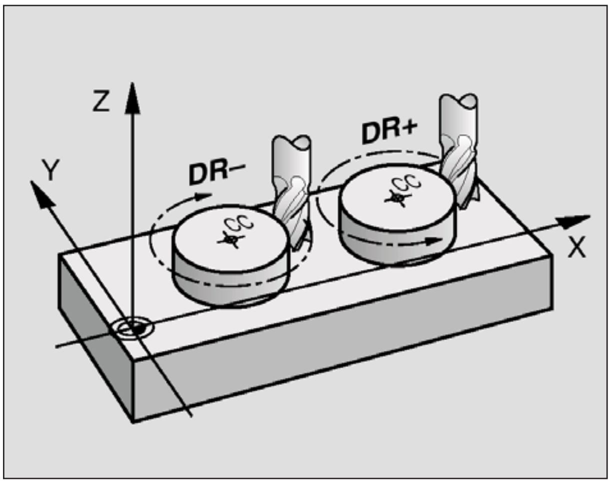
Circles and circular arcs

The WinNC traverses two machine axes simultaneously in circular movements: The tool moves in a circular path relative to the workpiece. For circular movements you can enter a circle center CC.

You program circles in the main planes by means of the path functions for circular arcs: the main plane is defined when you set the spindle axis during a TOOL CALL.

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

Note:
Circles that do not lie in the main plane are also programmed by using the Q parameters.
(see chapter G)



Direction of rotation DR for circular movements

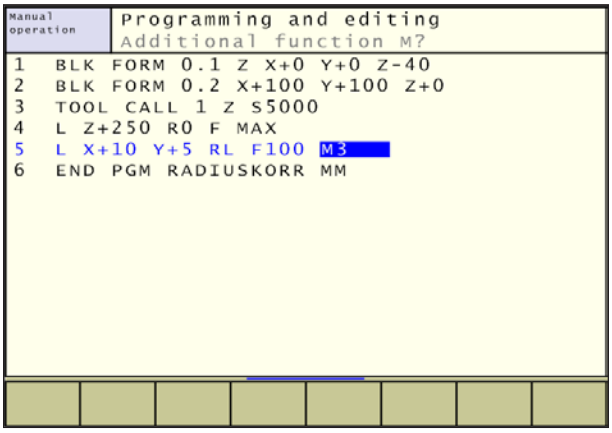
The direction of rotation DR has to be defined for circular movements.

clockwise direction of rotation: DR–
counterclockwise direction of rotation: DR+

Radius compensation

The radius compensation must be in the block in which you move to the first contour element. You must not begin radius compensation in a circle block. It must be programmed beforehand as a straight-line block (see "Path contours" - Cartesian coordinates) or as an approach block (APPR- block, see "Contour approach and departure").

Pre-positioning
Before running a part program, always pre-position the tool to prevent the tool and the workpiece from being damaged.



Creating program blocks with path function keys
The gray path function keys initiate the conversational format.
The WinNC asks you successively for all the necessary information and inserts the program block into the part program.

Example: Programming a straight line:
Initiate the programming dialogue, e.g. for a straight line

Enter the coordinates of the straight-line end point.

X 10
Y 5
ENT

R0 RL RR

Select the radius compensation:e.g press the RL soft key and the tool moves to the left of the contour

Enter the feed rate and confirm with the ENT key: e.g. 100 mm/min. For programming in inches, enter 100 for a feed rate of 10 ipm.

100 ENT

F MAX

Move at rapid traverse: Press the F MAX soft key.

Enter a miscellaneous function, e.g. M3, and terminate the dialogue with the END key.

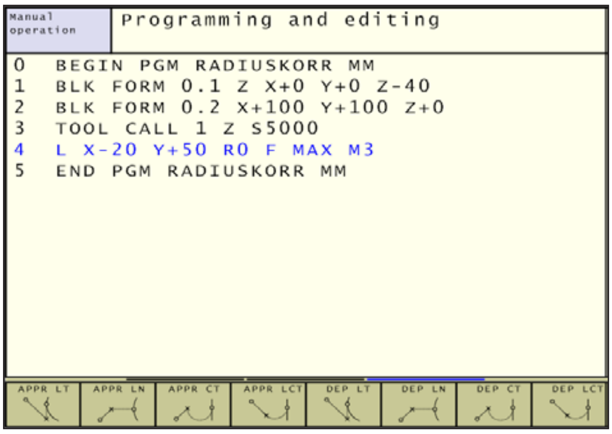
3 END

The part program now contains the following line:
L X+10 Y+5 RL F100 M3

Contour approach and departure

Overview: Types of paths for contour approach and departure

The functions APPR (approach) and DEP (departure) are activated with the APPR DEP key. You can then select the following path functions with the corresponding soft keys:

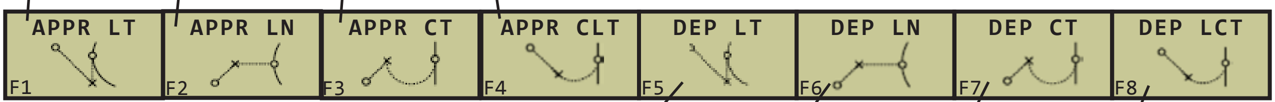


Approaching a straight line with tangential connection

Approaching a straight line perpendicular to a contour point

Approaching a circular arc with tangential connection

Approaching on a circular arc with tangential connection from a straight line to the contour.



Departing a straight line with tangential connection.

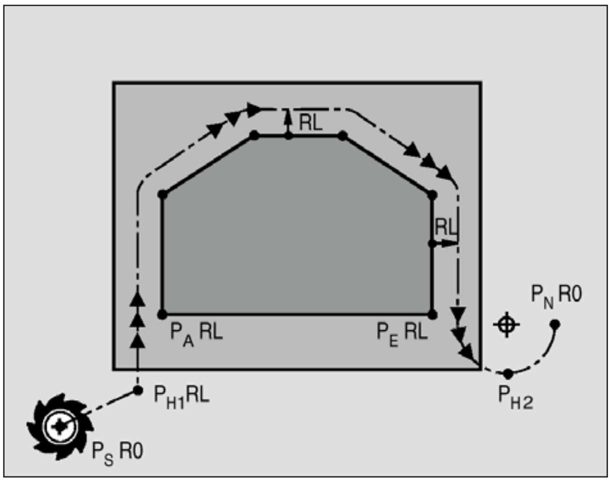
Departing a straight line perpendicular to a contour point

Departing a circular arc with tangential connection

Departing on a circular arc tangentially connecting the contour and a straight line.

Approaching and departing a helix

The tool approaches and departs a helix on its extension and connects to the contour on a tangential circular arc. You program helix approach and departure with the APPR CT and DEP CT functions.



Important positions for approach and departure

- Starting point P_S
  You program this position directly before the APPR block.
  P_s lies outside the contour and is approached without radius compensation (R0).
- Auxiliary point P_H
  Some of the paths for approach and departure go through an auxiliary point P_H that the WinNC calculates from your input in the APPR or DEP block.
- First contour point P_A and last contour point P_E
  You 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 also contains a Z axis coordinate, the WinNC will first move the tool to P_{H1} in the working plane and then move it to the entered depth in the tool axis.
- Endpoint P_N
  The position P_N lies outside of the contour and results from your input in the DEP block. If the DEP block also contains a Z axis coordinate, the WinNC will first move the tool to P_{H2} in the working plane and then move it to the entered height in the tool axis.

The WinNC does not check whether the programmed contour will be damaged when moving from the actual position to the auxiliary point P_H. Check before by using the graphics !

When using the functions APPR LT, APPR LN and APPR CT the WinNC moves the tool from the actual position to the auxiliary point P_H at the feed rate/ rapid traverse that was programmed last.

When using the APPR LCT function the WinNC moves the tool to the auxiliary point P_H at the feed rate that was programmed in the APPR block.

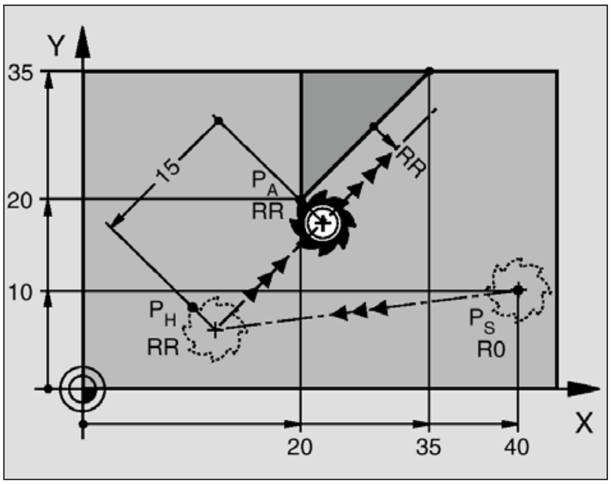
The coordinates can be entered in an absolute or incremental way in Cartesian coordinates or polar coordinates.

Radius compensation
The radius compensation is programmed together with the first contour point P_A in the APPR block. The DEP blocks automatically remove the radius compensation!
Approach without radius compensation: If you program the APPR block with R0, the WinNC will calculate the tool path for a tool radius of 0 mm and a radius compensation RR! The radius compensation is necessary to set the direction of contour approach and departure in the APPR/DEP LN and APPR/DEP CT functions.

Used abbreviations and their meaning:

APPR	Approach
DEP	Departure
L	Line
C	Circle
T	Tangential (continuous, smooth connection)
N	Normal (perpendicular)

Approaching on a straight line with tangential connection: APPR LT

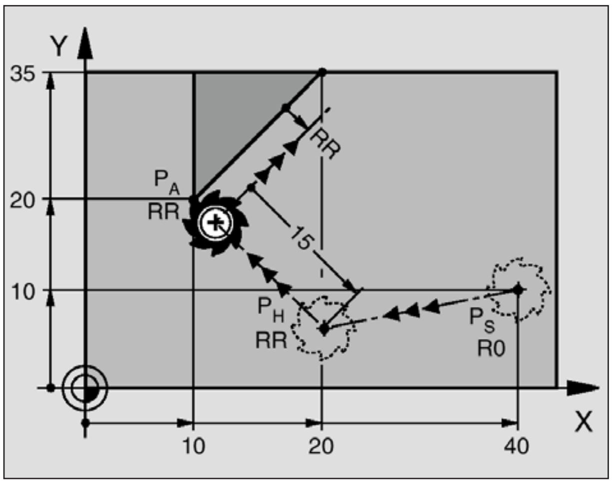


The WinNC moves the tool on a straight line from the starting point P_S to an auxiliary point P_H . From there it moves to the first contour point P_A on a straight line that connects tangentially to the contour. The auxiliary point P_H has the distance LEN to the first contour point P_A.

- Use any path function to approach the starting point P_S
- Initiate the dialogue with the APPR/DEP key and the APPR LT soft key:
- Coordinates of the first contour point P_A
- LEN: Distance from the auxiliary point P_H to the first contour point P_A
- Radius compensation RR/RL for machining

Example NC blocks	
7 L X+40 Y+10 RO FMAX M3	Approach P_S without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	P_A with radius comp. RR, distance P_H to P_A: LEN=15
9 L Y+35 Y+35	End point of the first contour element
10 L ...	Next contour element.

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

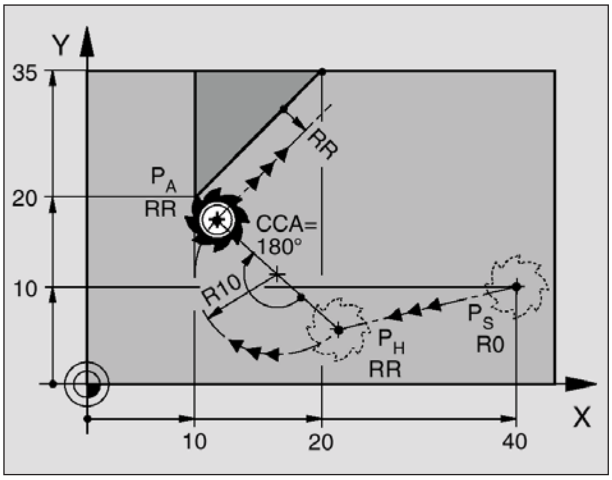


The WinNC moves the tool on a straight line from the starting point P_S to an auxiliary point P_H . From there it moves to the first contour point P_A on a straight line perpendicular to the contour. The auxiliary point P_H has the distance LEN plus the tool radius to the first contour point P_A.

- Use any path function to approach the starting point P_S
- Initiate the dialogue with the APPR/DEP key and the APPR LT soft key:
- Coordinates of the first contour point P_A
- Length: Distance to the auxiliary point P_H. Always enter LEN as a positive value.
- Radius compensation RR/RL for machining

Example NC blocks	
7 L X+40 Y+10 RO FMAX M3	Approach P_S without radius compensation
8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100	P_A with radius comp. RR, distance P_H to P_A: LEN=15
9 L Y+35 Y+35	End point of the first contour element
10 L ...	Next contour element

Approaching on a circular path with tangential connection: APPR CT

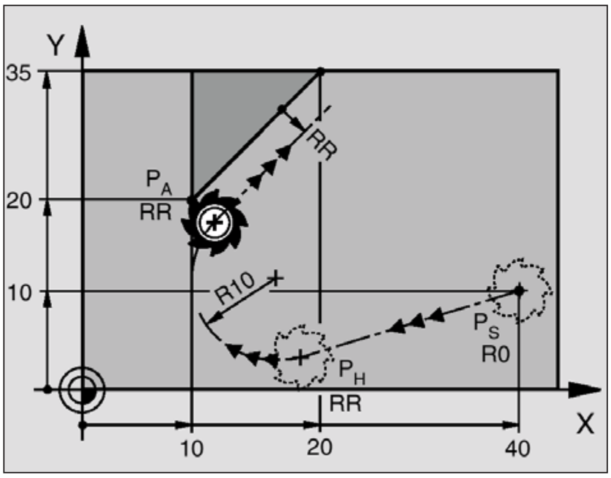


The WinNC moves the tool on a straight line from the starting point P_S to an auxiliary point P_H . From there it moves to the first contour point P_A following a circular arc that is tangential to the first contour element. The arc from P_H nach P_A is determined through the radius R and the center angle CCA. The direction of rotation of the circular arc is automatically derived from the tool path for the first contour element.

- Use any path function to approach the starting point P_S.
- Initiate the dialogue with the APPR/DEP key and the APPR CT softkey:
- Coordinates of the first contour point P_A
- Radius R of the circular arc
  - Approaching the workpiece in the direction defined by the radius compensation: Enter R as a positive value.
  - Approaching the workpiece opposite to the radius compensation: Enter R as a negative value
- Center angle CCA of the circular arc
  - Enter CCA only as a positive value
  - Maximum input value 360°
- Radius compensation RR/RL for machining

Example NC blocks	
7 L X+40 Y+10 RO FMAX M3	Approach P_S without radius compensation
8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100	P_A with radius compensation RR, radius R=10
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element

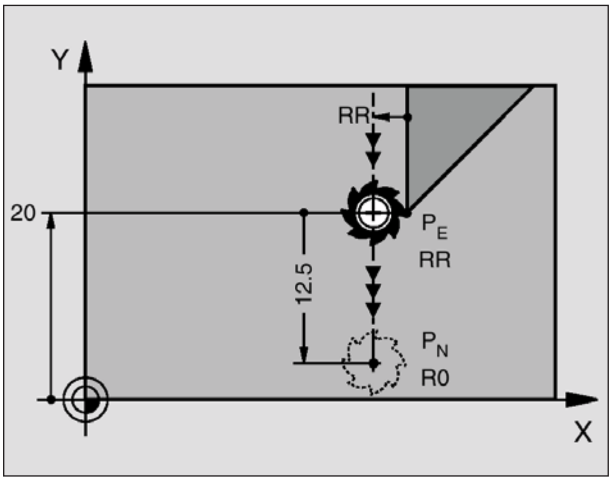
Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT



The WinNC moves the tool on a straight line from the starting point P_S to an auxiliary point P_H . From there it moves to the first contour point P_A on a circular arc. The feed rate programmed in the APPR block is active. The circular arc is tangentially connected both to the straight line P_S-P_H and to the first contour element. Herewith it is completely defined by the radius R.

- Usa any path function to approach the starting point P_S.
- Intitiate the dialogue with the APPR/DEP key and the APPR LCT soft key:
- Coordinates of the first contour point P_A
- Radius R of the circular arc: Enter R as a positive value.
- Radius compensation RR/RL for machining

Example NC blocks	
7 L X+40 Y+10 RO FMAX M3	Approach P_S without radius compensation
8 APPR LCT X+10 Y+20 Z-10 R10 RR F100	P_A with radius compensation RR, radius R=10
9 L X+20 Y+35	End point of the first contour element
10 L ...	Next contour element



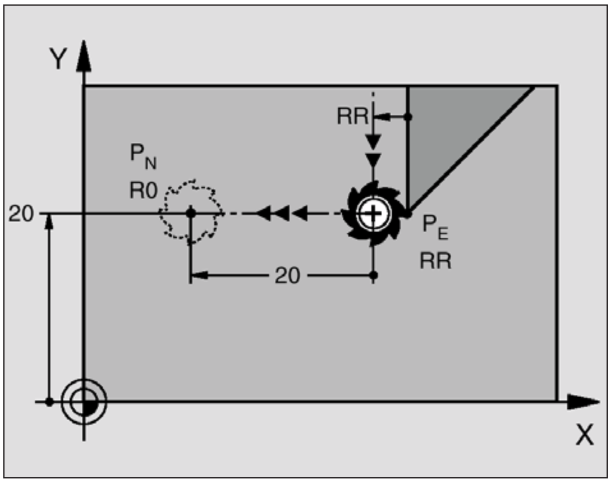
Departing on a straight line with tangential connection: DEP LT

The WinNC moves the tool on a straight line from the last contour point P_E to the end point P_N. The straight line lies in the extension of the last contour element. P_N has the distance LEN from P_E .

- Program the last contour element with the end point P_E and radius compensation
- Initiate the dialogue with the APPR/DEP key and the DEP LT soft key
- LEN: Enter the distance from the end point P_N to the last contour element P_E.

Example NC blocks

23 L Y+20 RR F100	Last contour element: P_E with radius compensation
24 DEP LT LEN12,5 F100	Depart contour by LEN=12,5 mm
25 L Z+100 FMAX M2	Retract in Z, return to block, end program



Departing on a straight line perpendicular to the last contour point: DEP LN

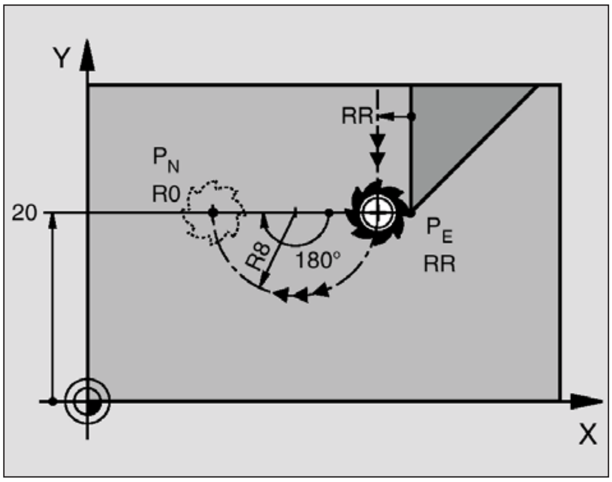
The WinNC moves the tool on a straight line from the last contour point P_E to the end point P_N. The straight line departs on a perpendicular path from the last contour point P_E. P_N has a distance LEN plus the tool radius
from P_E.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialogue with the APPR/DEP key and the DEP LT soft key
- LEN: Enter the distance from the end point P_N. Important: Always enter LEN as a positive value!

Example NC blocks

23 L Y+20 RR F100	Last contour element P_E with radius compensation
24 DEP LN LEN+20 F100	Depart perpendicular to contour by LEN=20 mm
25 L Z+100 FMAX M2	Retract in Z, return to block, end program

Departure on a circular arc with tangential connection: DEP CT



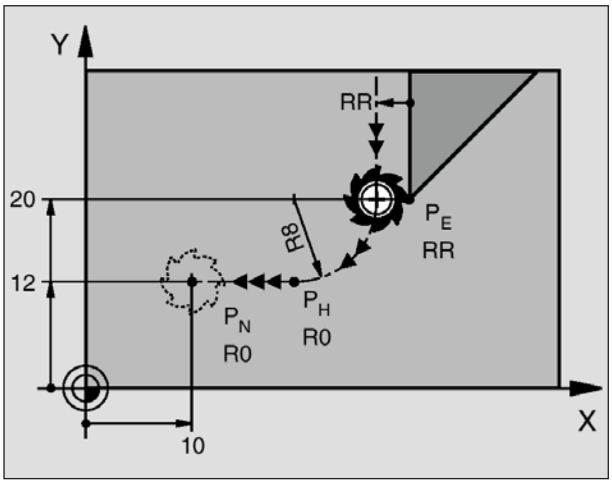
The WinNC moves the tool on a circular arc from the last contour point P_E to the end point P_N . The circular arc is tangentially connected to the last contour element.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialogue with the APPR/DEP key and the DEP CT softkey.
- Center angle CCA of the circular arc
- Radius R of the circular arc
- The tool should depart the workpiece in the direction that is determined by the radius compensation: Enter R as a positive value.
- The tool should depart the workpiece in the direction **opposite** to the radius compensation: Enter R as a negative value.

Example NC blocks

23 L Y+20 RR F100	Last contour element: P_E with radius compensation
24 DEP CT CCA 180 R+8 F100	Center angle=180°, arc radius=8 mm
25 L Z+100 FMAX M2	Retract in Z, return to block, end program

Departing on a circular arc tangentially connecting the contour and a straight line: DEP LCT



The WinNC moves the tool on a circular arc 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 circular arc is tangentially connected both to the last contour element and to the straight line from $P_H – P_N$.
Herewith it is completely defined by the radius R.

- Program the last contour element with the end point P_E and the radius compensation.
- Initiate the dialogue with the APPR/DEP key and the DEP LCT soft key:
- Enter the coordinates of the end point P_N
- Radius R of the circular arc: Enter R as a positive value

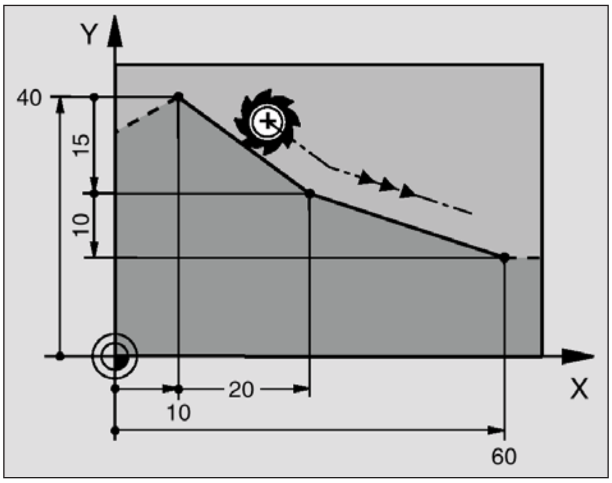
Example NC blocks

23 L Y+20 RR F10	Last contour element: P_E with radius compensation
24 DEP LCT X+10 Y+12 R+8 F10	Coordinates P_N, arc radius=8 mm
25 L Z+100 FMAX M2	Retract in Z, return to block, end program

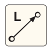
Path contours - Cartesian coordinates

Overview of path functions

Function	Path function key	Tool movement	Required input
Line **L**	L	Straight line	Coordinates of the end point of the straight line
Chamfer **CHF**	CHF	Chamfer between two straight lines	Chamfer length
Circle Center **CC**	CC	No tool movement	Coordinates of the circle center or pole
Circle **C**	C	Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation
Circle by Radius **CR**	CR	Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation
Circle Tangential **CT**	CT	Circular arc with tangential connection to the preceding and subsequent contour element	Coordinates of the arc endpoint
Corner Rounding **RND**	RND	Circular arc with tangential connection to the preceding and subsequent contour element	Rounding-off radius R
Free Contour Programming **FK**	FK	Straight line or circular arc with any connection to the preceding contour element	See „Free Contour programming FK“



Example NC blocks
7 L X+10 Y+40 RL F200 M3
8 L IX+20 IY-15
9 L X+60 IY-10

Straight line L 

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

- **Coordinates** of the end point of the straight line

Further entries, if necessary:

- **Radius compensation RL/RR/R0**
- **Feed rate F**
- **Miscellaneous function M**

Actual position capture

You can also generate a straight-line block (L block) by using the ACTUAL-POSITION-CAPTURE key:

- Move the tool to the position you wish to capture in the manual operation mode.
- Switch the screen display to Programming/ Editing
- Select the program block after which you want to insert the L block.
- Press the ACTUAL-POSITION-CAPTURE key: The WinNC generates an L block with the actual position coordinates.

Note:
You define the number of axes that the WinNC saves in an L block by means of the MOD functions.

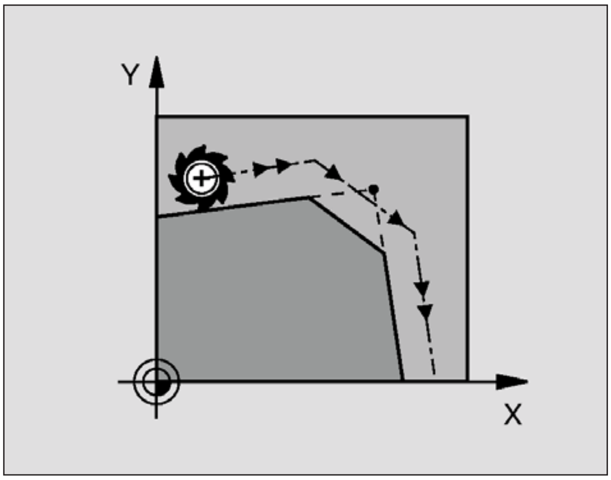
Inserting a chamfer CHF between two straight lines

The chamfer enables you to cut off corners at the intersection of two straight lines.

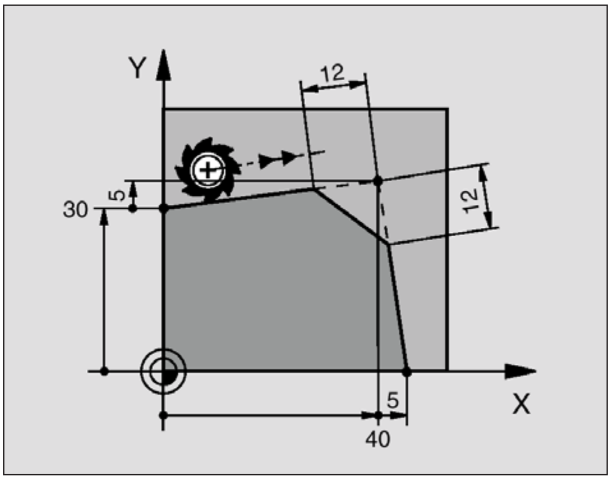
- In the blocks before and after the CHF block you program both coordinates of the plane, in which the chamfer is executed.
- The radius compensation before and after the CHF block must be the same.
- An inside chamfer must be large enough to accomodate the current tool.
- **Chamfer side length:** Length of the chamfer

Further entries, if necessary:

- **Feed rate F** (only effective in CHF block)

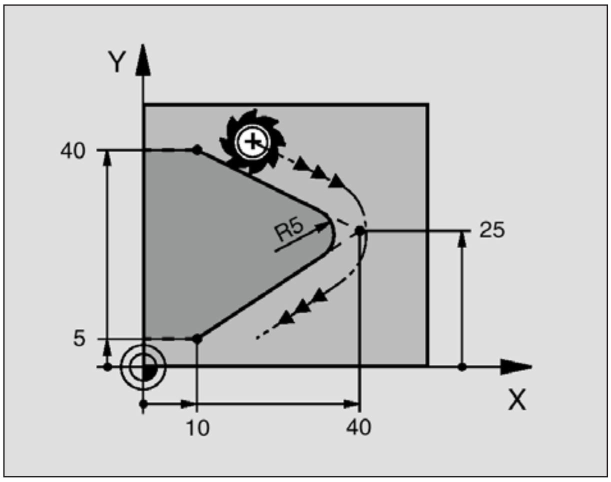


Example NC blocks
7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12 F250
10 L IX+5 Y+0



Note:
Do not start a contour with a CHF block.
A chamfer is only possible in the working plane.
The corner point that is cut off by the chamfer will not be approached.
A feed rate programmed in the CHF block is only effective in that block. After the CHF block the previous feed rate becomes effective again.

Corner Rounding RND



The RND function is used for rounding off contour corners.
The tool moves on a circular arc that is tangentially connected to both the preceding and the subsequent contour element.
The rounding arc must be executable with the called tool.

- **Rounding-off radius:** Radius of the circular arc

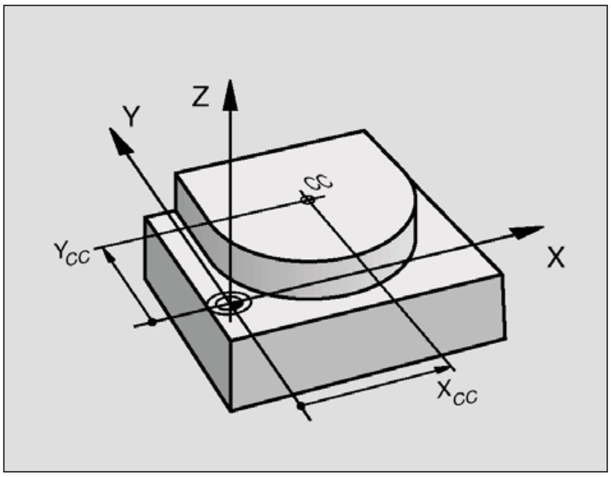
Further entries, if necessary:

- **Feed rate F** (only effective in RND block)

Example NC blocks
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 the subsequent contour element should contain both coordinates of the plane, in which the corner rounding is executed.
If you machine the contour without tool-radius compensation, you must program both coordinates of the working plane.
The corner point will not be approached.
A feed rate programmed in the RND block is only effective in that block. After the RND block the previous feed rate becomes effective again.
You can also use a RND block for a smooth contour approach if you do not want to use the APPR functions.

Circle center CC



Example NC blocks
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 illustration.

You can define a circle center CC for circular paths that are programmed with the C key (circular arc C). This is done in the following ways:

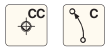
- Enter the Cartesian coordinates of the circle center, or
- Use the circle center defined in an earlier block, or
- Capture the coordinates with the ACTUAL-POSITION-CAPTURE key.
- **Coordinates** CC: Enter the circle center coordinates
  or
  If you want to use the last programmed position, do not enter any coordinates

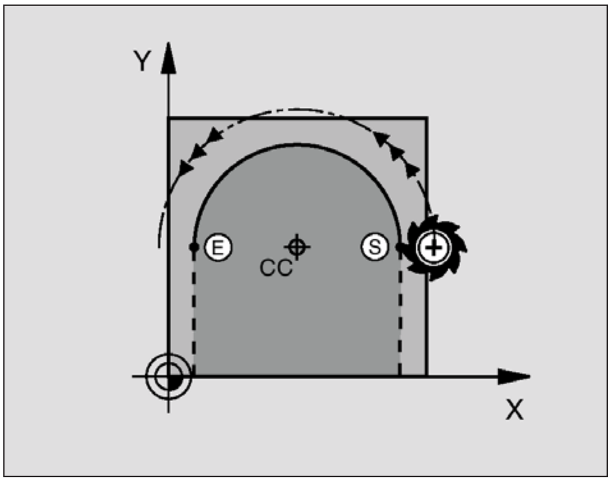
Duration of effect
The circle center definition remains in effect until a new circle center is programmed.

Entering the circle center CC incrementally
An incrementally entered coordinate for the circle center always refers to the last programmed tool position.

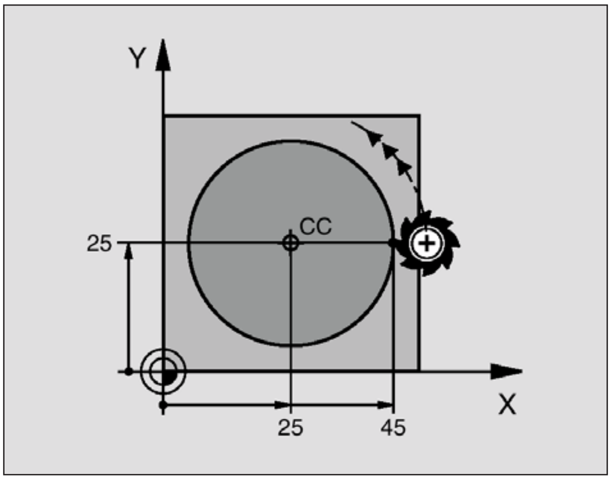
> **Note:**
> You define a position as circle center with CC: The tool does not move to this position.
> The circle center is also the pole for polar coordinates.

Circular path C around circle center CC





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



Before programming a circular path C, you have to define the circle center CC .
The last programmed tool position before the C block is the starting point for the circular path.

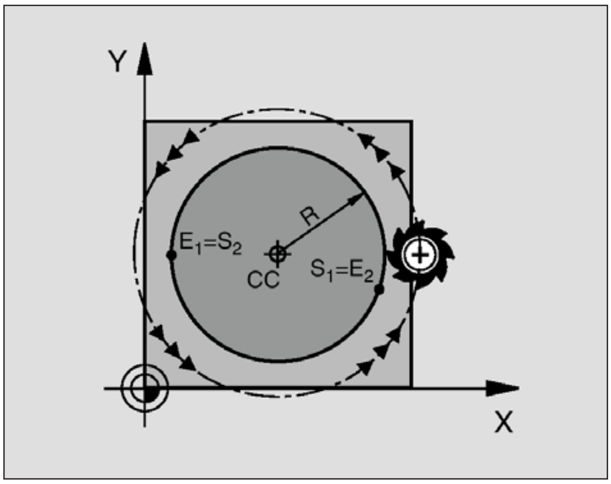
- Move the tool to the starting point of the circular path
- **Coordinates** of the circle center
- **Coordinates** of the arc end point
- **Direction of rotation DR**

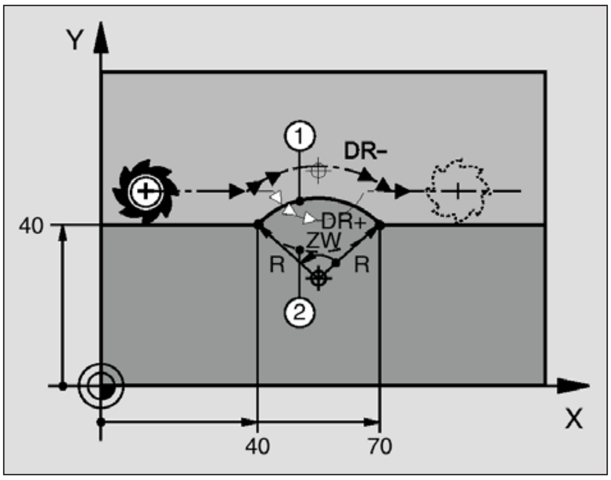
Further entries, if necessary

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

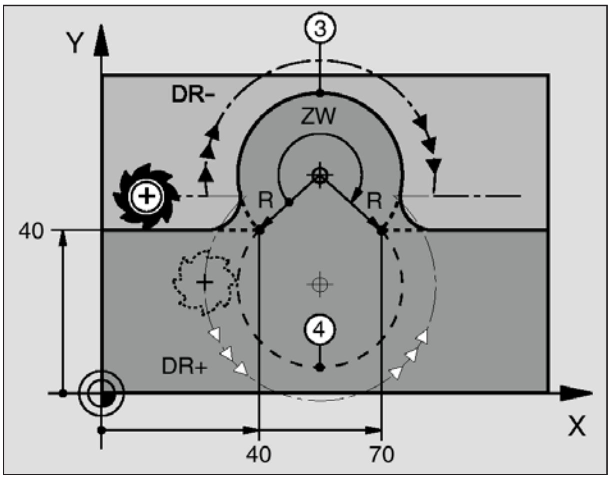
Full circle
Program the same coordinates for the end point that you entered for the starting point.

Note:
The starting and end points of the circular movement must lie on the circular path.
Input tolerance: 0.016 mm





Example NC blocks
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 NC blocks
11 CR X+70 Y+40 R-20 DR- (arc 3)
or
11 CR X+70 Y+40 R-20 DR+ (arc 4)

Circular path CR with defined radius

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

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

Note: The algebraic sign determines the size of the circular arc!

- **Direction of rotation DR**

Note: The algebraic sign determines whether the arc is concave or convex!

Further entries, if necessary:

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

Full circle

For a full circle, program two CR blocks in succession: 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.

Central angle CCA and circular arc radius R

The starting and end points on the contour can be connected with four different circular arcs of 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

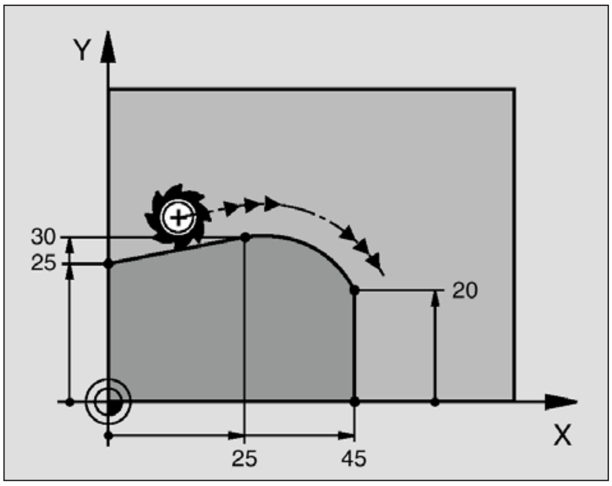
The direction of rotation determines whether the arc is convex or concave.

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

Conkave: Direction of rotation DR+ (with radius correction RL)

Note:
The distance between the starting and the end point of the arc diameter must not be greater than the diameter of the arc.
The maximum radius is 99.9999 m.

Circular path CT with tangential connection



Example NC blocks
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

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.

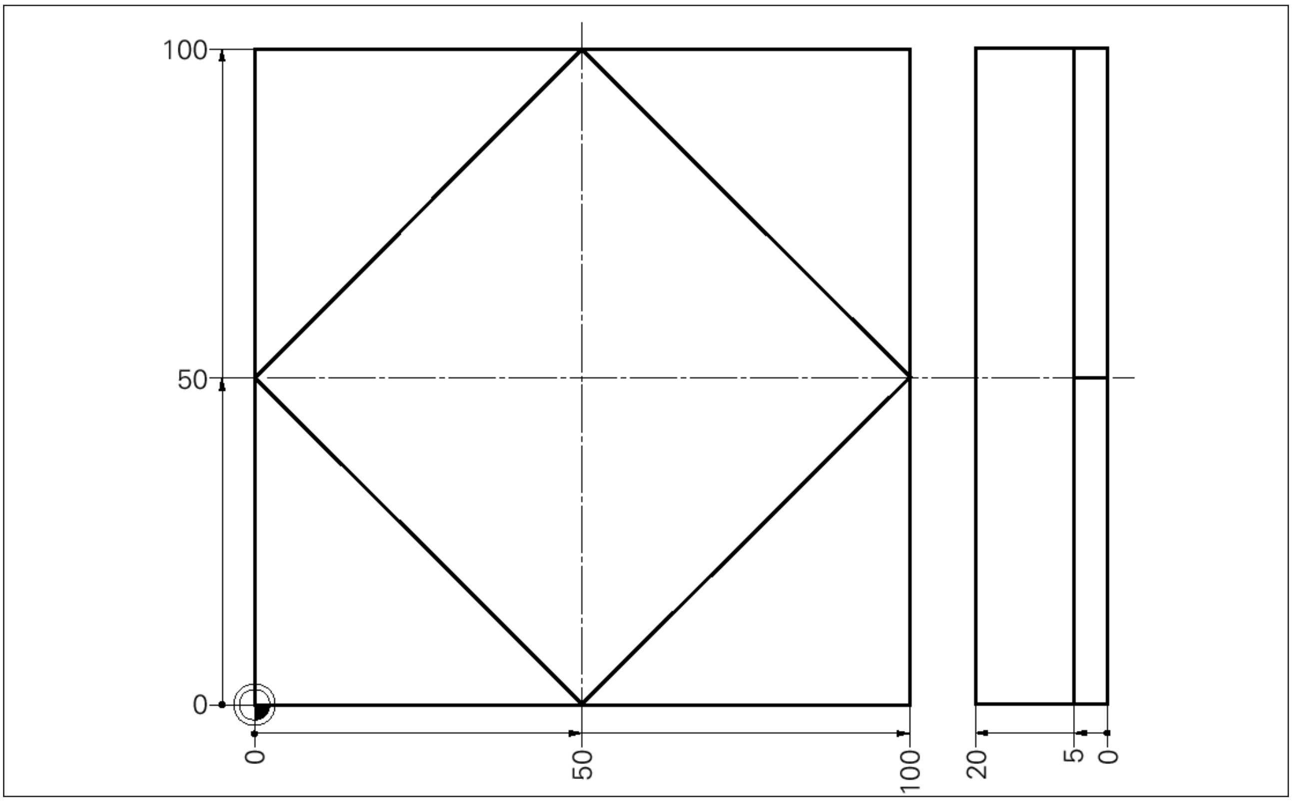
- **Coordinates** of the end point of the circular arc

Further entries, if necessary:

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

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

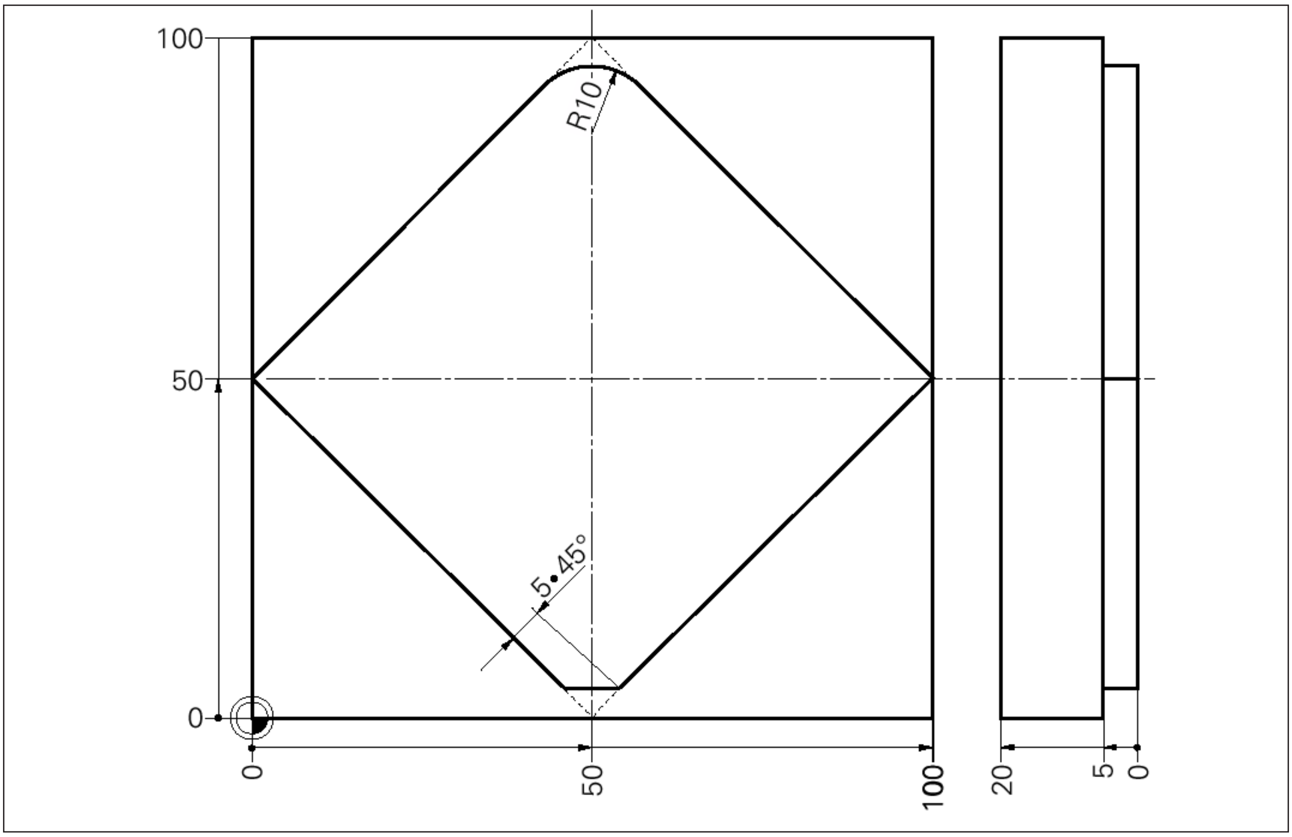
example: square



main program

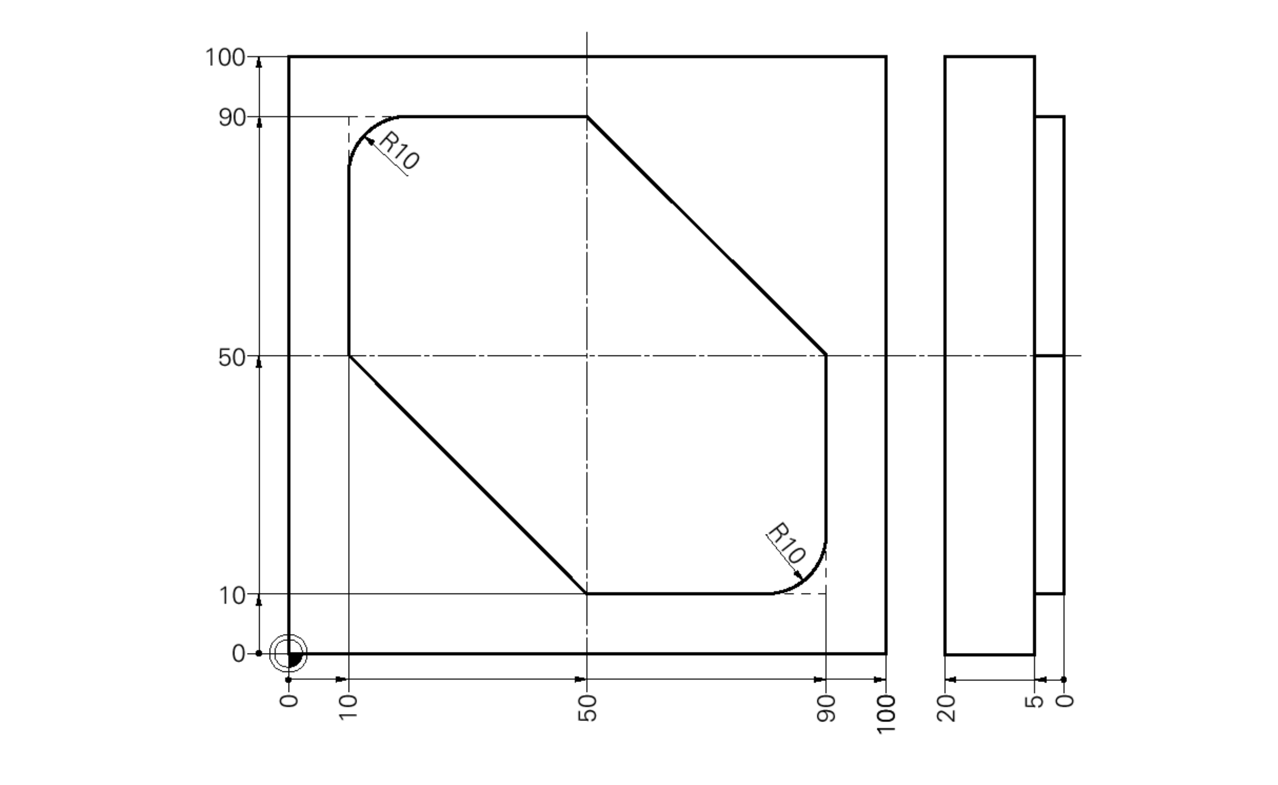
0 BEGIN PGM 152 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20
2 BLK FORM 0.2 X+100 Y+100 Z+0 .... raw-part-definition
3 TOOL DEF 1 L+0 R+8 .... tool-definition
4 TOOL CALL 1 Z S4000 .... tool-call
5 L Z+100 R0 F MAX.. secure height
6 L X-30 Y+50 R0 F MAX................................... auxiliary point (R0)
7 L Z-5 R0 F MAX M3 .... infeed depth
8 L X+0 Y+50 RL F400....................................... departure-contour (RL/RR)
9 L X+50 Y+100
10 L X+100 Y+50
11 L X+50 Y+0
12 L X+0 Y+50 RL.. last contour-point
13 L X-30 R0 F MAX M5 .... auxiliary point
14 L Z+100 R0 F MAX M2.................................. relieve/PGM-END
15 END PGM 152 MM

example: nooks round / chamfer 1



main program
0 BEGIN PGM 153 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20
2 BLK FORM 0.2 X+100 Y+100 Z+0.................. raw-part-definition
3 TOOL DEF 1 L+0 R+8..................................... tool-definition
4 TOOL CALL 1 Z S4000 .... tool call
5 L Z+100 R0 F MAX.. secure heigth
6 L X-30 Y+50 R0 F MAX................................... auxiliary point (R0)
7 L Z-5 R0 F MAX M3
8 L X+0 Y+50 RL F200....................................... departure contour RL
9 L X+50 Y+100
10 RND R10... nook round
11 L X+100 Y+50
12 L X+50 Y+0
13 CHF 5... chamfer
14 L X+0 Y+50 RL
15 L X-30 R0 M5 .... auxiliary point (RO)
16 L Z+100 R0 F MAX M2.................................. PGM-END
17 END PGM 153 MM

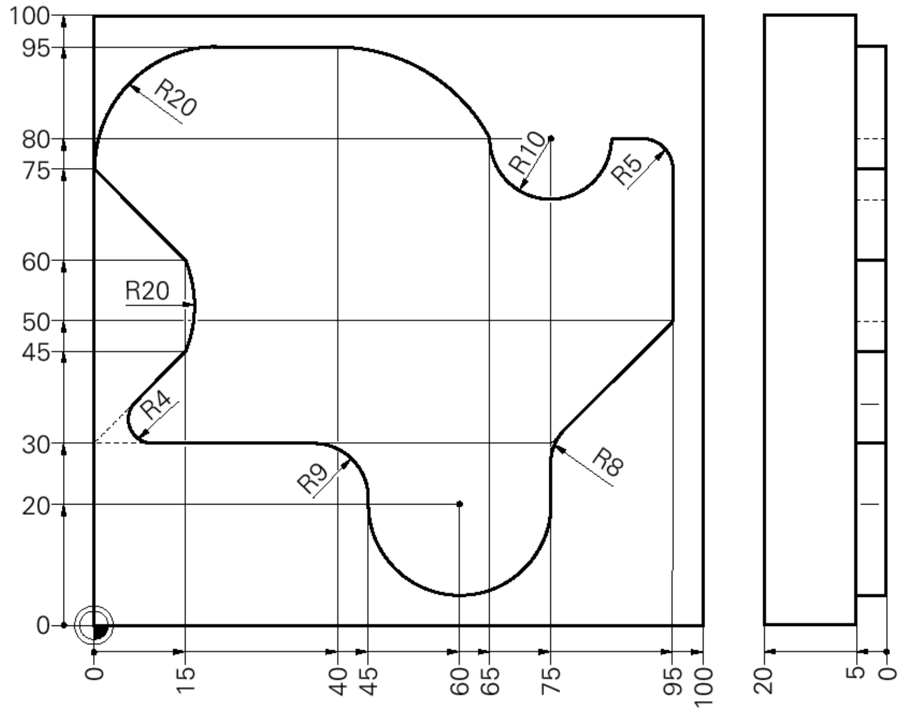
example: nooks round / chamfer 2



main program

0 BEGIN PGM 154 MM
1 BLK FORM 0.1 Z X-20 Y+0 Z-20
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL DEF 1 L+0 R+8
4 TOOL CALL 1 Z S4000
5 L Z+100 R0 F MAX
6 L X-30 Y+70 R0 F MAX .... auxiliary point (R0)
7 L Z-5 R0 F MAX M3
8 APPR LCT X+10 Y+70 R5 RL F400 .... soft departure point
9 L X+10 Y+90
10 RND R10
11 L X+50 Y+90
12 L Y+50 X+90
13 L X+90 Y+10
14 RND R10
15 L X+50 Y+10
16 L X+10 Y+50
17 L Y+70 .... last contour point RL
18 DEP LCT X-30 Y+70 R5 .... depart soft auxiliary point
19 L Z+100 R0 F MAX M2
20 END PGM 154 MM

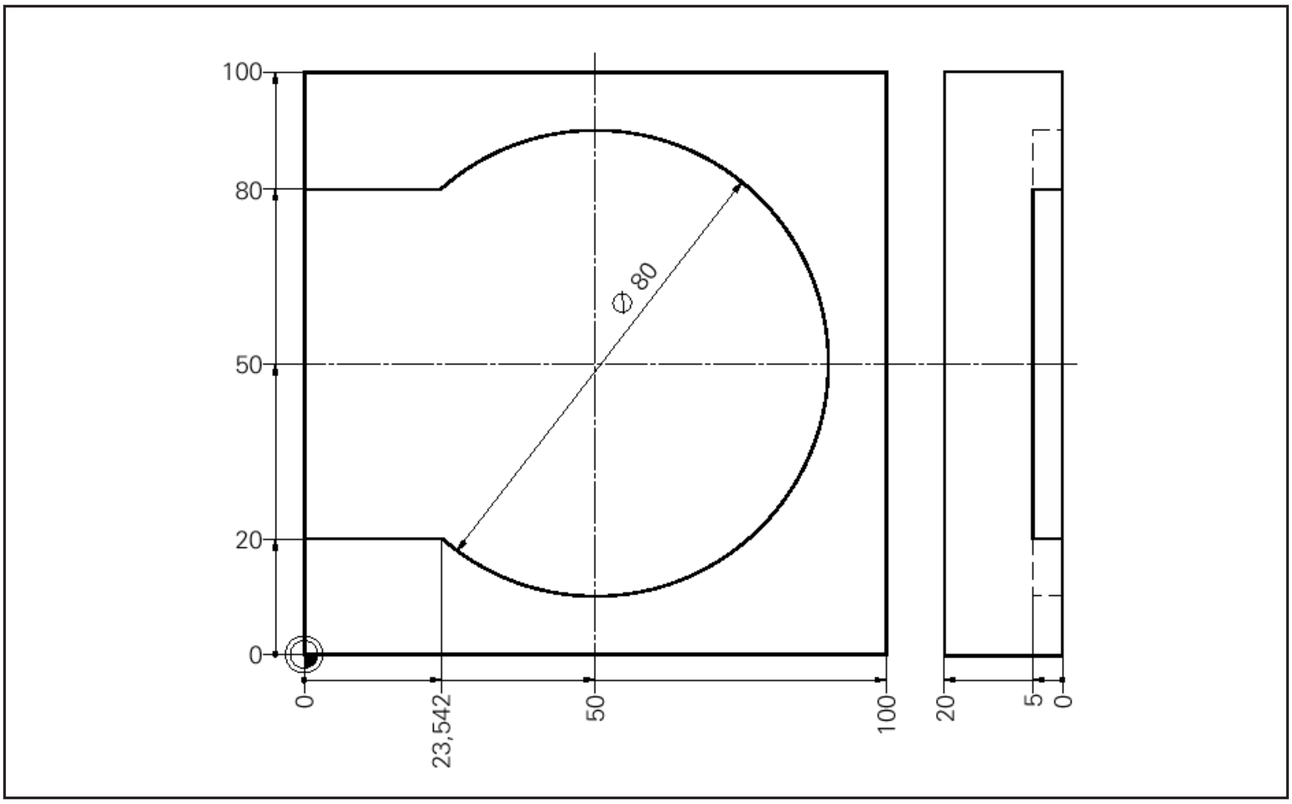
example: circular motions



main program
0 BEGIN PGM 251 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL CALL 7 Z S2500 .... R4
4 L Z+100 R0 F9999
5 L X+20 Y-20 .... auxiliary point (R0)
6 L Z+2 M3
7 L Z-5 F500
8 APPR LCT X+20 Y+30 R3 RL F300 .... 1st contour point
9 L X+0 (soft start up)
10 RND R4
11 L X+15 Y+45
12 CR X+15 Y+60 R+20 DR+
13 L X+0 Y+75
14 CR X+20 Y+95 R+20 DR-
15 L X+40
16 CT X+65 Y+80
17 CC X+75 Y+80
18 C X+85 Y+80 DR+
19 L X+95
20 RND R5
21 L Y+50
22 L X+75 Y+30
23 RND R8
24 L Y+20
25 CC X+60 Y+20

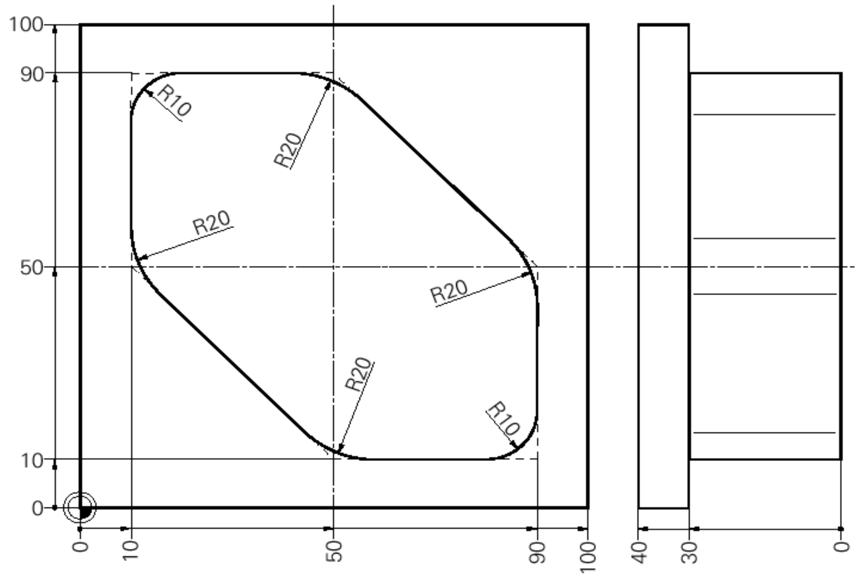
26 C X+45 Y+20 DR-
27 L Y+30
28 RND R9
29 L X+20 .... last contour point
30 DEP LCT X+20 Y-20 R3 F500 .... auxiliary point (R0)
31 L Z+100 R0 F MAX M2
32 END PGM 251 MM

example: circular arc with CC, C



main program
0 BEGIN PGM 206 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-20
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL CALL 13 Z S2500 .... R20
4 L Z+100 R0 F MAX
5 L X-30 Y+50 R0 F MAX .... auxiliary point
6 L Z-5 R0 F MAX M3
7 APPR LT X+0 Y+20 LEN10 RL F250 M8
8 L X+23,542 RL
9 CC X+50 Y+50 .... circle center
10 C Y+80 X+23,542 DR+ .... circular motion
11 L X+0 RL
12 DEP LT LEN10 .... soft depart (auxiliary point.)
13 L Z+100 R0 F MAX M2
14 END PGM 206 MM

example: milling with multi-infeed



main program

0 BEGIN PGM 223 MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-40
2 BLK FORM 0.2 X+100 Y+100 Z+0
3 TOOL CALL 13 Z S2500................................. R20
4 L Z+100 R0 F MAX M3
5 L X-30 Y+70 R0 F MAX.................................. startposition
6 L Z+0 F MAX
7 LBL 2
8 L IZ-5 R0 F MAX M3....................................... infeed
9 CALL LBL 1 .... Label call
10 CALL LBL 2 REP 5/5..................................... further contour-cuts
11 L Z+100 R0 F MAX M2.................................. relieve, end

subprogram, contour

12 LBL 1
13 APPR LCT X+10 Y+70 R5 RL F250 M3
14 L X+10 Y+90 RL
15 RND R10
16 L X+50 Y+90
17 RND R20
18 L X+90 Y+50
19 RND R20 .... contour
20 L X+90 Y+10
21 RND R10
22 L X+50 Y+10
23 RND R20
24 L X+10 Y+50
25 RND R20
26 L X+10 Y+70
27 DEP LCT X-20 Y+70 R5 F500
28 LBL 0
29 END PGM 223 MM....................................... subprogram-end

Path contours - Polar coordinates

Overview

With polar coordinates you can define a position in terms of its angle PA and its distance PR relative to a previously defined pole CC (see "FK free contur programming").

Polar coordinates are useful for:

- positions on circular arcs
- workpiece drawings with angle dimensions, e.g. for bolt hole circles

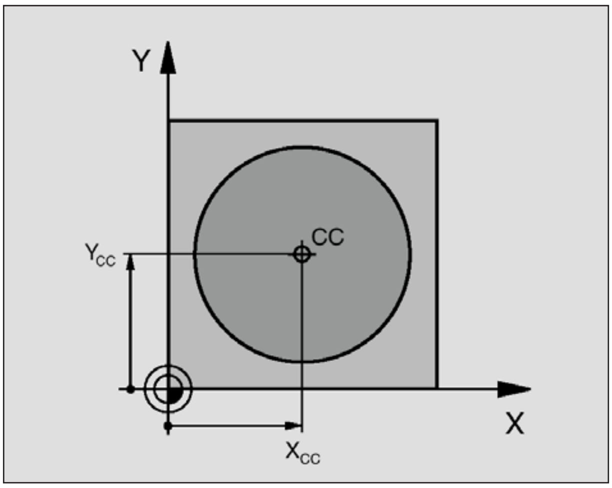
Overview of path functions with polar coordinates

Function	Path function key	Tool movement	Required input
Line **LP**	+ P	Straight line	Polar radius, polar angle of the straight line end point
Circular path **CP**	+ P	Circular path around circle center/ pole CC to arc end point	Polar angle of the arc end point, direction of rotation
Circular arc **CTP**	+ P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helical interpolation	+ P	Combination of a circular and a linear movement	Polar radius, polar angle of the arc end point, coordinate of the end point in the tool axis

Polar coordinate origin: Pole CC

You can define the pole CC anywhere in the part program before you enter blocks containing polar coordinates. Enter the pole in the same way as you program the circle center CC.

- **Coordinates** CC: Enter Cartesian coordinates for a pole. The pole CC remains in effect until you define a new pole CC.



Example NC blocks
12 CC X+45 Y+25

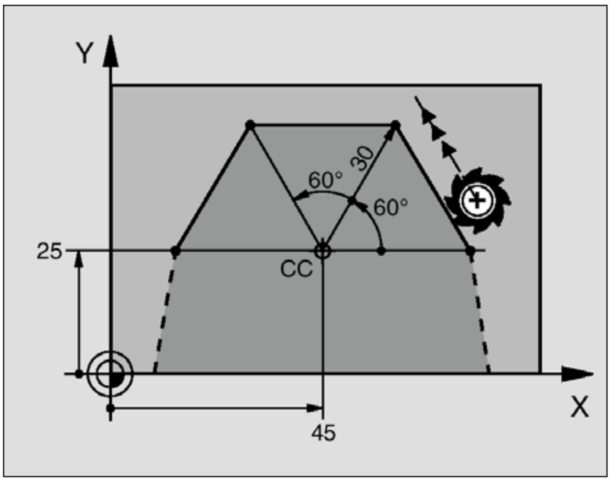
Straight line LP

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

- **Polar coordinate radius PR:** Enter the distance between the straight-line end point and the pole CC.
- **Polar coordinate angle PA:** Angular position of the straight-line end point between –360° and +360°.

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

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



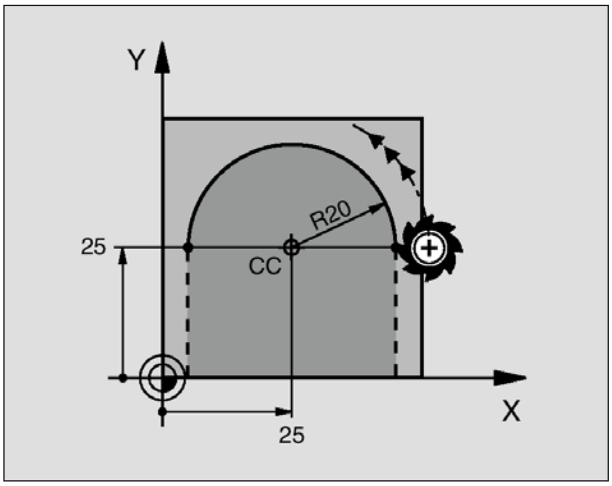
Example NC blocks
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F300 M3
14 LP PA+60
15 LP IPA+60
16 LP PA+180

Circular path CP around pole CC

The polar coordinate radius PR is also the radius of the circular arc. PR is defined by the distance between the starting point and the pole CC. The last programmed tool position before the CP block is the starting point of the circular path.

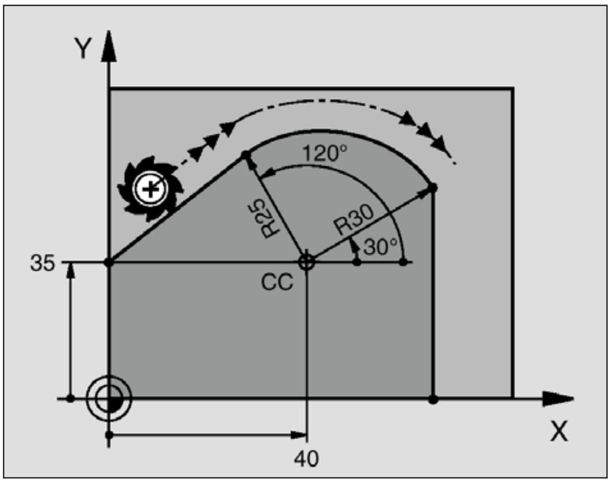
- **Polar coordinate angle PA:** Angular position of the circular-path end point between –5400° and +5400°
- **Direction of rotation DR**

Note:
For incremental coordinates enter the same sign for DR and PA.



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

Circular path CTP with tangential connection P



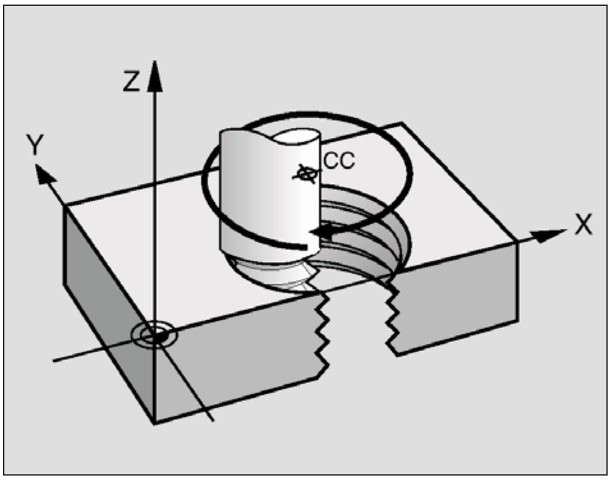
Example NC blocks
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0

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

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

Helix P



A helix is a combination of a circular movement in a main plane and a linear movement perpendicular to this plane.

A helix can only be programmed in polar coordinates.

Application

- Large diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix you need the incremental angle through which the tool moves on the helix, and the total height of the helix.

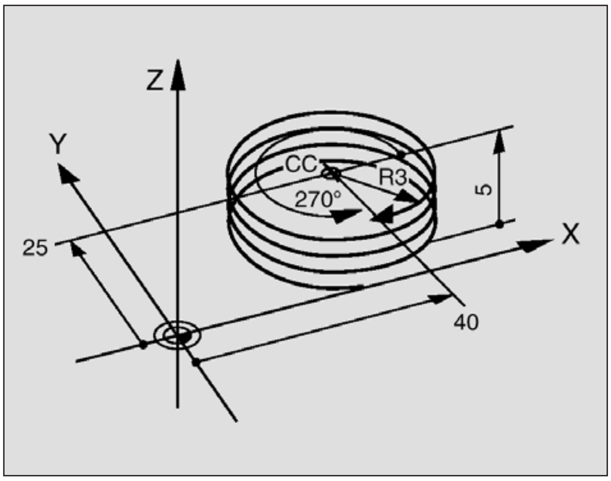
For calculating a helix that is to be cut in an upward direction you need the following data:

Thread revolutions n.......	Thread revolutions + thread overrun at the start and end of the thread
Total height h	Thread pitch P times thread revolutions n
Incremental total angle IPA	Number of revolutions times 360° + angle for start of thread + angle for thread overrun
Starting coordinate Z	Thread pitch P times (thread revolutions +thread overrun at start of thread)

Internal thread	Working direction	Direction	Radius comp.
right-handed	Z+	DR+	RL
left-handed	Z+	DR-	RR
right-handed	Z-	DR-	RR
left-handed	Z-	DR+	RL
External thread			
right-handed	Z+	DR+	RR
left-handed	Z+	DR-	RL
right-handed	Z-	DR-	RL
left-handed	Z-	DR+	RR

Shape of the helix

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



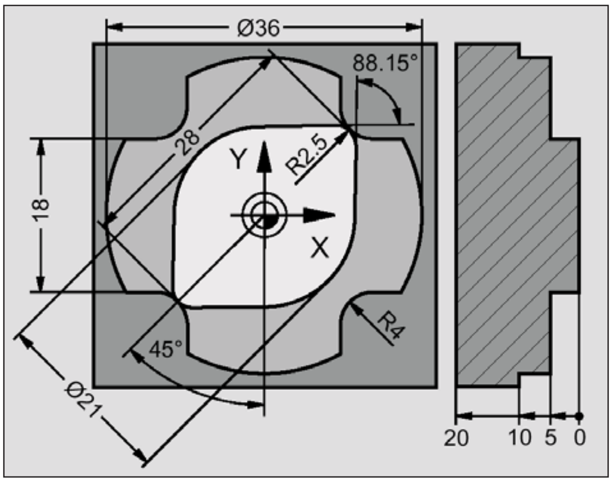
Example NC blocks
12 CC X+40 Y+25
13 L Z+0 F100 M3
14 LP PR+3 PA+270 RL F50
15 CP IPA-1800 IZ+5 DR-

Programming a helix

- **Polar coordinate angle:** Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle identify** the tool axis with an axis selection key.
- **Coordinate:** Enter the coordinate for the height of the helix in incremental dimensions.
- **Direction of rotation DR**
  Clockwise helix: DR–
  Counterclockwise helix: DR+
- **Radius compensation RL/RR/R0**
  Enter the radius compensation according to the table.

Note:
Enter the same algebraic sign for the direction of rotation DR and the incremental total angle IPA. Otherwise the tool may move in a wrong path.
For the total angle IPA you can enter a value from –5400° to +5400°. If a thread has more than 15 revolutions, you program the helix in a program section repeat.

Path contours – FK Free contour programming



Fundamentals

Workpiece drawings that are not dimensioned for NC often contain coordinate data that cannot be entered with the gray dialogue keys. 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.

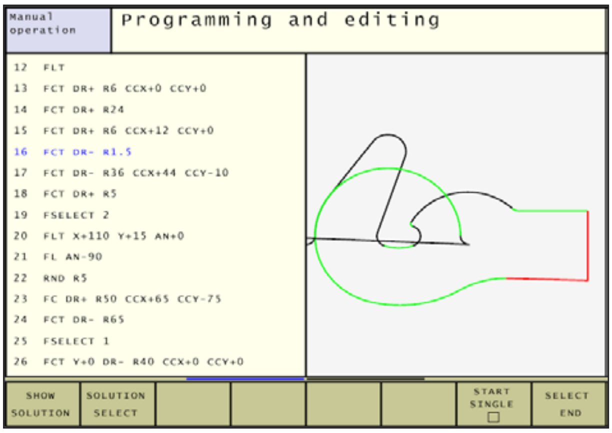
Pay attention to the following prerequisites for FK programming

- With the FK free contour programming you can only program contour elements that lie in the working plane. The working plane is defined in the first BLK FORM block of the part program.
- 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 except for elements with relative references (e.g. RX or RAN), or for elements that are referenced to other NC blocks.
- 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

Note:
Select the PGM+GRAPHICS screen layout to use graphic support during FK programming.



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

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

- Set Cursor on the green indicated element
- Press the SHOW soft key repeatedly until the correct contour element is displayed.
- If the displayed contour element matches the drawing, select it with the FSELECT 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.

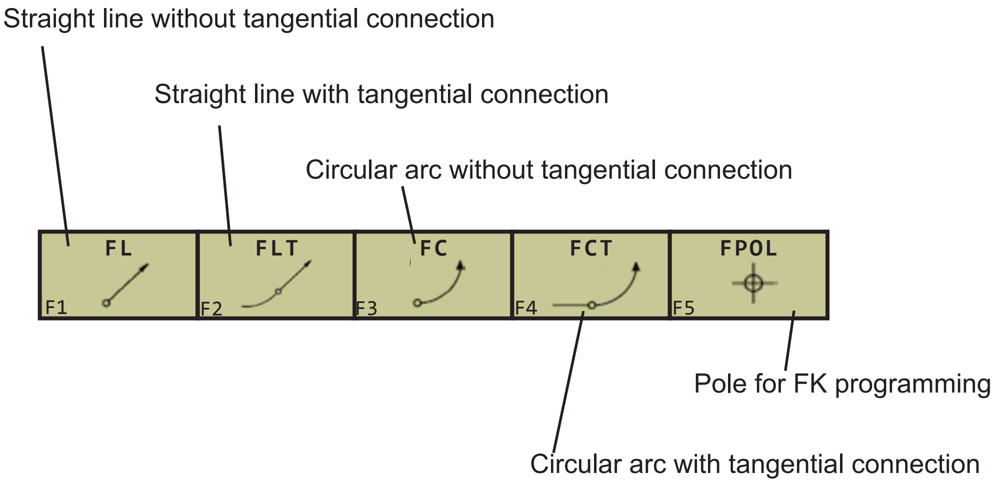
If you do not yet wish to select a green contour element, press the EDIT soft key to continue the FK dialogue.

Note:
Before a FK-program has to be run with included solution selections on an original Heidenhain control system make sure it is changed into a dialogue-program. Otherwise fixed contoures can not be overtaken correctly. PGM MGT -> More Function -> Convert FK -> H

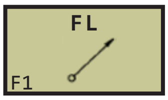
Initiating the FK dialogue

If you press the gray FK button, the WinNC shows the soft keys you can use to initiate an FK dialogue: See the following table. To deselect the soft keys, press the FK key a second time.

If you initiate the FK dialogue with one of these soft keys, the WinNC shows further soft-key rows that you can use for entering known coordinates, directional data and data regarding the course of the contour.

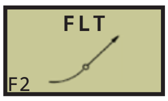


Free programming of straight lines



Straight line without tangential connection

- To display the soft keys for free contour programming, press the FK key.
- To initiate the dialogue for free programming of straight lines, press the FL soft key. The WinNC displays further soft keys.
- Enter all known data in the block by using soft keys. The FK graphic displays the programmed contour element in red until sufficient data are entered. If the entered data permit more solutions, the graphic displays the contour element in green.

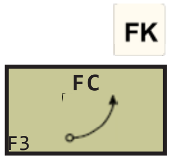


Straight line with tangential connection

If the straight line connects tangentially to another contour element, initiate the dialogue with the FLT soft key:

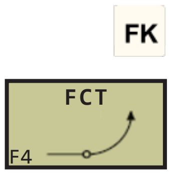
- To display the soft keys for free contour programming, press the FK key.
- To initiate the dialogue, press the FLT soft key
- Enter all known data in the block by using the soft keys

Free programming of circular arcs



Circular arc without tangential connection

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



Circular arc with tangential connection

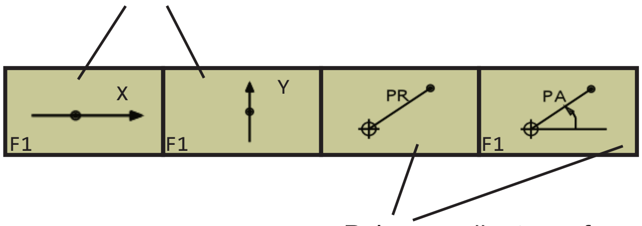
If the circular arc connects tangentially to another contour element, initiate the dialogue with the FCT soft key.

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

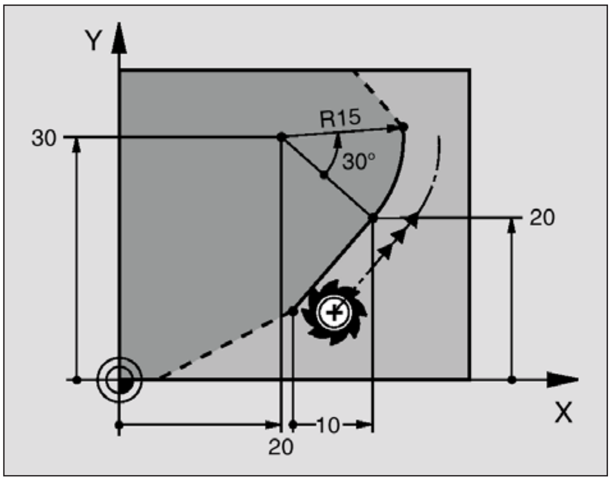
Input possibilites

End point coordinates

Cartesian coordinates X and Y



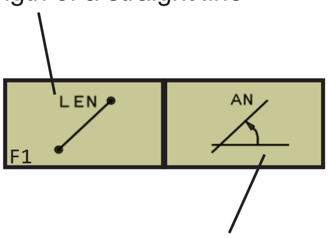
Polar coordinates referenced to FPOL



Example NC blocks
7 FPOL X+20 Y+30
8 FL IX+10 Y+20 RR F100
9 FCT PR+15 IPA+30 DR+ R15

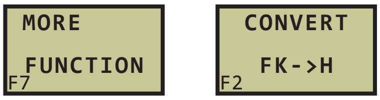
Direction and length of contour elements

Length of a straight line



Gradient angle of a straight line

Note:
LEN (chord length of the arc) and AN (gradient angle of the entry tangent) are for circuits not available.



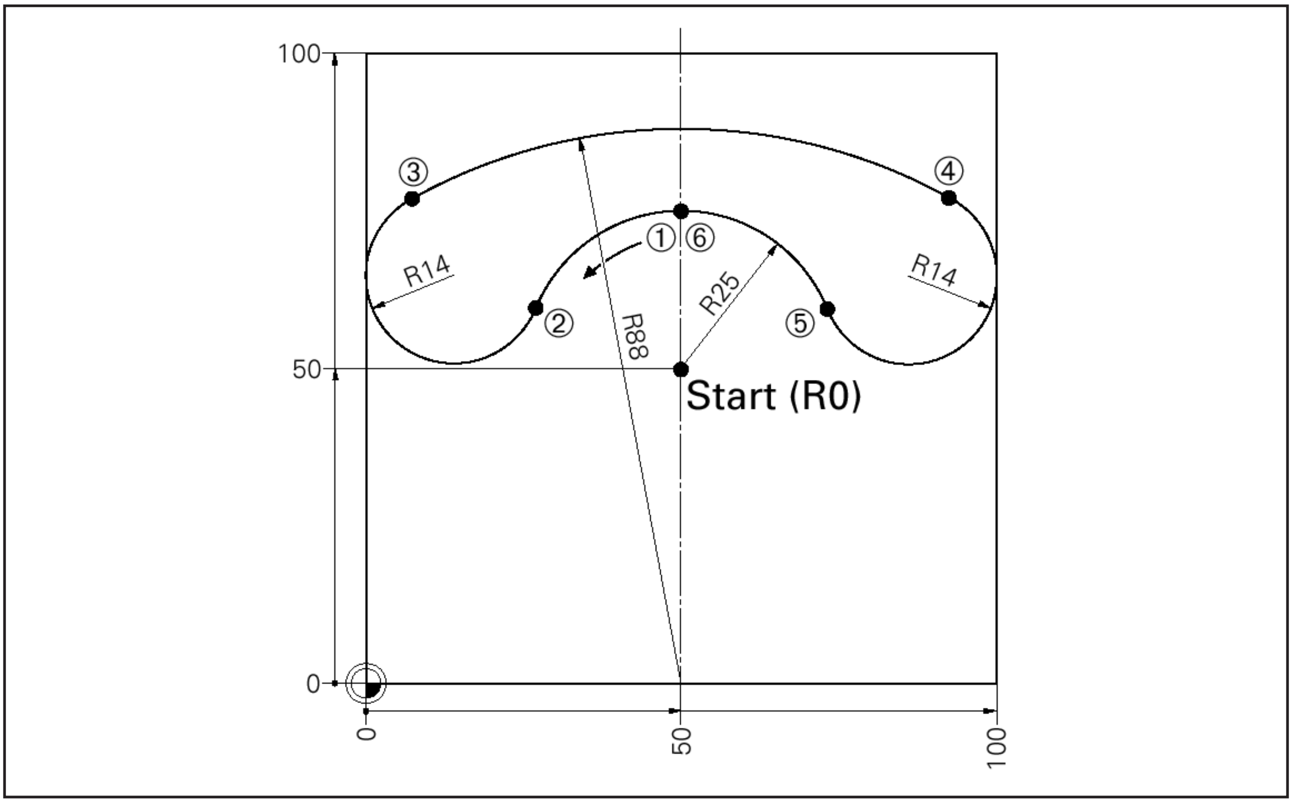
Converting FK programs

You can convert an FK program into HEIDENHAIN conversational format by using the file manager:

- Call the file manager and display the files.
- Move the highlight to the file you wish to convert.
- Press the soft keys MORE FUNCTIONS and then CONVERT FK->H. The WinNC converts all FK blocks into HEIDENHAIN dialog blocks.

Note:
Circle centers that you have entered before programming an FK contour may need to be redefined in the converted program. We recommend that you test the converted part program before executing it. FK programs with Q parameters cannot be converted.

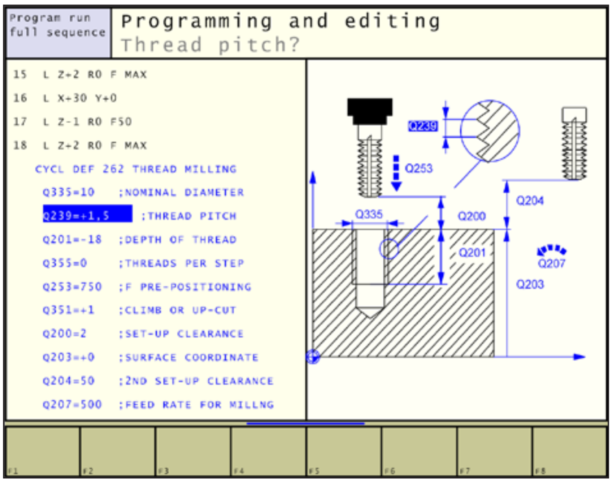
example: FK telephone



mainprogramm

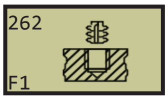
0 BEGIN PGM TELEPHONE MM
1 BLK FORM 0.1 Z X+0 Y+0 Z-10
2 BLK FORM 0.2 X+100 Y+100 Z+0 .... raw-part-definition
3 TOOL CALL 5 Z S3000 ; RADIUS 5 .... tool call
4 L Z+100 R0 F MAX M3.. secure heigth
5 L X+50 Y+50 R0 F MAX .... auxiliary point (R0)
6 L Z-2 1
7 APPR LCT X+50 Y+75 R2 RL F500 2
8 FC DR+ R25 CCX+50 CCY+50
9 FCT DR- R14.. 3
10 FSELECT 2
11 FCT DR- R88 CCX+50 CCY+0 4
12 FCT DR- R14... 5
13 FSELECT 1
14 FCT X+50 Y+75 DR+ R25 CCX+50 CCY+50 ... 6
15 DEP LCT X+50 Y+50 R2
16 L Z+100
17 END PGM TELEPHONE MM

Cycles



Working with cycles

Frequently recurring machining cycles that comprise several working steps are stored in the WinNC memory as standard cycles. Coordinate transformations and other special cycles are also provided as standard cycles.
Fixed cycles with numbers 200 and over use Q parameters as transfer parameters. Parameters with specific functions that are required in several cycles always have the same number: For example, Q200 is always assigned the setup clearance, Q202 the plunging depth, etc.



Defining a cycle using soft keys

- The soft-key row shows the available groups of cycles
- Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles
- Select the desired cycle, for example THREAD MILLING. The WinNC initiates the programming dialog and asks all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted
- Enter all parameters asked by the WinNC and conclude each entry with the ENT key 

- The WinNC terminates the dialog when all required data have been entered
- Press  for terminating the entry at any time

Example NC blocks

7 CYCL DEF 200 Drilling
Q200=2 ; SET-UP CLEARANCE
Q201=-20 ;DEPTH
Q206=150 ;FEED RATE FOR PLUNGING
Q202=5 ;PLUNGING DEPTH
Q210=0 ;DWELL TIME AT TOP
Q203=+0 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q211=0.25 ;DWELL TIME AT BOTTOM

Defining a cycle using the GOTO function

- The soft-key row shows the available groups of cycles
- Enter the cycle number and confirm with  . The WinNC then initiates the cycle dialog as described above.

Note:
If you use indirect parameter assignments in fixed cycles with numbers greater than 200 (e.g. Q210 = Q1), any change in the assigned parameter (e.g. Q1) will have no effect after the cycle definition. Define the cycle parameter (e.g. Q210) directly in such cases.

Calling a cycle

Note:
The following data must always be programmed before a cycle call:

- **BLK FORM** for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Cycle definition (CYCL DEF).

For some cycles, additional prerequisites must be observed. They are described with the individual cycle.

The following cycles become effective automatically as soon as they are defined in the part program. These cycles cannot and must not be called:

- Cycle 220 for circular and Cycle 221 for linear hole patterns
- SL Cycle 14 CONTOUR GEOMETRY
- SL Cycle 20 CONTOUR DATA
- Coordinate transformation cycles
- Cycle 9 DWELL TIME

All other cycles are called as described below:

If the WinNC is to execute the cycle once after the last programmed block, program the cycle call with the miscellaneous functionM99 or with CYCL CALL:

- To program the cycle call, press the CYCL CALL key.
- Press the CYCL CALL M soft key to enter a cycle call.
- Enter a miscellaneous function M or press END to end the dialog.

Point Tables

Function

You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.
If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.

Creating a point table

Select the **Programming and Editing** mode of operation:

PGM MGT

To call the file manager, press the PGM MGT key.

NEW.PNT

File-name?
Enter the name and file type of the point table and confirm your entry with the ENT key.

To select the unit of measure, press the MM or INCH soft key. The WinNC changes to the program blocks window and displays an empty point table.

F1 MM	F2 INCH

With the soft key LINE INSERT, insert new lines and enter the coordinates of the desired machining position.

F5 LINE INSERT

Repeat the process until all desired coordinates have been entered.

F1 X OFF/ON	F2 Y OFF/ON	F3 Z OFF/ON

Note:
With the soft keys X OFF/ON, Y OFF/ON, Z OFF/ON (second soft-key row), you can specify which coordinates you want to enter in the point table.

Selecting a point table in the program
In the Programming and Editing mode of operation, select the program for which you want to activate the point table:

PGM CALL

Press the PGM CALL key to call the function for selecting the point table.

PATTERN TABLE F3

Press the PATTERN TABLE soft key.

Enter the name of the point table and confirm your entry with the ENT key. If the point table is not stored in the same directory as the NC program, you must enter the complete path.

Example NC block
7 SEL PATTERN “TNC:\DIRKT5\MUST35.PNT“

Effect of the point tables with Cycles 1 to 5, 17 and 18
The WinNC interprets the points of the working plane as coordinates of the hole centers. The coordinate of the spindle axis defines the upper surface of the workpiece, so the WinNC can pre-position automatically (first in the working plane, then in the spindle axis).

Effect of the point tables with SL cycles and Cycle 12
The WinNC interprets the points as an additional datum shift.

Effect of the point tables with Cycles 200 to 208 and 262 to 267
The WinNC interprets the points of the working plane as coordinates of
the hole centers. If you want to use the coordinate defined in the point
table for the spindle axis as the starting point coordinate, you must
define the workpiece surface coordinate (Q203) as 0.

Effect of the point tables with Cycles 210 to 215
The WinNC interprets the points as an additional datum shift. If you want
to use the points defined in the point table as starting-point
coordinates, you must define the starting points and the workpiece
surface coordinate (Q203) in the respective milling cycle as 0.

Calling a cycle in connection with point tables

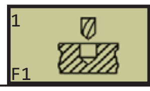
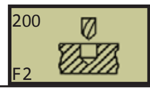
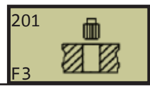
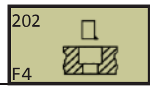
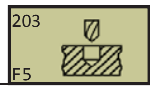
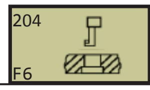
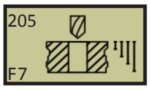
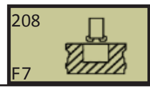
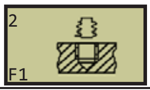
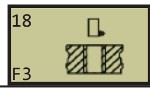
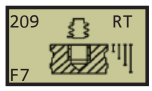
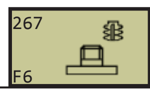
Note:
With CYCL CALL PAT the WinNC runs the points table that you last defined (even if you have defined the point table in a program that was nested with **CALL PGM**. The WinNC uses the coordinate in the spindle axis as the clearance height, where the tool is located during cycle call. A clearance height or 2nd setup clearance that is defined separately in a cycle must not be greater than the clearance height defined in the global pattern.

If you want the WinNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with **CYCLE CALL PAT**:

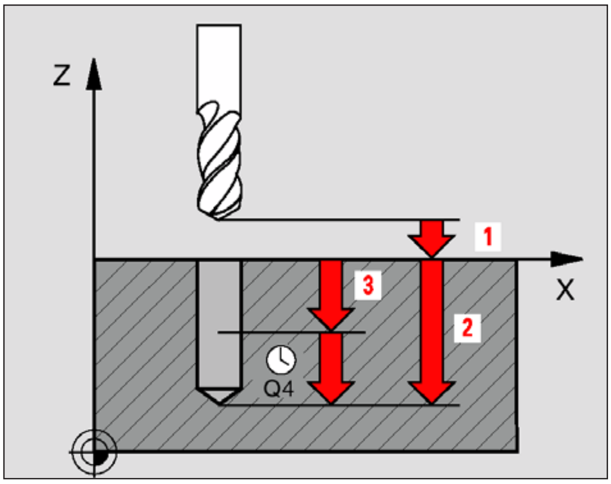
- To program the cycle call, press the CYCL CALL key.
- Press the CYCL CALL PAT soft key to call a point table.
- Enter the feed rate at which the WinNC is to move from point to point (if you make no entry the WinNC will move at the last programmed feed rate, FMAX not valid).
- If required, enter miscellaneous function M, then confirm with the END key.

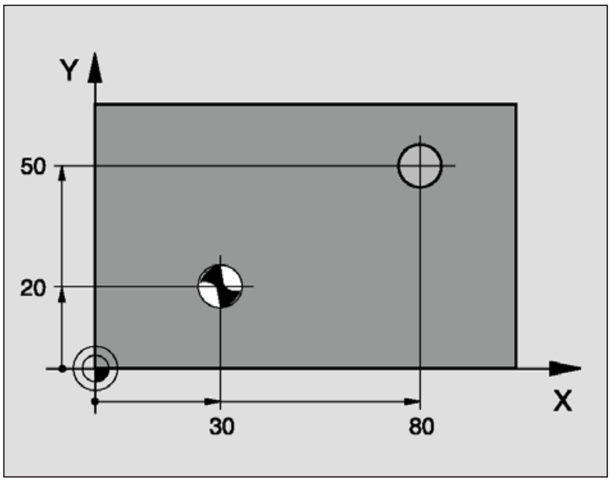
The WinNC moves the tool back to the safe height over each successive starting point (safe height = the spindle axis coordinate for cycle call). To use this procedure also for the cycles number 200 and greater, you must define the 2nd setup clearance (Q204) as 0. If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103 (see "Feed rate factor for plunging movements: M103" on page 169).

Cycles for Drilling, Tapping and Thread Milling

Cycle	Soft key
1 PECKING Without automatic pre-positioning	
200 DRILLING With automatic pre-positioning, 2nd set-up clearance	
201 REAMING With automatic pre-positioning, 2nd set-up clearance	
202 BORING With automatic pre-positioning, 2nd set-up clearance	
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrement	
204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	
205 UNIVERSAL PECKING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and advanced stop distance	
208 BORE MILLING With automatic pre-positioning, 2nd set-up clearance	
2 TAPPING With a floating tap holder	
17 RIGID TAPPING Without a floating tap holder	
18 THREAD CUTTING	
206 TAPPING NEW With a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	
207 RIGID TAPPING NEW Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance	
209 TAPPING WITH CHIP BREAKING Without a floating tap holder, with automatic pre-positioning, 2nd set-up clearance, chip breaking	
262 THREAD MILLING Cycle for milling a thread in pre-drilled material	
263 THREAD MILLING/COUNTERSINKING Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	
264 THREAD DRILLING/MILLING Cycle for drilling into the solid material with subsequent milling of the thread with a tool	
265 HELICAL THREAD DRILLING/MILLING Cycle for milling the thread into the solid material	
267 OUTSIDE THREAD MILLING Cycle for milling an external thread and machining a countersunk chamfer	

PECKING (Cycle 1)





Example: NC blocks

5 L Z+100 R0 FMAX
6 CYCL DEF 1.0 PECKING
7 CYCL DEF 1.1 SET UP 2
8 CYCL DEF 1.2 DEPTH -15
9 CYCL DEF 1.3 PECKG 7.5
10 CYCL DEF 1.4 V.DWELL 1
11 CYCL DEF 1.5 F80
12 L X+30 Y+20 FMAX M3
13 L Z+2 FMAX M99
14 L X+80 Y+50 FMAX M99
15 L Z+100 FMAX M2

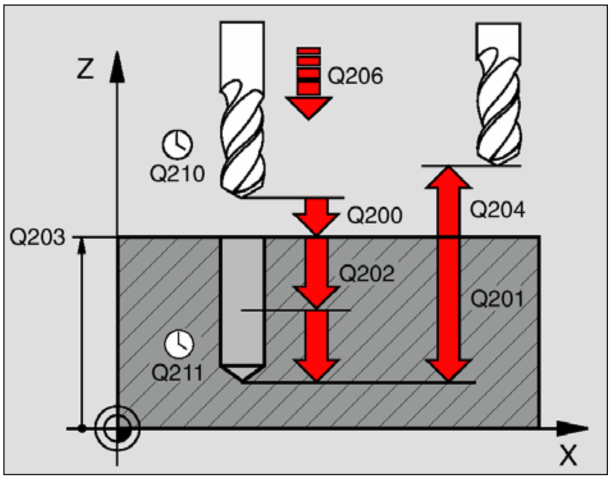
1 The tool drills from the current position to the first plunging depth at the programmed feed rate F.
2 When it reaches the first plunging depth, the tool retracts in rapid traverse FMAX to the starting position and advances again to the first plunging depth minus the advanced stop distance t.
3 The advanced stop distance is automatically calculated by the control:

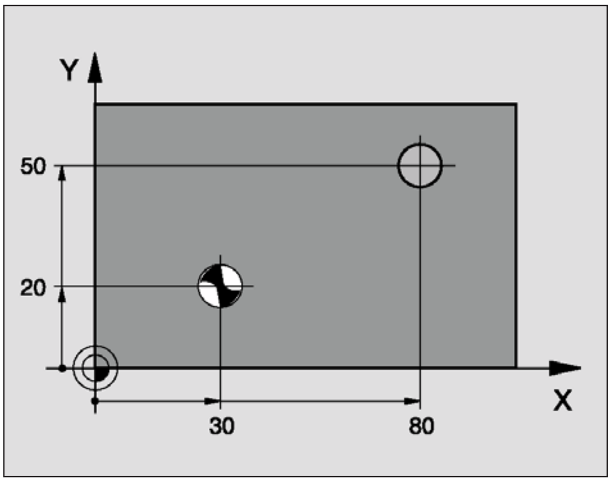
- At a total hole depth of up to 30 mm: t = 0.6 mm
- At a total hole depth exceeding 30 mm: t = hole depth / 50
- Maximum advanced stop distance: 7 mm

4 The tool then advances with another infeed at the programmed feed rate F.
5 The WinNC repeats this process (1 to 4) until the programmed total hole depth is reached.
6 After a dwell time at the hole bottom, the tool is returned to the starting position in rapid traverse FMAX for chip breaking.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface). The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.

- **1.1 Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **1.2 Depth 2** (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **1.3 Plunging depth 3** (incremental value): Infeed per cut The total hole depth does not have to be a multiple of the plunging depth. The tool will drill to the total hole depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the total hole depth
- **1.4 Dwell time in seconds 4**: Amount of time the tool remains at the total hole depth for chip breaking
- **1.5 Feed rate F**: Traversing speed of the tool during drilling in mm/min

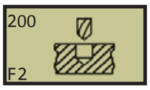




DRILLING (Cycle 200)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the set-up clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 The WinNC returns the tool at FMAX to the setup clearance, dwells there (if a dwell time was entered), and then moves at FMAX to the setup clearance above the first plunging depth.
4 The tool then advances with another infeed at the programmed feed rate F.
5 The WinNC repeats this process (2 to 4) until the programmed depth is reached.
6 At the hole bottom, the tool path is retraced to set-up clearance or—if programmed—to the 2nd set-up clearance in rapid traverse FMAX.

Note:
Before programming, note the following:
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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.

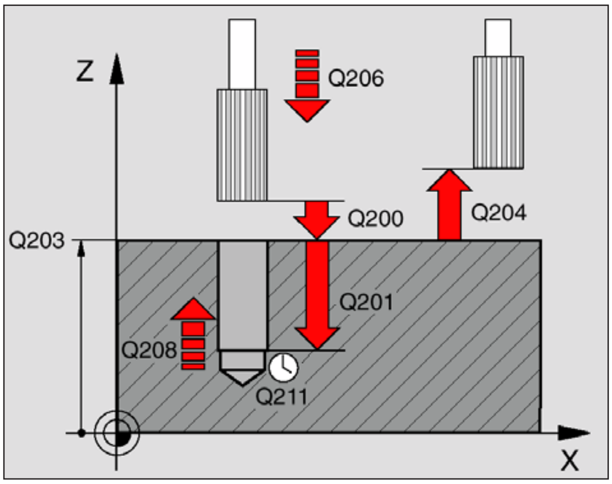


Example: NC blocks

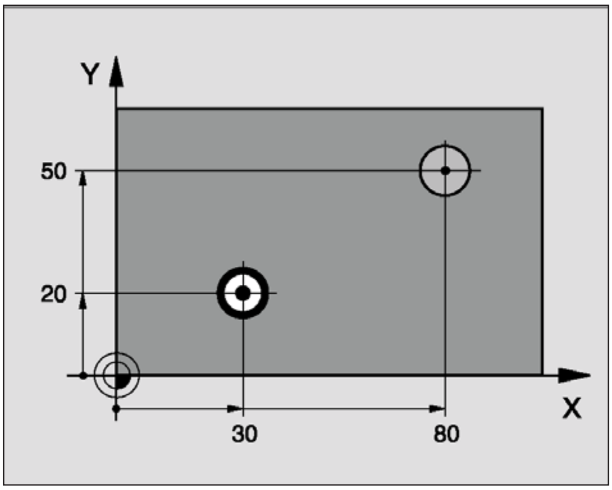
```
10 L Z+100 R0 FMAX
11 CYCL DEF 200 DRILLING
    Q200 = 2 ;SET-UP CLEARANCE
    Q201 = -15 ;DEPTH
    Q206 = 250 ;FEED RATE FOR PLUNGING
    Q202 = 5 ;PLUNGING DEPTH
    Q210 = 0 ;DWELL TIME AT TOP
    Q203 = +20 ;SURFACE COORDINATE
    Q204 = 100 ;2ND SET-UP CLEARANCE
    Q211 = 0.1 ;DWELL TIME AT BOTTOM
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
15 L Z+100 FMAX M2
```

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min
- **Plunging depth** Q202 (incremental value): Infeed per cut The depth does not have to be a multiple of the plunging depth. The WinNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip release.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom

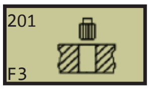
REAMING (Cycle 201)



1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.
2 The tool reams to the entered depth at the programmed feed rate F.
3 If programmed, the tool remains at the hole bottom (Q201) for the entered dwell time.
4 The tool then retracts to set-up clearance at the feed rate F, and from there—if programmed—to the 2nd set-up clearance in FMAX.



> **Note:**
> Before programming, note the following:
> 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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.

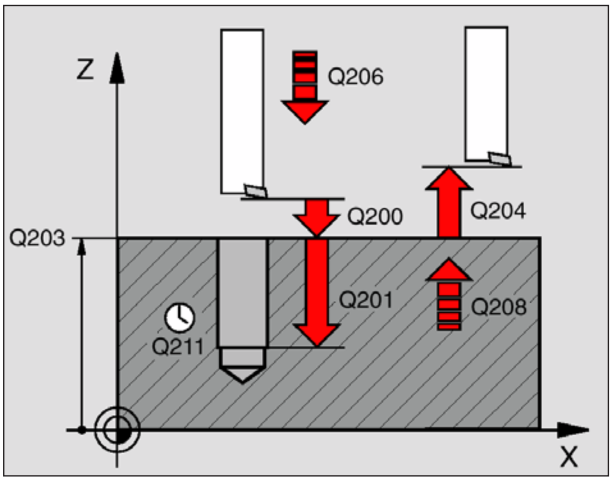


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole
- **Feed rate for plunging** Q206: Traversing speed of the tool during reaming in mm/min
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the plunging feed rate.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

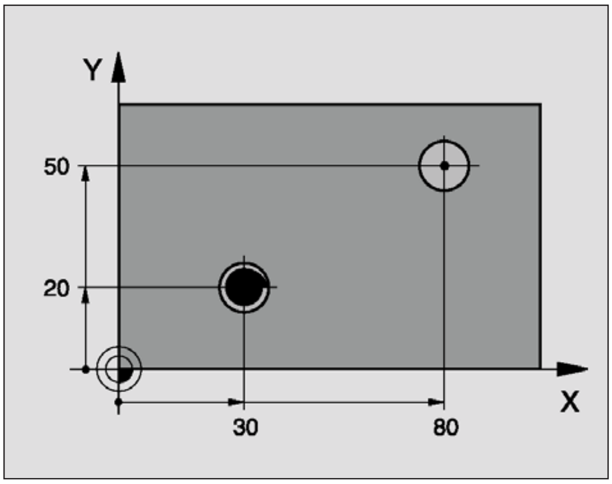
Example: NC blocks

```
10 L Z+100 R0 FMAX
11 CYCL DEF 201 REAMING
   Q200 = 2 ;SET-UP CLEARANCE
   Q201 = -15 ;DEPTH
   Q206 = 100 ;FEED RATE FOR PLUNGING
   Q211 = 0.5 ;DWELL TIME AT BOTTOM
   Q208 = 250 ;RETRACTION FEED TIME
   Q203 = +20 ;SURFACE COORDINATE
   Q204 = 100 ;2. SET-UP CLEARANCE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M9
15 L Z+100 FMAX M2
```

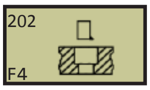
BORING (Cycle 202)



1 The WinNC positions the tool in the tool axis at rapid traverse FMAXto the set-up clearance above the workpiece surface.
2 The tool drills to the programmed depth at the feed rate for plunging.
3 If programmed, the tool remains at the hole bottom (Q201) for the entered dwell time with active spindle rotation for cutting free.
4 The WinNC then orients the spindle to the 0° position with an oriented spindle stop.
5 If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
6 The WinNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance with FMAX. If Q214=0 the tool point remains on the wall of the hole.



Note:
Before programming, note the following:
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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. After the cycle is completed, the WinNC restores the coolant and spindle conditions that were active before the cycle call.



Example:

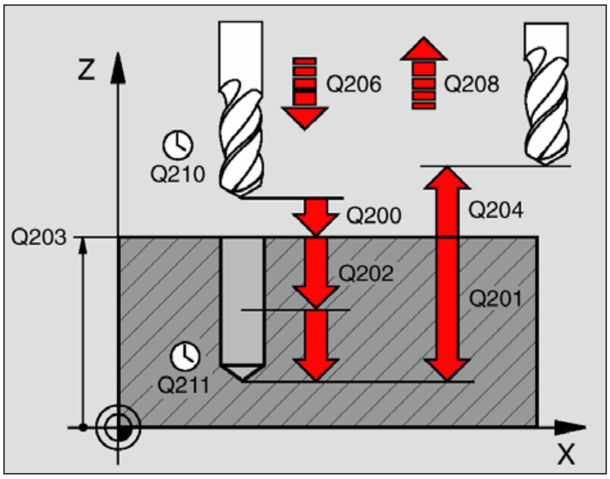
```
10 L Z+100 R0 FMAX
11 CYCL DEF 202 BORING
    Q200 = 2 ;SET-UP CLEARANCE
    Q201 = -15 ;DEPTH
    Q206 = 100 ;FEED RATE FOR PLUNGING
    Q211 = 0.5 ;DWELL TIME AT BOTTOM
    Q208 = 250 ;RETRACTION FEED TIME
    Q203 = +20 ;SURFACE COORDINATE
    Q204 = 100 ;2ND SET-UP CLEARANCE
    Q214 = 1 ;DISENGAGING DIRECTN
    Q336 = 0 ;ANGLE OF SPINDLE
12 L X+30 Y+20 FMAX M3
13 CYCL CALL
14 L X+80 Y+50 FMAX M99
```

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole
- **Feed rate for plunging** Q206: Traversing speed of the tool during boring in mm/min
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at feed rate for plunging.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Disengaging direction** (0/1/2/3/4) Q214: Determine the direction in which the WinNC retracts the tool at the hole bottom (after spindle orientation).
  0 Do not retract tool
  1 Retract tool in the negative reference axis direction
  2 Retract tool in the negative secondary axis direction
  3 Retract tool in the positive reference axis direction
  4 Retract tool in the positive secondary axis direction

Danger of collision!

Select a disengaging direction in which the tool moves away from the edge of the hole. Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis.

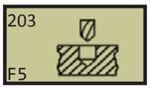
- **Angle for spindle orientation** Q336 (absolute value): Angle at which the WinNC positions the tool before retracting it.



UNIVERSAL DRILLING (Cycle 203)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool retracts at the retraction feed rate to setup clearance, remains there — if programmed — for the entered dwell time, and advances again in FMAX to the setup clearance above the first PLUNGING DEPTH.
4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5 The WinNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6 The tool remains at the hole bottom — if programmed — for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. and — if programmed — to the 2nd set-up clearance with FMAX.

Note:
Before programming, note the following:
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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.

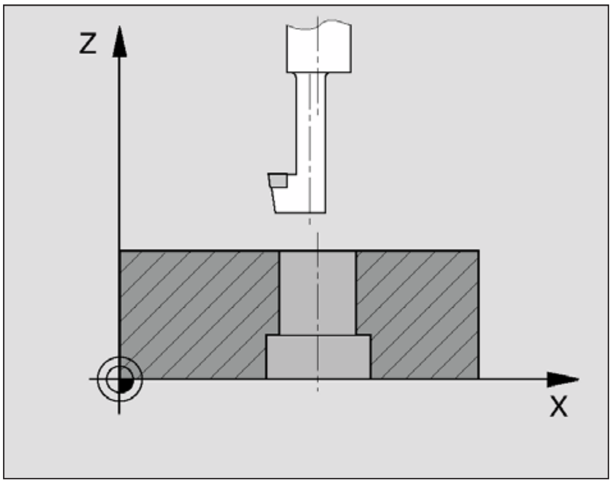


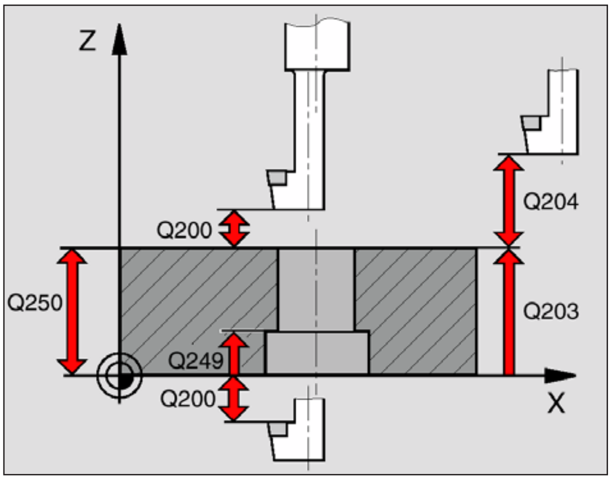
Example: NC blocks

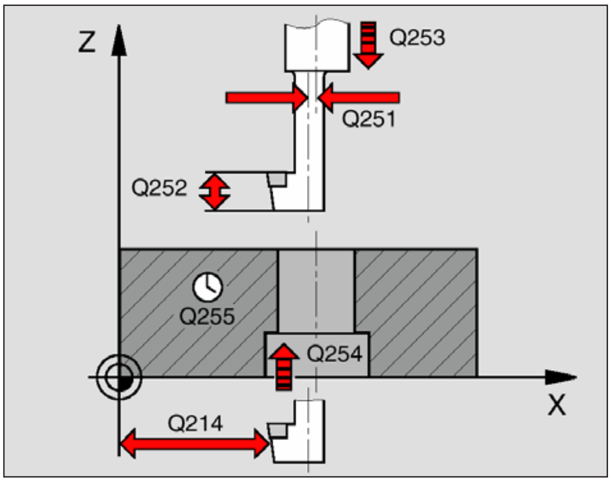
11 CYCL DEF 203 UNIVERSAL DRILLING
Q200=2 ;SET-UP CLEARANCE
Q201=-20 ;DEPTH
Q206=150 ;FEED RATE FOR PLUNGING
Q202=5 ;PLUNGING DEPTH
Q210=0 ;DWELL TIME AT TOP
Q203=+20 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q212=0.2 ;DECREMENT
Q213=3 ;BREAKS
Q205=3 ;MIN. PLUNGING DEPTH
Q211=0.25 ;DWELL TIME AT BOTTOM
Q208=500 ;RETRACTION FEED RATE
Q256=0.2 ;DIST FOR CHIP BRKNG

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min
- **Plunging depth** Q202 (incremental value): Infeed per cut The depth does not have to be a multiple of the plunging depth. The WinNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Dwell time at top** Q210: Time in seconds that the tool remains at set-up clearance after having been retracted from the hole for chip release.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Decrement** Q212 (incremental value): Value by which the WinNC decreases the plunging depth Q202 after each infeed.
- **No. of breaks before retracting** Q213: Number of chip breaks after which the WinNC is to withdraw the tool from the hole for chip release. For chip breaking, the WinNC retracts the tool each time by the value Q256.
- **Minimum plunging depth** Q205 (incremental value): If you have entered a decrement, the WinNC limits the plunging depth to the value entered with Q205.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the feed rate in Q206.
- **Retraction rate for chip breaking** Q256 (incremental value): Value by which the WinNC retracts the tool during chip breaking

BACK BORING (CYCLE 204)



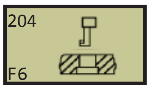




This cycle allows holes to be bored from the underside of the workpiece.

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the 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 setup 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 WinNC moves the tool at the pre-positioning feed rate to the setup clearance and then, if entered, to the 2nd setup clearance with FMAX.

Note:
Before programming, note the following:
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.



Example: NC blocks

11 CYCL DEF 204 BACK BORING
Q200=2 ;SET-UP CLEARANCE
Q249=+5 ;DEPTH OF COUNTERBORE
Q250=20 ;MATERIAL THICKNESS
Q251=3.5 ;OFF-CENTER DISTANCE
Q252=15 ;TOOL EDGE HEIGHT
Q253=750 ;F PRE-POSITIONING
Q254=200 ;F COUNTERBORING
Q255=0 ;DWELL TIME
Q203=+20 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q214=1 ;DISENGAGING DIRECTN
Q336=0 ;ANGLE OF SPINDLE

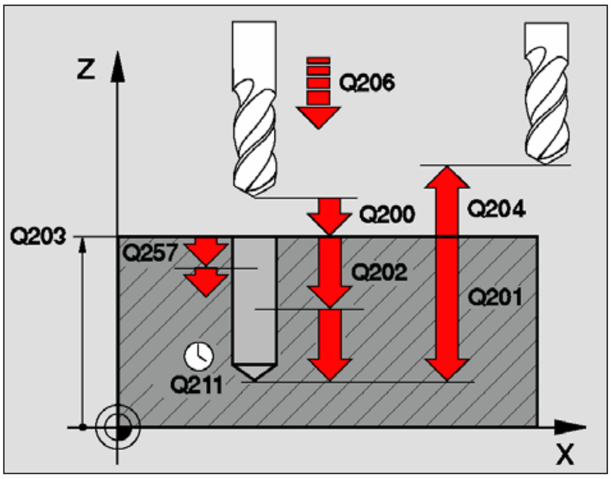
Danger of collision:

Check the position of the tool tip when you program a spindle orientation to the angle that you enter in Q336 (for example, in the Positioning with Manual Data Input mode of operation). Set the angle so that the tool tip is parallel to a coordinate axis. Select a disengaging direction in which the tool moves away from the edge of the hole.

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth of counterbore** Q249 (incremental value): 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.
- **Material thickness** Q250 (incremental value): Thickness of the workpiece
- **Off-center distance** Q251 (incremental value): Off-center distance for the boring bar; value from tool data sheet
- **Tool edge height** Q252 (incremental value): Distance between the underside of the boring bar and the main cutting tooth; value from tool data sheet
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min
- **Dwell time** Q255: Dwell time in seconds at the top of the bore hole
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Disengaging direction (0/1/2/3/4)** Q214: Determine the direction in which the WinNC displaces the tool by the off-center distance (after spindle orientation).

  Example:

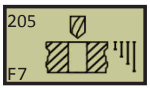
  1 Retract tool in the negative reference axis direction
  2 Retract tool in the negative secondary axis direction
  3 Retract tool in the positive reference axis direction
  4 Retract tool in the positive secondary axis direction
- **Angle for spindle orientation** Q336 (absolute value): Angle at which the WinNC positions the tool before it is plunged into or retracted from the bore hole.



UNIVERSAL PECKING (Cycle 205)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.
2 The tool drills to the first plunging depth at the programmed feed rate F.
3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to setup clearance and then at FMAX to the entered starting position above the first plunging depth.
4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.
5 The WinNC repeats this process (2 to 4) until the programmed total hole depth is reached.
6 The tool remains at the hole bottom — if programmed — for the entered dwell time to cut free, and then retracts to set-up clearance at the retraction feed rate. and — if programmed — to the 2nd set-up clearance with FMAX.

Note:
Before programming, note the following:
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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.



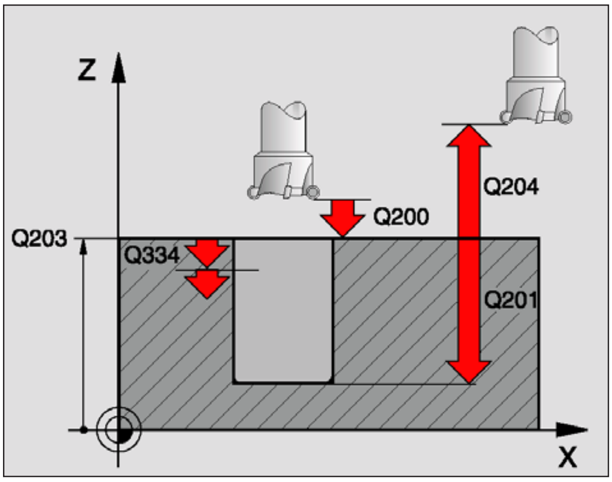
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min
- **Plunging depth** Q202 (incremental value): Infeed per cut The depth does not have to be a multiple of the plunging depth. The WinNC will go to depth in one movement if:
    - the plunging depth is equal to the depth
    - the plunging depth is greater than the depth
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Decrement** Q212 (incremental value): Value by which the WinNC decreases the plunging depth Q202.
- **Minimum plunging depth** Q205 (incremental value): If you have entered a decrement, the WinNC limits the plunging depth to the value entered with Q205.
- **Upper advanced stop distance** Q258 (incremental value): Setup clearance for rapid traverse positioning when the WinNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth
- **Lower advanced stop distance** Q259 (incremental value): Setup clearance for rapid traverse positioning when the WinNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth

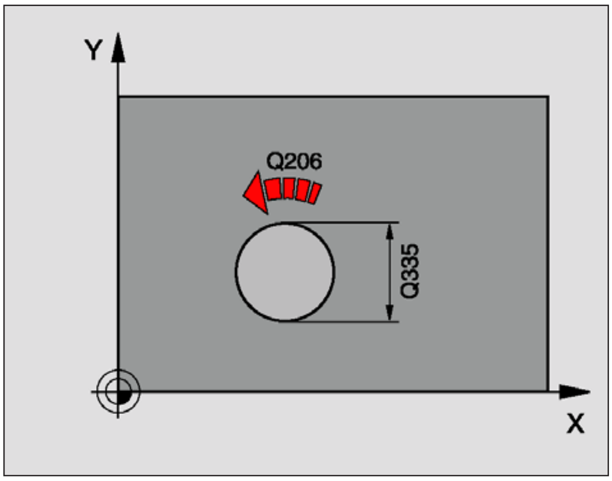
Example: NC blocks

11 CYCL DEF 205 UNIVERSAL PECKING
Q200=2 ;SET-UP CLEARANCE
Q201=-80 ;DEPTH
Q206=150 ;FEED RATE FOR PLUNGING
Q202=5 ;PLUNGING DEPTH
Q203=+100 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q212=0.5 ;DECREMENT
Q205=3 ;MIN. PLUNGING DEPTH
Q258=0.5 ;UPPER ADV STOP DIST
Q259=1 ;LOWER ADV STOP DIST
Q257=5 ;DEPTH FOR CHIP BRKNG
Q256=0.2 ;DIST FOR CHIP BRKNG
Q211=0.25 ;DWELL TIME AT BOTTOM

Note:
If you enter Q258 not equal to Q259, the WinNC will change the advance stop distances between the first and last plunging depths at the same rate.

- **Infeed depth for chip breaking** Q257 (incremental value): Depth at which the WinNC carries out chip breaking. There is no chip breaking if 0 is entered.
- **Retraction rate for chip breaking** Q256 (incremental value): Value by which the WinNC retracts the tool during chip breaking
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom





BORE MILLING (Cycle 208)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface and then moves the tool to the bore hole circumference on a rounded arc (if enough space is available).

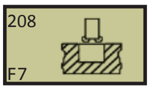
2 The tool mills in a helix from the current position to the first plunging depth at the programmed feed rate.

3 When the drilling depth is reached, the WinNC once again traverses a full circle to remove the material remaining after the initial plunge.

4 The WinNC then positions the tool at the center of the hole again.

5 Finally the WinNC returns to the setup clearance at FMAX. and — if programmed — to the 2nd set-up clearance with FMAX.

Note:
Before programming, note the following:
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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. If you have entered the bore hole diameter to be the same as the tool diameter, the WinNC will bore directly to the entered depth without any helical interpolation.



- **Set-up clearance** Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole
- **Feed rate for plunging** Q206: Traversing speed of the tool during helical drilling in mm/min
- **Infeed per helix** Q334 (incremental value): Depth of the tool plunge with each helix (=360°)

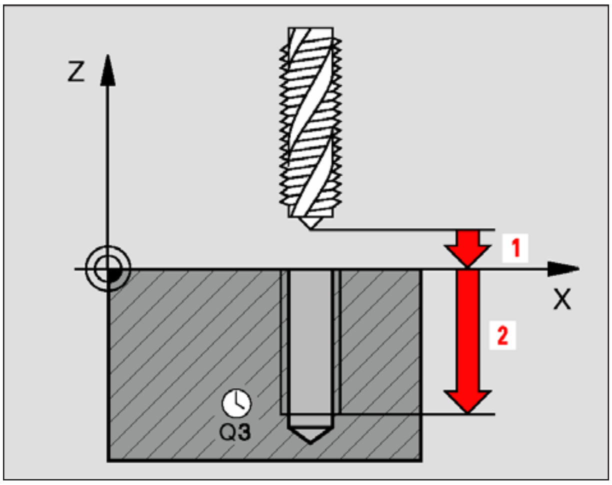
Example: NC blocks

```
12 CYCL DEF 208 BORE MILLING
   Q200=2 ;SET-UP CLEARANCE
   Q201=-80 ;DEPTH
   Q206=150 ;FEED RATE FOR PLUNGING
   Q334=1.5 ;PLUNGING DEPTH
   Q203=+100 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q335=25 ;NOMINAL DIAMETER
   Q342=0 ;ROUGHING DIAMETER
```

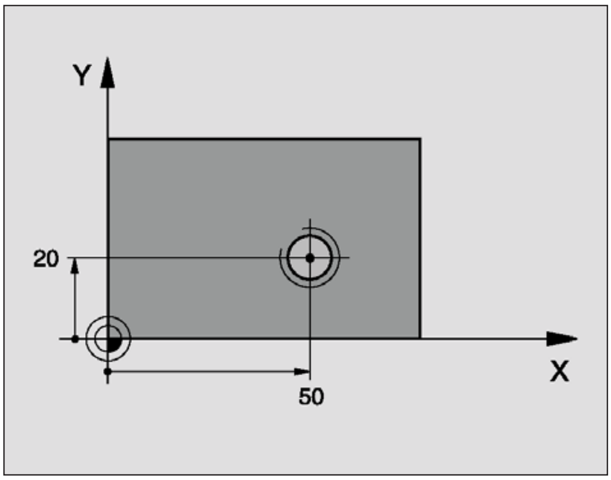
Note:
Note that if the infeed distance is too large, the tool or the workpiece may be damaged.

- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Nominal diameter** Q335. (absolute value): Borehole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the WinNC will bore directly to the entered depth without any helical interpolation.
- **Roughing diameter** Q342 (absolute value): As soon as you enter a value greater than 0 in Q342, the WinNC no longer checks the ratio between the nominal diameter and the tool diameter. This allows you to rough-mill holes whose diameter is more than twice as large as the tool diameter.

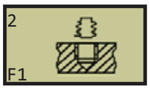
TAPPING with a floating tap holder (Cycle 2)



1 The tool drills to the total hole depth in one movement.
2 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the dwell time.
3 At the starting position, the direction of spindle rotation reverses once again.



Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface). The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process. When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual). For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

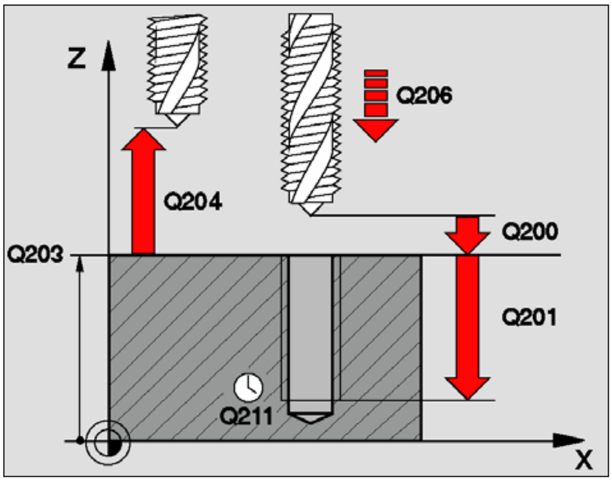


- **2.1 Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- **2.2 Total hole depth 2** (thread length, incremental value): Distance between workpiece surface and end of thread
- **2.3 Dwell time in seconds**: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.
- **2.4 Feed rate F**: Traversing speed of the tool during tapping

Example: NC blocks

```
24 L Z+100 R0 FMAX
25 CYCL DEF 2.0 TAPPING
26 CYCL DEF 2.1 SET UP 3
27 CYCL DEF 2.2 DEPTH -20
28 CYCL DEF 2.3 DWELL 0,4
29 CYCL DEF 2.4 F100
30 L X+50 Y+20 FMAX M3
31 L Z+3 FMAX M99
```

The feed rate is calculated as follows: F = S x p
F Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)



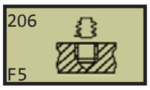
TAPPING NEW with floating tap holder (Cycle 206)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.
2 The tool drills to the total hole depth in one movement.
3 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. and — if programmed — to the 2nd set-up clearance with FMAX.
4 At the set-up clearance, the direction of spindle rotation reverses once again.

Note:
Before programming, note the following:
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. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process. When a cycle is being run, the spindle speed override knob is disabled. The feed rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual). For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

Example: NC blocks

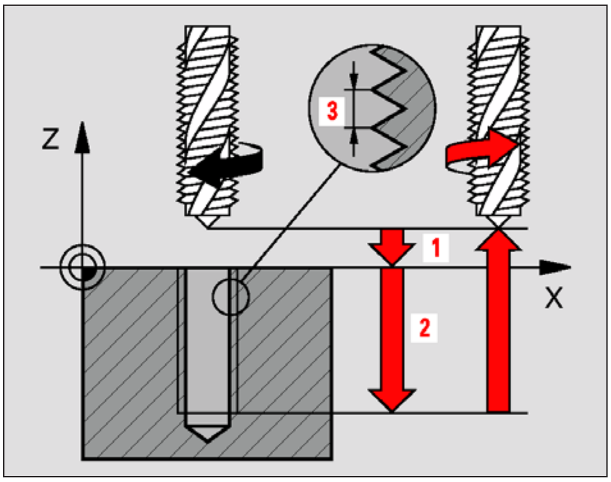
25 CYCL DEF 206 TAPPING NEW
Q200=2 ;SET-UP CLEARANCE
Q201=-20 ;DEPTH
Q206=150 ;FEED RATE FOR PLUNGING
Q211=0.25 ;DWELL TIME AT BOTTOM
Q203=+25 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE



- **Set-up clearance** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- **Total hole depth** Q201 (thread length, incremental value): Distance between workpiece surface and end of thread
- **Feed rate F** Q206: Traversing speed of the tool during tapping
- **Dwell time at bottom** Q211: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

The feed rate is calculated as follows: F = S x p
F Feed rate (mm/min)
S: Spindle speed (rpm)
p: Thread pitch (mm)

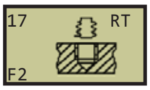
RIGID TAPPING (Cycle 17)



The WinNC cuts the thread without a floating tap holder in one or more passes.
Rigid tapping offers the following advantages over tapping with a floating tap holder:

- Higher machining speeds possible
- Repeated tapping of the same thread is possible; repetitions are enabled via spindle orientation to the 0° position during cycle call
- Increased traverse range of the spindle axis due to absence of a floating tap holder.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface). The algebraic sign for the parameter total hole depth determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The WinNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted. The feed-rate override knob is disabled. At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

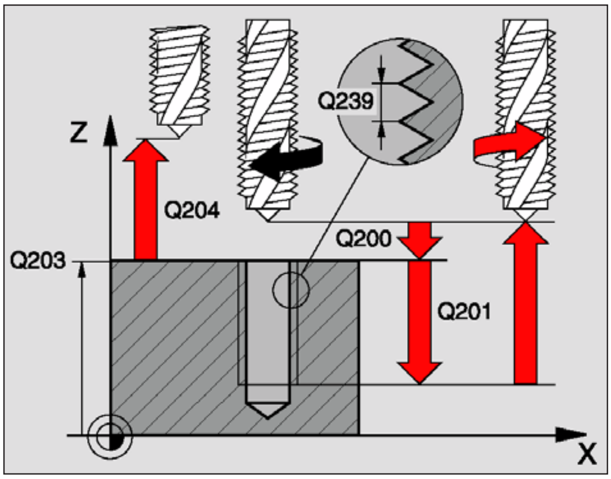


Example: NC blocks

18 CYCL DEF 17.0 RIGID TAPPING GS
19 CYCL DEF 17,1 SET UP 2
20 CYCL DEF 17.2 DEPTH -20
21 CYCL DEF 17,3 PITCH +1

- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Total hole depth 2** (incremental value): Distance between workpiece surface (beginning of thread) and end of thread
- **Pitch 3**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread

RIGID TAPPING without a floating tap holder TAPPING (Cycle 207)

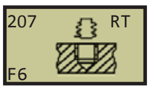


The WinNC cuts the thread without a floating tap holder in one or more passes.
Rigid tapping offers the following advantages over tapping with a floating tap holder:See ”RIGID TAPPING (Cycle 17)”.

1. The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.
2. The tool drills to the total hole depth in one movement.
3. Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the dwell time. and — if programmed — to the 2nd set-up clearance with FMAX.
4. The WinNC stops the spindle turning at set-up clearance.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign for the parameter total hole depth determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The WinNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted. The feed-rate override knob is disabled. At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

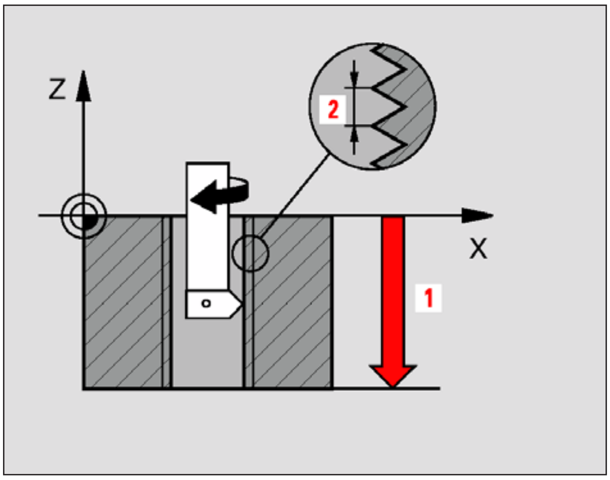
- **Set-up clearance** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Total hole depth** Q201 (incremental value): Distance between workpiece surface and end of thread
- **Pitch** Q239 Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.



Example: NC blocks

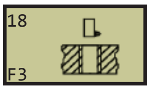
```
26 CYCL DEF 207 RIGID TAPPING NEW
   Q200=2 ;SET-UP CLEARANCE
   Q201=-20 ;DEPTH
   Q239=+1 ;THREAD PITCH
   Q203=+25 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
```

THREAD CUTTING (CYCLE 18)



Cycle 18 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign for the parameter thread depth determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The WinNC calculates the feed rate from the spindle speed. If the spindle speed override is used during thread cutting, the feed rate is automatically adjusted. The feed-rate override knob is disabled. The WinNC automatically activates and deactivates spindle rotation. Do not program M3 or M4 before cycle call.



- **Total hole depth 1**: Distance between current tool position and end of thread The algebraic sign for the total hole depth determines the working direction (a negative value means a negative working direction in the tool axis)
- **Pitch 2**: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread (M3 with negative depth)
  – = left-hand thread (M4 with negative depth)

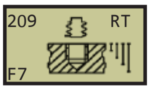
Example: NC blocks
22 CYCL DEF 18.0 THREAD CUTTING
23 CYCL DEF 18,1 DEPTH -20
24 CYCL DEF 18.2 PITCH +1

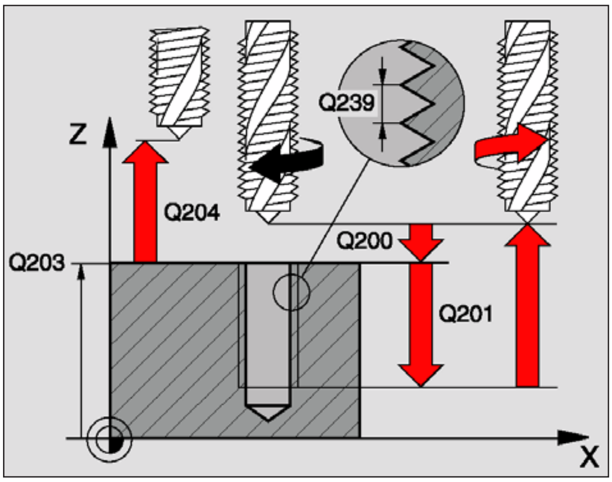
TAPPING WITH CHIP BREAKING (Cycle 209)

The tool machines the thread in several passes until it reaches the programmed depth. You can define in a parameter whether the tool is to be retracted completely from the hole for chip breaking.

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface. There it carries out an oriented spindle stop.
2 The tool moves to the programmed infeed depth, reverses the direction of spindle rotation and retracts by a specific distance or completely for chip release, depending on the definition .
3 It then reverses the direction of spindle rotation again and advances to the next infeed depth.
4 The WinNC repeats this process (2 to 3) until the programmed thread depth is reached.
5 The tool is then retracted to set-up clearance. and — if programmed — to the 2nd set-up clearance with FMAX.
6 The WinNC stops the spindle turning at set-up clearance.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign for the parameter thread depth determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The WinNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted. The feed-rate override knob is disabled. At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).





Example: NC blocks

26 CYCL DEF 209 TAPPING W/ CHIP BRKG
Q200=2 ;SET-UP CLEARANCE
Q201=-20 ;DEPTH
Q239=+1 ;THREAD PITCH
Q203=+25 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q257=5 ;DEPTH FOR CHIP BRKNG
Q256=+25 ;DIST FOR CHIP BRKNG
Q336=50 ;ANGLE OF SPINDLE

- **Set-up clearance** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and end of thread
- **Pitch** Q239 Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Infeed depth for chip breaking** Q257 (incremental value): Depth at which TNC carries out chip breaking
- **Retraction rate for chip breaking** Q256: The WinNC multiplies the pitch Q239 by the programmed value and retracts the tool by the calculated value during chip breaking. If you enter Q256 = 0, the WinNC retracts the tool completely from the hole (to set-up clearance) for chip release.
- **Angle for spindle orientation** Q336 (absolute value): Angle at which the WinNC positions the tool before machining the thread. This allows you to regroove the thread, as required.

Fundamentals of thread milling

Prerequisites

- Thread milling usually leads to distortions of the thread profile. To correct this effect, you need tool-specific compensation values which are given in the tool catalog or are available from the tool manufacturer. You program the compensation with the delta value for the tool radius DR in the tool call .
- The Cycles 262, 263, 264 and 267 can only be used with rightward rotating tools. For Cycle 265, you can use rightward and leftward rotating tools.
- The working direction is determined by the following input parameters: Algebraic sign Q239 (+ = right-hand thread /– = left-hand thread) and milling method Q351 (+1 = climb /–1 = up-cut). The table below illustrates the interrelation between the individual input parameters for rightward rotating tools.

Internal thread	Pitch	Climb/Up-cut	Work direction
Right-handed	+	+1 (RL)	Z+
Left-handed	-	-1 (RR)	Z+
Right-handed	+	-1 (RR)	Z-
Left-handed	-	+1 (RL)	Z-
External thread	**Pitch**	**Climb/Up-cut**	**Work direction**
Right-handed	+	+1 (RL)	Z-
Left-handed	-	-1 (RR)	Z-
Right-handed	+	-1 (RR)	Z+
Left-handed	-	+1 (RL)	Z+

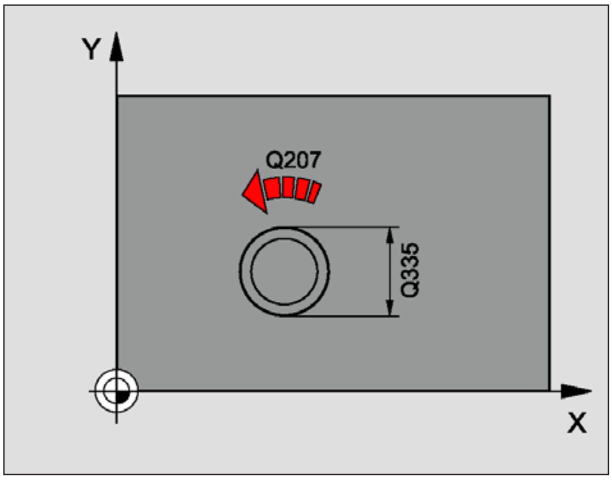
Note:
The WinNC references the programmed feed rate during thread milling to the tool cutting edge. Since the WinNC, however, always displays the feed rate relative to the path of the tool tip, the displayed value does not match the programmed value.

Danger of collision!

Always program the same algebraic sign for the infeeds: Cycles comprise several sequences of operation that are independent of each other. The order of precedence according to which the work direction is determined is described with the individual cycles. If you want to repeat specific machining operation of a cycle, for example with only the countersinking process, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

Procedure in the case of a tool break:

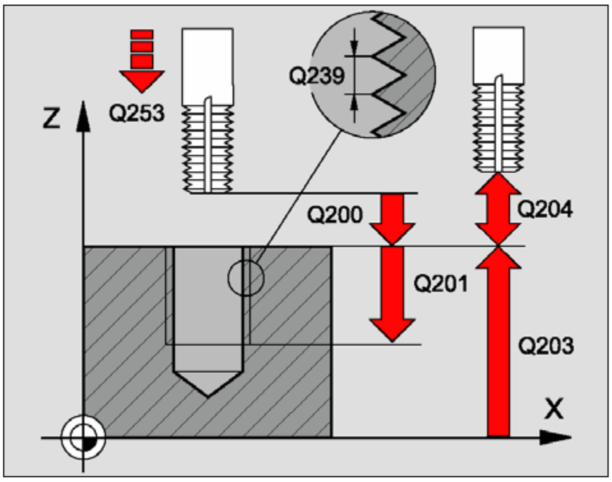
If a tool break occurs during thread cutting, stop the program run, change to the Positioning with manual data input (MDI) operating mode and move the tool in a linear path to the hole center. You can then retract the tool in the infeed axis and replace it.

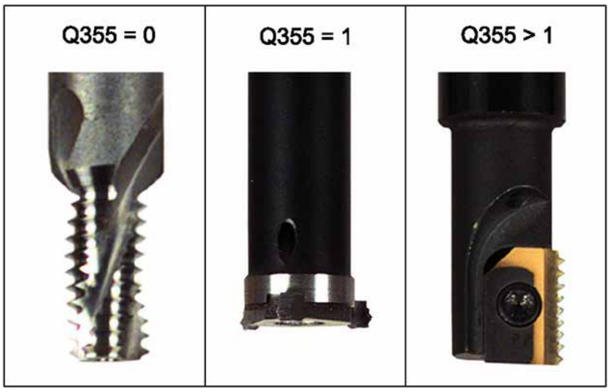


THREAD MILLING (Cycle 262)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.
2 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
3 The tool then approaches the thread diameter tangentially in a helical movement.
4 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one, in several spaced or in one continuous helical movement.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance or, if programmed, to the 2nd setup clearance.

Note:
Before programming, note the following:
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 thread depth determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program the thread depth = 0, the cycle will not be executed. The thread diameter is approached in a semi-circle from the center. A pre-positioning movement to the side is carried out if the pitch of the tool diameter is four times smaller than the thread diameter.

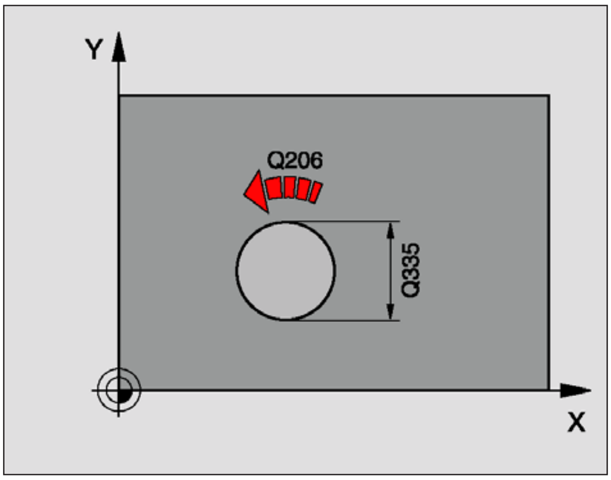




Example: NC blocks

```
25 CYCL DEF 262 THREAD MILLING
    Q335=10 ;NOMINAL DIAMETER
    Q239=+1,5 ;PITCH
    Q201=-20 ;THREAD DEPTH
    Q355=0 ;THREADS PER STEP
    Q253=750 ;F PRE-POSITIONING
    Q351=+1 ;CLIMB OR UP-CUT
    Q200=2 ;SET-UP CLEARANCE
    Q203=+30 ;SURFACE COORDINATE
    Q204=50 ;2ND SET-UP CLEARANCE
    Q207=500 ;FEED RATE FOR MILLING
```

- **Nominal diameter** Q335: Nominal thread diameter
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread
- **Threads per step** Q355: Number of thread revolutions by which the tool is offset, see figure at lower right
  0 = one 360° helical path to the depth of thread
  1 = continuous helical path over the entire length of the thread
  >1 = everal helical paths with approach and departure; between them, the WinNC offsets the tool by Q355, multiplied by the pitch
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min
- **Climb or up-cut** Q351: Type of milling operation
  +1 = climb milling
  –1 = up-cut milling
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.



Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. A negative sign bores in the direction of the neagative spindle axis. The working direction is defined in the following sequence:
1. Depth of thread
2. Countersinking depth
3. Depth at front
If you program a depth parameter to be 0, the WinNC does not execute that step. If you wish to countersink with the front of the tool, define the countersinking depth as 0. Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

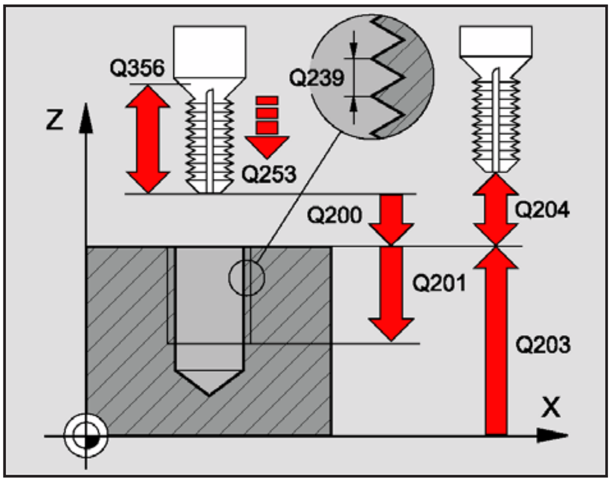
THREAD MILLING/COUNTERSINKING (Cycle 263)

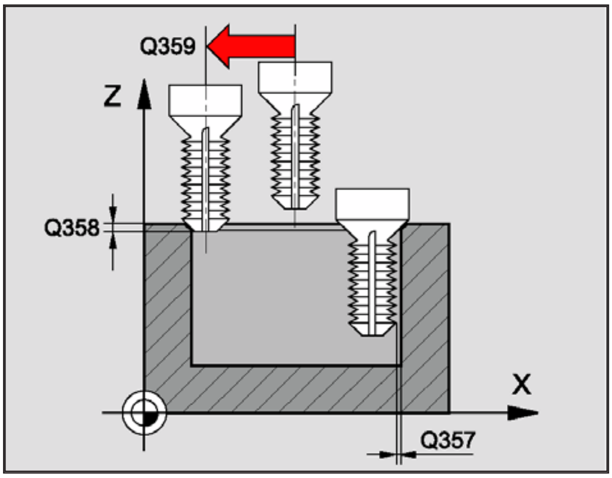
1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.

Countersinking
2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the setup clearance, and then at the feed rate for countersinking to the countersinking depth.
3 If a safety clearance to the side has been entered, the WinNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth
4 Then, depending on the available space, the WinNC makes a tangential approach to the core diameter, either tangentially from the center or with a pre-positioning move to the side, and follows a circular path.

Countersinking at front
5 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
6 The WinNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
7 The tool then moves in a semicircle to the hole center.

Thread milling
8 The WinNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).
9 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.
10 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
11 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance or, if programmed, to the 2nd setup clearance.



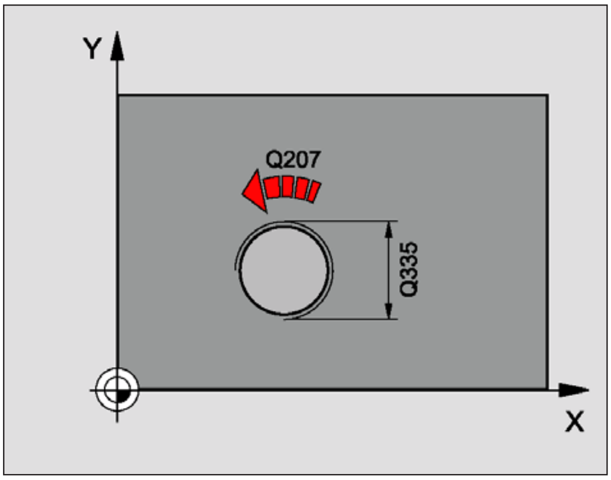


Example: NC blocks

```
25 CYCL DEF 263 THREAD MILLING/
   COUNTERSINKING
   Q335=10 ;NOMINAL DIAMETER
   Q239=+1,5 ;PITCH
   Q201=-16 ;THREAD DEPTH
   Q356=-20 ;COUNTERSINKING DEPTH
   Q253=750 ;F PRE-POSITIONING
   Q351=+1 ;CLIMB OR UP-CUT
   Q200=2 ;SET-UP CLEARANCE
   Q357=0.2 ;CLEARANCE TO SIDE
   Q358=+0 ;DEPTH AT FRONT
   Q359=+0 ;OFFSET AT FRONT
   Q203=+30 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q254=150 ;F COUNTERSINKING
   Q207=500 ;FEED RATE FOR MILLING
```

- **Nominal diameter** Q335: Nominal thread diameter
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread
- **Countersinking depth** Q356 (incremental value): Distance between tool point and the top surface of the workpiece
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min
- **Climb or up-cut** Q351: Type of milling operation
  +1 = climb milling
  –1 = up-cut milling
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface
- **Set-up clearance to the side** Q357 (incremental value): Distance between tool tooth and the wall
- **Depth at front** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool
- **Countersinking offset at front** Q359 (incremental value): Distance by which the WinNC moves the tool center away from the hole center
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

THREAD DRILLING/MILLING (Cycle 264)



1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.

Drilling

2 The tool drills to the first plunging depth at the programmed feed rate for plunging.

3 If you have programmed chip breaking, the tool then retracts by the entered retraction value. If you are working without chip breaking, the tool is moved at rapid traverse to setup clearance and then at FMAX to the entered starting position above the first plunging depth.

4 The tool then advances with another infeed at the programmed feed rate.

5 The WinNC repeats this process (2 to 4) until the programmed total hole depth is reached.

Countersinking at front

6 The tool moves at the feed rate for pre-positioning to the sinking depth at front.

7 The WinNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

8 The tool then moves in a semicircle to the hole center.

Thread milling

9 The WinNC moves the tool at the programmed feed rate for pre-positioning to the starting plane for the thread. The starting plane is determined from the thread pitch and the type of milling (climb or up-cut).

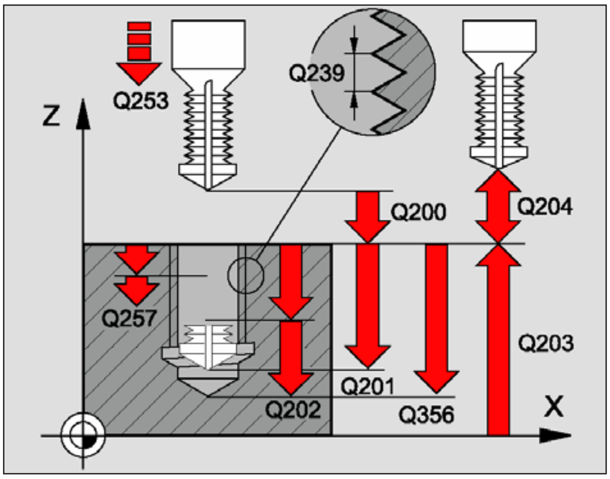
10 Then the tool moves tangentially on a helical path to the thread diameter and mills the thread with a 360° helical motion.

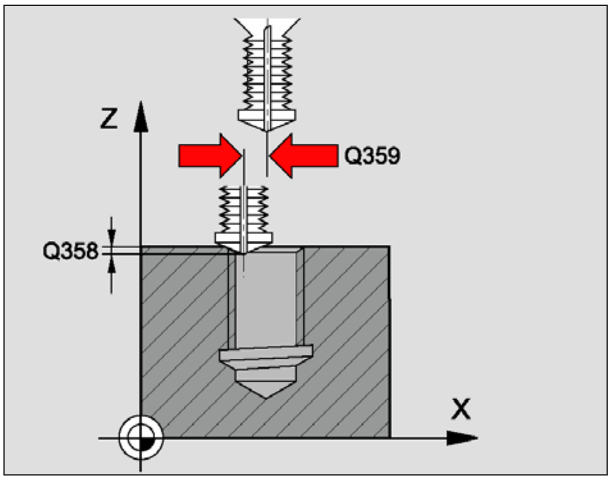
11 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

12 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance or, if programmed, to the 2nd setup clearance.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. A negative sign bores in the direction of the neagative spindle axis. The working direction is defined in the following sequence:
1. Depth of thread
2. Total hole depth
3. Depth at front
If you program a depth parameter to be 0, the WinNC does not execute that step. Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

- **Nominal diameter** Q335: Nominal thread diameter
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread





Example: NC blocks

```
25 CYCL DEF 264 THREAD DRILLNG/MLLNG
   Q335=10 ;NOMINAL DIAMETER
   Q239=+1,5 ;PITCH
   Q201=-16 ;THREAD DEPTH
   Q356=-20 ;TOTAL HOLE DEPTH
   Q253=750 ;F PRE-POSITIONING
   Q351=+1 ;CLIMB OR UP-CUT
   Q202=5 ;PLUNGING DEPTH
   Q258=0,2 ;ADVANCED STOP DISTANCE
   Q257=5 ;DEPTH FOR CHIP BRKNG
   Q256=0:2 ;DIST FOR CHIP BRKNG
   Q358=+0 ;DEPTH AT FRONT
   Q359=+0 ;OFFSET AT FRONT
   Q200=2 ;SET-UP CLEARANCE
   Q203=+30 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q206=150 ;FEED RATE FOR PLUNGING
   Q207=500 ;FEED RATE FOR MILLING
```

- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread
- **Total hole depth** Q356 (incremental value): Distance between workpiece surface and bottom of hole
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min
- **Climb or up-cut** Q351: Type of milling operation
  +1 = climb milling
  –1 = up-cut milling
- **Plunging depth** Q202 (incremental value): Infeed per cut The depth does not have to be a multiple of the plunging depth. The WinNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Upper advanced stop distance** Q258 (incremental value): Setup clearance for rapid traverse positioning when the WinNC moves the tool again to the current plunging depth after retraction from the hole
- **Infeed depth for chip breaking** Q257 (incremental value): Depth at which TNC carries out chip breaking. There is no chip breaking if 0 is entered.
- **Retraction rate for chip breaking** Q256 (incremental value): Value by which the WinNC retracts the tool during chip breaking
- **Depth at front** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool
- **Countersinking offset at front** Q359 (incremental value): Distance by which the WinNC moves the tool center away from the hole center
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for plunging** Q206: Traversing speed of the tool during drilling in mm/min
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

HELICAL THREAD DRILLING/MILLING (Cycle 265)

1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.

Countersinking at front

2 If countersinking is before thread milling, the tool moves at the feed rate for countersinking to the sinking depth at front. If countersinking is after thread milling, the tool moves at the feed rate for pre-positioning to the countersinking depth.

3 The WinNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.

4 The tool then moves in a semicircle to the hole center.

Thread milling

5 The tool moves at the programmed feed rate for pre-positioning to the starting plane for the thread.

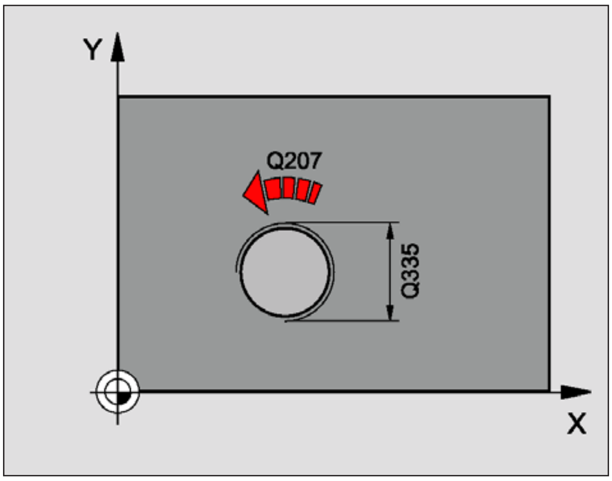
6 The tool then approaches the thread diameter tangentially in a helical movement.

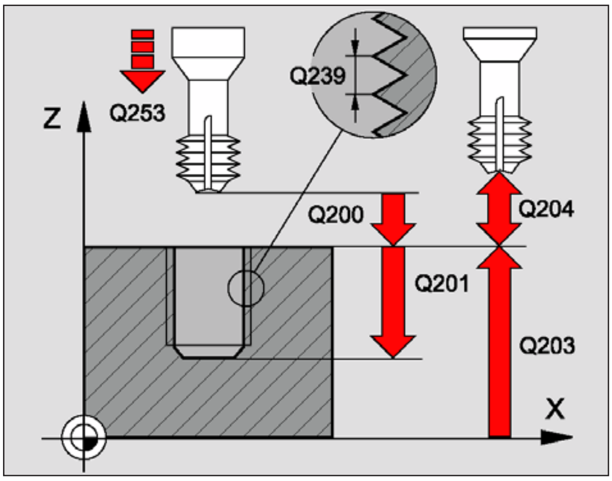
7 The tool moves on a continuous helical downward path until it reaches the thread depth.

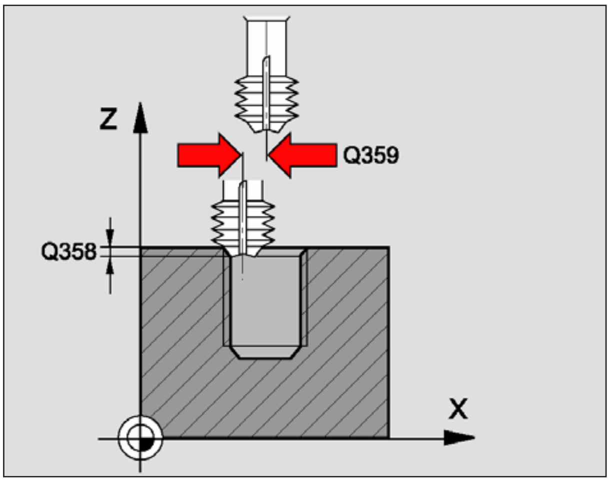
8 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.

9 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance or, if programmed, to the 2nd setup clearance.

Note:
Before programming, note the following:
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0. The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. A negative sign bores in the direction of the neagative spindle axis. The working direction is defined in the following sequence:
1. Depth of thread
2. Depth at front
If you program a depth parameter to be 0, the WinNC does not execute that step. The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.



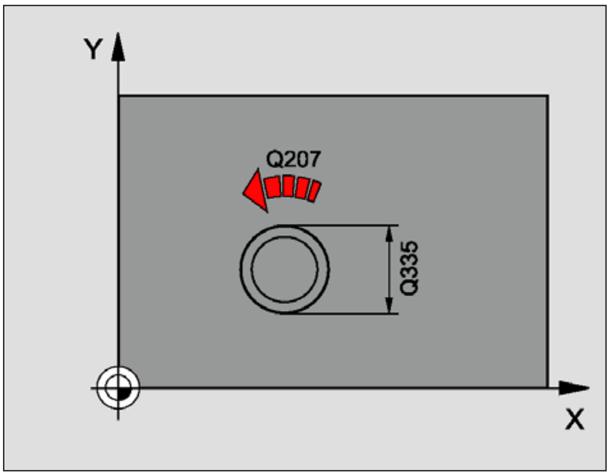




- **Nominal diameter** Q335: Nominal thread diameter
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min
- **Depth at front** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool
- **Countersinking offset at front** Q359 (incremental value): Distance by which the WinNC moves the tool center away from the hole center
- **Countersink** Q360: Execution of the chamfer
  0 = before thread machining
  1 = after thread machining
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
25 CYCL DEF 265 HEL. THREAD DRLG/MLG
    Q335=10 ;NOMINAL DIAMETER
    Q239=+1,5 ;PITCH
    Q201=-16 ;THREAD DEPTH
    Q253=750 ;F PRE-POSITIONING
    Q351=+1 ;CLIMB OR UP-CUT
    Q358=+0 ;DEPTH AT FRONT
    Q359=+0 ;OFFSET AT FRONT
    Q360=0 ;COUNTERSINKING
    Q200=2 ;SET-UP CLEARANCE
    Q203=+30 ;SURFACE COORDINATE
    Q204=50 ;2ND SET-UP CLEARANCE
    Q254=150 ;F COUNTERSINKING
    Q207=500 ;FEED RATE FOR MILLING
```



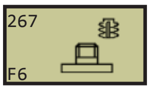
Note:
Before programming, note the following:
Program a positioning block for the starting point (stud center) in the working plane with radius compensation R0. The algebraic sign of the cycle parameters depth of thread, countersinking depth or sinking depth at front determines the working direction. A negative sign bores in the direction of the neagative spindle axis. The working direction is defined in the following sequence:
1. Depth of thread
2. Depth at front
If you program a depth parameter to be 0, the WinNC does not execute that step. The algebraic sign for the cycle parameter thread depth determines the working direction. If, for example, you program the thread depth = 0, the cycle will not be executed.

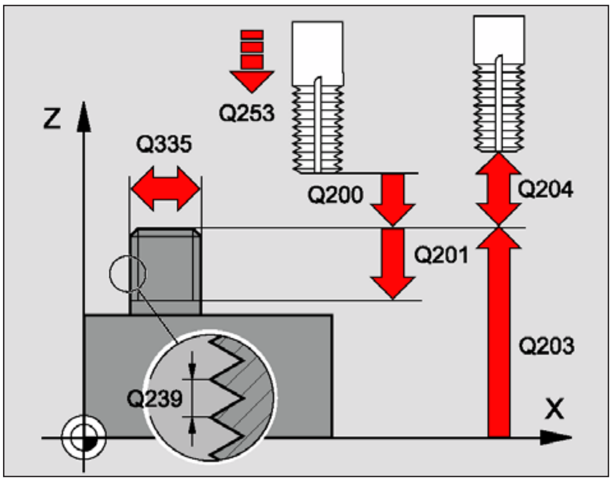
OUTSIDE THREAD MILLING (Cycle 267)

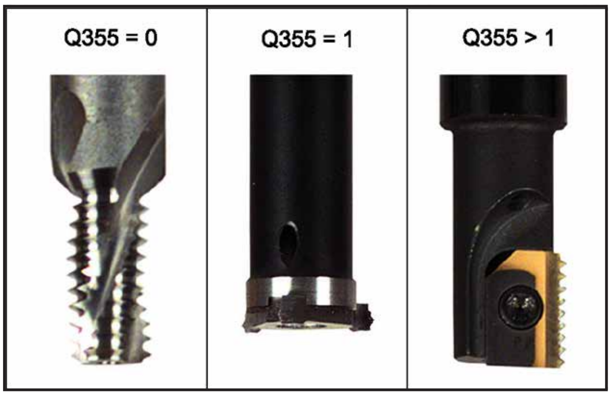
1 The WinNC positions the tool in the tool axis at rapid traverse FMAX to the programmed setup clearance above the workpiece surface.

Countersinking at front
2 The WinNC moves on the reference axis of the working plane from the center of the stud to the starting point for countersinking at front. The position of the starting point is determined by the thread radius, tool radius and pitch.
3 The tool moves at the feed rate for pre-positioning to the sinking depth at front.
4 The WinNC positions the tool without compensation from the center on a semicircle to the offset at front, and then follows a circular path at the feed rate for countersinking.
5 The tool then moves in a semicircle to the starting point.

Thread milling
6 The WinNC positions the tool to the starting point if there has been no previous countersinking at front. Starting point for thread milling = starting point for countersinking at front.
7 The tool moves at the programmed feed rate for pre-positioning to the starting plane. The starting plane is derived from the algebraic sign of the thread pitch, the milling method (climb or up-cut milling) and the number of threads per step.
8 The tool then approaches the thread diameter tangentially in a helical movement.
9 Depending on the setting of the parameter for the number of threads, the tool mills the thread in one, in several spaced or in one continuous helical movement.
10 After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
11 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance or, if programmed, to the 2nd setup clearance.





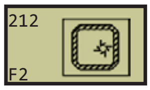
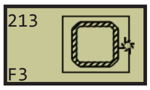
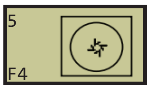
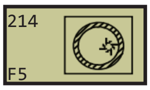
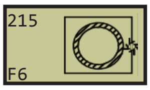
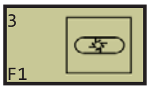
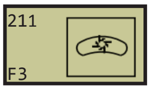


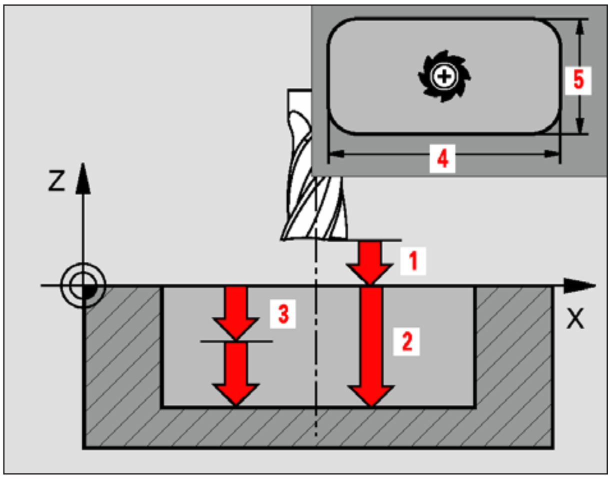
Example: NC blocks

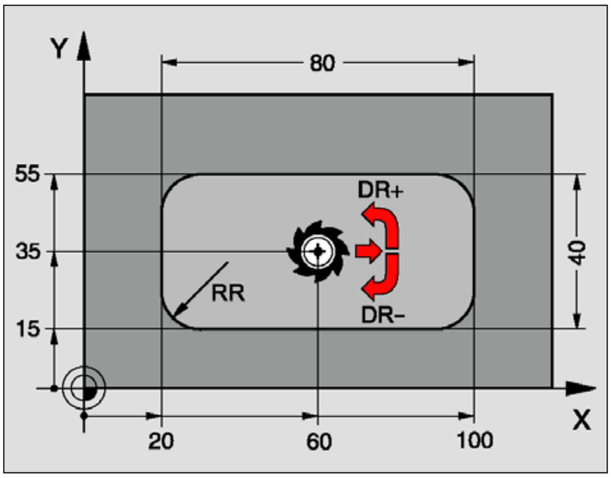
25 CYCL DEF 267 OUTSIDE THREAD MLLNG
Q335=10 ;NOMINAL DIAMETER
Q239=+1,5 ;PITCH
Q201=-20 ;THREAD DEPTH
Q355=0 ;THREADS PER STEP
Q253=750 ;F PRE-POSITIONING
Q351=+1 ;CLIMB OR UP-CUT
Q200=2 ;SET-UP CLEARANCE
Q358=+0 ;DEPTH AT FRONT
Q359=+0 ;OFFSET AT FRONT
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q254=150 ;F COUNTERSINKING
Q207=500 ;FEED RATE FOR MILLING

- **Nominal diameter** Q335: Nominal thread diameter
- **Thread pitch** Q239: Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread
- **Thread depth** Q201 (incremental value): Distance between workpiece surface and root of thread
- **Threads per step** Q355: Number of thread revolutions by which the tool is offset, see figure at lower right
  0 = one 360° helical path to the depth of thread
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the WinNC offsets the tool by Q355, multiplied by the pitch
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool when moving in and out of the workpiece, in mm/min
- **Climb or up-cut** Q351: Type of milling operation
  +1 = climb milling
  –1 = up-cut milling
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface
- **Depth at front** Q358 (incremental value): Distance between tool point and the top surface of the workpiece for countersinking at the front of the tool
- **Countersinking offset at front** Q359 (incremental value): Distance by which the WinNC moves the tool center away from the stud center
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Feed rate for counterboring** Q254: Traversing speed of the tool during counterboring in mm/min
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Cycles for milling pockets, studs and slots

Cycle	Soft key
4 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning	
212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
5 CIRCULAR POCKET Roughing cycle without automatic pre-positioning	
214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning, 2nd set-up clearance	
3 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical depth infeed	
210 SLOT with reciprocating plungecut Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed	
211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning, with reciprocating plunge infeed	





Example: NC blocks

11 L Z+100 R0 FMAX
12 CYCL DEF 4.0 POCKET MILLING
13 CYCL DEF 4.1 SET UP 2
14 CYCL DEF 4.2 DEPTH -10
15 CYCL DEF 4.3 PLNGNG 4 F80
16 CYCL DEF 4.4 X80
17 CYCL DEF 4.5 Y40
18 CYCL DEF 4.6 F100 DR+ RADIUS 10
19 L X+60 Y+35 FMAX M3
20 L Z+2 FMAX M99

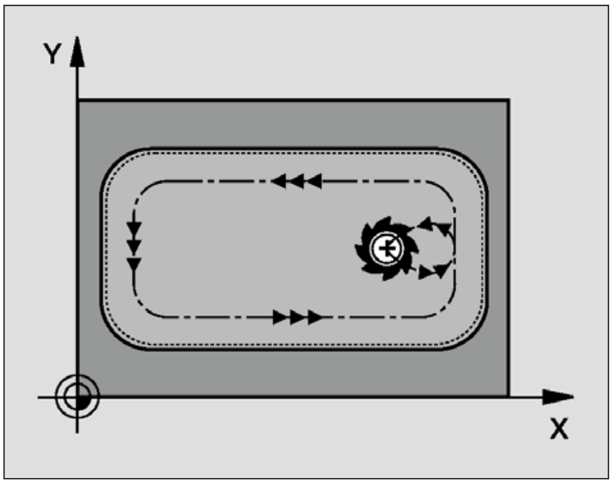
Calculations:
Stepover factor k = K x R
K: is the overlap factor = 1.9 (default value)
R is the cutter radius

POCKET MILLING (Cycle 4)

1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first plunging depth.
2 The cutter begins milling in the positive axis direction of the longer side (on square pockets, always starting in the positive Y direction) and then roughs out the pocket from the inside out.
3 This process (1 to 2) is repeated until the depth is reached.
4 At the end of the cycle, the WinNC retracts the tool to the starting position.

Note:
Before programming, note the following:
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center. Pre-position over the pocket center with radius compensation R0. Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface). The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The following prerequisite applies for the 2nd side length: 2nd side length greater than [(2 x rounding radius) + stepover factor k].

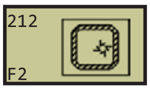
- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The WinNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging**: Traversing speed of the tool during penetration
- **First side length 4** (incremental value): Pocket length, parallel to the reference axis of the working plane
- **2nd side length 5**: Pocket width
- **Feed rate F**: Traversing speed of the tool in the working plane
- **Clockwise**
  DR **+** : Climb milling with M3
  DR **–** : Up-cut milling with M3
- **Rounding off radius**: Radius RR for the pocket corners. The Rounding off radius RR is always greater than or equal to the radius of the cutter.

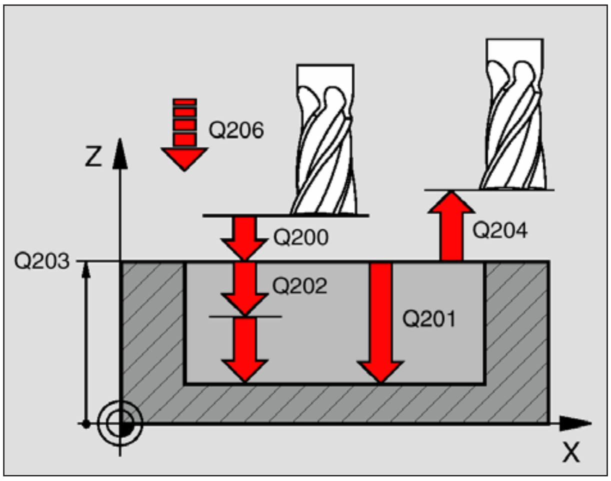


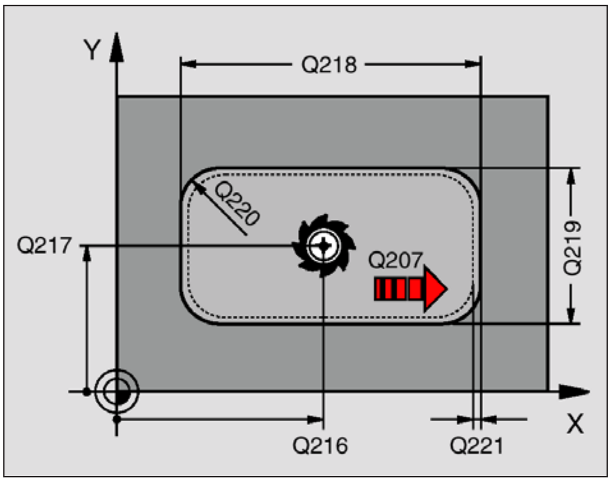
POCKET FINISHING (Cycle212)

1 The WinNC automatically moves the tool in the tool axis to set-up clearance, or — if programmed —to the 2nd set-up clearance, and subsequently to the center of the pocket.
2 From the pocket center, the tool moves in the working plane to the starting point for machining. The WinNC takes the allowance and tool radius into account for calculating the starting point. If necessary, the WinNC penetrates at the pocket center.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).

Note:
Before programming, note the following:
The WinNC automatically pre-positions the tool in the tool axis and working plane. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging. Minimum size of the pocket: 3 times the tool radius.



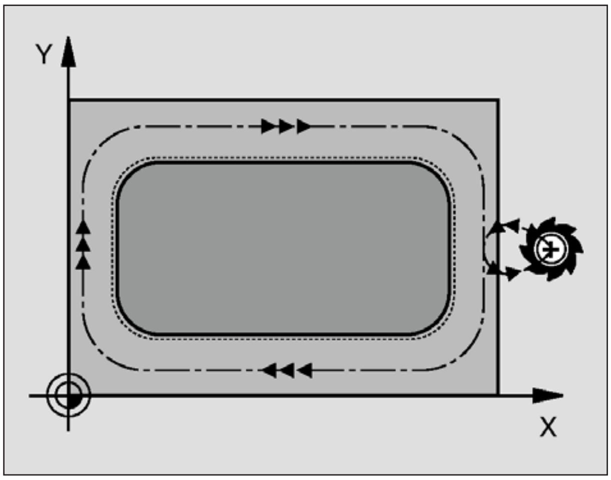




- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of pocket
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a value lower than that defined in Q207
- **Plunging depth** Q202 (incremental value): Infeed per cut; enter a value greater than 0.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the pocket in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the pocket in the minor axis of the working plane
- **First side length** Q218 (incremental value): Pocket length, parallel to the reference axis of the working plane
- **Second side length** Q219 (incremental value): Pocket length, parallel to the minor axis of the working plane
- **Corner radius** Q220: Radius of the pocket corner If you make no entry here, the WinNC assumes that the corner radius is equal to the tool radius.
- **Allowance in 1st axis** Q221 (incremental value): Allowance for pre-positioning in the reference axis of the working plane.

Example: NC blocks

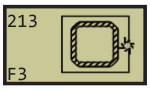
```
34 CYCL DEF 212 POCKET FINISHING
   Q200=2 ;SET-UP CLEARANCE
   Q201=-20 ;DEPTH
   Q206=150 ;FEED RATE FOR PLUNGING
   Q202=5 ;PLUNGING DEPTH
   Q207=500 ;FEED RATE FOR MILLING
   Q203=+30 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q216=+50 ;CENTER IN 1ST AXIS
   Q217=+50 ;CENTER IN 2ND AXIS
   Q218=80 ;FIRST SIDE LENGTH
   Q219=60 ;SECOND SIDE LENGTH
   Q220=5 ;CORNER RADIUS
   Q221=0 ;ALLOWANCE
```

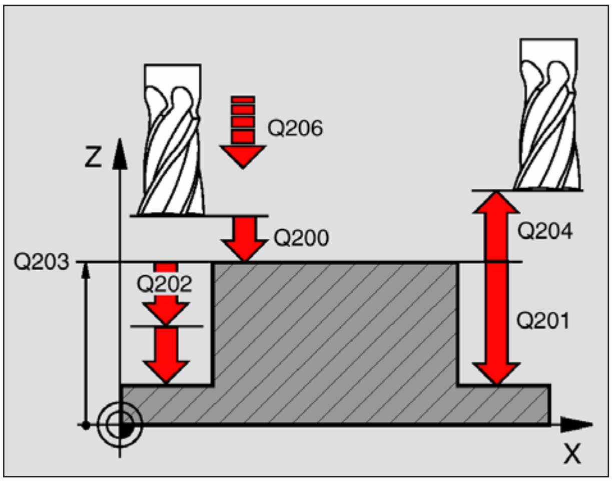


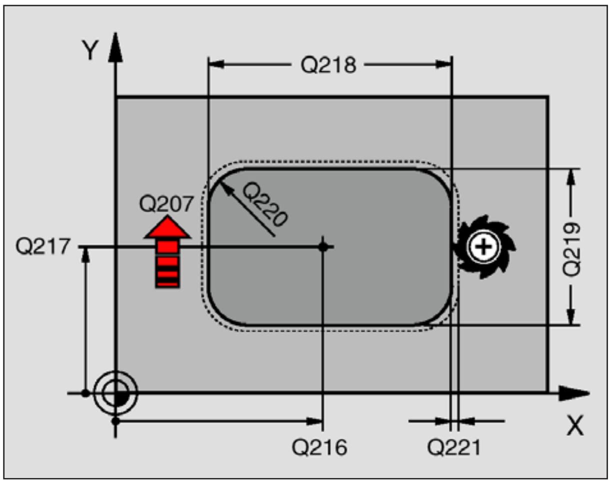
STUD FINISHING (Cycle 213)

1 The WinNC moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3.5 times the tool radius.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the WinNC retracts the tool in FMAX to set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).

Note:
Before programming, note the following:
The WinNC automatically pre-positions the tool in the tool axis and working plane. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



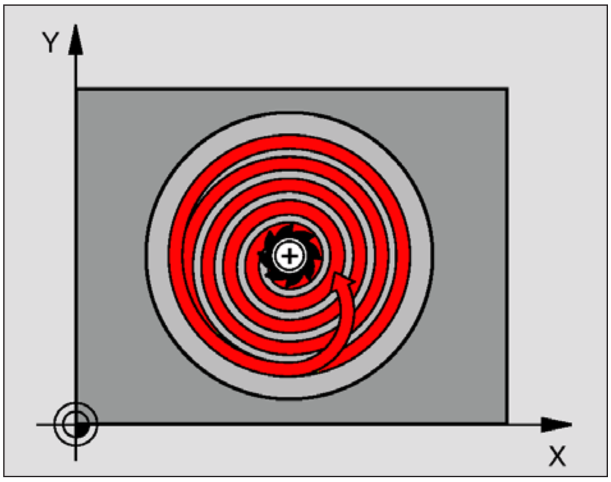


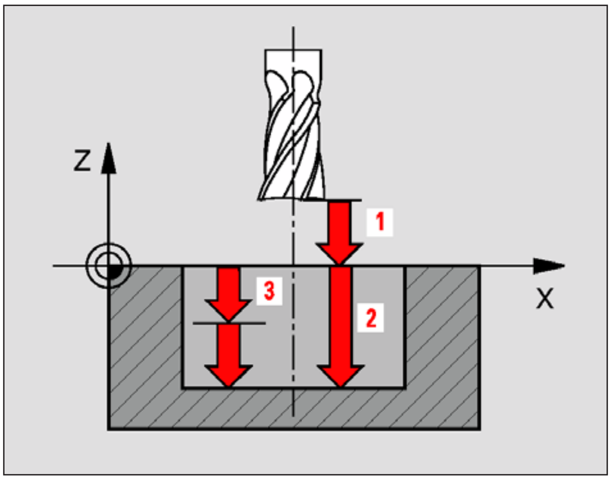


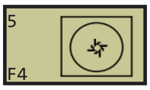
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- **Plunging depth** Q202 (incremental value): Infeed per cut Enter a value greater than 0.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the stud in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the stud in the minor axis of the working plane
- **First side length** Q218 (incremental value): Length of stud parallel to the reference axis of the working plane
- **Second side length** Q219 (incremental value): Length of stud parallel to the secondary axis of the working plane
- **Corner radius** Q220: Radius of the stud corner
- **Allowance in 1st axis** Q221 (incremental value): Allowance for pre-positioning in the reference axis of the working plane.

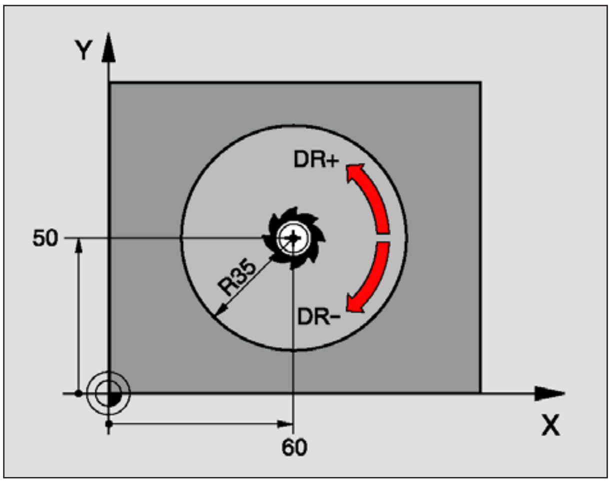
Example: NC blocks

```
35 CYCL DEF 213 STUD FINISHING
    Q200=2 ;SET-UP CLEARANCE
    Q201=-20 ;DEPTH
    Q206=150 ;FEED RATE FOR PLUNGING
    Q202=5 ;PLUNGING DEPTH
    Q207=500 ;FEED RATE FOR MILLING
    Q203=+30 ;SURFACE COORDINATE
    Q204=50 ;2ND SET-UP CLEARANCE
    Q216=+50 ;CENTER IN 1ST AXIS
    Q217=+50 ;CENTER IN 2ND AXIS
    Q218=80 ;FIRST SIDE LENGTH
    Q219=60 ;SECOND SIDE LENGTH
    Q220=5 ;CORNER RADIUS
    Q221=0 ;ALLOWANCE
```









CIRCULAR POCKET MILLING (Cycle 5)

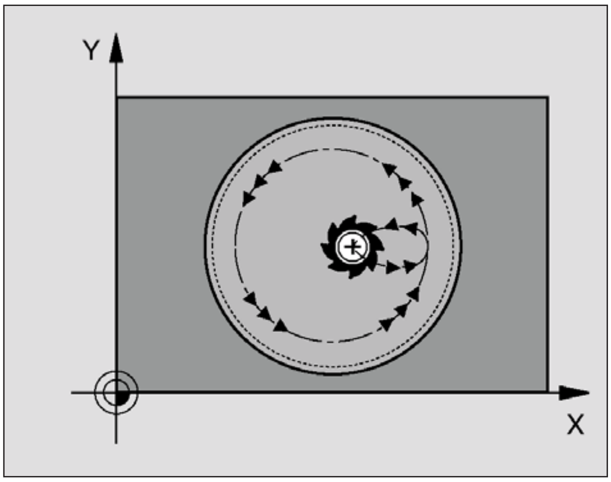
1 The tool penetrates the workpiece at the starting position (pocket ncenter) and advances to the first plunging depth.
2 The tool subsequently follows a spiral path at the feed rate F - see figure at right. For calculating the stepover factor k, see Cycle 4 POCKET MILLING.
3 This process is repeated until the depth is reached.
4 At the end of the cycle, the WinNC retracts the tool to the starting position.

Note:
Before programming, note the following:
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center. Pre-position over the pocket center with radius compensation R0. Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface). The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.

- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2**: Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut The WinNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging**: Traversing speed of the tool during penetration
- **Circular radius**: Radius of the circular pocket
- **Feed rate** F: Traversing speed of the tool in the working plane
- **Clockwise**
  DR **+** : Climb milling with M3
  DR **–** : Up-cut milling with M3

Example: NC blocks

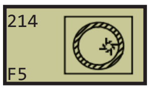
16 L Z+100 R0 FMAX
17 CYCL DEF 5.0 CIRCULAR POCKET
18 CYCL DEF 5.1 SET UP 2
19 CYCL DEF 5.2 DEPTH -12
20 CYCL DEF 5.3 PLNGNG 6 F80
21 CYCL DEF 5.4 RADIUS 35
22 CYCL DEF 5.5 F100 DR+
23 L X+60 Y+50 FMAX M3
24 L Z+2 FMAX M99

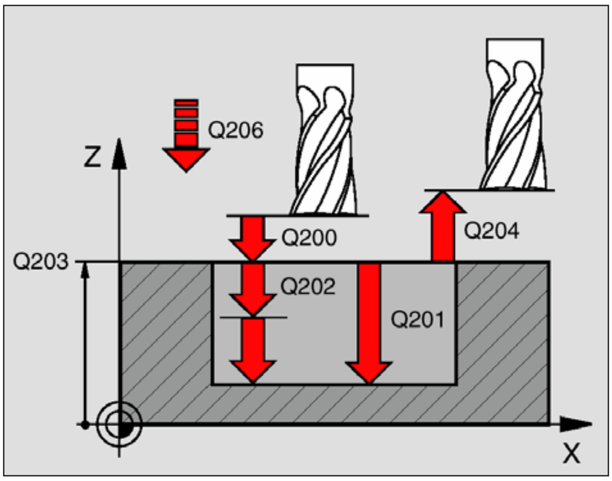


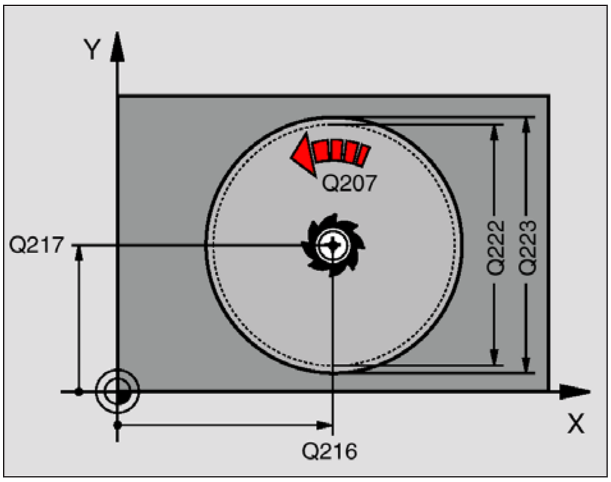
CIRCULAR POCKET FINISHING (Cycle 214)

1 The WinNC automatically moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
2 From the pocket center, the tool moves in the working plane to the starting point for machining. The WinNC takes the workpiece blank diameter and tool radius into account for calculating the starting point. If you enter a workpiece blank diameter of 0, the WinNC plunge-cuts into the pocket center.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the WinNC retracts the tool in rapid traverse to set-up clearance, or, if programmed, to the 2nd set-up clearance and then to the center of the pocket (end position = starting position)

Note:
Before programming, note the following:
The WinNC automatically pre-positions the tool in the tool axis and working plane. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



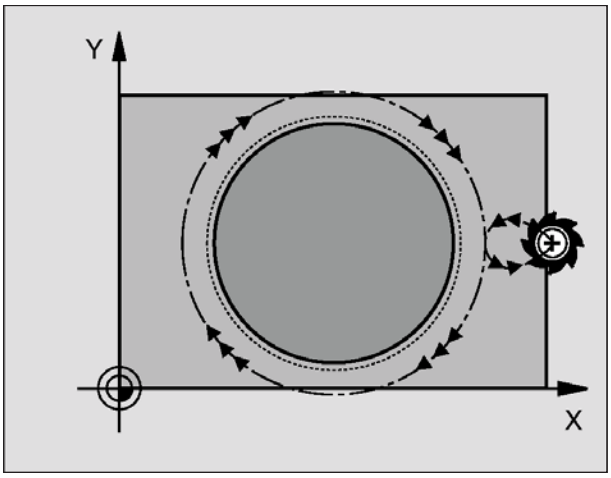




- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of pocket
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a value lower than that defined in Q207
- **Plunging depth** Q202 (incremental value): Infeed per cut
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the pocket in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the pocket in the minor axis of the working plane
- **Workpiece blank diameter** Q222: Diameter of the premachined pocket for calculating the pre-position. Enter the workpiece blank diameter to be less than the diameter of the finished part
- **Finished part diameter** Q223: Diameter of the finished pocket. Enter the diameter of the finished part to be greater than the workpiece blank diameter.

Example: NC blocks

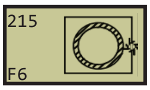
```
42 CYCL DEF 214 CIRCULAR POCKET FINISHING
   Q200=2 ;SET-UP CLEARANCE
   Q201=-20 ;DEPTH
   Q206=150 ;FEED RATE FOR PLUNGING
   Q202=5 ;PLUNGING DEPTH
   Q207=500 ;FEED RATE FOR MILLING
   Q203=+30 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q216=+50 ;CENTER IN 1ST AXIS
   Q217=+50 ;CENTER IN 2ND AXIS
   Q222=79 ;WORKPIECE BLANK DIA.
   Q223=80 ;FINISHED PART DIA.
```

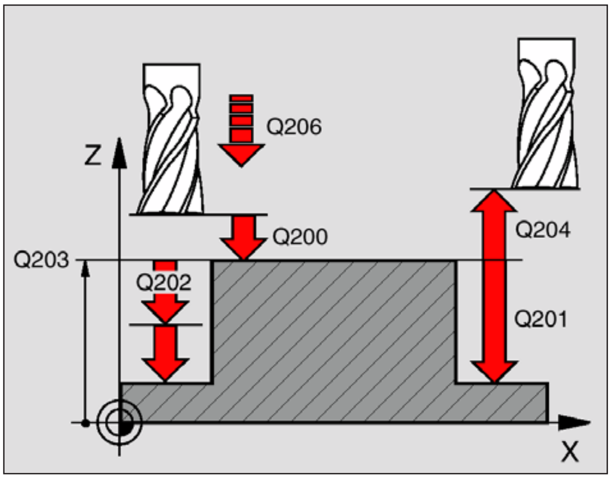


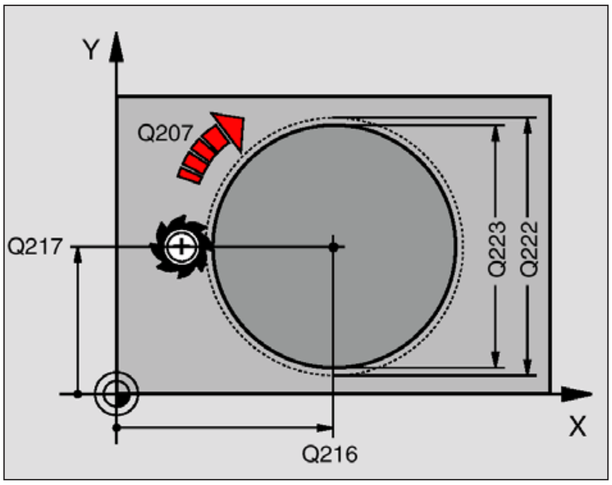
CIRCULAR STUD FINISHING (Cycle 215)

1 The WinNC automatically moves the tool in the tool axis to set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the stud.
2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3.5 times the tool radius.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse FMAX to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.
5 The tool then departs the contour on a tangential path and returns to the starting point in the working plane.
6 This process (3 to 5) is repeated until the programmed depth is reached.
7 At the end of the cycle, the WinNC retracts the tool in FMAX to set-up clearance, or — if programmed — to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).

Note:
Before programming, note the following:
The WinNC automatically pre-positions the tool in the tool axis and working plane. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



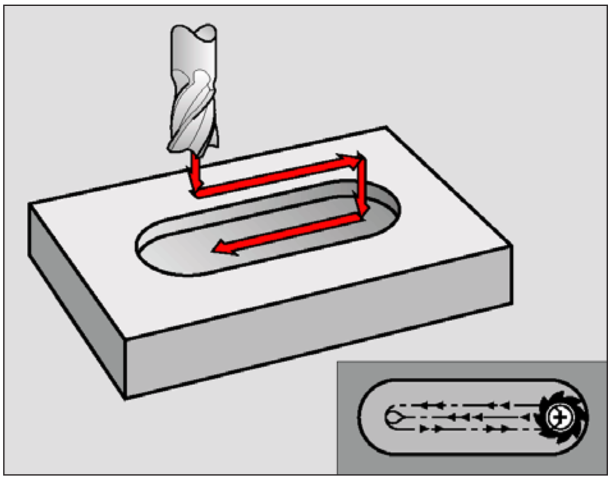




- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of stud
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- **Plunging depth** Q202 (incremental value): Infeed per cut; enter a value greater than 0.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Center in 1st axis** Q216 (absolute value): Center of the stud in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the stud in the minor axis of the working plane
- **Workpiece blank diameter** Q222: Diameter of the premachined stud for calculating the pre-position. Enter the workpiece blank diameter to be greater than the diameter of the finished part
- **Diameter of finished part** Q223: Diameter of the finished stud. Enter the diameter of the finished part to be less than the workpiece blank diameter.

Example: NC blocks

```
43 CYCL DEF 215 C. STUD FINISHING
   Q200=2 ;SET-UP CLEARANCE
   Q201=-20 ;DEPTH
   Q206=150 ;FEED RATE FOR PLUNGING
   Q202=5 ;PLUNGING DEPTH
   Q207=500 ;FEED RATE FOR MILLING
   Q203=+30 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q216=+50 ;CENTER IN 1ST AXIS
   Q217=+50 ;CENTER IN 2ND AXIS
   Q222=81 ;WORKPIECE BLANK DIA.
   Q223=80 ;FINISHED PART DIA.
```



SLOT MILLING (Cycle 3)

Roughing process

1 The WinNC moves the tool inward by the milling allowance (half the difference between the slot width and the tool diameter). From there it plunge-cuts into the workpiece and mills in the longitudinal direction of the slot.

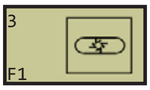
2 After downfeed at the end of the slot, milling is performed in the opposite direction. This process is repeated until the programmed milling depth is reached.

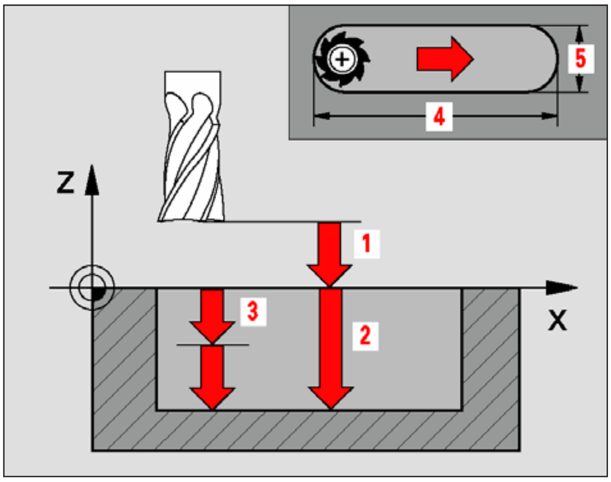
Finishing process

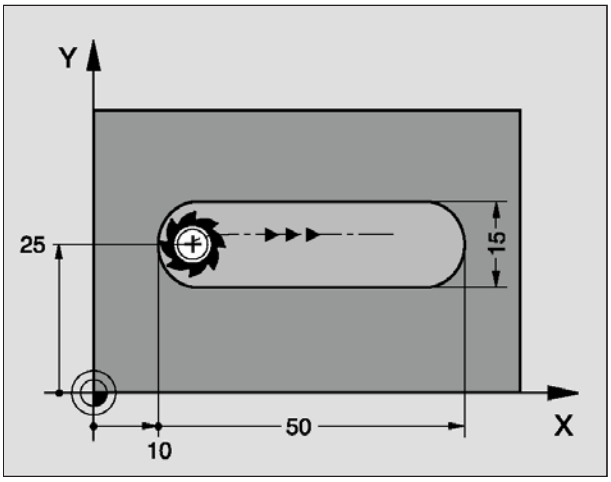
3 The WinNC advances the tool at the slot bottom on a tangential arc to the outside contour. The tool subsequently climb mills the contour (with M3).

4 At the end of the cycle, the tool is retracted in rapid traverse FMAX to set-up clearance. If the number of infeeds was odd, the tool returns to the starting position at the level of the set-up clearance.

Note:
Before programming, note the following:
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point. Pre-position to the center of the slot and offset by the tool radius into the slot with radius compensation R0. The cutter diameter must be not be larger than the slot width and not smaller than half the slot width. Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface). The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed.



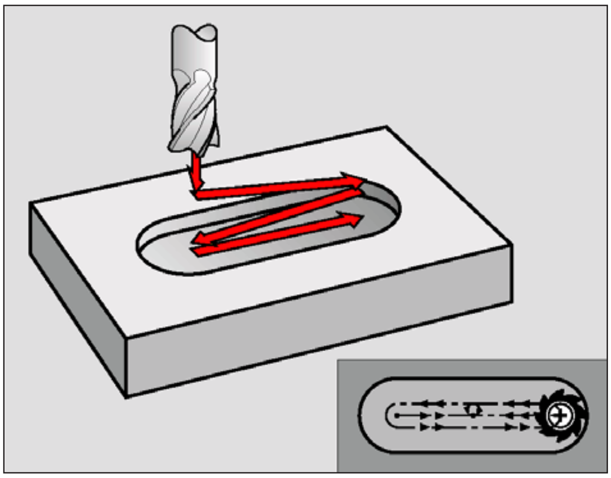




- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface
- **Milling depth 2**: (incremental value): Distance between workpiece surface and bottom of slot
- **Plunging depth 3** (incremental value): Infeed per cut. The tool will drill to the depth in one operation if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth
- **Feed rate for plunging**: Traversing speed during penetration
- **1st side length 4**: Länge der Nut; 1. Schnittrichtung durch Vorzeichen festlegen
- **2nd side length 5**: Breite der Nut
- **Feed rate F**: Traversing speed of the tool in the working plane

Example: NC blocks

```
9 L Z+100 R0 FMAX
10 TOOL DEF 1 L+0 R+6
11 TOOL CALL 1 Z S1500
12 CYCL DEF 3.0 SLOT MILLING
13 CYCL DEF 3.1 SET UP 2
14 CYCL DEF 3.2 DEPTH -15
15 CYCL DEF 3.3 PLNGNG 5 F80
16 CYCL DEF 3.4 X50
17 CYCL DEF 3.5 Y15
18 CYCL DEF 3.6 F120
19 L X+16 Y+25 R0 FMAX M3
20 L Z+2 M99
```



SLOT (oblong hole) with reciprocating plungecut (Cycle 210)

Roughing process

1 At rapid traverse, the WinNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the left circle. From there, the WinNC positions the tool to set-up clearance above the workpiece surface.

2 The tool moves at the feed rate for milling to the workpiece surface. From there, the cutter advances in the longitudinal direction of the slot—plunge-cutting obliquely into the material—until it reaches the center of the right circle.

3 The tool then moves back to the center of the left circle, again with oblique plunge-cutting. This process is repeated until the programmed milling depth is reached.

4 At the milling depth, the WinNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.

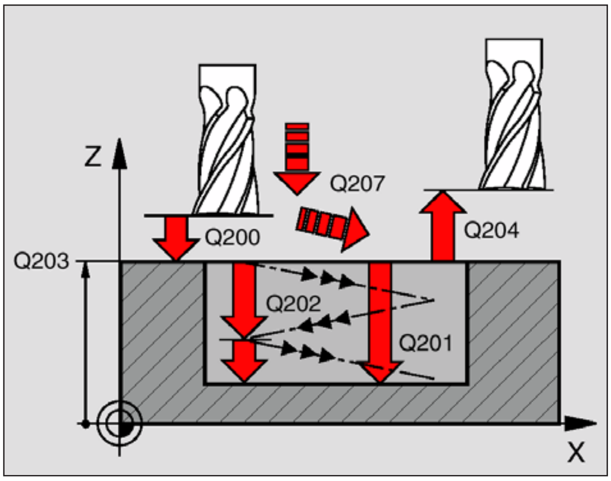
Finishing process

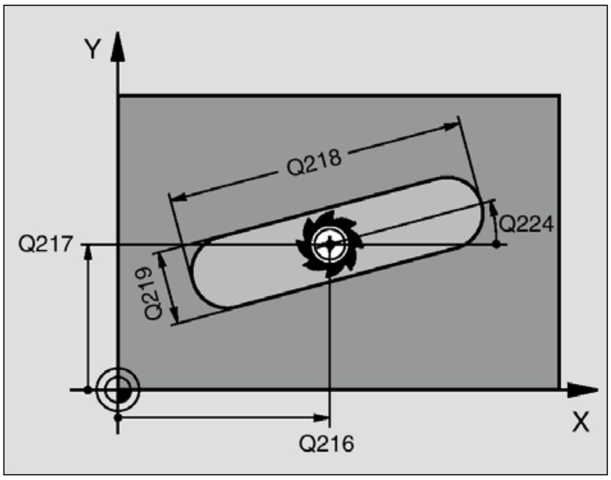
5 The WinNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed.

6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.

7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to set-up clearance and—if programmed—to the 2nd set-up clearance.

Note:
Before programming, note the following:
The WinNC automatically pre-positions the tool in the tool axis and working plane. During roughing the tool plunges into the material with a sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary.
The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width. The cutter diameter must be smaller than half the slot length. The WinNC otherwise cannot execute this cycle.

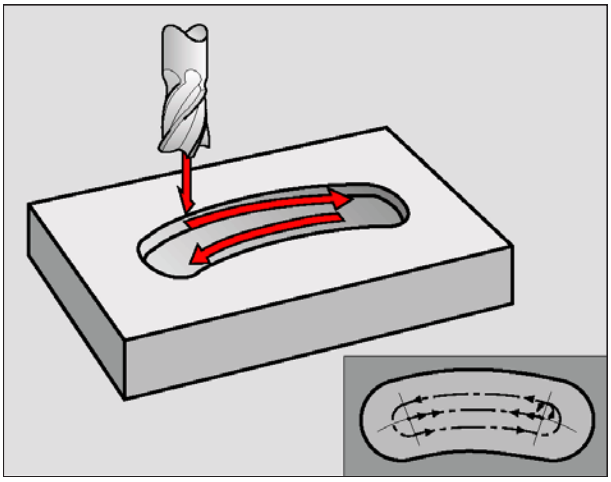




Example: NC blocks

```
51 CYCL DEF 210 SLOT RECIP. PLNG
   Q200=2 ;SET-UP CLEARANCE
   Q201=-20 ;DEPTH
   Q207=500 ;FEED RATE FOR MILLING
   Q202=5 ;PLUNGING DEPTH
   Q215=0 ;MACHINING OPERATION
   Q203=+30 ;SURFACE COORDINATE
   Q204=50 ;2ND SET-UP CLEARANCE
   Q216=+50 ;CENTER IN 1ST AXIS
   Q217=+50 ;CENTER IN 2ND AXIS
   Q218=80 ;FIRST SIDE LENGTH
   Q219=12 ;SECOND SIDE LENGTH
   Q224=+15 ;ANGLE OF ROTATION
   Q338=5 ;INFEED FOR FINISHING
```

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of slot
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Plunging depth** Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement
- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd setup clearance** Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur
- **Center in 1st axis** Q216 (absolute value): Center of the slot in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the slot in the minor axis of the working plane
- **First side length** Q218 (value parallel to the reference axis of the working plane): Enter the length of the slot
- **Second side length** Q219 (value parallel to the secondary axis of the working plane): Enter the slot width. If you enter a slot width that equals the tool diameter, the WinNC will carry out the roughing process only (slot milling).
- **Angle of rotation** Q224 (absolute value): Angle by which the entire slot is rotated. The center of rotation lies in the center of the slot.
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.



CIRCULAR SLOT (oblong hole) with reciprocating plunge-cut (Cycle 211)

Roughing process

1 At rapid traverse, the WinNC positions the tool in the tool axis to the 2nd set-up clearance and subsequently to the center of the right circle. From there, the tool is positioned to the programmed set-up clearance above the workpiece surface.

2 The tool moves at the milling feed rate to the workpiece surface. From there, the cutter advances—plunge-cutting obliquely into the material—to the other end of the slot.

3 The tool then moves at a downward angle back to the starting point, again with oblique plunge-cutting. This process (2 to 3) is repeated until the programmed milling depth is reached.

4 At the milling depth, the WinNC moves the tool for the purpose of face milling to the other end of the slot.

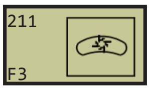
Finishing process

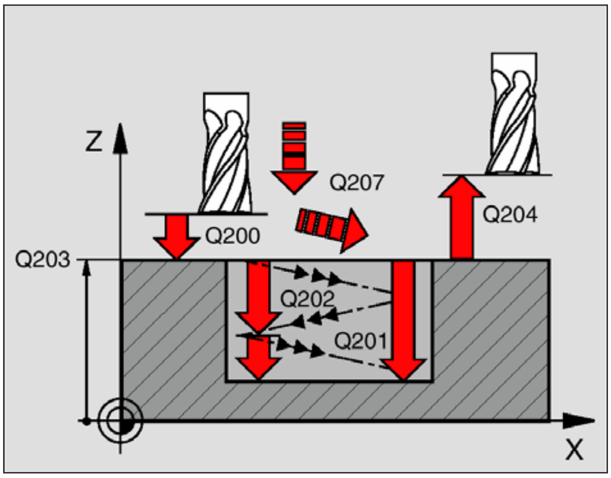
5 The WinNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3), and if so entered, in more than one infeed. The starting point for the finishing process is the center of the right circle.

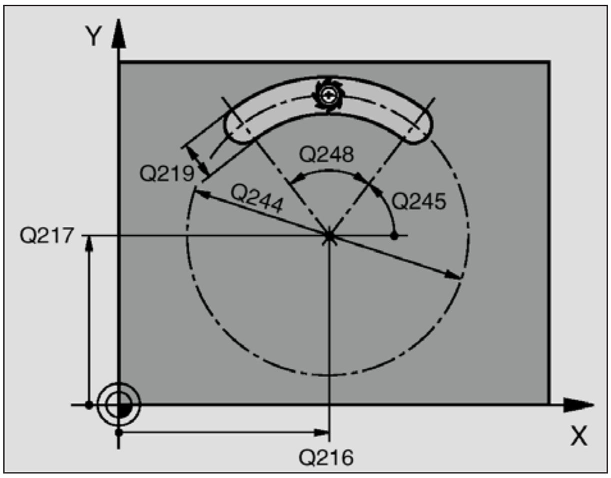
6 When the tool reaches the end of the contour, it departs the contour tangentially.

7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to set-up clearance and—if programmed—to the 2nd set-up clearance.

Note:
Before programming, note the following:
The WinNC automatically pre-positions the tool in the tool axis and working plane. During roughing the tool plunges into the material with a helical sideward reciprocating motion from one end of the slot to the other. Pilot drilling is therefore unnecessary. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width. The cutter diameter must be smaller than half the slot length. The WinNC otherwise cannot execute this cycle.







Example: NC blocks

52 CYCL DEF 211 CIRCULAR SLOT

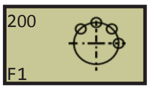
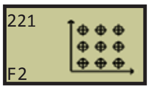
```
Q200=2 ;SET-UP CLEARANCE
Q201=-20 ;DEPTH
Q207=500 ;FEED RATE FOR MILLING
Q202=5 ;PLUNGING DEPTH
Q215=0 ;MACHINING OPERATION
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q216=+50 ;CENTER IN 1ST AXIS
Q217=+50 ;CENTER IN 2ND AXIS
Q244=80 ;PITCH CIRCLE DIAMETR
Q219=12 ;SECOND SIDE LENGTH
Q245=+45 ;STARTING ANGLE
Q248=90 ;ANGULAR LENGTH
Q338=5 ;INFEED FOR FINISHING
```

- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of slot
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Plunging depth** Q202 (incremental value): Total extent by which the tool is fed in the tool axis during a reciprocating movement
- **Machining operation (0/1/2)** Q215: Define the machining operation:
  0: Roughing and finishing
  1: Only roughing
  2: Only finishing
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd setup clearance** Q204 (incremental value): Z coordinate at which no collision between tool and workpiece (clamping devices) can occur
- **Center in 1st axis** Q216 (absolute value): Center of the slot in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the slot in the minor axis of the working plane
- **Pitch circle diameter** Q244: Enter the diameter of the pitch circle
- **Second side length** Q219: Enter the slot width. If you enter a slot width that equals the tool diameter, the WinNC will carry out the roughing process only (slot milling).
- **Starting angle** Q245 (absolute value): Enter the polar angle of the starting point
- **Angular length** Q248 (incremental value): Enter the angular length of the slot
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.

Cycles for Machining Hole Patterns

Overview

The WinNC provides two cycles for machining hole patterns directly:

Cycle	Soft key
220 CIRCULAR PATTERN	
221 LINEAR PATTERN	

Note:
If you have to machine irregular hole patterns, use **CYCL CALL PAT** (see ”Point Tables”) to develop point tables.

You can combine Cycle 220 and Cycle 221 with the following fixed cycles:

Cycle 1	PECKING
Cycle 2	TAPPING with a floating tap holder
Cycle 3	SLOT MILLING
Cycle 4	POCKET MILLING
Cycle 5	CIRCULAR POCKET MILLING
Cycle 17	RIGID TAPPING without a floating tap holder
Cycle 18	THREAD CUTTING
Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL DRILLING
Cycle 204	BACK BORING
Cycle 205	UNIVERSAL PECKING
Cycle 206	TAPPING NEW with a floating tap holder
Cycle 207	RIGID TAPPING NEW without a floating tap holder
Cycle 208	BORE MILLING
Cycle 209	TAPPING WITH CHIP BREAKING
Cycle 212	POCKET FINISHING
Cycle 213	STUD FINISHING
Cycle 214	CIRCULAR POCKET FINISHING
Cycle 215	CIRCULAR STUD FINISHING
Cycle 262	THREAD MILLING
Cycle 263	THREAD MILLING/COUNTERSINKING
Cycle 264	THREAD DRILLING/MILLING
Cycle 265	HELICAL THREAD DRILLING/MILLING
Cycle 267	OUTSIDE THREAD MILLING

CIRCULAR PATTERN (Cycle 220)

1 At rapid traverse, the WinNC moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:

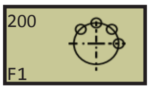
- 2. Move to 2nd setup clearance (spindle axis)
- Approach the starting point in the spindle axis
- Move to setup clearance above the workpiece surface (spindle axis)

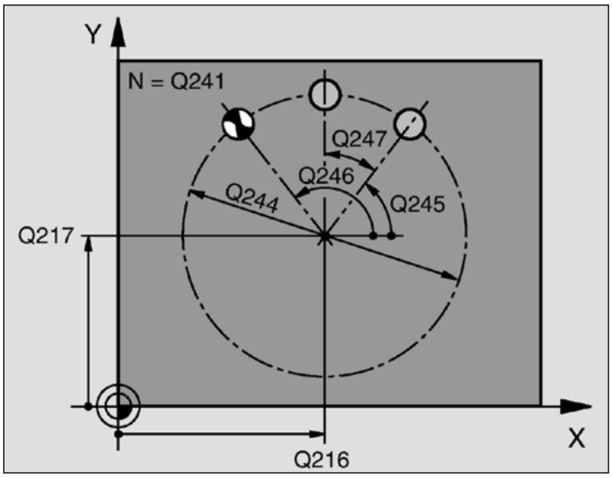
2 From this position, the WinNC executes the last defined fixed cycle.

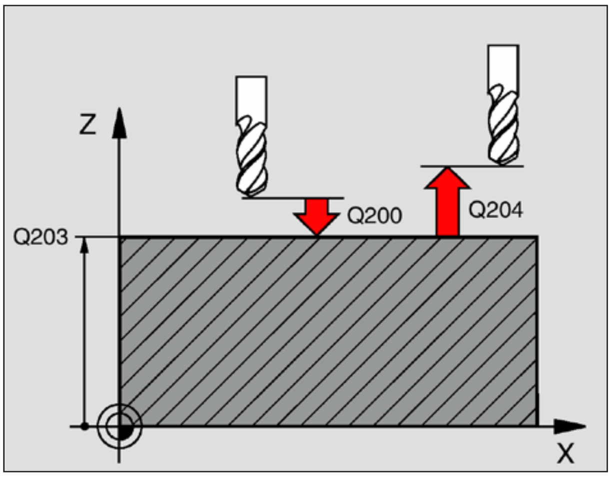
3 The tool then approaches the starting point for the next machining operation on a straight line at set-up clearance (or 2nd set-up clearance).

4 This process (1 to 3) is repeated until all machining operations have been executed.

Note:
Before programming, note the following:
Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle. If you combine Cycle 220 with one of the fixed cycles 200 to 208 and 212 to 215, the set-up clearance, workpiece surface and 2nd setup clearance from Cycle 220 will be effective for the selected fixed cycle.





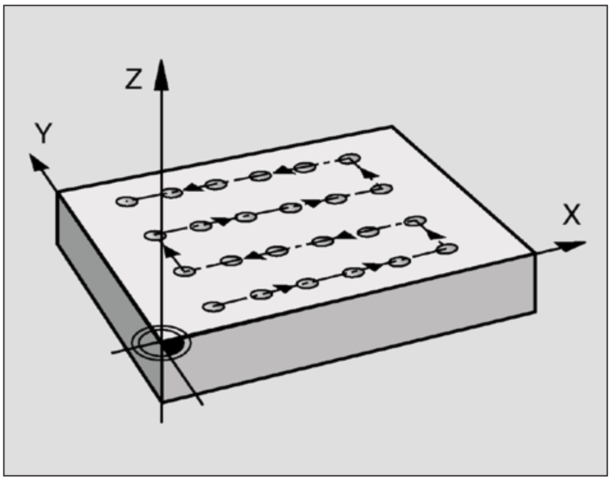


Example: NC blocks

53 CYCL DEF 220 POLAR PATTERN
Q216=+50 ;CENTER IN 1ST AXIS
Q217=+50 ;CENTER IN 2ND AXIS
Q244=80 ;PITCH CIRCLE DIAMETR
Q245=+0 ;STARTING ANGLE
Q246=+360 ;STOPPING ANGLE
Q247=+0 ;STEPPING ANGLE
Q241=8 ;NR OF REPETITIONS
Q200=2 ;SET-UP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q301=1 ;TRAVERSE TO CLEARANCE HEIGHT

- **Center in 1st axis** Q216 (absolute value): Center of the pitch circle in the reference axis of the working plane
- **Center in 2nd axis** Q217 (absolute value): Center of the pitch circle in the minor axis of the working plane
- **Pitch circle diameter** Q244: Diameter of the pitch circle
- **Starting angle** Q245 (absolute value): Angle between the reference axis of the working plane and the starting point for the first machining operation on the pitch circle
- **Stopping angle** Q246 (absolute value): Angle between the reference axis of the working plane and the starting point for the last machining operation on the pitch circle (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise.

- **Stepping angle** Q247 (incremental value): Angle between two machining operations on a pitch circle. If you enter an angle step of 0, the WinNC will calculate the angle step from the starting and stopping angles and the number of pattern repetitions. If you enter a value other than 0, the WinNC will not take the stopping angle into account. The sign for the angle step determines the working direction (– = clockwise).
- **Number of repetitions** Q241: Number of machining operations on a pitch circle
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Traversing to clearance height** Q301: Definition of how the tool is to move between machining processes:
  0: Move to setup clearance between operations.
  1: Move to 2nd setup clearance between machining operations.



LINEAR PATTERN (Cycle 221)

Note:
Before programming, note the following:
Cycle 221 is DEF active, which means that Cycle 221 calls the last defined fixed cycle automatically. If you combine Cycle 221 with one of the fixed cycles 200 to 208 and 212 to 215, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 221 will be effective for the selected fixed cycle.

1 The WinNC automatically moves the tool from its current position to the starting point for the first machining operation.

The tool is positioned in the following sequence:

- 2. Move to 2nd setup clearance (spindle axis)
- Approach the starting point in the spindle axis
- Move to setup clearance above the workpiece surface (spindle axis)

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

3 The tool then approaches the starting point for the next machining operation in the positive reference axis direction at set-up clearance (or 2nd set-up clearance).

4 This process (1 to 3) is repeated until all machining operations on the first line have been executed. The tool is located above the last point on the first line.

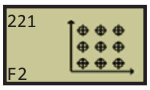
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

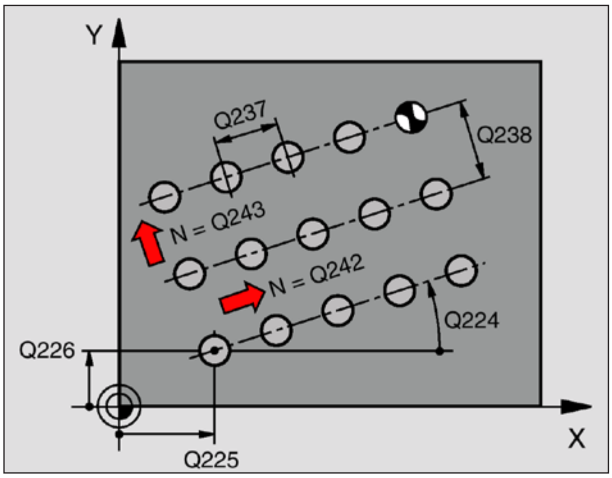
6 From this position, the tool approaches the starting point for the next machining operation in the negative reference axis direction.

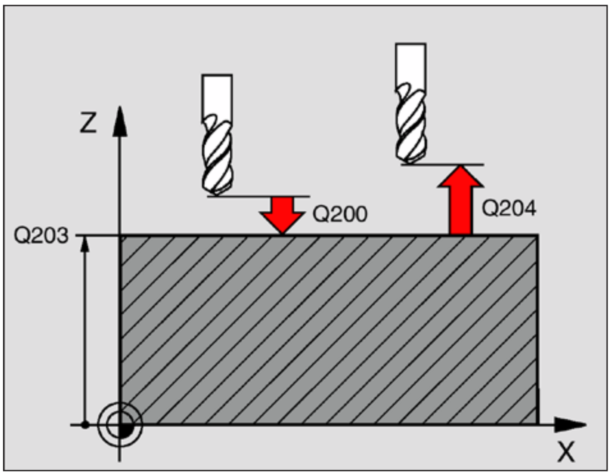
7 This process (6) is repeated until all machining operations in the second line have been executed.

8 The tool then moves to the starting point of the next line.

9 All subsequent lines are processed in a reciprocating movement.







- **Starting point 1st axis** Q225 (absolute value): Coordinate of the starting point in the reference axis of the working plane
- **Starting point 2nd axis** Q226 (absolute value): Coordinate of the starting point in the minor axis of the working plane
- **Spacing in 1st axis** Q237 (incremental value): Spacing between the individual points on a line
- **Spacing in 2nd axis** Q238 (incremental value): Spacing between the individual lines
- **Number of columns** Q242: Number of machining operations on a line
- **Number of lines** Q243: Number of passes
- **Angle of rotation** Q224 (absolute value): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point.
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface
- **Workpiece surface coordinate** Q203 (absolute value): Coordinate of the workpiece surface
- **2nd set-up clearance** Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- **Traversing to clearance height** Q301: Definition of how the tool is to move between machining processes:
  0: Move to setup clearance between operations.
  1: Move to 2nd setup clearance between the measuring points.

Example: NC blocks

54 CYCL DEF 221 CARTESIAN PATTRN
Q225=+15 ;STARTNG PNT 1ST AXIS
Q226=+15 ;STARTNG PNT 2ND AXIS
Q237=+10 ;SPACING IN 1ST AXIS
Q238=+8 ;SPACING IN 2ND AXIS
Q242=6 ;NUMBER OF COLUMNS
Q243=4 ;NUMBER OF LINES
Q224=+15 ;ANGLE OF ROTATION
Q200=2 ;SET-UP CLEARANCE
Q203=+30 ;SURFACE COORDINATE
Q204=50 ;2ND SET-UP CLEARANCE
Q301=1 ;TRAVERSE TO CLEARANCE HEIGHT

SL-Cycles

Fundamentals

SL cycles enable you to form complex contours by combining up to 12 subcontours (pockets or islands). You define the individual subcontours in subprograms. The TNC calculates the total contour from the subcontours (subprogram numbers) that enter in Cycle 14 CONTOUR GEOMETRY.

Characteristics of the subprograms

- Coordinate transformations are permitted. If they are programmed within the subcontour they are also effective in the following subprograms, but they need not be reset after the cycle call.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation RR.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation RL.
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram.

Characteristics of the fixed cycles

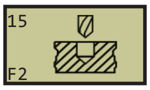
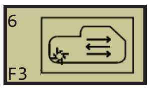
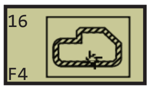
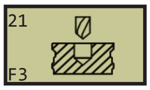
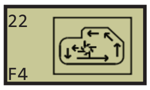
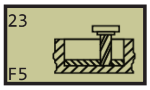
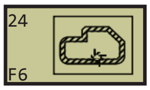
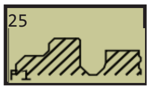
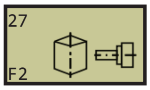
- The TNC automatically positions the tool to set-up clearance before a cycle.
- Each level of infeed depth is milled without interruptions since the cutter traverses around islands instead of over them.
- The radius of "inside corners" can be programmed—the tool keeps moving to prevent surface blemishes at inside corners (this applies for the outermost pass in the Rough-out and Side-Finishing cycles).
- The contour is approached in a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece in a tangential arc (for tool axis Z, for example, the arc may be in the Z/X plane).
- The contour is machined throughout in either climb or up-cut milling.

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle 20.

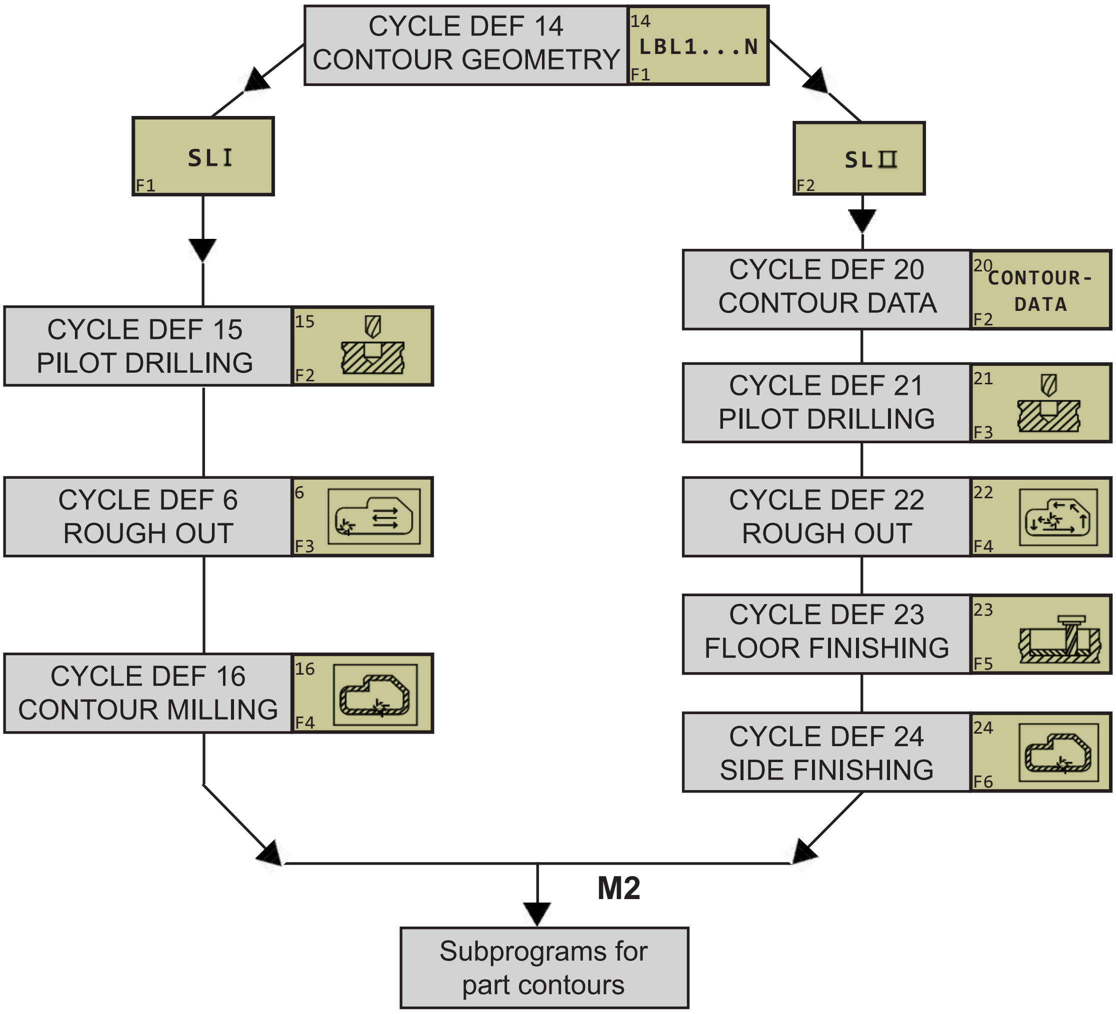
Example: Program structure: Machining with SL cycles

```
0 BEGIN PGM SL2 MM
...
12 CYCL DEF 14.0 CONTOUR GEOMETRY ...
13 CYCL DEF 20.0 CONTOUR DATA ...
...
16 CYCL DEF 21.0 PILOT DRILLING
17 CYCL CALL
...
18 CYCL DEF 22.0 ROUGH-OUT
19 CYCL CALL
...
22 CYCL DEF 23.0 FLOOR FINISHING ...
23 CYCL CALL
...
26 CYCL DEF 24.0 SIDE FINISHING ...
27 CYCL CALL
...
50 L Z+250 R0 FMAX M2
51 LBL 1
...
55 LBL 0
56 LBL 2
...
60 LBL 0
...
99 END PGM SL2 MM
```

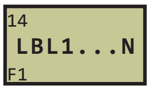
Overview of SL-Cycles

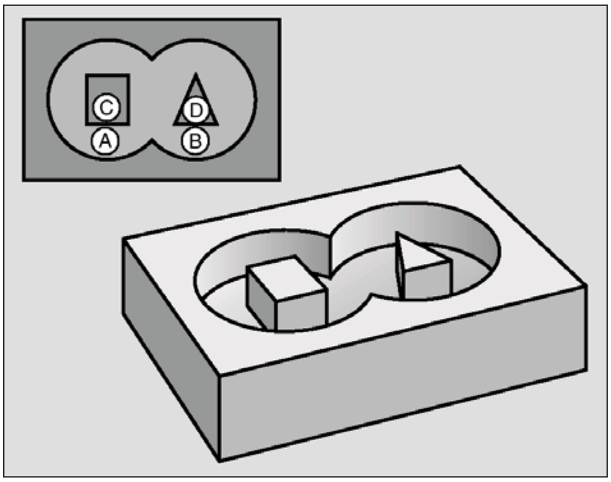
Cycles	SL-group	Soft key
14 CONTOUR GEOMETRY (essential)	SL I (F1) / SL II (F2)	14 LBL1...N F1
15 PILOT DRILLING (optional)		
6 ROUGH OUT (essential)	SL I (F1)	
16 CONTOUR MILLING (optional)		
20 CONTOUR DATA (essential)		20 KONTUR-DATEN F2
21 PILOT DRILLING (optional)		
22 ROUGH-OUT (essential)		
23 FLOOR FINISHING (optional)	SL II (F2)	
24 SIDE FINISHING (optional)		
25 CONTOUR TRAIN		
27 CYLINDER SURFACE		

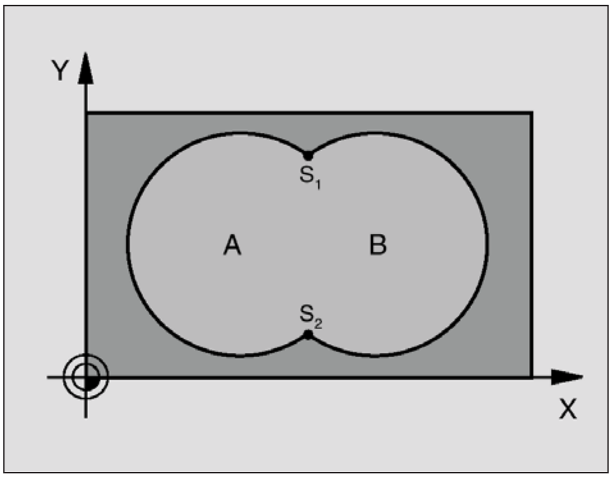
SL-cycles, program flowchart



Note:
The SL I-Cycles 6, 15 and 16 are processed like SL II-Cycles 21, 22 and 24.







Example: NC blocks

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

CONTOUR GEOMETRY (Cycle 14)

All subprograms that are superimposed to define the contour are listed in Cycle 14 CONTOUR GEOMETRY.

Note:
Before programming, note the following:
Cycle 14 is DEF active which means that it becomes effective as soon as it is defined in the part program. You can list up to 12 subprograms (subcontours) in Cycle 14.

- **Label numbers for the contour**: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.

Overlapping contours

Pockets and islands can be overlapped to form a new contour. You can thus enlarge the area of a pocket by another pocket or reduce it by an island.

Subprograms: Overlapping pockets

Note:
The subsequent programming examples are contour subprograms that are called by Cycle 14 CONTOUR GEOMETRY in a main program.

Pockets A and B overlap.

The TNC calculates the points of intersection S1 and S2 (they do not have to be programmed).

The pockets are programmed as full circles.

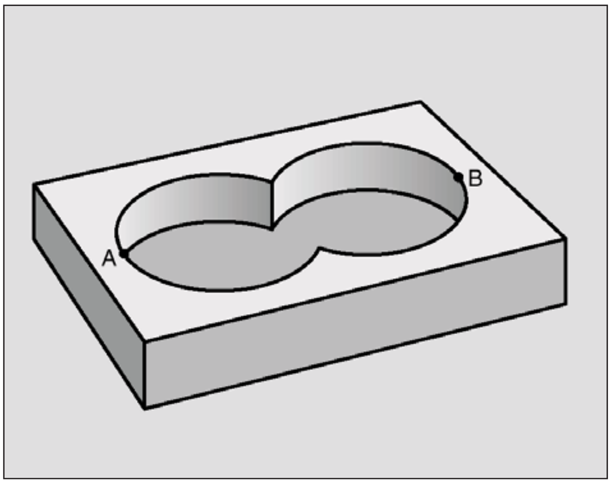
Subprogram 1: Pocket A
Example: NC blocks

```
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
Example: NC blocks

```
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
```

Area of inclusion



Both surfaces A and B are to be machined, including the mutually overlapped area:

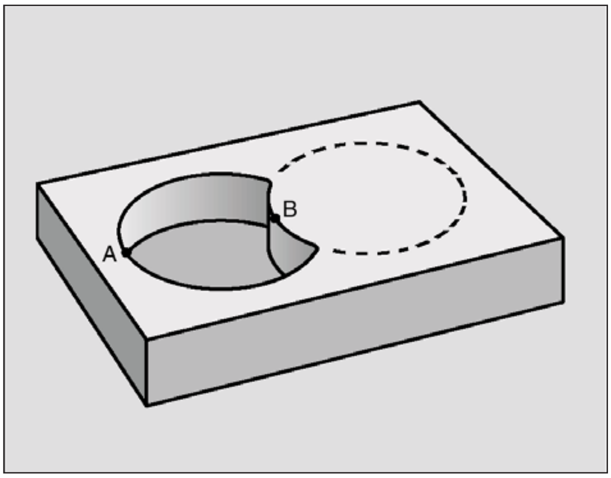
- The surfaces A and B must be pockets.
- The first pocket (in Cycle 14) must start outside the second pocket.

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
```



Area of exclusion

Surface A is to be machined without the portion overlapped by B

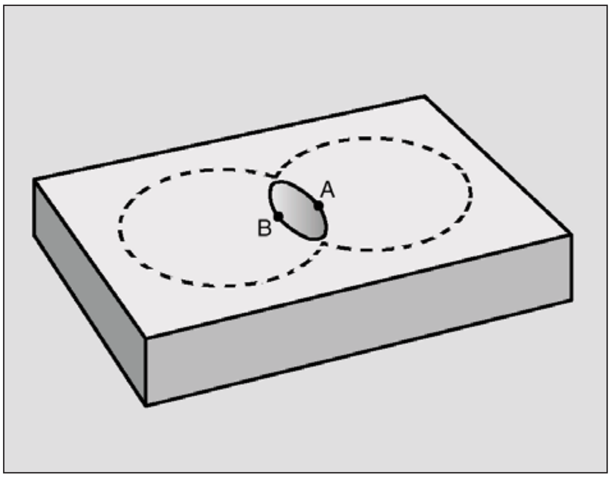
- Surface A must be a pocket and B an island.
- A must start outside of B.

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



Area of intersection

Only the area overlapped by both A and B is to be machined. (The areas covered by A or B alone are to be left unmachined.)

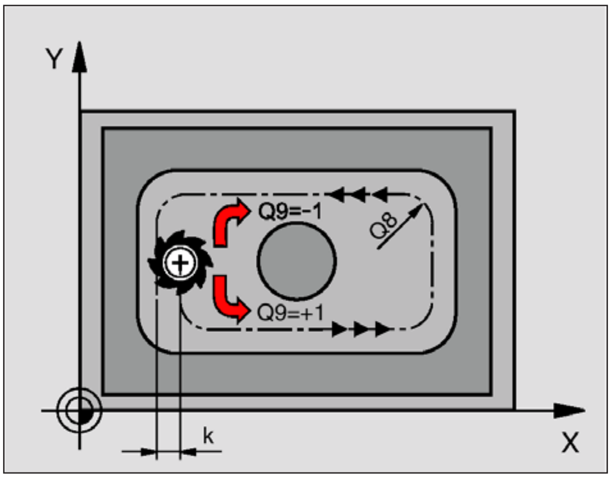
- A and B must be pockets.
- A must start inside of B.

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:

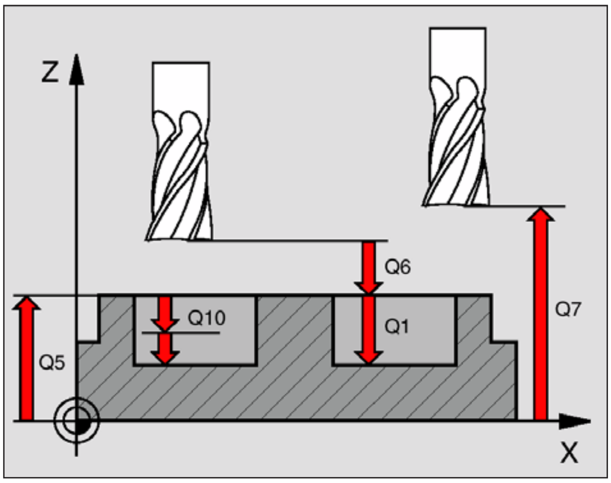
56 LBL 2
57 L X+90 Y+50 RR
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0



CONTOUR DATA (Cycle 20)

Machining data for the subprograms describing the subcontours are entered in Cycle 20.

Note:
Before programming, note the following:
Cycle 20 is DEF active which means that it becomes effective as soon as it is defined in the part program. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program depth = 0, the TNC does not execute that next cycle. The machining data entered in Cycle 20 are valid for Cycles 21 to 24. If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q19 cannot be used as program parameters.



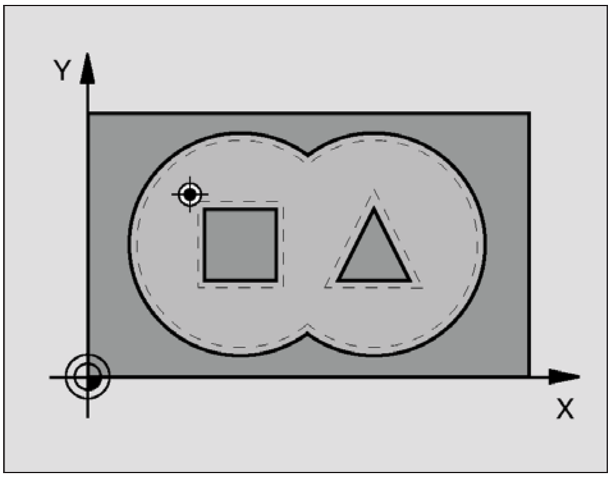
Example: NC blocks

```
57 CYCL DEF 20.0 CONTOUR DATA
    Q1=-20 ;MILLING DEPTH
    Q2=1 ;TOOL PATH OVERLAP
    Q3=+0.2 ;ALLOWANCE FOR SIDE
    Q4=+0.1 ;ALLOWANCE FOR FLOOR
    Q5=+30 ;SURFACE COORDINATE
    Q6=2 ;SET-UP CLEARANCE
    Q7=+80 ;CLEARANCE HEIGHT
    Q8=0.5 ;ROUNDING RADIUS
    Q9=+1 ;DIRECTION OF ROTATION
```

- **Milling depth** Q1 (incremental value): Distance between workpiece surface and bottom of pocket
- **Path overlap** factor Q2: Q2 x tool radius = stepover factor k
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the working plane
- **Finishing allowance for floor** Q4 (incremental value): Finishing allowance in the tool axis
- **Workpiece surface coordinate** Q5 (absolute value): Absolute coordinate of the workpiece surface
- **Set-up clearance** Q6 (incremental value): Distance between tool tip and workpiece surface
- **Clearance height** Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle)
- **Inside corner radius** Q8: Inside "corner" rounding radius; entered value is referenced to the tool midpoint path.
- **Direction of rotation ? Clockwise = -1** Q9: Machining direction for pockets
    - Clockwise (Q9 = –1 up-cut milling for pocket and island with M03)
    - Counterclockwise (Q9 = +1 climb milling for pocket and island with M03)

You can check the machining parameters during a program interruption and overwrite them if required.

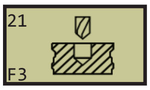
REAMING (Cycle 21)



Note:
When calculating the infeed points, the TNC does not account for the delta value DR programmed in a TOOL CALL block. In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

Process
Same as Cycle 1, Pecking.

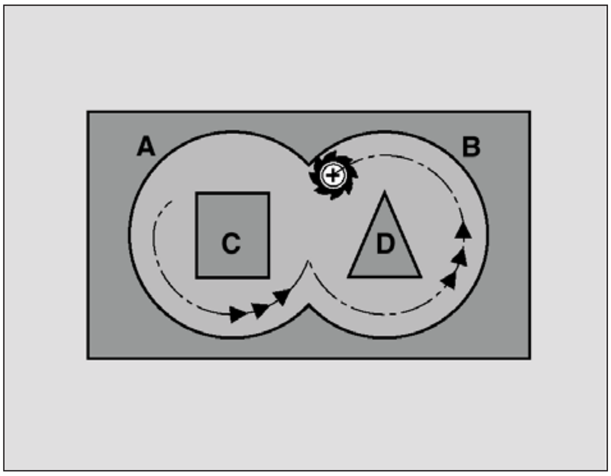
Application
Cycle 21 is for PILOT DRILLING of the cutter infeed points. It accounts for the allowance for side and the allowance for floor as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for roughing.



- **Plunging depth** Q10 (incremental value): Dimension by which the tool drills in each infeed (negative sign for negative working direction)
- **Feed rate for plunging** Q11: Traversing speed in mm/min during drilling
- **Rough-out tool number** Q13: Tool number of the roughing mill

Example: NC blocks

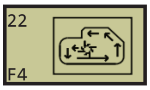
```
58 CYCL DEF 21.0 PILOT DRILLING
   Q10=+5 ;PLUNGING DEPTH
   Q11=100 ;FEED RATE FOR PLUNGING
   Q13=1 ;ROUGH-OUT TOOL
```



ROUGH-OUT (Cycle 22)

1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2. In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate Q12.
3. The island contours (here: C/D) are cleared out with an approach toward the pocket contour (here: A/B).
4. Then the TNC rough-mills the pocket contour retracts the tool to the clearance height.

Note:
Before programming, note the following:
This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

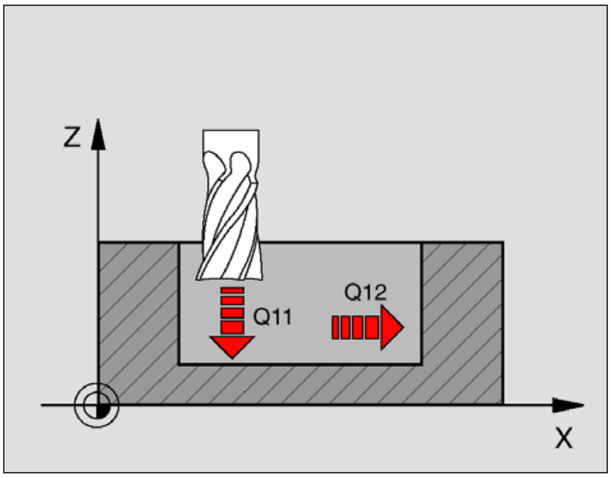


Example: NC blocks

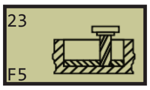
59 CYCL DEF 22.0 ROUGH-OUT
Q10=+5 ;PLUNGING DEPTH
Q11=100 ;FEED RATE FOR PLUNGING
Q12=350 ;FEED RATE FOR MILLING
Q18=1 ;COARSE ROUGHING TOOL
Q19=150 ;RECIPROCATION FEED RATE

- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed
- **Feed rate for plunging** Q11: Traversing speed of the tool in mm/min during penetration
- **Feed rate for milling** Q12: Traversing speed for milling in mm/min
- **Coarse roughing tool number** Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If there was no coarse roughing, enter ”0”; if you enter a value other than zero, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool. (Only for contours without islands available.)
- **Reciprocation feed rate** Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut (does not consider)

FLOOR FINISHING (Cycle 23)



Note:
The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.



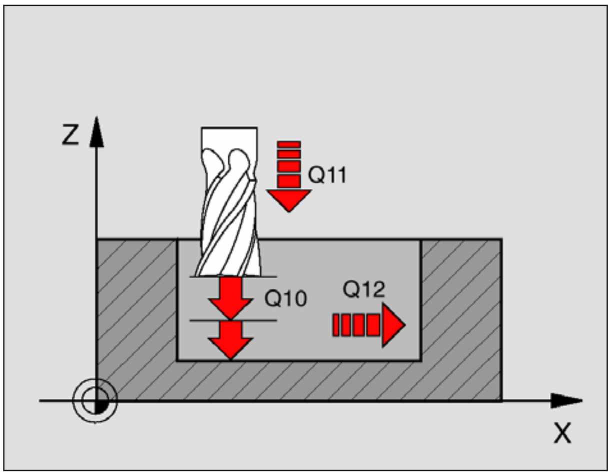
The tool approaches the machining plane smoothly (in a vertically tangential arc). The tool then clears the finishing allowance remaining from rough-out.

- **Feed rate for plunging**: Traversing speed of the tool during penetration
- **Feed rate for milling** Q12: Traversing speed for milling

Example: NC blocks

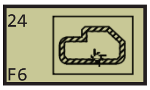
```
60 CYCL DEF 23.0 FLOOR FINISHING
   Q11=100 ;FEED RATE FOR PLUNGING
   Q12=350 ;FEED RATE FOR MILLING
```

SIDE FINISHING (Cycle 24)



The subcontours are approached and departed on a tangential arc. Each subcontour is finish-milled separately.

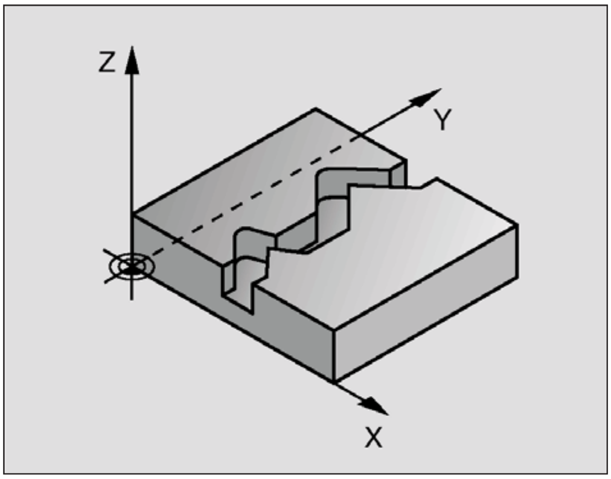
Note:
Before programming, note the following:
The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle 20) and the radius of the rough mill. This calculation also holds if you run Cycle 24 without having roughed out with Cycle 22; in this case, enter ”0” for the radius of the rough mill. The TNC automatically calculates the starting point for finishing. The starting point depends on the available space in the pocket.



Example: NC blocks
61 CYCL DEF 24.0 SIDE FINISHING
Q9=+1 ;DIRECTION OF ROTATION
Q10=+5 ;PLUNGING DEPTH
Q11=100 ;FEED RATE FOR PLUNGING
Q12=350 ;FEED RATE FOR MILLING
Q14=+0 ;ALLOWANCE FOR SIDE

- **Direction of rotation ? Clockwise = –1** Q9: Machining direction:
  +1:Counterclockwise (with M03)
  –1:Clockwise (with M03)
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed
- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration
- **Feed rate for milling** Q12: Traversing speed for milling
- **Finishing allowance for side** Q14 (incremental value): Enter the allowed material for several finish-milling operations. If you enter Q14 = 0, the entire smooth remainder is finished at one time.

CONTOUR TRAIN (Cycle 25)



In conjunction with Cycle 14 CONTOUR GEOMETRY, this cycle facilitates the machining of open contours (i.e., where the starting point of the contour is not the same as its end point).

Cycle 25 CONTOUR TRAIN offers considerable advantages over machining an open contour using positioning blocks:

- The TNC monitors the operation to prevent undercuts and surface blemishes. It is recommended that you run a graphic simulation of the contour before execution.
- If the radius of the selected tool is too large, the corners of the contour may have to be reworked.
- The contour can be machined throughout by up-cut or by climb milling. The type of milling even remains effective when the contours are mirrored.
- The tool can traverse back and forth for milling in several infeeds: This results in faster machining.
- Allowance values can be entered in order to perform repeated rough-milling and finish-milling operations.

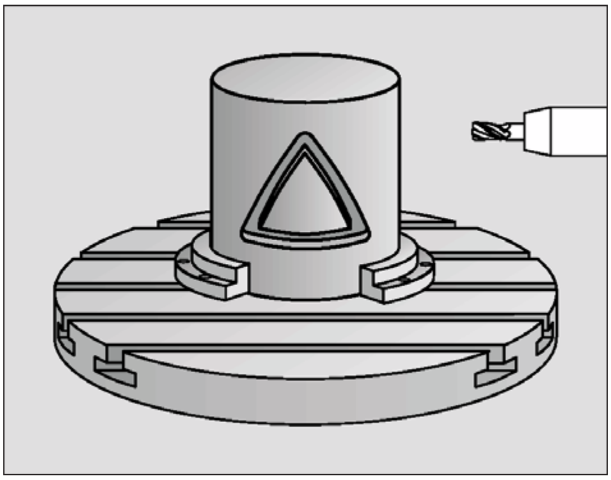
Note:
Before programming, note the following:
The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis.
If you program DEPTH = 0, the cycle will not be executed. The TNC takes only the first label of Cycle 14 CONTOUR GEOMETRY into account.
Cycle 20 CONTOUR DATA is not required. Positions that are programmed in incremental dimensions immediately after Cycle 25 are referenced to the position of the tool at the end of the cycle.

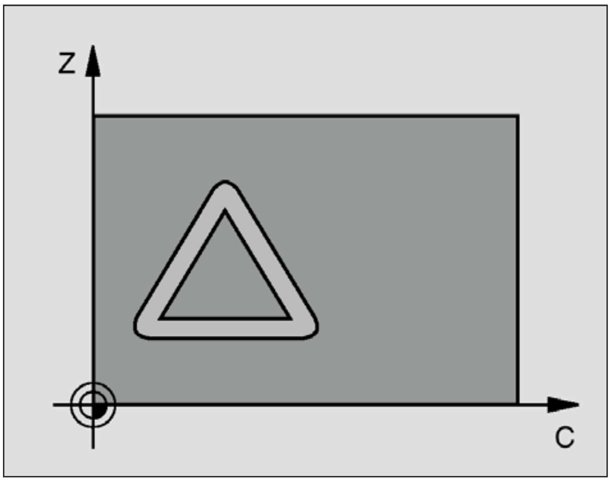
- **Milling depth** Q1 (incremental value): Distance between workpiece surface and contour floor
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the working plane
- **Workpiece surface coordinate** Q5 (absolute value): Absolute coordinate of the workpiece surface referenced to the workpiece datum
- **Clearance height** Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece. Position for tool retraction at the end of the cycle.
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed
- **Feed rate for plunging** Q11: Traversing speed of the tool in the tool axis
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane
- **Climb or up-cut ?= –1** Q15:
  Climb milling: Input value = +1
  Up-cut milling: Input value = –1
  To enable climb milling and up-cut milling alternately in several infeeds:Input value = 0

Example: NC blocks

```
62 CYCL DEF 25.0 CONTOUR TRAIN
    Q1=-20 ;MILLING DEPTH
    Q3=+0 ;ALLOWANCE FOR SIDE
    Q5=+0 ;WORKPIECE SURFACE COORD.
    Q7=+50 ;CLEARANCE HEIGHT
    Q10=+5 ;PLUNGING DEPTH
    Q11=100 ;FEED RATE FOR PLUNGING
    Q12=350 ;FEED RATE FOR MILLING
    Q15=-1 ;FRAESART
```

CYLINDER SURFACE (Cycle 27)





This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. Use Cycle 28 if you wish to mill guide notches onto the cylinder surface.

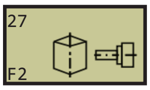
The contour is described in a subprogram identified in Cycle 14 CONTOUR GEOMETRY.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, is parallel to the Z axis. The path functions L, CHF, CR, RND APPR (except APPR LCT) and DEP are available.

The dimensions in the rotary axis can be entered as desired either in degrees or in mm (or inches). You can select the desired dimension type in the cycle definition.

1 The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.
3 At the end of the contour, the TNC returns the tool to the setup clearance and returns to the point of penetration;
4 Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.
5 Then the tool moves to the setup clearance.

Note:
Before programming, note the following:
The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle. The algebraic sign for the cycle parameter DEPTH determines the working direction. A negative sign bores in the direction of the neagative spindle axis. If you program DEPTH = 0, the cycle will not be executed. This cycle requires a center-cut end mill (ISO 1641). The cylinder must be set up centered on the rotary table. The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.
This cycle can also be used in a tilted working plane. The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis.



Example: NC blocks

```
63 CYCL DEF 27.0 CYLINDER SURFACE
    Q1=-8 ;MILLING DEPTH
    Q3=+0 ;ALLOWANCE FOR SIDE
    Q6=+0 ;SET-UP CLEARANCE
    Q10=+3 ;PLUNGING DEPTH
    Q11=100 ;FEED RATE FOR PLUNGING
    Q12=350 ;FEED RATE FOR MILLING
    Q16=25 ;RADIUS
    Q17=0 ;DIMENSION TYPE (ANG/LIN)
```

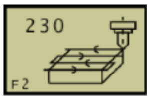
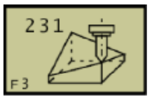
- **Milling depth** Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour
- **Finishing allowance for side** Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation
- **Set-up clearance** Q6 (incremental value): Distance between the tool tip and the cylinder surface
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed
- **Feed rate for plunging** Q11: Traversing speed of the tool in the tool axis
- **Feed rate for milling** Q12: Traversing speed of the tool in the working plane
- **Cylinder radius** Q16: Radius of the cylinder on which the contour is to be machined
- **Dimension type ? ang./lin.** Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)

Cycles for multipass milling

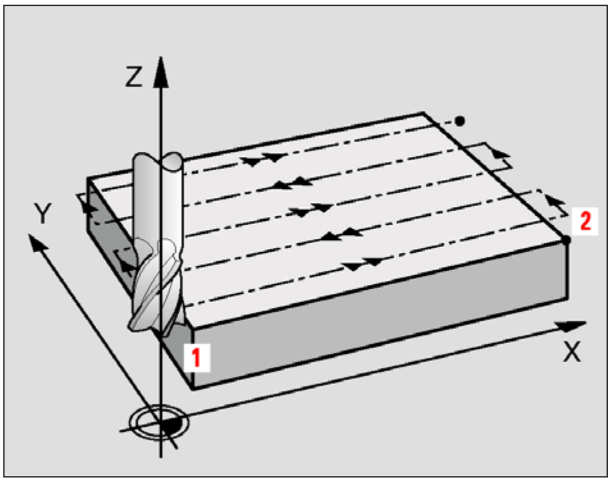
Overview

The WinNC offers two cycles for machining the following surface types:Die TNC stellt zwei Zyklen zur Verfügung, mit denen Sie Flächen mit folgenden Eigenschaften bearbeiten können:

- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

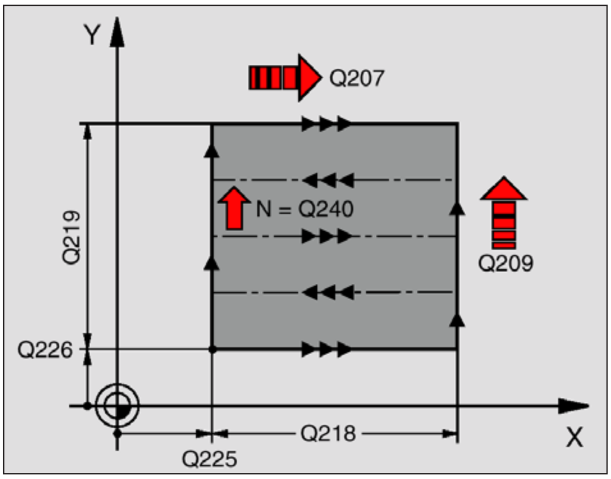
Cycle	Soft key
230 MULTIPASS MILLING For flat rectangular surfaces	
231 RULED SURFACE For oblique, inclined or twisted surfaces	

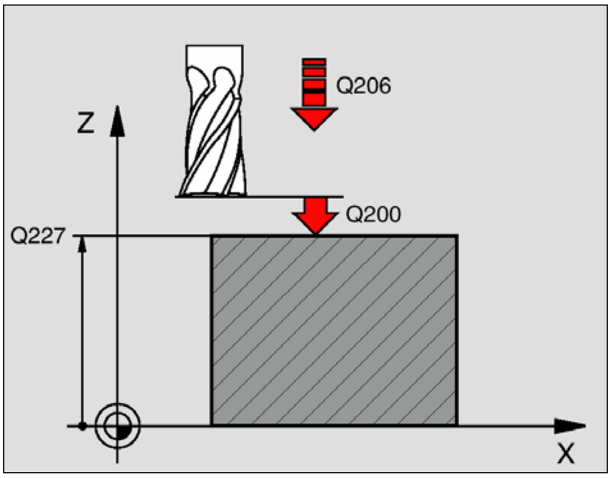
MULTIPASS MILLING (Cycle 230)



1 From the current position in the working plane, the WinNC positions the tool in rapid traverse FMAX to the starting point 1; the WinNC moves the tool by its radius to the left and upward.
2 The tool then moves in FMAX in the tool axis to set-up clearance. From there it approaches the programmed starting position in the tool axis at the feed rate for plunging.
3 The tool then moves as the programmed feed rate for milling to the end point 2. The WinNC calculates the end point from the programmed starting point, the program length, and the tool radius.
4 The WinNC offsets the tool to the starting point in the next pass at the stepover feed rate. The offset is calculated from the programmed width and the number of cuts.
5 The tool then returns in the negative direction of the first axis.
6 Multipass milling is repeated until the programmed surface has been completed.
7 At the end of the cycle, the tool is retracted in FMAX to set-up clearance.

Note:
Before programming, note the following:
From the current position, the WinNC positions the tool at the starting point 1, first in the working plane and then in the tool axis. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

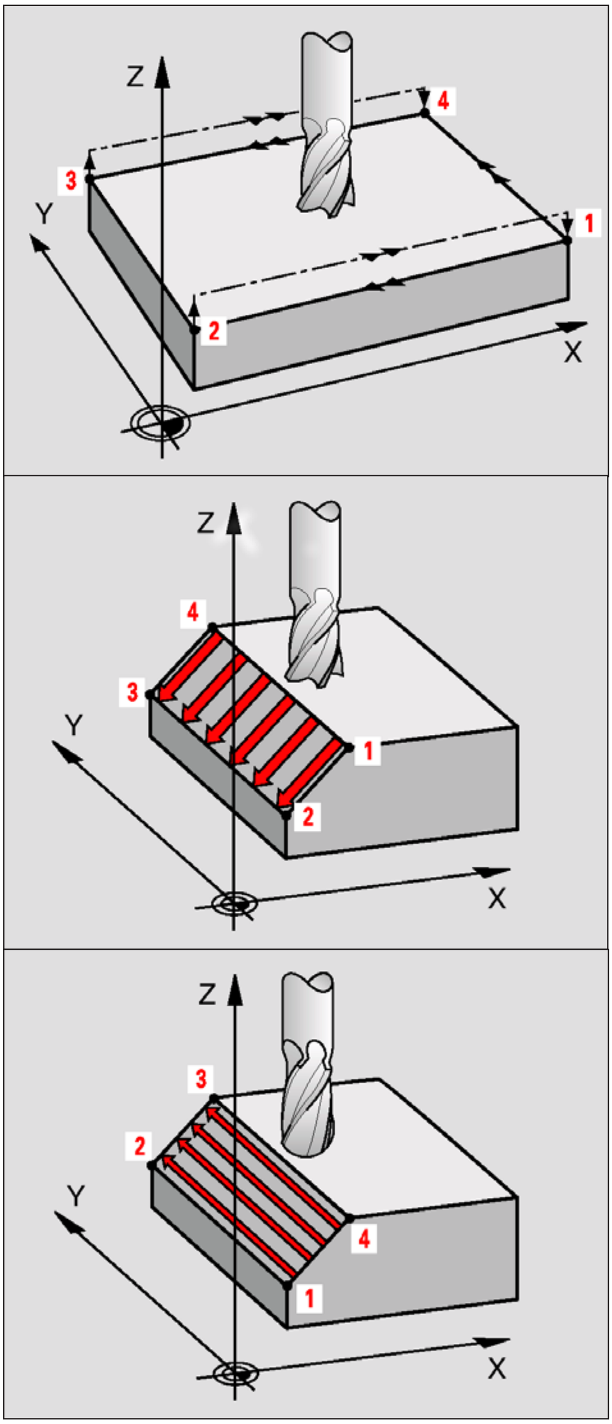




- **Starting point in 1st axis** Q225 (absolute value): Minimum point coordinate of the surface to be multipass-milled in the reference axis of the working plane
- **Starting point in 2nd axis** Q226 (absolute value): Minimum-point coordinate of the surface to be multipass-milled in the minor axis of the working plane
- **Starting point in 3rd axis** Q227 (absolute value): Height in the spindle axis at which multipass-milling is carried out
- **First side length** Q218 (incremental value): Length of the surface to be multipass-milled in the reference axis of the working plane, referenced to the starting point in 1st axis
- **Second side length** Q219 (incremental value): Length of the surface to be multipass-milled in the minor axis of the working plane, referenced to the starting point in 2nd axis
- **Number of cuts** Q240: Number of passes to be made over the width
- **Feed rate for plunging** 206: Traversing speed of the tool in mm/min when moving from set-up clearance to the milling depth
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Stepover feed rate** Q209: Traversing speed of the tool in mm/min when moving to the next pass. If you are moving the tool transversely in the material, enter Q209 to be smaller than Q207 If you are moving it transversely in the open, Q209 may be greater than Q207.
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and milling depth for positioning at the start and end of the cycle.

Example: NC blocks

```
71 CYCL DEF 230 MULTIPASS MILLNG
   Q225=+10 ;STARTNG PNT 1ST AXIS
   Q226=+12 ;STARTNG PNT 2ND AXIS
   Q227=+2.5 ;STARTNG PNT 2ND AXIS
   Q218=150 ;FIRST SIDE LENGTH
   Q219=75 ;SECOND SIDE LENGTH
   Q240=25 ;NUMBER OF CUTS
   Q206=150 ;FEED RATE FOR PLUNGING
   Q207=500 ;FEED RATE FOR MILLING
   Q209=200 ;STEPOVER FEED RATE
   Q200=2 ;SET-UP CLEARANCE
```



RULED SURFACE (Cycle 231)

1 From the current position, the WinNC positions the tool in a linear 3-D movement to the starting point **1**.
2 The tool subsequently advances to the stopping point **2** at the feed rate for milling.
3 From this point, the tool moves in rapid traverse FMAX by the tool diameter in the positive tool axis direction, and then back to starting point **1**.
4 At the starting point **1** the WinNC moves the tool back to the last traversed Z value.
5 Then the WinNC moves the tool in all three axes from point **1** in the direction of point **4** to the next line.
6 From this point, the tool moves to the stopping point on this pass. The WinNC calculates the end point from point **2** and a movement in the direction of point **3**.
7 Multipass milling is repeated until the programmed surface has been completed.
8 At the end of the cycle, the tool is positioned above the highest programmed point in the tool axis, offset by the tool diameter.

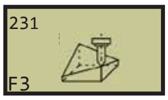
Cutting motion
The starting point, and therefore the milling direction, is selectable because the WinNC always moves from point **1** to point **2** and in the total movement from point **1** / **2** to point **3** / **4**. You can program point **1** at any corner of the surface to be machined.
If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways

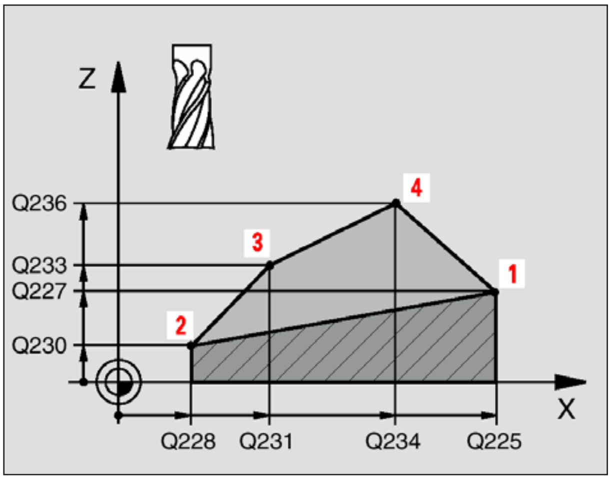
- A shaping cut (spindle axis coordinate of point **1** greater than spindle-axis coordinate of point **2**) for slightly inclined surfaces.
- A drawing cut (spindle axis coordinate of point **1** smaller than spindle-axis coordinate of point **2**) for steep surfaces.
- When milling twisted surfaces, program the main cutting direction (from point **1** to point **2**) parallel to the direction of the steeper inclination.

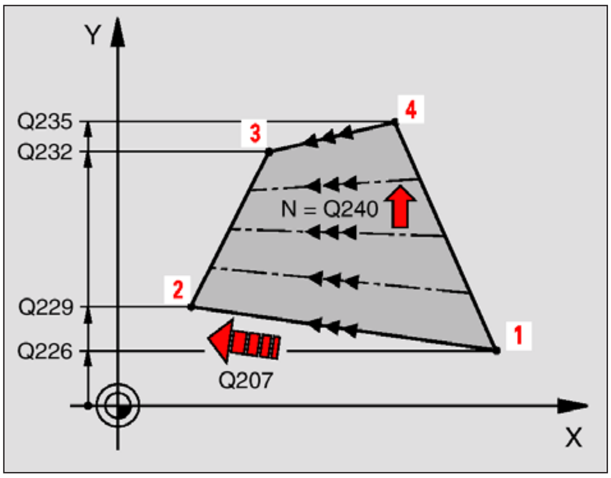
If you are using a spherical cutter for the machining operation, you can optimize the surface finish in the following way

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

Note:
Before programming, note the following:
The WinNC positions the tool from the current position in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur. The WinNC moves the tool with radius compensation R0 to the programmed positions.
If required, use a center-cut end mill (ISO 1641).







Example: NC blocks

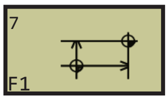
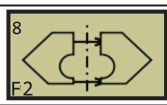
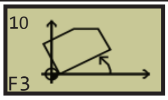
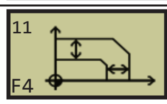
72 CYCL DEF 231 RULED SURFACE
Q225=+0 ;STARTNG PNT 1ST AXIS
Q226=+5 ;STARTNG PNT 2ND AXIS
Q227=-2 ;STARTING PNT 3RD AXIS
Q228=+100 ;2ND POINT 1ST AXIS
Q229=+15 ;2ND POINT 2ND AXIS
Q230=+5 ;2ND POINT 3RD AXIS
Q231=+15 ;3RD POINT 1ST AXIS
Q232=+125 ;3RD POINT 2ND AXIS
Q233=+25 ;3RD POINT 3RD AXIS
Q234=+15 ;4TH POINT 1ST AXIS
Q235=+125 ;4TH POINT 2ND AXIS
Q236=+25 ;4TH POINT 3RD AXIS
Q240=40 ;NUMBER OF CUTS
Q207=500 ;FEED RATE FOR MILLING

- **Starting point in 1st axis** Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the reference axis of the working plane
- **Starting point in 2nd axis** Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the minor axis of the working plane
- **Starting point in 3rd axis** Q227 (absolute value): Starting point coordinate of the surface to be multipass-milled in the tool axis
- **2nd point in 1st axis** Q228 (absolute value): Stopping point coordinate of the surface to be multipass milled in the reference axis of the working plane
- **2nd point in 2nd axis** Q229 (absolute value): Stopping point coordinate of the surface to be multipass milled in the minor axis of the working plane
- **2nd point in 3rd axis** Q230 (absolute value): Stopping point coordinate of the surface to be multipass milled in the tool axis
- **3rd point in 1st axis** Q231 (absolute value): Coordinate of point 3 in the reference axis of the working plane
- **3rd point in 2nd axis** Q232 (absolute value): Coordinate of point 3 in the minor axis of the working plane
- **3rd point in 3rd axis** Q233 (absolute value): Coordinate of point 3 in the tool axis
- **4th point in 1st axis** Q234 (absolute value): Coordinate of point 4 in the reference axis of the working plane
- **4th point in 2nd axis** Q235 (absolute value): Coordinate of point 4 in the minor axis of the working plane
- **4th point in 3rd axis** Q236 (absolute value): Coordinate of point 4 in the tool axis
- **Number of cuts** Q240: Number of passes to be made between points 1 and 4, 2 and 3.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling. The WinNC performs the first step at half the programmed feed rate.

Coordinate Transformation Cycles

Overview

Once a contour has been programmed, you can position it on the workpiece at various locations and in different sizes through the use of coordinate transformations. The WinNC provides the following coordinate transformation cycles:

Cycle	Soft key
7 DATUM SHIFT For shifting contours directly within the program or from datum tables	
8 MIRROR IMAGE Mirroring contours	
10 ROTATION For rotating contours in the working place	
11 SCALING FACTOR For increasing or reducing the size of contours	

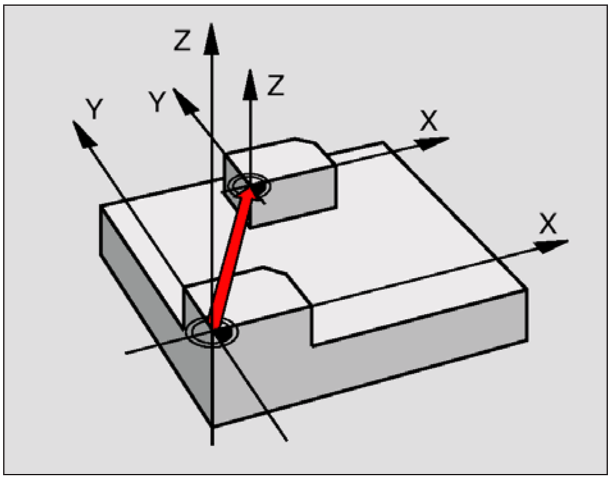
Effect of coordinate transformation

Beginning of effect: A coordinate transformation becomes effective as soon as it is defined — it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M02, M30, or an END PGM block
- Select a new program

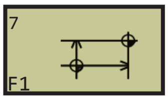
DATUM SHIFT (Cycle 7)



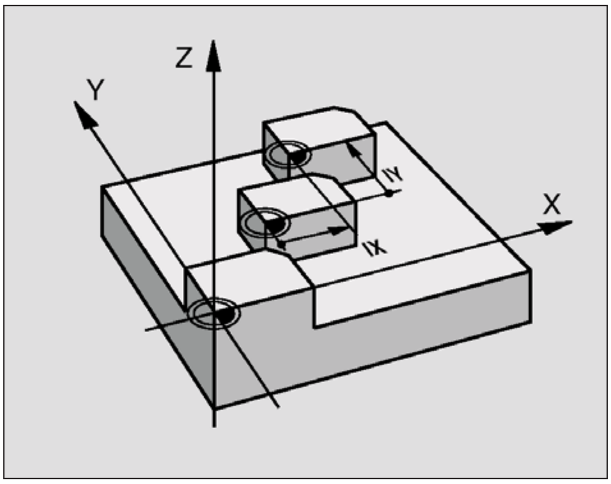
A datum shift allows machining operations to be repeated at various locations on the workpiece. The coordinate system is displaced at an appropriate point in the machinig room.

The work piece datum shift can be displaced within a subroutine optional often.

Effect
When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The WinNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.



- **Datum shift**: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid — this can be a datum which has already been shifted.

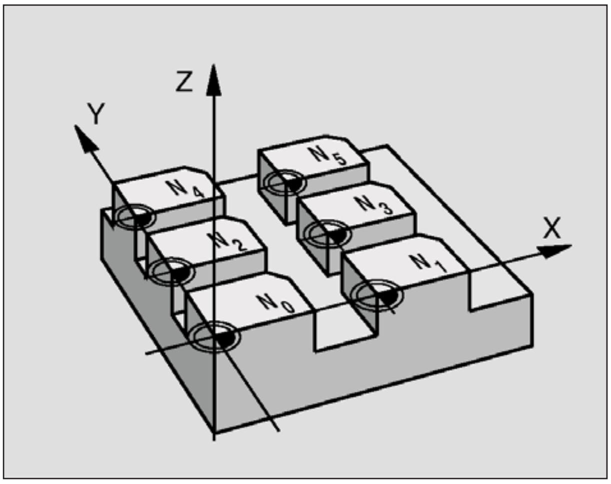


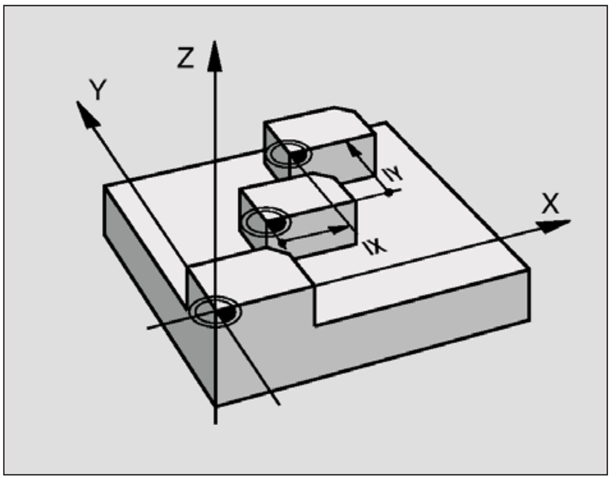
Cancellation
A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Status Displays
- The actual position values are referenced to the active (shifted) datum.
- All of the position values shown in the additional status display are referenced to the manually set datum.

Example: NC blocks
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
16 CYCL DEF 7.3 Z-5
15 CYCL DEF 7.2 Y+40

DATUM SHIFT with datum tables (Cycle 7)





Note:
If you are using datum shifts with datum tables, then use the SEL TABLE function to activate the desired datum table from the NC program. If you work without SEL-TABLE, then you must activate the desired datum table before the test run or the program run. (This applies also for the programming graphics):

- Use the file management to select the desired table for a test run in the **Test Run** operating mode: The table receives the status S.
- Use the file management in a program run mode to select the desired table for a program run: The table receives the status M.

Datums from a datum table can be referenced either to the current datum or to the machine datum. The coordinate values from datum tables are only effective with absolute coordinate values. New lines can only be inserted at the end of the table.

Example: NC blocks
77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1 #5

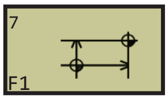
Function

Datum tables are used for

- frequently recurring machining sequences at various locations on the workpiece
- frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.

- **Datum shift**: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the WinNC activates the datum number found in the Q parameter.



Cancellation

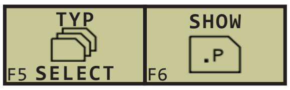
- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table
- Execute a datum shift to the coordinates X=0; Y=0 etc. directly with a cycle definition.

Selecting a datum table in the part program

With the **SEL TABLE** function you select the table from which the WinNC takes the datums:

- To select the functions for program call, press the PGM CALL key.
- Press the TOOL TABLE soft key.
- Enter the complete path name of the datum table and confirm your entry with the ENT key.

> **Note:**
> Program a SEL TABLE block before Cycle 7 Datum Shift. A datum table selected with SEL TABLE remains active until you select another datum table with SEL TABLE or through PGM MGT.

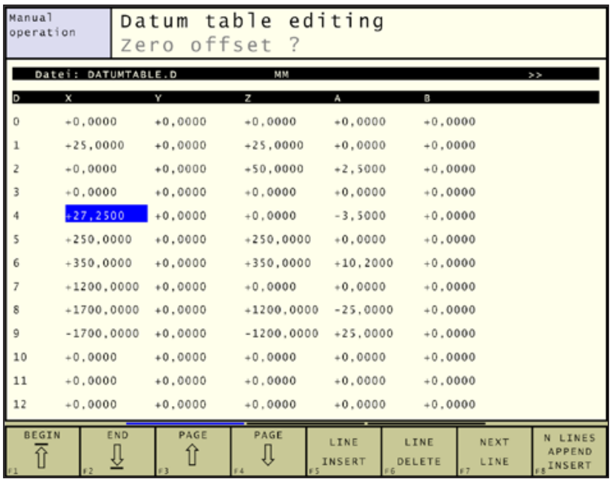


Editing a datum table

Select the datum table in the **Programming and Editing** mode of operation.

- To call the file manager, press the PGM MGT key, see "File Management: Fundamentals" (capture C).
- Display the datum tables: Press the soft keys SELECT TYPE and SHOW .D.
- Select the desired table or enter a new file name.
- Edit the file. The soft-key row comprises the following functions for editing:

Function	Soft key
Select beginning of table	BEGIN ⇧ (F1)
Select end of table	END ⇩ (F2)
Go to the previous page	PAGE ⇧ (F1)
Go to the next page	PAGE ⇩ (F4)
Insert line (only possible at the end of table)	LINE INSERT (F5)
Delete line	LINE DELETE (F6)
Confirm the entered line and go to the beginning of the next line	NEXT LINE (F7)
Add the entered number of lines (reference points) to the end of the table	N LINES APPEND INSERT (F8)



Edit a pocket table in a Program Run operating mode.

In a program run mode you can select the active datum table. Press the DATUM TABLE soft key. You can then use the same editing functions as in the **Programming and Editing** mode of operation.

Configuring the datum table

On the second and third soft-key rows you can define for each datum table the axes for which you wish to set the datums. In the standard setting all of the axes are active. If you wish to exclude an axis, set the corresponding soft key to OFF. The WinNC then deletes that column from the datum table. If you do not wish to define a datum table for an active axis, press the NO ENT key. The WinNC then enters a dash in the corresponding column.

To leave a datum table

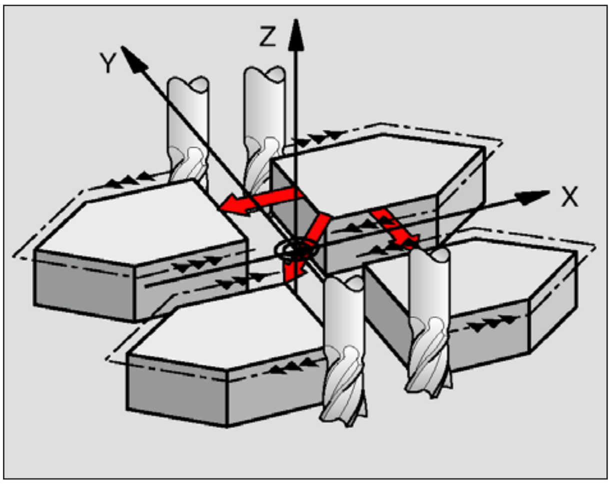
Select a different type of file in file management and choose the desired file.

Status Displays

If datums in the table are referenced to the machine datum, then:

- The actual position values are referenced to the active (shifted) datum.
- All of the position values shown in the additional status display are referenced to the machine datum, whereby the WinNC accounts for the manually set datum.

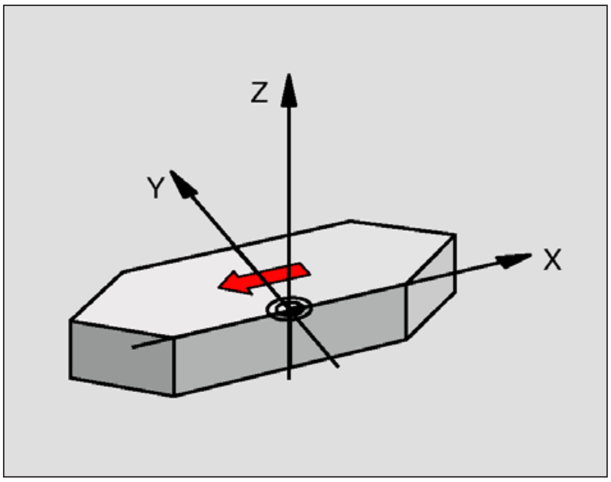
MIRROR IMAGE (Cycle 8)



The WinNC can machine the mirror image of a contour in the working plane.

Effect
The mirror image cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

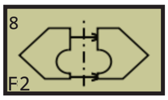
- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.



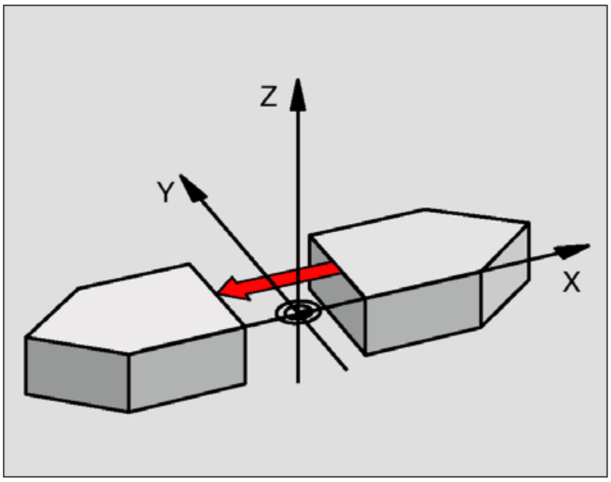
The result of the mirror image depends on the location of the datum:

- If the datum lies on the contour to be mirrored, the element simply flips over.
- If the datum lies outside the contour to be mirrored, the element also ”jumps” to another location.

Note:
If you mirror only one axis, the machining direction is reversed for the machining cycles (cycles 2xx). The machining direction remains the same for older machining cycles, such as Cycle 4 POCKET MILLING:



- **Mirrored axis?**: Enter the axis to be mirrored. You can mirror all axes, including rotary axes, except for the spindle axis and its auxiliary axes. You can enter up to three axes.

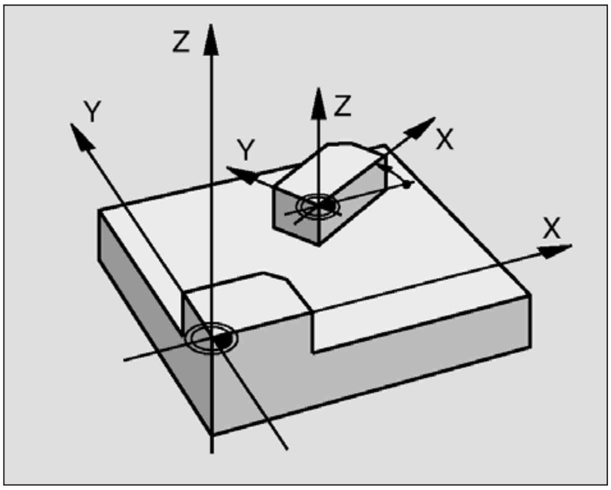


Reset
Program the MIRROR IMAGE cycle once again with NO ENT.

Example: NC blocks
79 CYCL DEF 8.0 MIRROR IMAGE
80 CYCL DEF 8.1 X Y U

ROTATION (CYCLE 10)

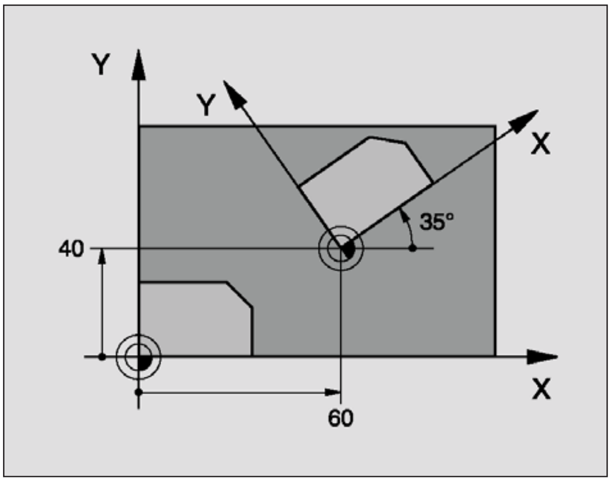
The WinNC can rotate the coordinate system about the active datum in the working plane within a program.



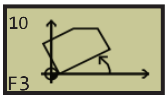
Effect
The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

Reference axis for the rotation angle:

- X/Y plane X axis
- Y/Z plane Y axis
- Z/X planeZ axis



> **Note:**
> Before programming, note the following:
> An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary. After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

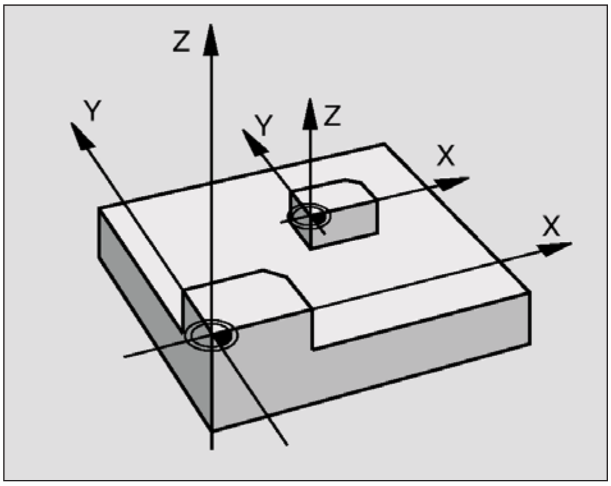


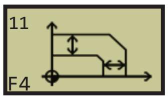
Example: NC blocks

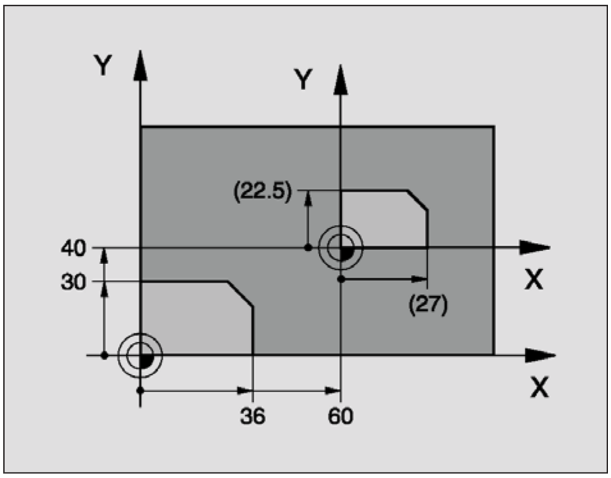
12 CALL LBL1
13 CYCL DEF 7.0 DATUM SHIFT
14 CYCL DEF 7.1 X+60
15 CYCL DEF 7.2 Y+40
16 CYCL DEF 10.0 ROTATION
17 CYCL DEF 10.1 ROT+35
18 CALL LBL1

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

Cancellation
Program the ROTATION cycle once again with a rotation angle of 0°.







Example: NC blocks

11 CALL LBL1
12 CYCL DEF 7.0 DATUM SHIFT
13 CYCL DEF 7.1 X+60
14 CYCL DEF 7.2 Y+40
15 CYCL DEF 11.0 SCALING
16 CYCL DEF 11.1 SCL 0.75
17 CALL LBL1

SCALING FACTOR (Cycle 11)

The WinNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.

Effect
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

Scaling factor

- in the working plane, or on all three coordinate axes at the same time
- to the 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.

- **Scalingfactor?**: Enter the scaling factor SCL. The WinNC multiplies the coordinates and radii by the SCL factor (as described under „Effect“ above)

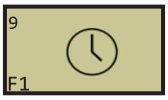
Enlargement: SCL greater than 1 (up to 99.999 999)
Reduction: SCL less than 1 (down to 0.000 001)

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

Special Cycles

Overview

Cycle	Soft key
9 DWELL TIME	
12 PROGRAM CALL	
13 ORIENTED SPINDLE STOP	



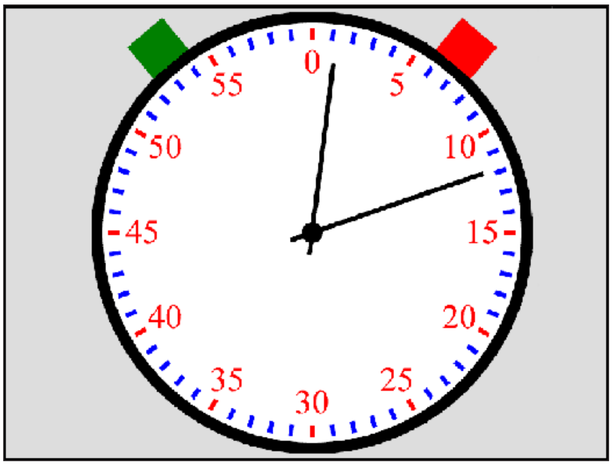
DWELL TIME (Cycle 9)

This causes the execution of the next block within a running program to be delayed by the programmed dwell time. A dwell time can be used for such purposes as chip breaking.

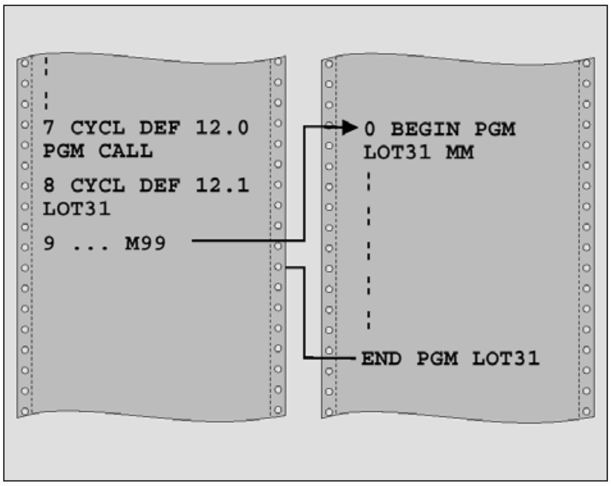
Effect
Cycle 9 becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.

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

Input range 0 to 3600 s (1 hour) in 0.001 s steps



Example: NC blocks
89 CYCL DEF 9.0 DWELL TIME
90 CYCL DEF 9.1 DWELL 1.5



PROGRAM CALL (Cycle 12)

Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs and then called like fixed cycles.

Note:
Before programming, note the following:
If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only to enter the program name. If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path (for example TNC:\KLAR35\FK1\50.H. If you want to define an ISO program to be a cycle, enter the file type .I behind the program name.

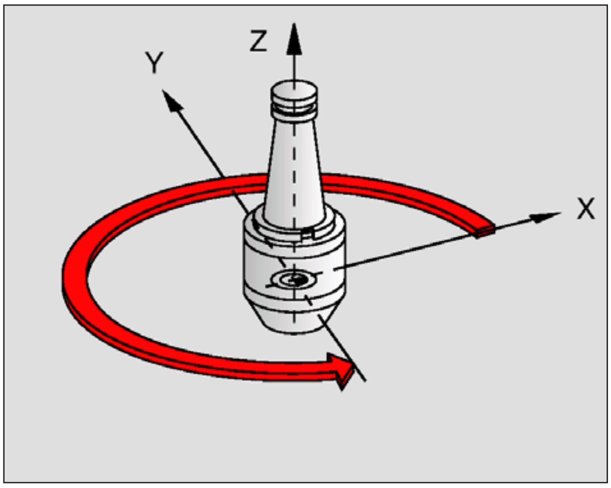
- **Program name**: Enter the name of the program you want to call and, if necessary, the directory it is located in.

Call the program with
- CYCL CALL (separate block) or
- M99 (blockwise)

Example: NC blocks
55 CYCL DEF 12.0 PGM CALL
56 CYCL DEF 12.1 PGM TNC:\KLAR35\FK1\50.H
57 L X+20 Y+50 FMAX M99

Example: Program call
A callable program 50 is to be called into a program via a cycle call.

ORIENTED SPINDLE STOP (Cycle 13)



Note:
Cycle 13 is used internally for machining cycles 202, 204 and 209. Please note that, if required, you must program Cycle 13 again in your NC program after one of the machining cycles mentioned above.

The control can control the machine tool spindle and rotate it to a given angular position.

- **Angle of orientation**: Enter the angle according to the reference axis of the working plane
  Input range: 0 to 360°
  Input resolution: 0,1°

Oriented spindle stops are required for tool changing systems with a defined tool change position.

Example: NC blocks
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180

Subprograms

Labeling subprograms and program section repeats

Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as desired.

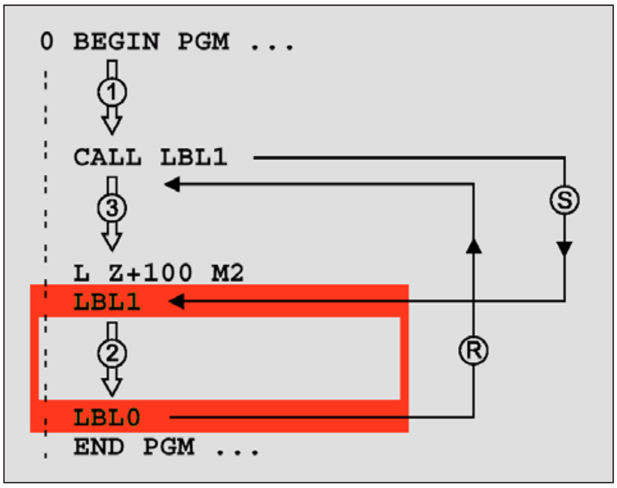
Labels

The beginning of subprograms and program section repeats are marked in a part program by labels.

A label is identified by a number between 1 and 254. Each label number can be set only once in a program with LABEL SET.

Note:
If a label is set more than once, the WinNC sends an error message at the end of the LBL SET block.

LABEL 0 (LBL 0) is used to mark the end of a subprogram and can therefore be used as often as desired.



Subprograms

Operating sequence

1 The WinNC executes the part program until a subprogram is called with CALL LBL.
2 The subprogram is then executed from beginning to end. The subprogram end is marked LBL 0.
3 The WinNC then resumes the part program from the block that follows the subprogram call.

Programming notes

- A main program can contain up to 254 subprograms.
- You can call subprograms in any sequence and as often as desired.
- A subprogram must not call itsself.
- Write subprograms at the end of the main program (behind the block with M2 or M30).
- If subprograms are located before the block with M02 or M30 in the part program, they will be executed at least once even if they are not called.

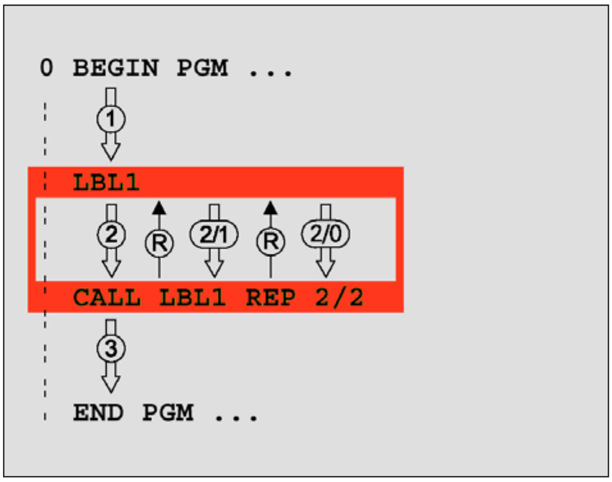
Programming a subprogram

- To mark the beginning, press the LBL SET key and enter a label number.
- Enter a subprogram number
- To mark the end, press the LBL SET key and enter the label number „ 0“.

Calling a subprogram

- To call a subprogram, press the LBL CALL key.
- **Label number**: Enter the label number of the subprogram you wish to call.
- **Repeat REP**: Ignore the dialogue with the NO ENT key. Repeat REP is only used for program section repeats.

Note:
CALL LBL 0 is not permitted since label 0 is only used to mark the end of a subprogram.



Program section repeats

Label LBL

The beginning of a program section repeat is marked by the label LBL.
A program section repeat ends with CALL LBL/ REP.

Operating sequence

1 The WinNC executes the part program until the end of the program section (CALL LBL /REP). This means, that the LABEL is executed once by the WinNC without any separat call.
2 The WinNC then repeats the program section between the called LABEL and the label call CALL LBL/ REP as many times as you entered at REP.
3 The WinNC then resumes the part program after the last repetition.

Programming notes

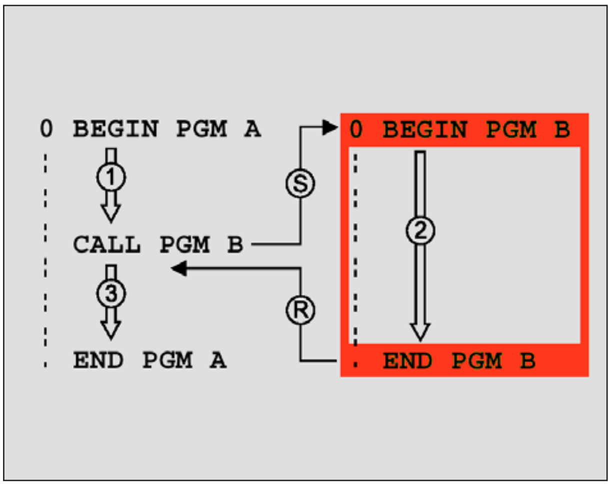
- You can repeat a program section up to 65 534 times in succession.
- The number behind the slash after REP indicates the number of program section repetitions that are still to run.
- The total number of times the program section is executed is always one more than the programmed number of repeats.

Programming a program section repeat:

- To mark the beginning, press the LBL SET key and enter a LABEL number for the program section you wish to repeat.
- Enter the program section.

Calling a program section repeat

- Press the LBL CALL key and enter the label number of the program section you want to repeat as well as the number of repeats REP.



Separate any program as subprogram

Operating sequence

1. The WinNC executes the part program until you call another program with CALL PGM.
2. The called program is then executed until its end.
3. The WinNC then resumes the (calling) part program with the block that follows the program call.

Programming notes

- No labels are needed to use any desired program as a subprogram.
- The called program must not contain the miscellaneous functions M2 or M30.
- The called program must not contain a program call CALL PGM into the calling program, otherwise an infinite loop will result.

Calling any program as a subprogram

- To select the functions for the program call, press the PGM CALL key.
- Press the PROGRAM soft key.
- Enter the complete path name of the program you want to call and confirm your entry with the ENT key.

Note:
The program you are calling must be stored in the hard disk of the WinNC.
If you only enter the program name, the program you call must be located in the same directory as the program you are calling it from.
If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path name, e.g.: TNC:\ZW35\SCHRUPP\PGM1.H

Nesting

Types of nesting

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Repeated subprograms
- Program section repeats within a subprogram

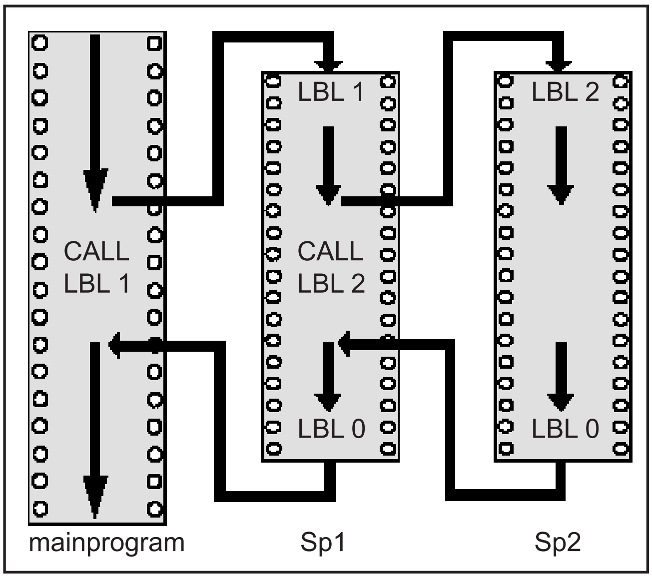
Nesting depth

The nesting depth determines how often program sections or subprograms may contain further subprograms or program section repeats.

- Maximum nesting depth for subprograms: 8
- Maximum nesting depth for main program calls: 4
- You can nest program section repeats as often as desired.

Subprogram within a subprogram

Example NC blocks	
0 BEGIN PGM UPGMS MM	Calling a subprogram at LBL 1
...	
17 CALL LBL 1	Calling a subprogram at LBL 1
...	
35 L Z+100 R0 FMAX M2	Last program block of the main program (with M2)
36 LBL 1	Beginning of subprogram 1
...	
39 CALL LBL 2	Subprogram marked with LBL2 is called
...	
45 LBL 0	End of subprogram 1
46 LBL 2	Beginning of subprogram 2
...	
62 LBL 0	End of subprogram 2
63 END PGM UPGMS MM	



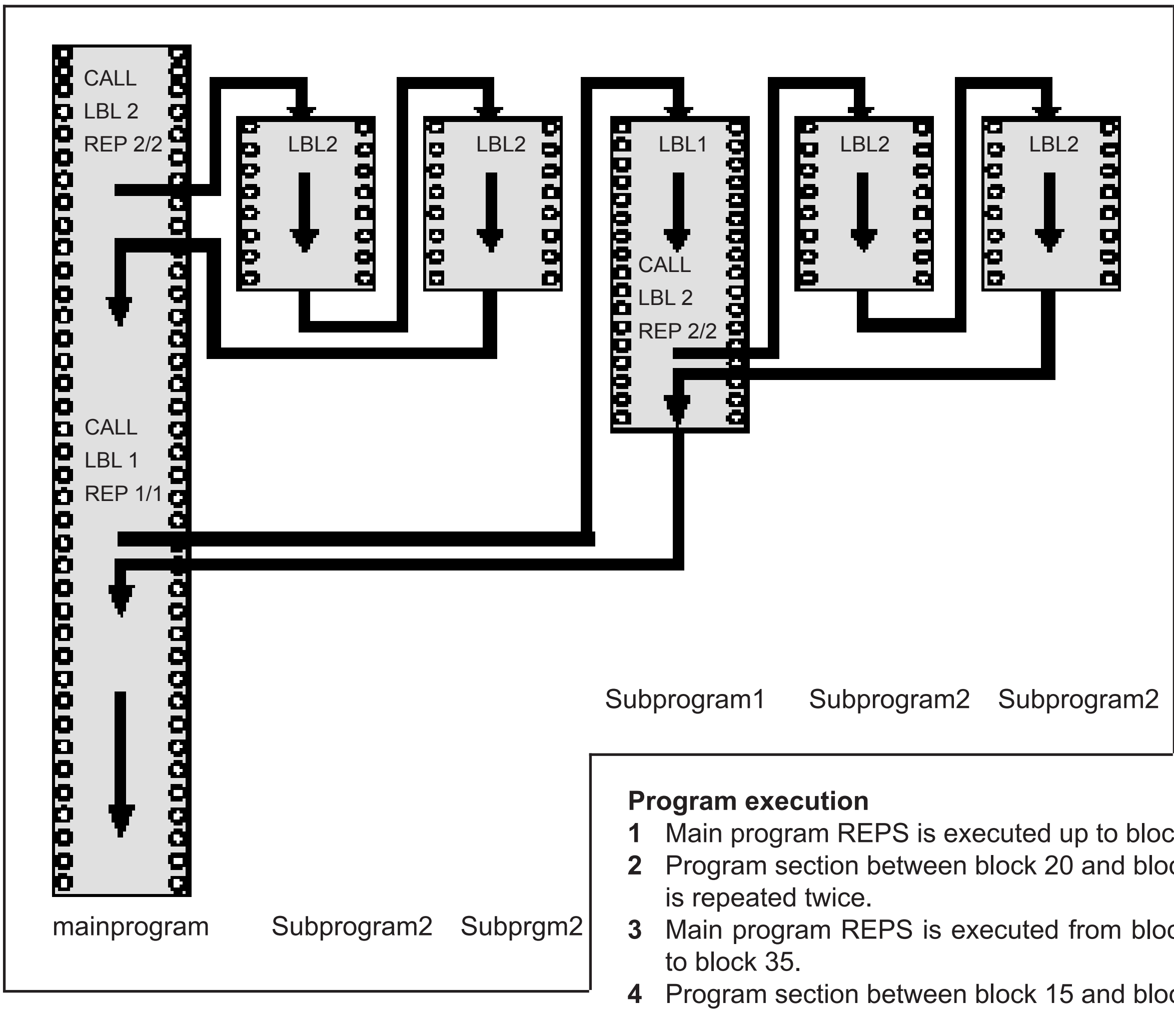
Program execution

1. Main program UPGMS is executed up to block 17.
2. Subprogram 1 (Sp1) is called and executed up to block 39.
3. Subprogram 2 (Sp2) is called and executed up to block 62. End of subprogram 2 and return jump to the subprogram from which it was called.
4. Subprogram 1 is executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program UPGMS.
5. Main program UPGMS is executed from block 18 up to block 35. Return jump to block 0 and end of program.

Repeating program section repeats

Example NC blocks

0 BEGIN PGM REPS MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2/2	The program section between this block and LBL 2 (block 20) is repeated twice.
...	
35 CALL LBL 1 REP 1/1	The program section between this block and LBL 1 (block 15) is repeated once.
...	
50 END PGM REPS MM	



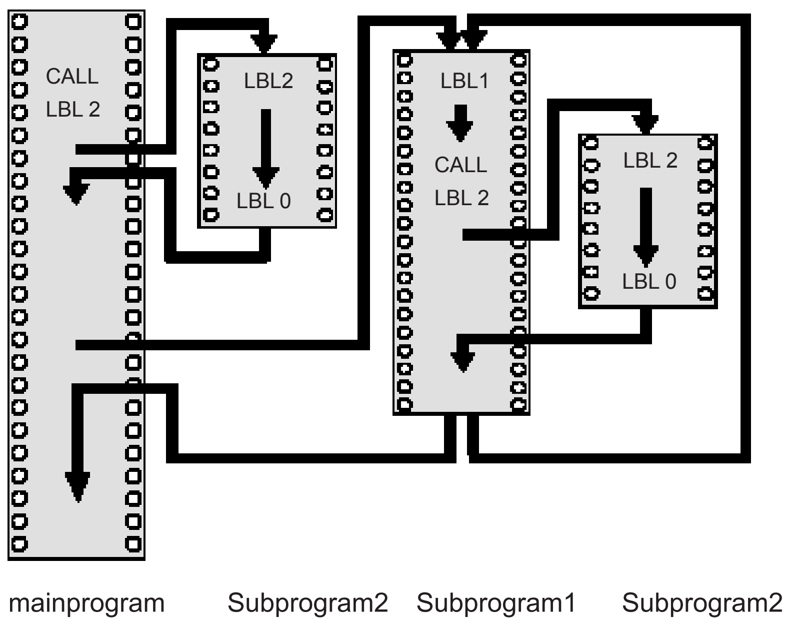
Program execution

1. Main program REPS is executed up to block 27.
2. Program section between block 20 and block 27 is repeated twice.
3. Main program REPS is executed from block 28 to block 35.
4. Program section between block 15 and block 35 is repeated once (including 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).

Repeating a subprogram

Example NC blocks

0 BEGIN PGM UPGREP MM	
...	
10 LBL 1	Beginning of program section repeat 1
11 CALL LBL 2	Subprogram call
12 CALL LBL 1 REP 2/2	The program section between this block and LBL1
...	(block 10) is repeated twice
19 L Z+100 R0 FMAX M2	Last block of the main program with M2
20 LBL 2	Beginning of the subprogram
...	
28 LBL 0	End of the subprogram
29 END PGM UPGREP MM	



Program execution

1. Main program UPGREP is executed up to block 11.
2. Subprogram 2 is called and executed.
3. Program section between block 10 and block 12 is repeated twice: subprogram 2 is repeated twice.
4. Main program UPGREP is executed from block 13 to block 19. End of program.

E: Tool programming

Entering tool-related data

Feed rate F

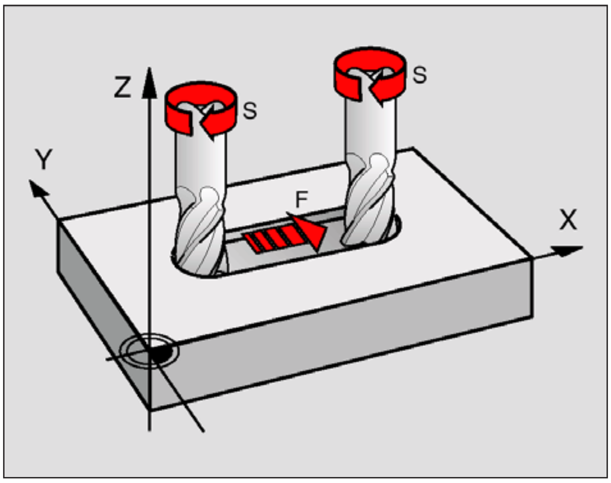
The feed rate **F** is the speed in milimeters per minute or inches per minute at which the tool center moves on its path. The maximum feed rate can be different for every individual axis and is defined by machine parameters.

Input
You can enter the feed rate in the **TOOL CALL** block and in every positioning block (see "Creating the program blocks with the path function keys in chapter D).

Radid traverse
If you wish to program rapid traverse, enter **F MAX** or F9999.
To enter **F MAX**, press the ENT key or the F MAX soft key when the dialogue question **Feed rate**=? appears on the screen.

Duration of effect
A feed rate entered as a numerical value remains in effect until a block is programmed with a different feed rate. **F MAX** is only effective in the block in which it is programmed. After the block with **F MAX** the feed rate will return to the last feed rate entered as a numerical value. F9999 is a selfholding radid traverse. It is erased by entering a feed rate number.

Change during program run
During program run you can adjust the feed rate with the feed-rate override knob F.



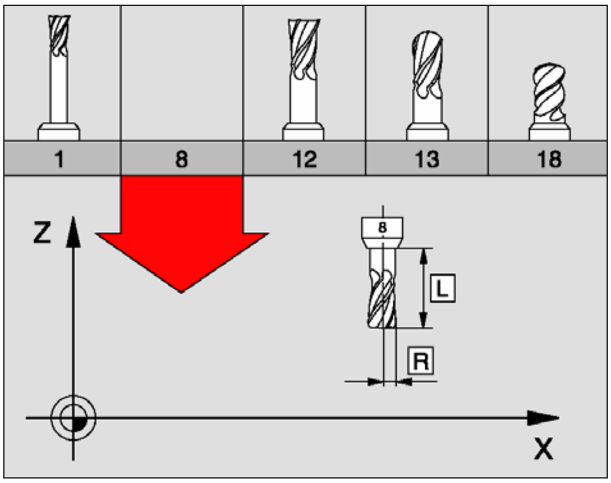
Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a **TOOL CALL** block.

Programmed change
In the part program you can change the spindle speed in a TOOL CALL block by entering only the new spindle speed:

- To program spindle speed, press the TOOL CALL key TOOL CALL.
- Ignore the dialogue question for **Tool number?** with the NO ENT key.
- Ignore the dialogue question for **Working spindle axis X/Y/Z** with the NO ENT key.
- Enter the new spindle speed for the dialogue question **Spindle speed S=?** and confirm with the END key.

Change during program run
During program run you can adjust the spindle speed with the spindle-speed override knob.



Tool Data

Requirements for tool compensation

Usually, you program the coordinates of the path contours as they are dimensioned in the workpiece drawing. To allow the WinNC to calculate the tool center path, i.e. a tool compensation, you have to enter the length and radius of every tool you are using.

Tool data can be entered either directly in the part program with the function TOOL DEF or separately in a tool table. In a tool table, you can also enter additional data on the specific tool. The WinNC considers only the data entered for T, Name, L, R, DL and DR when executing the part program.

Tool number, tool name

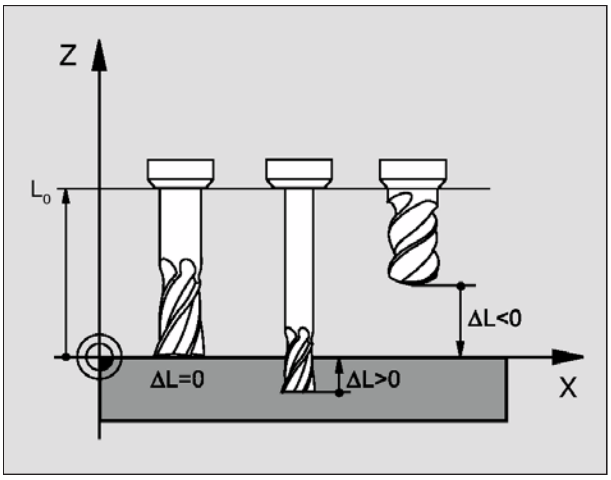
Each tool is identified by a number. If you work with tool tables, you can use higher numbers and also enter tool names.

The tool number 0 is defined as the zero tool with the length L=0 and the radius R=0. The tool T0 is not callabel. In tool tables, tool 0 should also be defined with L=0 and R=0.

Tool length L

There are two ways to determine the tool length L:

Difference between the tool length and the length of a zero tool L0



Algebraic sign:
L>L0: The tool is longer than the zero tool
L<L0: The tool is shorter than the zero tool

Determining the length:

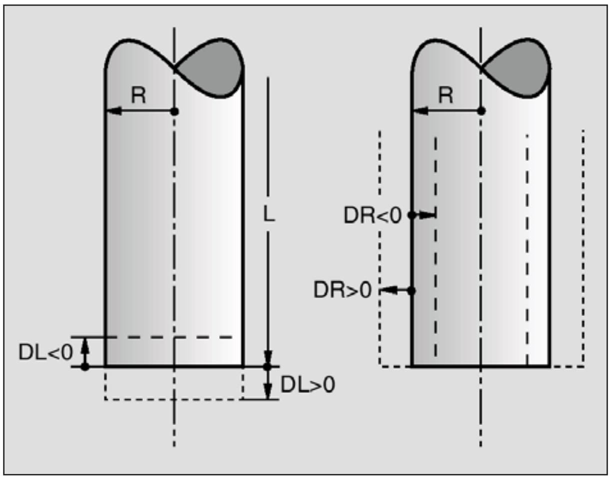
- Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).
- Set the display of the tool axis to zero (setting the reference point)
- Insert the next tool.
- Move the tool to the same reference position as the zero tool.
- The display of the tool axis shows the difference in length between the tool and the zero tool.
- Enter the value in the TOOL DEF block or in the tool table with the "Actual position capture" key.

Determining the length L with a tool presetter
Enter the determined value directly in the TOOL DEF tool definition or in the tool table.

Tool radius R

The tool radius R is entered directly.

Delta values for lengths and radii



Delta values are deviations from the length and radius of a tool.

A positive delta value describes a tool allowance (DL, DR, DR2>0). If you program the machining data with an allowance, enter the oversize value in the TOOL CALL block of the part program.

A negative delta value describes a tool undersize (DL, DR, DR2>0). An undersize is entered in the tool table for wear.

Delta values are entered as numerical values. In a TOOL CALL block you can also assign the value to a Q parameter.

Input range: Delta values can be entered with ± 99.999 mm at maximum.

Entering tool data into the program

The number, length and radius of a specific tool are defined in the TOOL DEF block of the part program:

- To select the tool definition, press the TOOL DEF key.
- Tool number: Each tool is clearly identified by its tool number.
- Tool length: Compensation value for the tool length
- Tool radius: Compensation value for the radius

Example
4 TOOL DEF 5 L+10 R+5

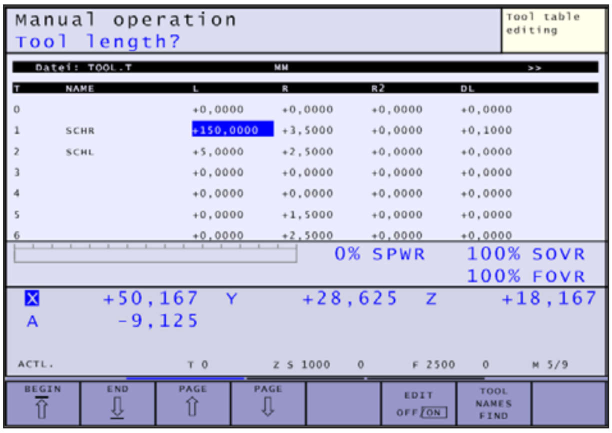
Entering tool data in tables

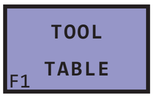
You can define and store tools and their tool data in a tool table. Please also refer to the editing functions at a later stage in this chapter.

You have to use a tool table if indexed tools such as stepped drills with more than one length compensation value are used.

Tool table: Standard tool data

Abbr.	Input	Dialogue
T	Number by which the tool is called in the program (e.g. 5, indexed: 5.2)	—
NAME	Name by which the tool is called in the program	**Tool name?**
L	Value for tool length compensation L	**Tool length?**
R	Compensation value for the tool radius R	**Tool radius R?**
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical illustration of the machining operation with toroid cutters)	**Tool radius R2?**
DL	Delta value for tool length L	**Tool length oversize?**
DR	Delta value for tool radius R	**Tool radius oversize?**
DR2	Delta value for tool radius R2	**Tool radius R2 oversize?**





Editing tool tables

The tool table being active during program run has the file name **TOOL.T**. TOOL T must be stored in the WinNC:\ directory and can only be edited in one of the machining modes. Tool tables that you wish to archive or use for test runs are given any other file name with the extension **.T** .

Opening the tool table TOOL.T:

- Select any machining mode
- To select the tool table, press the TOOL TABLE key.
- Set the EDIT soft key to ON.

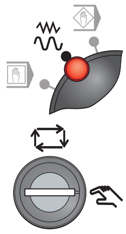
Opening any other tool table:

- Select the programming and editing mode.
- Call the file manager.
- To select the file type, press the SELECT TYPE soft key.
- To show type .T files, press the SHOW .T soft key.
- Select a file or enter a new file name. Confirm with the ENT key or the SELECT soft key.

When you have opened a tool table for editing, you can move the highlight to any desired position within the table by using the arrow keys or the soft keys. You can overwrite the stored values or enter new values at any position. Please refer to the illustration below for further editing functions.

If the WinNC cannot show all positions in the tool table on one screen page, the highlighted bar at the top of the table will display the symbol „**>>**“ or „**<<**“.

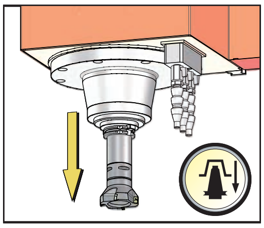
Load the tools into the magazine (random tool system)

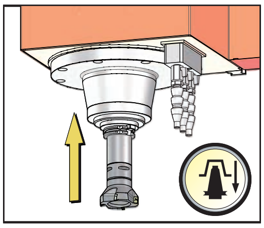


To fit the tool drum manually, the machine must be moved into the following condition:

- JOG mode
- Key switch in "Hand" position
- The tool must be placed in the control system

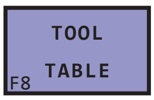
Clamp the tool in the milling spindle

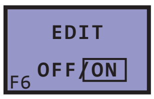




- Open machine doors.
- Hold any clamped tool and remove it by pressing the button for the tool clamp.
- Press and hold down the tool clamp button.
- Push the toolholder with the tool fitted into the right position in the milling spindle and hold it there.
- Release the tool clamp button and the tool is now clamped.

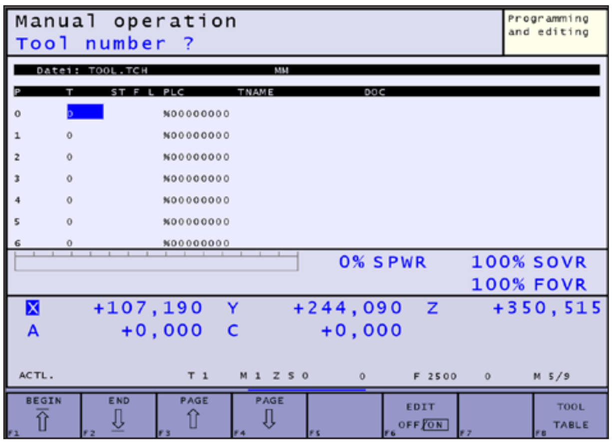
Call up the tool place table





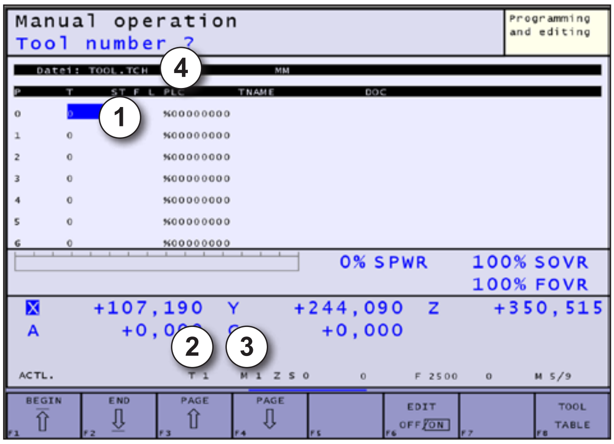
- Select any machine mode
- Select tool table: Press the TOOL TABLE softkey
- Set EDIT softlkey to "ON"

Files for tool place tables have the extension *.tch



Danger:

- Tools have sharp cutting edges! Accordingly, always wear proper protective gloves, if you are handling tools.
- Also note the weight of the tool if you are clamping/unclamping it!



1 Enter active tool

2 T1: Tool number

3 M1: Magazine place

4 TOOL.TCH: Tool place table filename

Decide tool and magazine place.

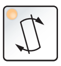
- Close machine doors.
- Enter at the current cursor position (1) "Active Tool" which tool has been clamped.
  At this point, enter the tool number with which the tool was created.
- Press the Enter key.
  The tool entered is now displayed in the place table on the screen at place "0" (0 = tool spindle).
  If the tool number is wrong, repeat the entry and press the softkey again.

- Use the Swivel keys to swivel into the corresponding tool place at which the new tool in the magazine is to be stored.
  The position into which you have now swung appears on screen under "Magazine Place" (3).

Place the tool in the magazine

- The machine doors must be closed.



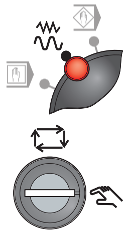
- Press "Tool Change" key.
  The tool change cycle now starts and the tool is taken out of the spindle and placed in the tool magazine at the stipulated position.
- The place table on screen is updated:
  - The newly clamped tool is displayed at the selected position.
  - Tool "0" (= no tool) is displayed at tool place "0" (= spindle).

Tool maintenance

When unclamping tools for inspection or maintenance, please take care that:

- The removed tool is no longer displayed in the place table.
- After clamping the tool in the spindle again, it is essential to re-enter the tool number, because otherwise the tool is no longer administered properly! (see "Deciding tool and magazine place")

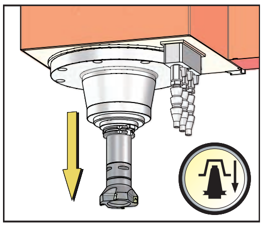
Loading the tools in the magazine with a random tool system



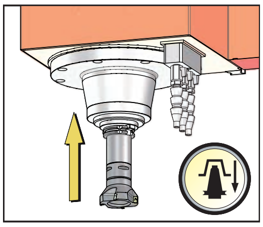
To fit the tool drum manually, the machine must be moved into the following condition:

- JOG mode
- Key switch in "Hand" position
- The tool must be placed in the control system (see WinNC description, Chapter E)

Clamp the tool in the milling spindle



- Open machine doors.
- Hold any clamped tool and remove it by pressing the button for the tool clamp.
- Press and hold down the tool clamp button.



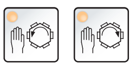
- Push the toolholder with the tool fitted into the right position in the milling spindle and hold it there.
- Release the tool clamp button and the tool is now clamped.

Danger:

- Tools have sharp cutting edges! Accordingly, always wear proper protective gloves, if you are handling tools.
- Also note the weight of the tool if you are clamping/unclamping it!

Decide tool and magazine place

- Close machine doors.



- Use the Swivel keys to swivel into the corresponding tool place at which the new tool in the magazine is to be stored.
  The position into which you have now swung appears on screen under "M" (3).

In a non-random tool system, a tool that is no longer needed is always placed at the point in the magazine from which it was withdrawn.

Place the tool in the magazine

- The machine doors must be closed.

- Press "Tool Change" key.
  The tool change cycle now starts and the tool is taken out of the spindle and placed in the tool magazine at the stipulated position.

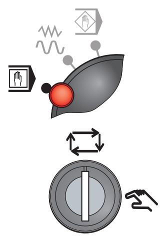
Caution:
If a tool has been swivelled into the wrong position, proceed as follows:

- Unclamp the tool
- Clamp the tool back in again
- Place the tool in the magazine (as described earlier)

Only in this way can you guarantee that the right tool is positioned in the right place.

Unload the tool

When unloading the tool, the latter must be removed from the tool list and from the magazine.

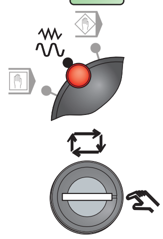


- Activate MDA mode.
- Key switch in "Automatic" position.

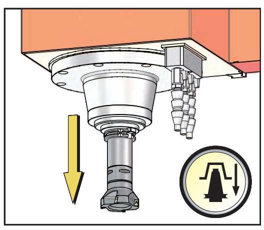
TOOL CALL 6 z

- Program tool call-up for the corresponding tool (in this case tool T6).

- Start program; tool T6 is loaded into the spindle.



- JOG mode
- Key switch in "Hand" position
- Open machine doors.



- Hold tool
- Press the tool clamp button and remove the tool.

Danger:

- Tools have sharp cutting edges! Accordingly, always wear proper protective gloves, if you are handling tools.
- Also note the weight of the tool if you are clamping/unclamping it!

Pre-positioning of the tool (only random tool system)

With random tool systems there is also the possibility of swivelling the next tool that should be changed in to the change position.
This happens during the processing.

```
0  BEGIN PGM PROG MM
1  BLK FORM 0.1 Z X+0 Y+0 Z-10
2  BLK FORM 0.2 X+50 Y+50 Z+0
3  TOOL CALL 1 Z S1000
4  L X+0 Y+0 Z+5 F MAX M3 M6

5  TOOL DEF 5
6  L Z-2 F200
7  L X+100
```

Tool pre-selection for T5
The tool T5 swivels into the change position (movement of the tool magazine).
While this happens, the processing with the active tool T1 is not interrupted.

```
8  TOOL CALL 5 Z S2000
```

The tool T5 is changed

```
9  L X+0 Y+0 Z+2 F MAX M3 M6
10  END PGM PROG MM
```

Information:
To avoid errors when programming for the pre-positioning, proceed as follows:

- First of all program the parts program without pre-positioning of the tools (as for a non-random tool system).
- Then insert the commands for the pre-positioning (tool call-up) in the parts program from bottom to top.

Leaving the tool table

- Call the file manager and select a file of a different type, e.g. a part program.

Select beginning of table

Select end of table

Select previous page in table

Select next page in table

Look for the tool name in the table

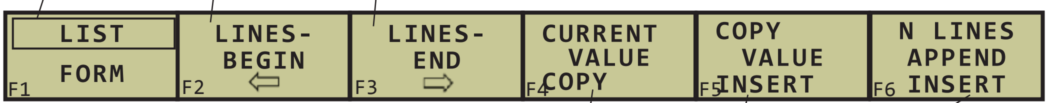
Delete current line (tool)

Insert a line with indexed tool number after the active line. The function is only active if you are allowed to store various compensation data for a tool. The WinNC inserts a copy of the tool data after the last available index and increases the index by 1.
Applicationexample: Stepped drill with more than one length compensation value; another compensation should be added to tool T4: move highlight to line 4, Softkey LINE INSERT results new line 4.1

Show tool information in columns or show all information on a tool on one screen

Move to beginning of line

Move to end of line



Add the entered number of lines (tools) to the end of the table.

Insert the copied field

Copy the highlighted field

Calling tool data

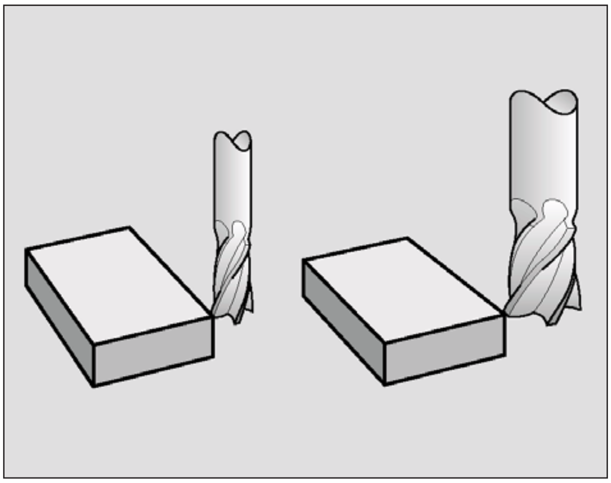
A **TOOL CALL** block in the part program is defined with the following data:

- Select the tool call function with the TOOL CALL key.
- **Tool number:** Enter the number or name of the tool. The tool must already be defined in a TOOL DEF block or in the tool table. A tool name is always entered between quotation marks. The tool name always refers to the entry in the active tool table TOOL.T. If you want to call a tool with other compensation values, enter the index that is defined in the tool table after the decimal point. e.g.: 4.1
- **Working spindle axis X/Y/Z**: Enter the tool axis
- **Spindle speed S**: Enter the spindle speed directly
- **Feed rate F**: Enter the feed rate directly. F remains in effect until you program a new positioning block or until you program a new feed rate in the TOOL CALL block.
- **Tool length oversize DL**: Enter the delta value for the tool length.
- **Tool radius oversize DR**: Enter the delta value for the tool radius.
- **Tool radius oversize DR2:** Enter the delta value for the tool radius 2.

Example: Tool call
Call tool number 5 with it´s three compensation values (5, 5.1, 5.2) in the tool axis Z with a spindle speed of 2500 rpm and a feed rate of 350 mm/min. The oversize for the tool length and the tool radius is 0.2 or 0.05 mm, the undersize for the tool radius is 1 mm.

20 TOOL CALL 5.2 Z S2500 F350 DL+0,2 DR-1 DR2+0,05

The character **D** before **L** and **R** designates the delta value.



Tool Compensation

Introduction

The WinNC adjusts the tool path in the spindle axis by the compensation value for the tool length. In the working plane, it compensates the tool radius.

If you are writing the part program directly on the WinNC, the tool radius compensation is only effective in the working plane. The WinNC considers the compensation in up to five axes including the rotary axes.

Tool length compensation

The tool length compensation becomes effective as soon as you call a tool and move along the spindle axis. To cancel length compensation, call a tool with a length L=0.

Note:
If you cancel a positive length compensation with **TOOL CALL 0**, the distance between the tool and the workpiece will be reduced.
After **TOOL CALL** the programmed tool path in the spindle axes is adjusted by the difference between the length of the previous tool and that of the new one.

For tool length compensation the WinNC considers delta values from both the **TOOL CALL** block and the tool table.

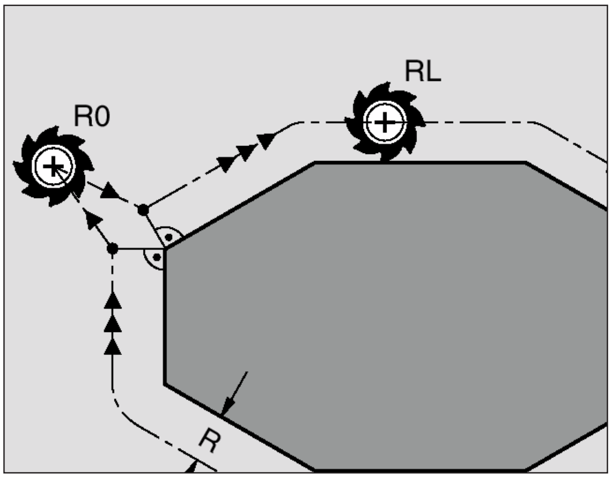
Compensation value = **L** + $\mathbf{DL}_{\text{TOOL CALL}}$ + $\mathbf{DL}_{\text{TAB}}$ with

L: tool length **L** from **TOOL DEF** block or tool tables

$\mathbf{DL}_{\text{TOOL CALL}}$: oversize for length **DL** in the **TOOL CALL** block (not taken into account by the position display)

$\mathbf{DL}_{\text{TAB}}$: oversize for length **DL** in the tool table

emco



Tool radius compensation

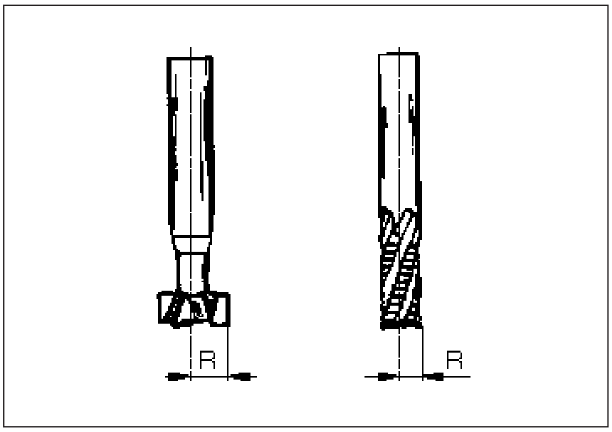
The NC block for programming a tool movement contains:

- **RL** or **RR** for radius compensation
- **R+** or **R–**, for radius compensation in single-axis movements
- **R0**, if there is no radius compensation

The radius compensation becomes effective as soon as a tool is called and is moved in the working plane with RL or RR.

Note:
The WinNC cancels the radius compensation if you:

- program a positioning block with **R0**
- depart the contour with the **DEP** function
- program a **PGM CALL**
- select a new program with PGM MGT



For the tool radius compensation the WinNC considers the delta values from both the **TOOL CALL** block and the tool table.

Compensation value = **R** + $\mathbf{DR}_{\text{TOOL CALL}}$ + $\mathbf{DR}_{\text{TAB}}$ with

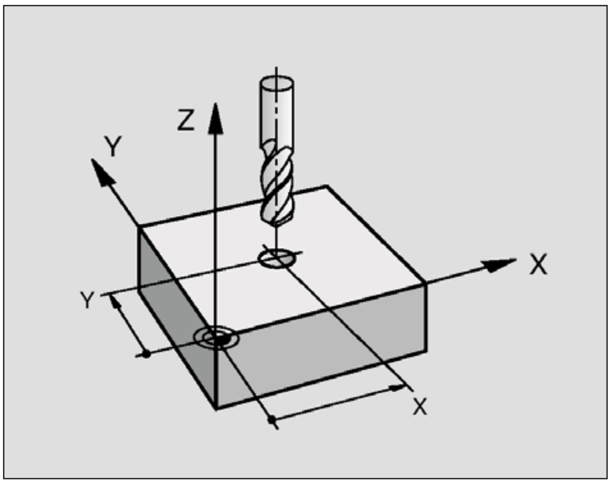
R: tool radius **R** from **TOOL DEF** block or tool table

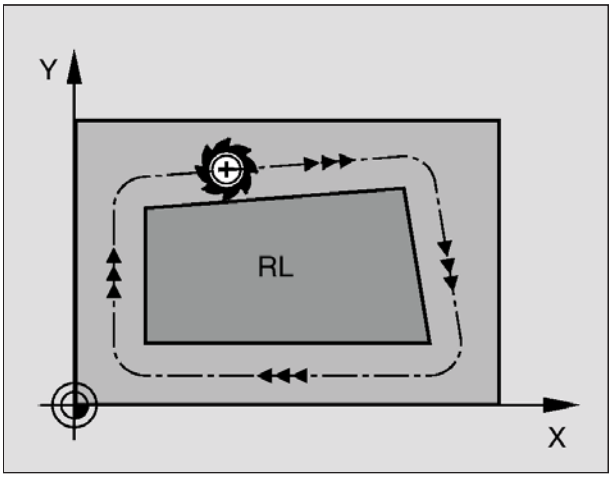
$\mathbf{DR}_{\text{TOOL CALL}}$: oversize for radius **DR** in the **TOOL CALL** block (not taken into account by the position display)

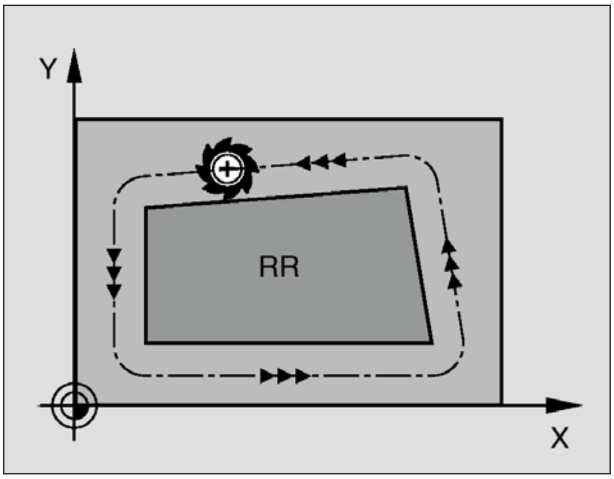
$\mathbf{DR}_{\text{TAB}}$: oversize for radius **DR** in the tool table

Contouring without radius compensation: R0
The tool center moves in the working plane on the programmed path or to the programmed coordinates.

Applications: Drilling and pre-positioning







Path movements with radius compensation: RR und RL

RR The tool moves to the right of the contour.
RL The tool moves to the left of the contour.

The tool center moves along the programmed contour at a distance equal to the radius. "Right" and "left" are to be understood as based on the direction of tool movement along the workpiece contour. See illustrations on the left.

Note:
Between two program blocks with different radius compensations **RR** and **RL** you must program at least one traversing block in the working plane without radius compensation (that is, with **R0**).

Radius compensation becomes effective at the end of the block, in which it was first programmed.

You can also activate the radius compensation for secondary axes in the working plane (only for Concept machines available). Program the secondary axes also in each following block, since otherwise the WinNC will execute the radius compensation in the main axis.

Whenever the radius compensation is activated with **RR/RL** or canceled with **R0**, the WinNC always positions the tool perpendicular to the programmed starting and end point. Position the tool at a sufficient distance from the first or last contour point to prevent the contour from being damaged.

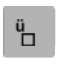
Entering radius compensation

Program any desired path function, enter the coordinates of the target point and confirm with the  key.

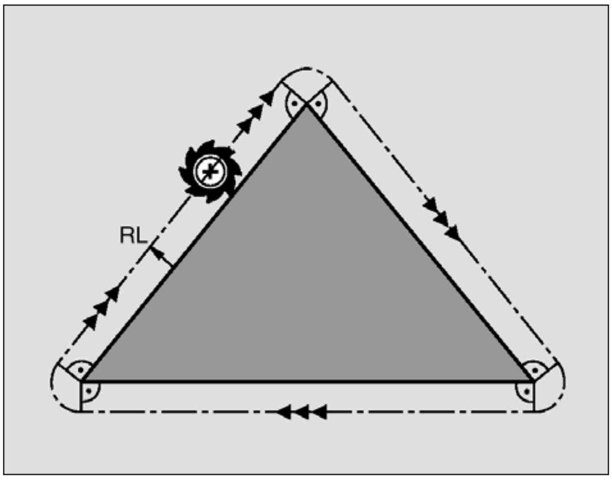
Radius comp.: RL/RR/ no comp.?
To select tool movement to the left of the programmed contour, press the RL soft key, or

To select tool movement to the right of the programmed contour, press the RR soft key, or

To select tool movement without radius compensation or to cancel radius compensation, press the  key or the softkey R0.



To terminate the block, press the END key.



Radius compensation: Machining corners

- Outside corners:
  If you have programmed a radius compensation, the WinNC moves the tool around outside corners on a transitional arc. If necessary, the WinNC reduces the feed rate at outside corners, for example at very great changes of direction.
- Inside corners:
  The WinNC calculates the intersection of paths, on which the tool center traverses under compensation at inside corners. From this point the tool moves along the next contour element. This prevents the inside corners of the workpiece from being damaged. Therefore, the permissible tool radius is limited by the geometry of the programmed contour.

Note:
To prevent the contour from being damaged, do not program the starting or end point for machining inside corners at a corner of the contour.

F: Program run

Requirements

Datum setting or cyclus 7

The used datums must be measured and entered.

Tools

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

Reference point

The reference point must be traversed in all axes.

Machine

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

Alarms

There must not be any alarms activated.

Program start, Program stop

Select program run, full sequence or single block

Select a program for machining.

Change into the automatic mode of operation.

Press the  key.

Stop program with , continue with .

Abort program with .

G: Flexible NC programming

Q parameters

By using Q parameters you can define an entire family of parts in a single part program. You do this by entering variables called Q parameters instead of numerical values.

For example, Q parameters may stand for

- Coordinate values
- Feed rates
- RPM
- Cycle data

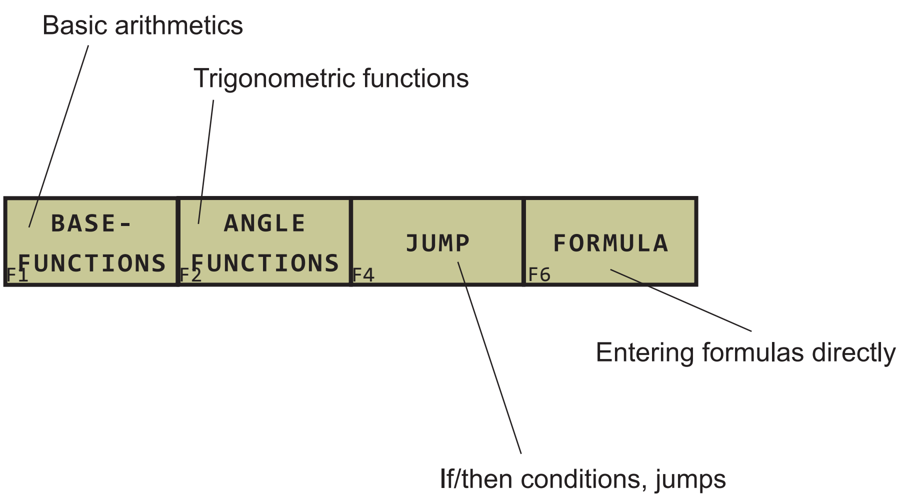
Q parameters also enable you to program contours that are defined through mathematical functions. You can also use Q parameters to make the execution of machining steps dependent on logical conditions. In conjunction with FK programming you can also combine contours that do not have NC-compatible dimensions with Q parameters.

Q parameters are designates by the letter Q and a number between 0 and 399. They are grouped in three ranges:

Range	Variable type
Q0 to Q199	Freely applicable paramters, effective for all programs in the TNC memory
Q200 to Q399	Parameters that are premarily used for cycles, effective for all programs being stored in

Calling Q parameter functions

Press the Q key in the Programming/ Editing mode of operation. The WinNC then displays the following soft keys:



Calculating with Q parameters

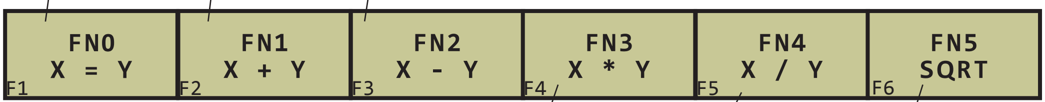
Q parameters enable you to program basic mathematical functions in a part program.

- Select a Q parameter function by pressing the **Q** key (in the numerical keypad on the right). The Q parameter functions are displayed in a soft key row.
- To select basic mathematical functions, press the BASIC ARITHMETIC soft key. The WinNC then displays the following soft keys:

FN0: ASSIGN
Example: **FN0: Q5 = +60**
Assigns a numerical value

FN1: ADDITION
Example: **FN1: Q1 = –Q2 + -5**
Calculates and assigns the sum of two values.

FN2: SUBTRACTION
Example: **FN2: Q1 = +10 – +5**
Calculates and assigns the difference of two values.



FN5: SQUARE ROOT
Example: **FN5: Q20 = SQRT 4**
Calculates and assigns the square root of a number
Not permitted: Square root of a negative value!

FN4: DIVISION
Example: **FN4: Q4 = +8 DIV +Q2**
Calculates and assigns the quotient of two values.
Not permitted: division by 0!

FN3: MULTIPLICATION
Example: **FN3: Q2 = +3 * +3**
Calculates and assigns the product of two values.

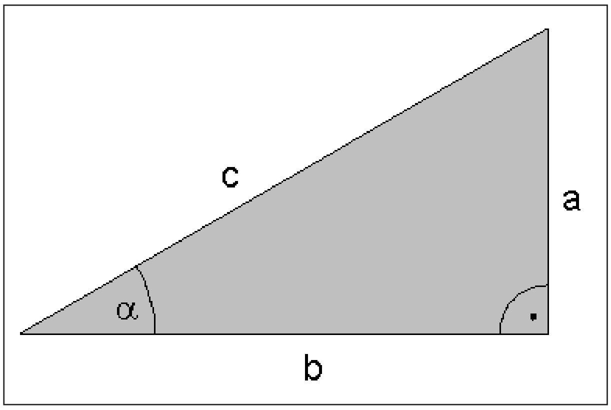
On the right of the "=" character you can enter the following:

- two numbers
- two Q parameters
- a number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Trigonometric functions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. For a triangle, the trigonometric functions of the angle are defined by the following equations:



Sine: sin α = a / c
Cosine cos α = b / c
Tangent: tan α = a / b = sin α / cos α

where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

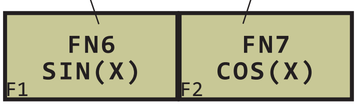
The WinNC can find the angle from the tangent:
α = arctan (a / b) = arctan (sin α / cos α)

Example:
a = 25 mm
b = 50 mm
α = arc tan (a / b) = arc tan 0,5 = 26,57°
Furthermore:
$a^2 + b^2 = c^2$ (mit a^2 = a x a)

Press the TRIGONOMETRY softkey to call the trigonometric functions. The WinNC shows the soft keys in the illustration below.

FN6: SINE
Example: **FN6: Q20 = SIN–Q5**
Calculate the sine of an angle in degrees (°) and assign it to a parameter

FN7: COSINE
Example: **FN7: Q21 = COS–Q5**
Calculate the cosine of an angle in degrees(°) and assign it to a parameter



If-Then decisions with Q parameters

The WinNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the WinNC continues the part program at the LABEL that is programmed after the condition. If it is not fulfilled, the WinNC continues with the next block.
If you want to call another program as a subprogram, program PGM CALL after 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 decisions

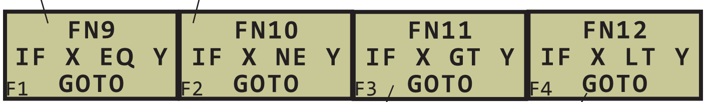
Press the JUMP soft key to call the If-Then decisions. The WinNC shows the following soft keys:

Abbreviations used:

IF	If
EQU	Equals
NE	Not equal
GT	Greater than
LT	Less than
GOTO	Go to

FN9: IF EQUAL, JUMP
Example: **FN9: IF +Q1 EQU +Q3 GOTO LBL 5**
If both values or parameters are equal, jump to the given label.

FN10: IF NOT EQUAL, JUMP
Example: **FN10: IF +10 NE –Q5 GOTO LBL 10**
If both values or parameters are not equal, jump to the given label.



FN12: IF LESS THAN, JUMP
Example: **FN12: IF+Q5 LT+0 GOTO LBL 1**
If the first value or parameter is less than the second value or parameter, jump to the given label.

FN11: IF GREATER THAN, JUMP
Example: **FN11: IF+Q1 GT+10 GOTO LBL 5**
If the first value or parameter is greater than the second value or parameter, jump to the given label.

Entering formulas directly

You can enter mathematical formulas that include several operations directly into the part program by soft keys.
Press the FORMULA soft key to call the formula functions. The WinNC displays the following soft keys in several soft-key rows:

Mathematical function	Softkey
Addition	+ F1
Subtraction	- F2
Multiplication	* F3
Division	/ F4
Opening parenthesis	(F5
Closing parenthesis	) F6
Square of a value	SQ F7
Square root	SQRT F8
Sine of an angle	SIN F1
Cosine of an angle	COS F2
Tangent of an angle	TAN F3
Arc Sine Inverse of the sine; determine the angle from the ratio of the opposite side to the hypotenuse.	ASIN F4
Arc cosine Inverse of the cosine; determine the angle from the ratio of the adjacent side to the hypotenuse.	ACOS F5
Arc tangent Inverse of the tangent; determine the angle from the ratio of the opposite side to the adjacent side.	ATAN F6
Powers of values	^ F1

Mathematical function	Softkey
Circular graduation number (constant) pi (3.14159265359)	PI F2
Natural logarithm (LN) of a number Base 2.7183	LN F3
Logarithm of a number, base 10	LOG F4
Exponential function, 2.7183 to the power of n	EXP F5
Negate (multiplication by -1)	NEG F1
Truncate decimal places Form an integer	INT F2
Absolute value of a number	ABS F3
Truncate places before the decimal point Form a fraction	FRAC F4

Example:
Q1 = 5 * 3 + 2 * 10 = 35

Note:
The following rules are valid:
Higher-level operations are performed first. The distributive law is valid. If you enter formulas directly, always enter a positive/ negative sign or a blank before a number!
Instead of entering Softkeys also characters and symbols as shown in the Softkeys can be fed in.

H: Alarms and Messages

Machine Alarms 6000 - 7999

These alarms will be triggered by the machines. There are different alarms for the different machines.

The alarms 6000 - 6999 normally must be confirmed with RESET. The alarms 7000 - 7999 are messages which normally will disappear when the releasing situation is finished.

PC MILL 50 / 55 / 100 / 105 / 125 / 155
Concept MILL 55 / 105 / 155

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

6001: PLC-CYCLE TIME EXCEEDING
Contact EMCO Service.

6002: PLC - NO PROGRAM CHARGED
Contact EMCO Service.

6003: PLC - NO DATA UNIT
Contact EMCO Service.

6004: PLC - RAM MEMORY FAILURE
Contact EMCO Service.

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

6006: OVERLOAD BRAKE RESISTOR
see 6005

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

6008:MISSING CAN SUBSCRIBER
Check fuses or EMCO customer service.
Contact EMCO Service.

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

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

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

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

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

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

6019: VICE TIME EXCEED
The electric vice has not reached a stop position within 30 seconds.
The control or the clamping device board are defective, the vice is stuck. Adjust the proximity switches of the stop position.

6020: VICE FAILURE
When the electric vice is closed, the signal "clamping device clamped" of the clamping device board has failed.
The control, the clamping device board or the wiring are defective.

I 2014-04

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

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

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

6028: DOOR TIMEOUT
The automatic door 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

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

7001: NO M6 PROGRAMMED
For an automatic tool change you also have to program a M6 after the T word.

7007: FEED STOP!
The axes have been stopped by the robotics interface (robotics entry FEEDHOLD).

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

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

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

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

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

7022: INITIALIZE TOOL TURRET !
see 7021

7023: WAITING TIME MAIN DRIVE!
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

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

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

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

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

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

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

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

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

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

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

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

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

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

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

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
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.

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

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

6078 MOTOR PROTECTION TOOL MAGAZINE RELEASED
Cause: Swing intervals are too short.
Remedy: Raise swing intervals.

6079 MOTOR PROTECTION TOOL CHANGER RELEASED
see 6068

6080 PRESSURE SWITCH FOR TANI MISSING
Cause: The pressure switch fails to active when the clamping closes. No pressurised air or mechanical problem.
Remedy: Check pressurised air.

6081 CLAMPING FOR TANI NOT OPEN
see 6080

6082 FAULT AS/SIGNAL
Cause: Active Safety-Signal X/Y-controller is faulty.
Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

6083 FAULT AS/SIGNAL
Cause: Active Safety-Signal main spindle/Z-controller is faulty.
Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

6084 FAULT AS/SIGNAL UE-MODUL
Cause: Active Safety-Signal Uncontrolled power supply module is faulty.
Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO.

6085 N=0 RELAY NOT DE-ENERGIZED
Cause: Rotation zero relay did not drop.
Remedy: Delete alarm using the RESET key and/or switch the machine on/off. If this error reoccurs, contact EMCO (replace relay).

6086 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC
Cause: ACC-PLC and USBSPS receive different door status reports.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6087 DRIVE A-AXIS NOT READY
see 6010

6088 PROTECT SWITCH DOOR CONTROL UNIT RELEASED
Cause: Door drive overload.
Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace motor, drive).

6089 DRIVE B-AXIS NOT READY
see 6010

6090 CHIP CONVEYOR CONTACTOR NOT DE-ENERGIZED
Cause: Chip conveyor guard not down.
Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace guard).

6091 AUTOMATIC DOOR CONTACTOR NOT DE-ENERGIZED
Cause: Automatic door guard not down.
Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO (replace guard).

6092 EMERGENCY-OFF EXTERNAL

6093 FAULT AS/SIGNAL A-AXIS
Cause: Active Safety-Signal A control element faulty.
Remedy: Cancel alarm with RESET button or switch machine on/off. If the problem occurs several times, contact EMCO.

6095 OVERHEATING IN THE SWITCHGEAR CABINET
Cause: Temperature monitoring responded.
Remedy: Check switchgear cabinet filter and fan, raise triggering temperature, switch machine on and off.

6096 SWITCHGEAR CABINET DOOR OPEN
Cause: Switchgear cabinet door opened without key switch release.
Remedy: Close switchgear cabinet door, switch machine off and on.

6900 USBPLC not available
Cause: USB communication with the safety board could not be established.
Remedy: Switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

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

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

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

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

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

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

6907 Error enable pulses ER-module
Cause: ACC-PLC did not shutdown the input/negative feeder-module.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6908 Error standstill monitoring main drive
Cause: Unexpeced warm up of the main spindle in the operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6909 Error main drive enable without spindle start
Cause: The release of the control unit of the main spindle was given by the ACC-PLC without the spindle-start key being pressed.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6910 Error standstill monitoring Y
Cause: Unauthorized movement of the Y axis in the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6911 Error standstill axes
Cause: Unauthorized movement of the axis in the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6912 Error overspeed axis
Cause: The feed of the axes exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6913 Error overspeed X
Cause: The feed of the X axis exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6914 Error overspeed Y
Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6915 Error overspeed Z
Cause: The feed of the Y axis exceeds the maximum permissible value for the current operating condition.
Remedy: Delete the alarm with the RESET button and switch the machine off and on again. Please contact the EMCO after-sales service in case the error occurs repeatedly.

6916 ERROR: X-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by X axis Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6917 ERROR: Y-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by Y axis Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6918 ERROR: Z-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by Z axis Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6919 ERROR: SPINDLE-INDUCTIVE PROXIMITY SWITCH DEFECT
Cause: No signal is delivered by main spindle Bero.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6920 INVERSION OF X-DIRECTION TOO LONG "1"
Cause: The change in direction of X axis was being sent to USBSPS for more than three seconds.
Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6921 INVERSION OF Y-DIRECTION TOO LONG "1"
Cause: The change in direction oY axis was being sent to USBSPS for more than three seconds.
Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6922 INVERSION OF Z-DIRECTION TOO LONG "1"
Cause: The change in direction of Z axis was being sent to USBSPS for more than three seconds.
Remedy: Delete alarm using the RESET key. Avoid driving back and forth using the manual wheel. If this error reoccurs, contact EMCO.

6923 DIFFERENT DOOR-SIGNALS FROM USBPLC AND ACC-PLC
Cause: ACC-PLC and USBSPS receive different door status reports.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

6924 ERROR ENABLE PULSES MAIN DRIVE
Cause: The pulse release on the main spindle control element was interrupted by the USBSPS, as the PLC did not shut it down in a timely fashion.
Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

6925 GRID PROTECTION ERROR!
Cause: Grid protection does not drop out in current operating state, or does not engage.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6926 MOTOR PROTECTION ERROR!
Cause: Motor protection drops out in current operating state.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6927 EMERGENCY OFF ACTIVE ERROR!
Cause: Emergency off button was pressed.
Remedy: Restart the machine.

6928 TOOL CHANGER SHUTDOWN MONITORING ERROR
Cause: Unauthorised tool changer movement in the current operating state.
Remedy: Clear alarm with emergency off button and restart the machine. Contact EMCO Customer Service if the error occurs on several occasions.

6929 MACHINE DOOR CLOSING/LOCKING ERROR

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

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

6930 BEROS MAIN SPINDLE PLAUSIBILITY ERROR

Cause: Beros main spindle signal different.

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

6931 MAIN DRIVE QUICK STOP FUNCTION PLAUSIBILITY ERROR

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

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

6999 USB-EXTENSION FOR ROBOTIK NOT AVAILABLE

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

Remedy: Contact EMCO.

7000: INVALID TOOL NUMBER PROGRAMMED

The tool position was programmed larger than 8. The CNC program will be stopped.
Interrupt program with RESET and correct the program.

7007: FEED HOLD

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

7016: SWITCH ON AUXILIARY DRIVES

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

7017: REFERENCE MACHINE

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

7018: TURN KEY SWITCH

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

7019: PNEUMATIC LUBRICATION MONITORING!

Refill pneumatic oil

7020: SPECIAL OPERATION MODE ACTIVE

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

7021: TOOL TURRET NOT LOCKED

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

7022: COLLECTION DEVICE MONITORING

Time exceed of the swivelling movement.
Check the pneumatics, respectively whether the mechanical system is jammed (possibly a workpiece is jammed).

7023: ADJUST PRESSURE SWITCH !

During opening and closing of the clamping device the pressure switch has to turn off and on once.
Adjust the pressure switch. This alarm does not exist any more for versions starting with PLC 3.10.

7024: ADJUST CLAMPING DEVICE PROXIMITY SWITCH !

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

7025 WAITING TIME MAIN DRIVE !
The LENZE frequency converter has to be separated from the mains supply for at least 20 seconds before you are allowed to switch it on again. This message will appear when the door is quickly openend/ closed (under 20 seconds).

7026 PROTECTION MAIN MOTOR FAN RELEASED!

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

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

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

7041: GEARBOX COVER OPEN
The main spindle cannot be switched on and NC start cannot be activated.
Close the gearbox cover in order to start a CNC program.

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

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

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

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

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

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

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

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

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

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

7060 RETRACT SLEEVE LIMIT SWITCH !
The axis arrived in the tailstock sleeve. Drive the travel off the tailstock sleeve.

7061 RETRACT X AXIS LIMIT SWITCH !
The axis arrived to the end switch. Drive the axis off the end switch again.

7062 RETRACT Z AXIS LIMIT SWITCH !
see 7061

7063 OIL LEVEL CENTRAL LUBRICATION !
Low oil level in central lubrication. Refill oil as per maintenance instructions to the machine.

7064 CHUCK GUARD OPEN !
The chuck guard is open. Close the chuck guard.

7065 MOTOR PROTECTION COOLANT PUMP RELEASED !
Overheated coolant pump. Check the coolant pump for ease of motion and presence of dirt. Ensure sufficient amount of coolant fluid in the coolant facility.

7066 CONFIRM TOOL !
To confirm the tool change, press T after the change has been completed.

7067 MANUAL OPERATING MODE
The Special Operation key switch is in the Set position (manual).

7068 X AXIS HANDWHEEL ACTIVE
The safety wheel is locked for manual travel movement. The safety wheel locking is monitored by contactless switches. With the manual wheel locked, the axis feed cannot be switched on. For automatic processing of a program, the manual wheel must be released again.

7069 Y AXIS HANDWHEEL ACTIVE
see 7068

7070 Z AXIS HANDWHEEL ACTIVE
see 7068

7071 VERTICAL TOOL CHANGE
The sheath for manual clamping of the tool holder is monitored by a switch. The switch reports a unaccepted socket wrench or a sheath which was left open. Remove the socket wrench after clamping the tool and close the sheath.

7072 HORIZONTAL TOOL CHANGE
The turning knob for manual tool clamping on the horizontal spindle is monitored by a switch. The switch reports a tightened turning knob. The spindle gets locked. Release the turning knob after clamping the tool.

7073 RETRACT Y AXIS LIMIT SWITCH !
see 7061

7074 CHANGE TOOL
Clamp programmed tool.

7076: SWIVEL UNIT VOR MILLING HEAD UNLOCKED
The milling head is not fully swung. Fix the milling head mechanically (the end switch must be pushed).

7077: ADJUST TOOL TURRET
No valid machine data for tool change are available. Contact EMCO.

7078: POCKET NOT IN HOME POSITION
Cancel during tool change. Swing back tool recessed in setup operation.

7079: TOOL ARM NOT IN HOME POSITION
see 7079

7080: INCORRECT TOOL CLAMPED !
The tool cone lies beyond tolerance. The clamped tool is twisted by 180°. Bero tool clamping is displaced. Check the tool and clamp it again. If this problem occurs with more tools, contact EMCO.

7082: MOTOR PROTECTION CHIP CONVEYOR RELEASED
The scrap belt is overloaded. Check the conveyor belt for ease of motion and remove jammed scrap.

7083: MAGAZINE IS ACTIVE !
A tool has been removed from the non-chaotic tool administration from the main spindle. Fill the tool drum.

7084: VICE OPEN !
The vice is not clamped. Clamp the vice.

7085 ROUNDAXIS A MOVE TO 0 DEGRE !
Cause: The MOC only shuts down if the A Round axis is at 0°.
When 4.5. is present, a round axis must be made each time before the machine is switched off.
Remedy: Move round axis to 0°.

7088 SWITCHGEAR CABINET OVERHEATING
Cause: Temperature monitoring responded.
Remedy: Check switchgear cabinet filter and fan, raise trigger temperature.

7089 SWITCHGEAR CABINET DOOR OPEN
Cause: Switchgear cabinet door open.
Remedy: Close switchgear cabinet door.

7900 INITIALIZE EMERGENCY OFF!
Cause: The emergency off button must be initialized.
Remedy: Press and then release emergency off button.

7901 INITIALIZE MACHINE DOORS!
Cause: The machine doors must be initialized.
Remedy: Open the machine doors and close them again.

Inputunit alarms 1700 - 1899

These alarms and messages are raised by the control keyboard.

1701 Error in RS232
Cause: Serial port settings are invalid or the connection to the serial keyboard were interrupted.
Remedy: Check the settings of the serial interface and/or turn keyboard off/on and check the control cable connection.

1703 Ext. keyboard not available
Cause: Connection with the external keyboard can not be made.
Remedy: Check the settings of the external keyboard and/or check the cable connection.

1704 Ext. keyboard: checksum error
Cause: Error in the transmission.
Remedy: The connection to the keyboard is automatically restored. If this fails, turn off or on the keyboard.

1705 Ext. keyboard: general error
Cause: The attached keyboard reported an error.
Remedy: Plug the keyboard off and on again.Contact EMCO Customer Service if the error occurs on several occasions.

1706 General USB error
Cause: Error in the USB communication.
Remedy: Plug the keyboard off and on again.Contact EMCO Customer Service if the error occurs on several occasions

1707 Ext. Keyboard: no LEDs
Cause: Fehlerhaftes LED-Kommando wurde an die Tastatur gesandt.
Remedy: EMCO-Service kontaktieren.

1708 Ext. Keyboard: unknown command
Cause: Unknown command was sent to the keyboard.
Remedy: Contact EMCO Customer Service

1710 Installation of Easy2control is damaged!
Cause: Incorrect installation of Easy2control
Remedy: Reinstall software and/or contact EMCO Customer Service

1711 Initialization of Easy2Control failed!
Cause: Configuration file onscreen.ini for Easy2control is missing.
Remedy: Reinstall software and/or contact EMCO Customer Service.

1712 USB-Dongle for Easy2control could not be found!
Cause: USB-Dongle for Easy2control is not connected. Easy2control is displayed but can not be operated.
Remedy: Connect USB-Dongle for Easy2control.

1801 Keytable not found!
Cause: The file with the keytable couldn't be found.
Remedy: Reinstall software and/or contact EMCO Customer Service.

1802 Connection to keyboard lost
Cause: Connection to the serial keyboard was interrupted.
Remedy: Turn keyboard off/on and check the cable connection.

Axis Controller Alarms 8000 - 9000, 22000 - 23000, 200000 - 300000

8000 Fatal Error AC

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

8101 Fatal init error AC
see 8101.

8102 Fatal init error AC
see 8101.

8103 Fatal init error AC
see 8101.

8104 Fatal system error AC
see 8101.

8105 Fatal init error AC
see 8101.

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

8107 PC-COM card not working
see 8106.

8108 Fatal error on PC-COM card
see 8106.

8109 Fatal error on PC-COM card
see 8106.

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

8111 Wrong configuration of PC-COM
see 8110.

8113 Invalid data (pccom.hex)
see 8110.

8114 Programming error on PC-COM
see 8110.

8115 PC-COM packet acknowledge missing
see 8110.

8116 PC-COM startup error
see 8110.

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

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

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

8120 PC interrupt no. unmaskable
see 8119

8121 Invalid command to PC-COM
Cause: Internal error or defective cable
Remedy: Check cables (screw it); Restart software or reinstall when necessary, report to EMCO, if repeatable.

8122 Internal AC mailbox overrun
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8123 Open error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8124 Write error on record file
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8125 Invalid memory for record buffer
Cause: Insufficient RAM, record time exceeding.
Remedy: Restart software, ev. remove drivers etc. to gain more RAM, reduce record time.

8126 AC Interpolation overrun
Cause: Ev. insufficient computer performance.
Remedy: Set a longer interrupt time in WinConfig. This may result in poorer path accuracy.

8127 Insufficient memory
Cause: Insufficient RAM
Remedy: Close other programs, restart software, ev. remove drivers etc. to gain more RAM.

8128 Invalid message to AC
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8129 Invalid MSD data - axisconfig.
see 8128.

8130 Internal init error AC
see 8128.

8130 Internal init error AC
see 8128.

8132 Axis accessed by multiple channels
see 8128.

8133 Insufficient NC block memory AC
see 8128.

8134 Too much center points programmed
see 8128.

8135 No centerpoint programmed
see 8128.

8136 Circle radius too small
see 8128.

8137 Invalid for Helix specified
Cause: Wrong axis for helix. The combination of linear and circular axes does not match.
Remedy: Program correction.

8140 Maschine (ACIF) not responding
Cause: Machine off or not connected.
Remedy: Switch on machine or connect.

8141 Internal PC-COM error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8142 ACIF Program error
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable.

8143 ACIF packet acknowledge missing
see 8142.

8144 ACIF startup error
see 8142.

8145 Fatal init data error (acif.hex)
see 8142.

8146 Multiple request for axis
see 8142.

8147 Invalid PC-COM state (DPRAM)
see 8142.

8148 Invalid PC-COM command (CNo)
see 8142.

8149 Invalid PC-COM command (Len)
see 8142.

8150 Fatal ACIF error
see 8142.

8151 AC Init Error (missing RPG file)
see 8142.

8152 AC Init Error (RPG file format)
see 8142.

8153 FPGA program timeout on ACIF
see 8142.

8154 Invalid Command to PC-COM
see 8142.

8155 Invalid FPGA packet acknowledge
see 8142 or hardware error on ACIF board (contact EMCO Service).

8156 Sync within 1.5 revol. not found
see 8142 or Bero hardware error (contact EMCO Service).

8157 Data record done
see 8142.

8158 Bero width too large (referencing)
see 8142 or Bero hardware error (contact EMCO Service).

8159 Function not implemented
Bedeutung: In normal operation this function can not be executed

8160 Axis synchronization lost axis 3..7
Cause: Axis spins or slide is locked, axis synchronisation was lost
Remedy: Approach reference point

8161 X-Axis synchronization lost
Step loss of the step motor. Causes:
- Axis mechanically blocked
- Axis belt defective
- Distance of proximity detector too large (>0,3mm) or proximity detector defective
- Step motor defective

8162 Y-Axis synchronization lost
see 8161

8163 Z-Axis synchronization lost
see 8161

8164 Software limit switch max axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8168 Software limit overtravel axis 3..7
Cause: Axis is at traverse area end
Remedy: Retract axis

8172 Communication error to machine
Cause: Internal error
Remedy: Restart software or reinstall when necessary, report to EMCO, if repeatable. Check connection PC - machine, eventually eliminate distortion sources.

8173 INC while NC program is running
Remedy: Stop the program with NC stop or with Reset. Traverse the axis.

8174 INC not allowed
Cause: At the moment the axis is in motion.
Remedy: Wait until the axis stops and then traverse the axis.

8175 MSD file could not be opened
Cause: Internal error
Remedy: Restart software oder bei Bedarf neu installieren, report to EMCO, if repeatable.

8176 PLS file could not be opened
see 8175.

8177 PLS file could not be accessed
see 8175.

8178 PLS file could not be written
see 8175.

8179 ACS file could not be opened
see 8175.

8180 ACS file could not be accessed
see 8175.

8181 ACS file could not be written
see 8175.

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

8184 Invalid interpolaton command

8185 Forbidden MSD data change
see 8175.

8186 MSD file could not be opened
see 8175.

8187 PLC program error
see 8175.

8188 Gear command invalid
see 8175.

8189 Invalid channel assignement
see 8175.

8190 Invalid channel within message
see 8175.

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

8192 Invalid axis in command
see 8175.

8193 Fatal PLC error
see 8175.

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

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

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

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

8198 Internal error (to manny threads)
see 8175.

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

8200 Thread without spindle on
Remedy: Switch on spindle

8201 Internal thread error (IPO)
see 8199.

8201 Internal thread error (IPO)
see 8199.

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

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

8205 PLC Runtime exceeded
Cause: Insufficient computer performance

8206 Invalid PLC M-group initialisation
see 8199.

8207 Invalid PLC machine data
see 8199.

8208 Invalid application message
see 8199.

8212 Rotation axis not allowed
see 8199.

8213 Circle and rotation axis can't be interpolated

8214 Thread and rotation axis cant't be interpolated

8215 Invalid state
see 8199.

8216 No rotation axis for rotation axis switch
see 8199.

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

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

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

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

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

8222 New master spindle is not valid
Cause: The indicated master spindle is not valid when switching over to the master spindle.
Remedy: Correct the spindle number.

8224 Invalid stop mode
see 8199.

8225 Invalid parameter for BC_MOVE_TO_IO!
Cause: The machine is not configurated for touch probes. A traversing movement with rotary axis is not allowed during touch probe operating mode.
Remedy: Remove the rotary axis movement from the traversing movement.

8226 Rotary axis switch not valid (MSD data)!
Cause: The indicated spindle does not have a rotary axis.

8228 Rotary axis switch not allowed while axis move!
Cause: The rotary axis has moved during switching over to the spindle operating mode.
Remedy: Stop the rotary axis before switching.

8229 Spindle on not allowed while rotary axis is active!

8230 Program start not allowed due to active spindle rotation axis!

8231 Axis configuration (MSD) for TRANSMIT not valid!
Cause: Transmit is not possible at this machine.

8232 Axis configuration (MSD) for TRACYL not valid!
Cause: Tracyl is not possible at this machine.

8233 Axis not available while TRANSMIT/ TRACYL is active!
Cause: Programming of the rotary axis is not allowed during Transmit/ Tracyl.

8234 Axis control grant removed by PLC while axis interpolates!
Cause: Internal error
Remedy: Delete error with reset and inform EMCO.

8235 Interpolation invalid while axis control grant is off by PLC!
see 8234.

8236 TRANSMIT/TRACYL activated while axis or spindle moves!
see 8234.

8237 Motion through pole in TRANSMIT!
Cause: It is not allowed to move through the coordinates X0 Y0 inTransmit.
Remedy: Alter the traversing movement.

8238 Speed limit in TRANSMIT exceeded!
Cause: The traversing movement gets too close to the coordinates X0 Y0. In order to observe the programmed feed rate, the maximum speed of the rotary axis would have to be exceeded.
Remedy: Reduce the feed rate. Set the value of the C-axis feed limitation in WinConfig, machine data settings / general machine data/ to 0.2. Thus, the feed rate will be automatically reduced near the coordinates X0 Y0.
The distance to the center is calculated with the following formula:
for CT155/CT325/CT450:
F[mm/min] * 0.0016 = distance [mm]
for CT250:
F[mm/min] * 0.00016 = distance [mm]
This applies for rapid traverse in transmit:
CT155/250/325: 4200 mm/min
CT450: 3,500 mm/min

8239 DAU exceeded 10V limit!
Cause: Internal error
Remedy: Start the software again or install it anew. Report the error to EMCO.

8240 Function not valid during active transformation (TRANSMIT/TRACYL)!
Cause: The Jog and INC operating mode are not possible during Transmit in X/C and during Tracyl in the rotary axis.

8241 TRANSMIT not enabled (MSD)!
Cause: Transmit is not possible at this machine.

8242 TRACYL not enabled (MSD)!
Cause: Tracyl is not possible at this machine.

8243 Round axis invalid during active transformation!
Cause: It is not allowed to program the rotary axis during Transmit/Tracyl.

8245 TRACYL radius = 0!
Cause: When selecting Tracyl, a radius of 0 was used.
Remedy: Correct the radius.

8246 Offset alignment not valid for this state!
see 8239.

8247 Offset alignment: MSD file write protected!

8248 Cyclic supervision failed!
Cause: The communication with the machine keyboard is interrupted.
Remedy: Start the software again or install it anew. Report the error to EMCO.

8249 Axis motion check alarm!
see 8239

8250 Spindle must be rotation axis !
see 8239

8251 Lead for G331/G332 missing !
Cause: The threading pitch is missing or the starting coordinates are identical to the target coordinates.
Remedy: Program the threading pitch.
Correct the target coordinates.

8252 Multiple or no linear axis programmed for G331/G332 !
Remedy: Program exactly one linear axis.

8253 Speed value for G331/G332 and G96 missing !
Cause: No cutting speed has been programmed.
Remedy: Program the cutting speed.

8254 Value for thread starting point offset not valid!
Cause: The thread starting point offset is not within the range of 0 to 360°.
Remedy: Correct the thread starting point offset.

8255 Reference point not in valid software limits!
Cause: The reference point has been defined outside the software limit switches.
Remedy: Correct the reference points in WinConfig.

8256 Spindle speed too low while executing G331/G332!
Cause: During tapping the spindle speed has decreased. Perhaps the incorrect threading pitch was used or the core drilling is not correct.
Remedy: Correct the threading pitch. Adapt the diameter to the core drilling.

8257 Real Time Module not active or PCI card not found!
Cause: ACC could not be started correctly or the PCI card in the ACC was not recognized.
Remedy: Report the error to EMCO.

8258 Error allocating Linux data!
see 8257.

8259 Current thread in sequence not valid!
Cause: One block of a thread in sequence has been programmed without thread G33.
Remedy: Correct the program.

8261 Missing thread in sequence !
Cause: A successive thread has not been programmed for a thread in sequence, the number has to be in accordance with the SETTHREADCOUNT () that has been defined before.
Remedy: Correct the number of threads in the thread in sequence and add a thread.

8262 Reference marks are not close enough !
Cause: The settings of the linear scale have been changed or the linear scale is defective.
Remedy: Correct the settings. Contact EMCO.

8263 Reference marks are too close together!
see 8262.

8265 No or wrong axis in axis switch command!
Cause: Internal error.
Remedy: Please contact the EMCO after-sales service.

8266 Invalid tool
Cause: Programmed tool is not set in magazine.
Remedy: Correct tool number and/or load tool in magazine.

8267 Speed difference to high
Cause: Die Soll- und Istgeschwindigkeit der Achse weichen zu stark voneinander ab.
Remedy: Run the program again with reduced feed. If this does not remedy the problem, contact EMCO.

8269 USBSPS and ACC speed values or override are different
Cause: USBSPS and ACC have diferent rotations saved.
Remedy: Delete alarm using the RESET key. If this error reoccurs, contact EMCO.

8270 Reference switch defective
Cause: The reference switch did not switch within the specified range.
Remedy: Cancel alarm with RESET button. If the problem occurs several times, contact EMCO.

8271 Tool load in locked place not possible
Cause: There was an attempt to swing a tool into a locked place in the magazine.
Remedy: Choose a free, unlocked place in the magazine and then swing the tool into the magazine.

8272 Old PLC version, update necessary
Cause: The PLC version is too old to fully support randomised tool management.
Remedy: Update the PLC.

8273 Spindle overload
Cause: The spindle was overloaded and during processing the speed fell (to half of the target speed for more than 500ms).
Remedy: Cancel alarm with RESET button. Change the cut data (feed, speed, infeed).

8274 Define tool before loading
Cause: The tool must be defined in the tool list before it is possible to transfer the tool into the spindle.
Remedy: Create the tool in the tool list, then load.

8277 Sinamics error
Cause: Error in Sinamics drive.
Remedy: Turn off and on the machine. Contact EMCO if this doesn't help.

8704 Feed override absent, REPOS is not executed
Cause: The REPOS command is not executed because the feed override is set to 0%.
Remedy: Change the feed override and restart REPOS.

8705 Tool sorting active
Cause: The tools will be re-sorted with random tool management to facilitate non-random operation (tool 1 at place 1, tool 2 at place 2, etc.).
Remedy: Wait until sorting has finished. The controller will delete the report independently.

8706 Check new controller - tool table
Cause: The controller was changed with random tool management active.
Remedy: Check the tool or place table to clear the alarm.

8707 Ending with auxiliary drives switched on not possible
Cause: An attempt was made to end the controller, although the auxiliary drives are still switched on.
Remedy: Switch off the auxiliary drives and then end the controller.

22000 Gear change not allowed
Cause: Gear step change when the spindle is active.
Remedy: Stop the spindle and carry out a gear step change.

22270 Feed too high (thread)
Cause: Thread pitch too large / missing, Feed for thread reaches 80% of rapid feed
Remedy: Program correction, lower pitch or lower spindle speed for thread

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 for OPERATE
Fanuc 31i

2200 Syntax error in line %s, column %s
Cause: Syntax error in the program code.

2001 Circle end point invalid
Cause: The start-middle point and middle-end point distances differ by more than 3 µm.
Remedy: Correct circular arc point.

2300 tracyl without corresponding round-axis invalid
Cause: Maybe the machine has no rotary axis.

3000 Traverse feed axis manually to position %s
Remedy: Move the axis manually to the required position.

3001 Tool T.. change!
Cause: A new tool was programmed in the NC program.
Remedy: Clamp the required tool in the machine.

4001 slot width too small
Cause: The tool radius is too large for the slot to be milled.

4002 slot length to small
Cause: The slot length is too small for the slot to be milled.

4003 length equal zero
Cause: Pocket length, pocket width, stud length, stud width are zero.

4004 slot width too big
Cause: The programmed slot width is larger than the slot length.

4005 depth equal zero
Cause: No machining takes place since no effective cutting depth has been defined.

4006 corner radius too big
Cause: The corner radius is too large for the size of the pocket.

4007 diameter too big
Cause: The remaining material (nominal diameter - diameter of the prebore) /2 is larger than the tool diameter.

4008 diameter too small
Cause: The tool diamter is too large for the intended bore.
Remedy: Enlarge the nominal diameter and use a smaller milling cutter.

4009 length too small
Cause: Width and length must be larger than the double tool radius.

4010 diameter less equal zero
Cause: The pocket diameter, the stud diameter, etc. must not be zero.

4011 blank diameter too big
Cause: The diameter of the machined pocket must be larger than the diameter of the premachined pocket.

4012 blank diameter too small
Cause: The diameter of the machined stud must be smaller than the diameter of the premachined stud.

4013 start angle equal to end angle
Cause: Start angle and end angle for hole pattern are identical.

4014 tool radius 0 not permitted
Cause: Tool radius zero is not permitted.
Remedy: Select a valid tool.

4015 no outer contour defined
Cause: The contour file indicated in the cycle was not found.

4017 tool radius too big
Cause: For the programmed machining, a tool being too large was selected. Therefore, machining is not possible.

4018 allowance must not be 0
Cause: There were programmed finishing operations without finishing offset.

4019 too many iterations
Cause: The contour definitions are too complex for the roughing-out cycle.
Remedy: Simplify the contour.

4020 illegal radian correction
Cause: An error has occured during the programming of the radius compensation.
Remedy: Check the cycle parameters.

4021 can't calculate parallel contour
Cause: The control was not able to calculate the tool radius compensation.
Remedy: Check the programmed contour for plausibility. Maybe contact EMCO.

4022 illegal contour definition
Cause: The programmed contour is not suited to the selected machining.
Remedy: Check the programmed contour.

4024 no contour definition
Cause: The contour file being defined in the cyle has not been found.

4025 internal calculation error
Cause: An unexpected error has occured during calculation of the cycle movements.
Remedy: Please inform the EMCO after-sales service.

4026 allowance too big
Cause: A part of the finishing offset (for several finishing passes) is larger than the total finishing offset.
Remedy: Correc the finishing offsets.

4028 pitch 0 not permitted
Cause: The thread was programmed with pitch zero.

4029 undefinded working mode
Cause: Internal error (invalid machining type for the thread).

4030 function not yet supported
Cause: Roughing out with pockets is not implemented yet.
Remedy: Please inform the EMCO after-sales service.

4031 value not permitted
Cause: An invalid retracting direction was programmed during inside turning.

4032 plunging must be defined
Cause: For the programmed cycle no cutting depth has been programmed.

4033 radius/chamfer too big
Cause: The radius, respectively the chamfer, cannot be inserted in the programmed contour.
Remedy: Reduce the radius, respectively the chamfer.

4034 diameter too big
Cause: The programmed starting point and the machining diameter are contradictory.

4035 diameter too small
Cause: The programmed starting point and the machining diameter are contradictory.

4036 unknown working direction
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4037 unknown working type
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4038 unknown sub cycle
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4039 rounding not possible
Cause: The programmed radius contradicts the rest of the cycle parameters.

4042 illegal tool width
Cause: The tool width for the cutting-off cycle must be defined.

4043 groove width too small
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4044 distance not defined
Cause: The distance for the multiple grooving cycle must not be zero.

4045 illegal allowance type
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4046 invalid speed
Cause: The spindle speed must be nonzero.

4047 invalid end point
Cause: The programmed end point contradicts the rest of the cycle definition.

4048 tool cut width too small
Cause: The cutting edge is too small for the programmed cutting depth.

4050 invalid distance
Cause: The hole patterns do not tally with the selected distance.

4052 working pattern not possible
Cause: Error in the definition of the hole pattern. The number of bores is contradictory.

4053 invalid start point
Cause: Internal error.
Remedy: Please inform the EMCO after-sales service.

4055 illegal working direction
Cause: The machining direction is contradictory to the rest of the cycle definition.

4057 plunging angle less equal zero
Cause: The plunging angle must be between 0 and 90 degree.

4058 chamfer too large
Cause: The programmed chamfer is too large for the pocket cycle.

4062 radius/chamfer too small
Cause: The radius, respectively the chamfer, cannot be machined with the current tool radius.

4066 invalid mill step
Cause: The mill step must be greater than zero.

4069 invalid angle
Cause: An angle of zero degree is not permitted.

4072 plunging too small
Cause: For the cycle, a cutting depth has been selected that leads to extra-long machining time.

4073 invalid clearance angle
Cause: The clearance angle indicated for the tool cannot be machined.
Remedy: Correct the clearance angle for the tool.

4074 contour-file not found
Cause: The contour file indicated in the cycle has not been found.
Remedy: Please select the contour file for the cycle.

4075 not machinable with selected tool
Cause: The tool is too wide for the programmed groove.

4076 reciprocating plunge cut impossible (initial move too short)
Cause: The first movement of the contour is shorter than the double tool radius and cannot be therefore used for the swinging delivery.
Remedy: Extend the first movement of the contour.

4077 wrong tool type in grooving or cut-off cylce
Cause: The wrong tool type was used in the cutting cycle.
Remedy: Use only grooving and punch tools in the cutting cycles.

4078 radius of helix too small
Cause: The pitch of the helix is ≤ 0.
Remedy: Program the radius > 0.

4079 pitch of helix too small
Cause: The radius of the helix is ≤ 0.
Remedy: Program the pitch > 0..

4080 radius of helix or tool too large
Cause: The helical approach cannot be executed with the selected data for the helix and the current tool radius without a contour breach.
Remedy: Use a tool with a smaller radius or reduce the radius of the helix.

4200 leaving movement is missing
Cause: No movement after the tool radius compensation was deactivated in the current plane.
Remedy: Insert the departing movement in the current plane after having deactivated the tool radius compensation.

4201 TPC off missing
Cause: The tool radius compensation has not been deactivated.
Remedy: Deactivate the tool radius compensation.

4202 TPC requires at least three movements
Cause: The tool radius compensation requires at least 3 movements in the current plane in order to calculate the tool radius compensation.

4203 approaching movement not possible
Cause: It was not possible to calculate the approaching movement.

4205 leaving movement not possible
Cause: It was not possible to calculate the departing movement.

4208 TPC curve could not be calculated
Cause: It was not possible to calculate the tool radius compensation for the programmed contour.

4209 switching the plane is not allowed when TPC is switched on
Cause: The programmed plane must not be changed during the tool radius compensation.
Remedy: Remove the change of planes during the tool radius compensation.

4210 tool path compensation already activated
Cause: G41 is active and G42 was programmed or G42 is active and G41 was programmed.
Remedy: Switch tool radius compensation off with G40 before programming the radius compensation again.

4211 Bottleneck detected
Cause: In the radius correction calculation some parts of the contour were omitted, as too large a milling cutter tool was used.
Remedy: Use a smaller milling cutting tool to process the contour completely.

4212 Infeed has been programmed twice during approach
ause: After the approach movement a second infeed has been programmed, without previously moving to the work plane.
Remedy: First program a movement to the work plane before programming a second infeed.

5000 drill manually now

5001 contour has been adjusted to the programmed clearance angle
Cause: The programmed contour was adapted to the programmed clearance angle. Maybe there will remain rest material that cannot be machined with this tool.

5500 3D simulation: Internal error
Cause: Internal error inside the 3D simulation.
Remedy: Restart the software or, if necessary, report the error to EMCO customer service.

5502 3D simulation: Tool place invalid
Cause: Tool place unavailable on the machine used.
Remedy: Correct tool call-up.

5503 3D simulation: Chuck invalid owing to the unmachined part definition

Cause: The distance from the front of the unmachined part is > the unmachined part length.
Remedy: Change the distance.

5505 3D simulation: Unmachined part definition invalid

Cause: Implausibility in the unmachined part geometry (e.g. expansion in one axis ≤ 0, inside diameter > outside diameter, unmachined part contour not closed, etc.).
Remedy: Correct unmachined part geometry.

5506 3D simulation: STL chuck file has auto-overcuts

Cause: Error in the chuck description.
Remedy: Correct file.

5507 3D simulation: Pole transit on TRANSMIT!

Cause: Travel comes too close to the X0 Y0 coordinates.
Remedy: Change travel.

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

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

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 (only with WinNC SINUMERIK 810D/840D and SINUMERIK Operate) 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 810D/840D T and M
- FANUC Series 0-TC and 0-MC
- FANUC Series 21 TB and MB
- 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.

Only for WinNC SINUMERIK 810D/840D:
Setting the serial DNC interface parameter is done as with the data transfer via the serial interface in the "SERVICES" operating area via the Softkeys "V24 USER" and "SETTINGS", in which the DNC serial interface must be selected.
The DNC format "Full Binary" requires 8 databits for the data transfer.

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

General

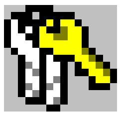
Note:
The settings which are available in EMConfig are depending on the machine and the control that is used.

EMConfig is a configuration software for WinNC. EMConfig helps you to alter the settings of WinNC.

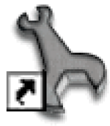
The most important settings are:

- Control language
- System of measurement mm - inch
- Activate accessories
- Selection of interface for control keyboard

Using EMConfig you can also activate diagnostic functions in case of troubles - that way you get help immediately.



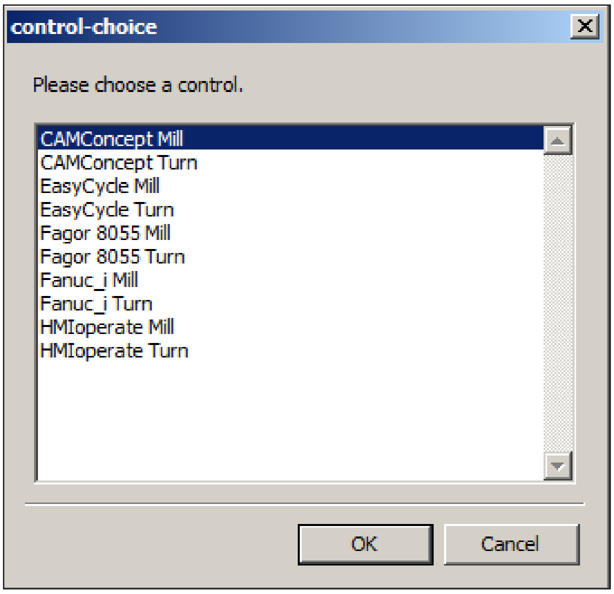
Safety-related parameters are protected by a password. They can only be activated by set-up technicans or by technical support representatives.



Icon for EMConfig

How to start EMConfig

Open EMConfig.



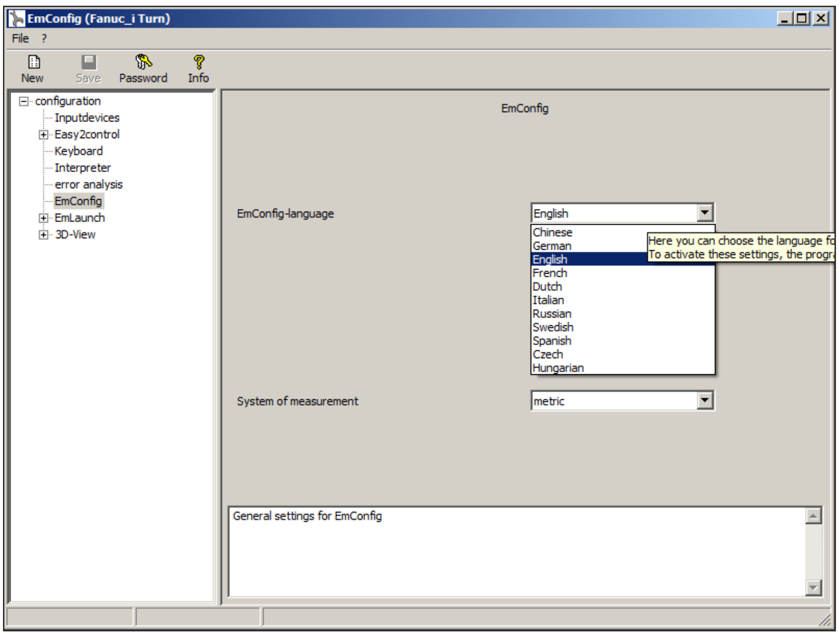
Selection box for control type

In case several control types are installed, a selection box will appear on the screen.

Select the required control type and click OK.

The following settings are only valid for the selected control type.

The window for EMConfig appears on the screen.



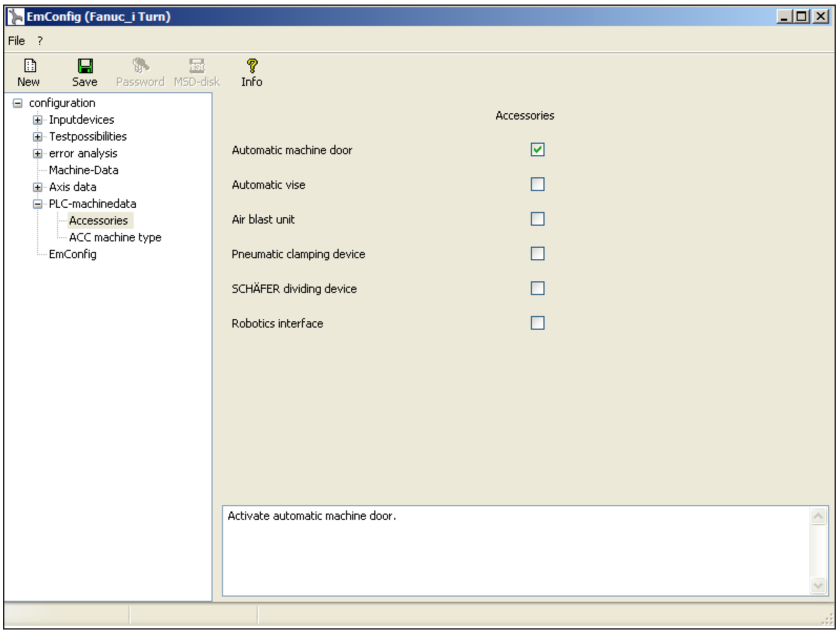
Change the language of EMConfig

Here you can change the language of EMConfig. In order to activate the settings, restart the program.

Note:
Select the desired menu item. The appropriate function is explained in the text box.

How to activate accessories

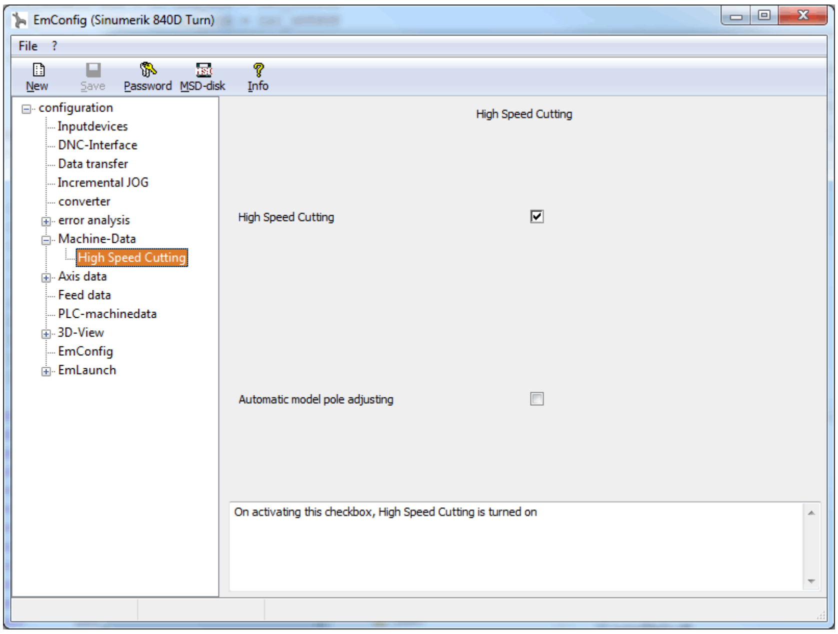
When you install accessories on your machine, you need to activate them here.



Activate accessories

High Speed Cutting

On activating this checkbox, High Speed Cutting is turned on.

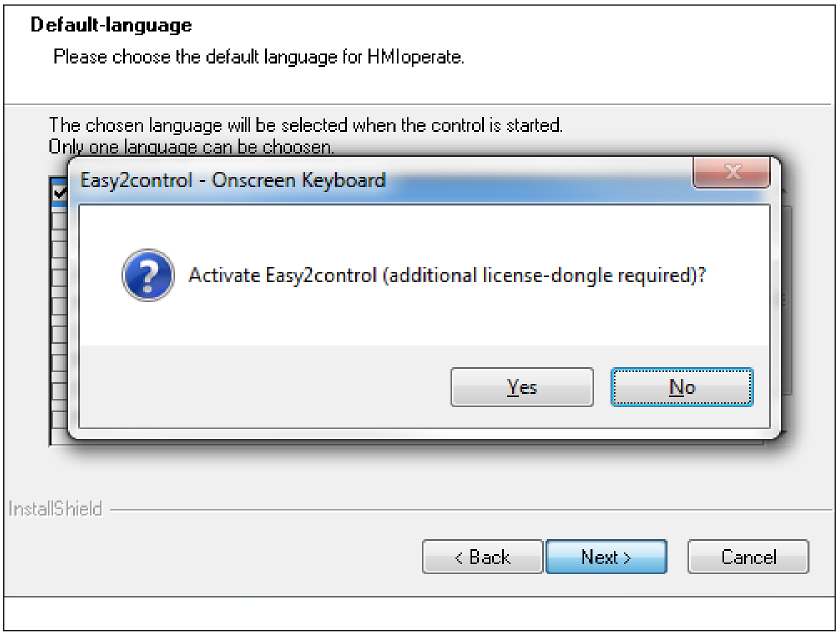


Activate High Speed Cutting

By using high speed cutting, the setting of the axis controller is adjusted. This gain is only effective until the programmed feed rate of 2500 mm/min and allows contour faithful retraction of the tool path and generating of sharp edges.
If the feed is set up to higher than 2500 mm/min, it is automatically reset to the normal operating mode and sanded and rounded edges are created.

Easy2control on screen operation

Installation and activation using the example of WinNC for Sinumerik Operate.

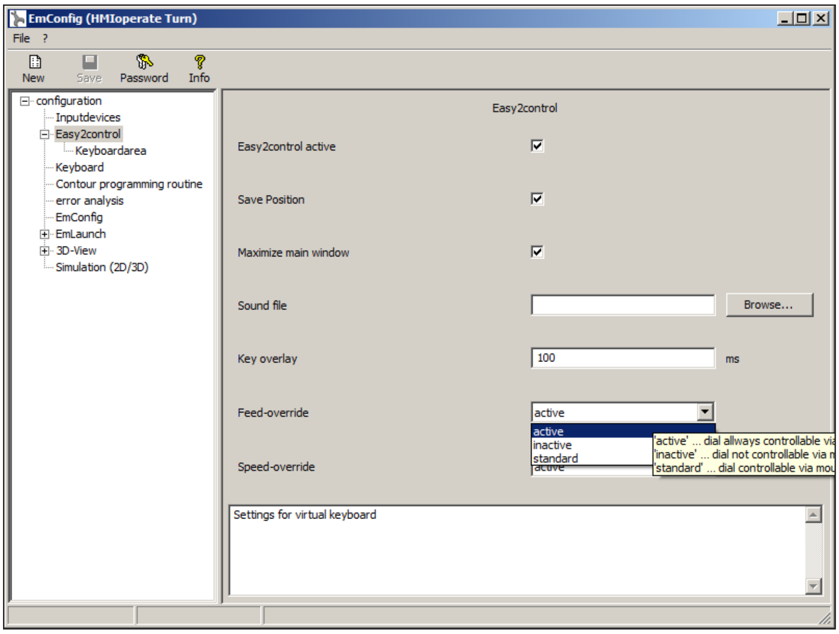


Activating Easy2control

When installing the software WinNC for Sinumerik Operate you will be prompted to activate Easy2control. In order to use the Software without restriction, the supplied dongle must be connected to a free USB port.

Settings

This mask allows you to enable or disable Easy-2control and make settings.



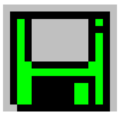
Easy2control settings

Dial feed-override and dial speed-override:

- **Aktive**: dial always controllable via mouse/ touchscreen (even with available hardware-version).
- **Inaktive**: dial not controllable via mouse/ touchscreen.
- **Standard**: dial controllable via mouse/touch-screen when no hardware-version is available.

Note:
If Easy2control is used without the hardware-dongle, the controls are diactivated and an appropriate alarm is output by the controller. However, the virtual keyboard is displayed completely.

How to save changes



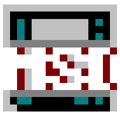
After the settings, the changes must be saved.

Select "Save" or click on the icon.

Note:
Input fields highlighted in red indicate inadmissible values. Inadmissible values are not saved in EMConfig.

After saving the changes, create a machine data floppy disk (MSD) or a machine data USB flash drive.

How to create machine data floppy disk or machine data USB flash drive



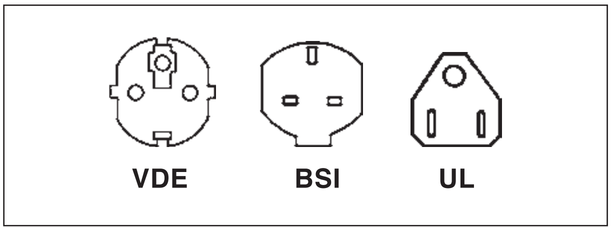
After having changed the machine data, the machine data floppy disk or the machine data USB flash drive must be in the appropriate drive.
Otherwise your changes cannot be saved and get lost.

Y: External Input Devices

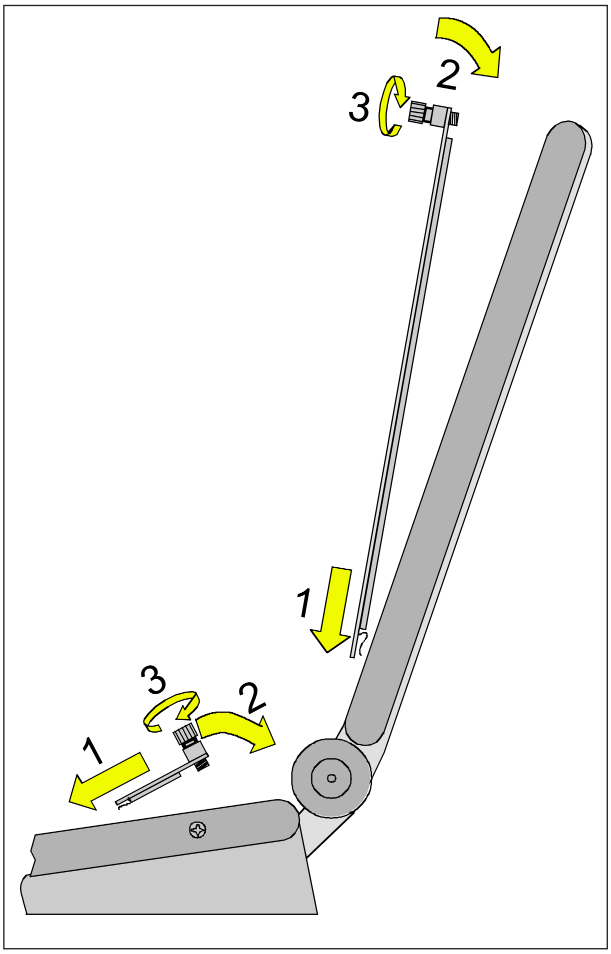
EMCO Control Keyboard USB

Scope of supply

The scope of supply for a complete control keyboard consists of two parts:
• Basic case
• Key module



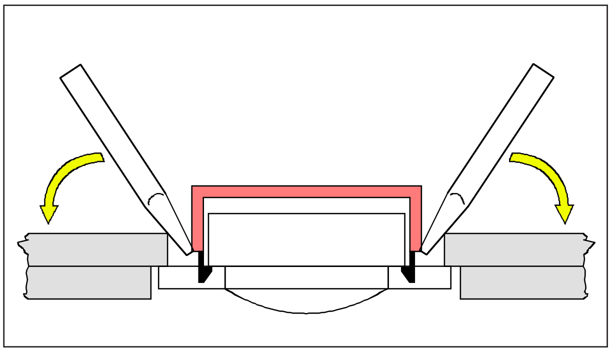
Ref. No.	Description
X9B 000	Basic unit with USB cable
X9Z 600	TFT Display with screen cable and power supply unit
A4Z 010	Mains cable VDE
A4Z 030	Mains cable BSI
A4Z 050	Mains cable UL
X9Z 040N	Key module SINUMERIK 840 2 key sheets with keys 1 package exchange keys
X9Z 050N	Key module FAGOR 8055 TC 2 key sheets with keys
X9Z 055N	Key module FAGOR 8055 MC 2 key sheets with keys
X9Z 110N	Key module FANUC 0 2 key sheets with keys 1 package exchange keys
X9Z 130N	Key module FANUC 21 2 key sheets with keys 1 package exchange keys
X9Z 426N	Key module HEIDENHAIN 426/430 2 key sheets with keys 1 package exchange keys
X9Z 060	Key module WinNC for SINUMERIK OPERATE 2 key sheets with keys
X9Z 030	Key module WinNC for FANUC 31i 2 key sheets with keys 1 package exchange keys



Assembling

- Place the correseponding key sheet with the clips in the basic case (1).
- Pull the key sheet into the basic case, it must be insertet plainly (2).
- Fix the key sheet with the two knurled screws (3).

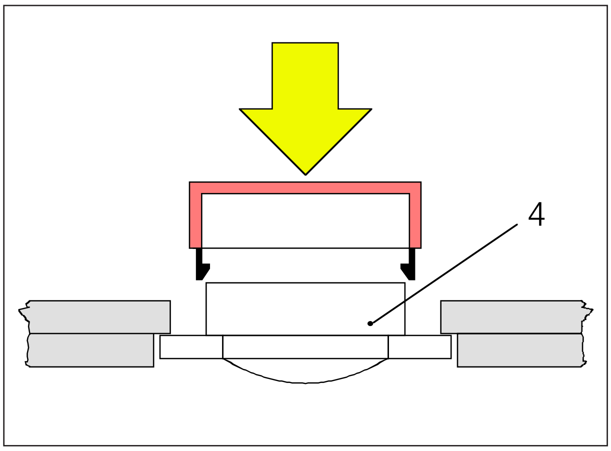
Note:
The key sheets must not be bended, otherwise the switching function can not be warranted.



Exchange of single key caps

Off works the keyboards are equipped with the keys for turning.
The scope of supply includes a package of exchange key caps to equip the keyboard for milling.
If you want to use the control keyboard for milling, you have to exchange a part of the key caps. Exchange them as shown on the following pages.

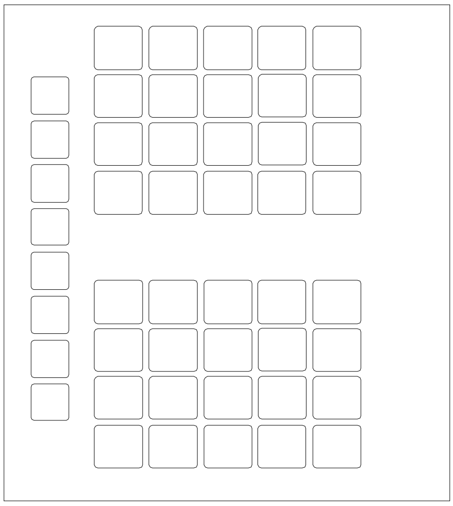
Note:
For the control type Heidenhain 426/430 only a milling version is available.

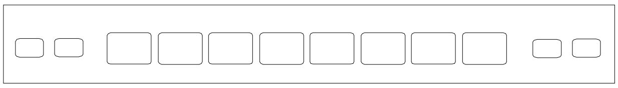


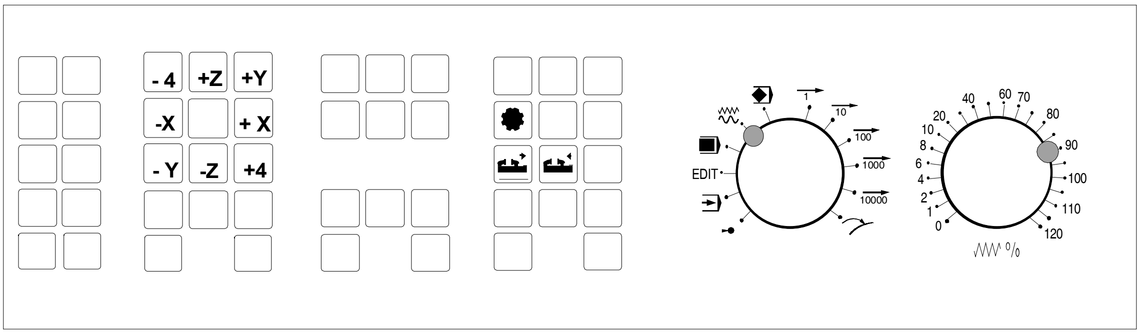
Take off
Pull out carefully the key caps to be exchanged with a fine screw driver or a knife.

Clip on
Move the key body in the middle of the recess. Push the key cap vertically down onto the key body, until the key cap snaps in tactily.

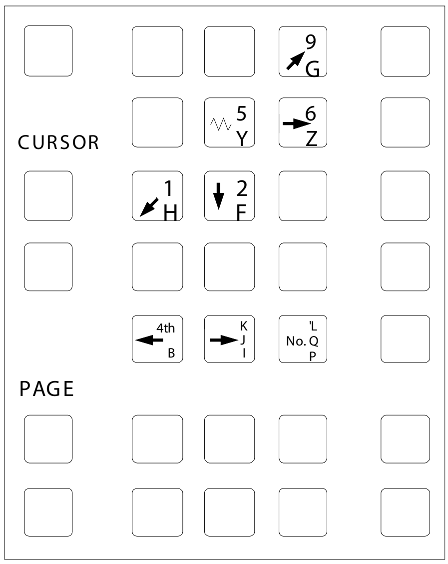
SINUMERIK 840D
Exchange key caps
for milling

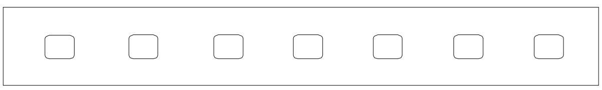


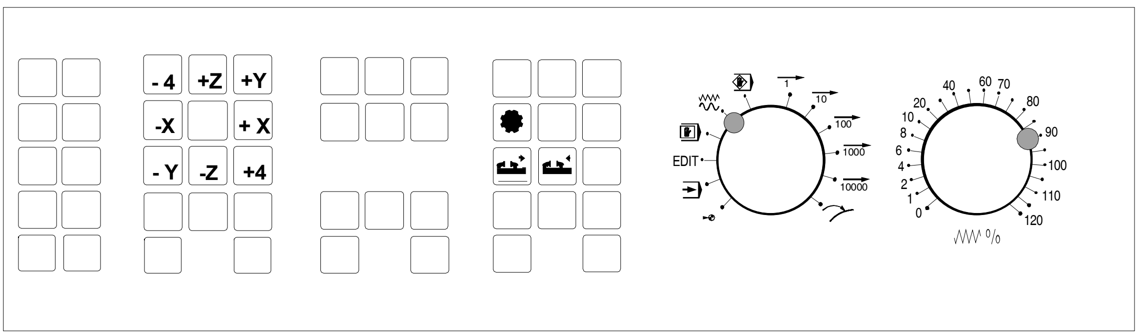




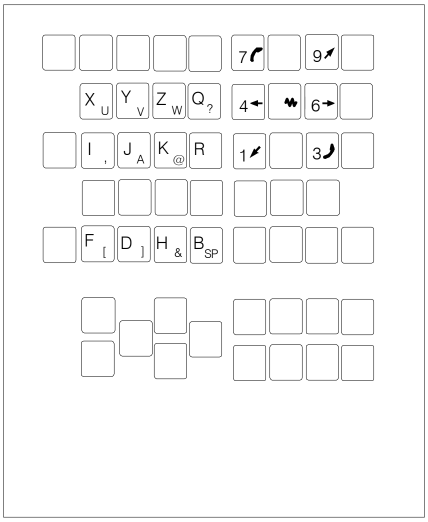
FANUC 0M
Exchange key caps
for milling

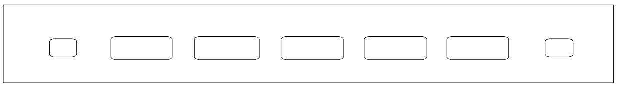


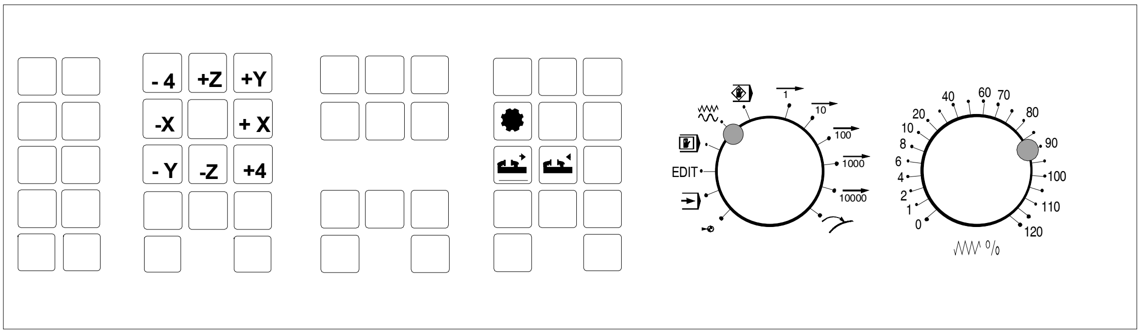




FANUC 21M, FANUC 31iM
Exchange key caps
for milling







Connection to the PC

The control keyboard is connected via USB interface to the PC.
The connection cable USB taking over at the same time the energy supply of the control keyboard is situated at the rear side of the control keyboard.

Settings at the PC software

Setting during new installation of the PC software

During the installation indicate the control keyboard and the respective USB interface.

Setting in case of PC software already installed

Select in EMConfig at the INI data settings the USB control keyboard as means of entry and the respective interface USB.
Furthermore, set the keyboard type to “New”.
Don’t forget to memorize the settings.

Easy2control On Screen operation

Easy2control adds a range of attractive applications to the successful interchangeable control system used in EMCO training machines. Suitable for use in machines and simulation workplaces alike, it displays additional control elements directly on the screen and, when used together with a touchscreen monitor, provides the ideal input interface.

Scope of supply

The software for Easy2control is part of the control software.
The dongle for a workstation license is delivered:

Ref. No. X9C 111

Technical data for the screen:

16:9 Full-HD Monitor (1920x1080) at the minimum

Currently available controls (T and M):

- Sinumerik Operate
- Fanuc 31i
- Sinumerik 840D
- Heidenhain 426 (M only)
- Fagor 8055

Note:
If a Full HD monitor is used without touchscreen function, the control is operated just with mouse and keyboard.

Operating areas

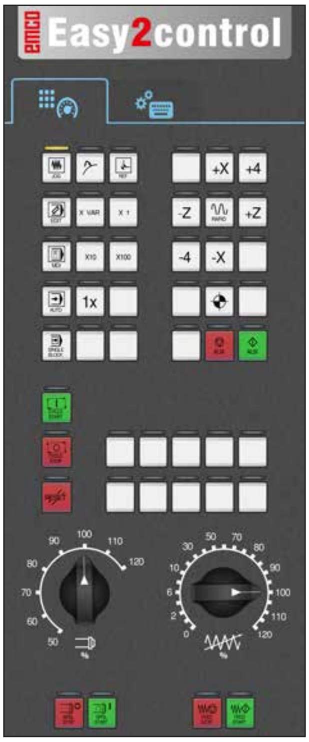
Sinumerik Operate

Machine control panel

Controller-specific operation

Control operation complete

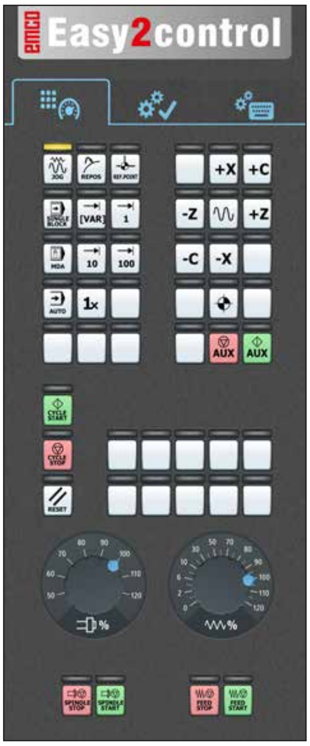
Fanuc 31i



Machine control panel

Control operation complete

Sinumerik 840D

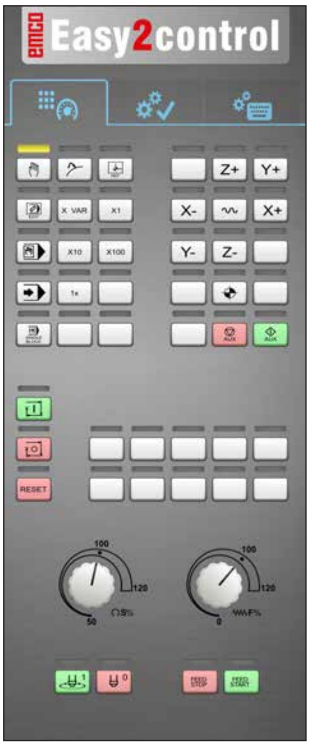


Machine control panel

Controller-specific operation

Control operation complete

Heidenhain 426

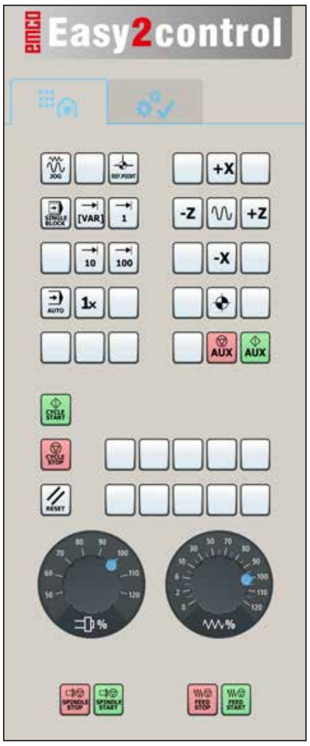


Machine control panel

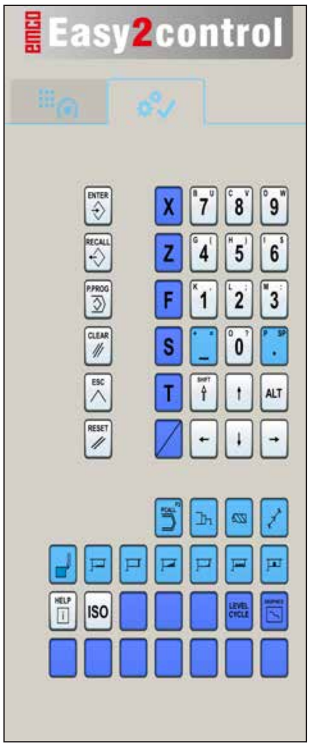
Controller-specific operation

Control operation complete

Fagor 8055



Machine control panel



Control operation complete

Refer to the chapter "Key Description" of the respective control description for operation and key function.

Note:
The screen display, based on customer-specific configurations, may look different.

Z: Software Installation Windows

System prerequisites

Machine with integrated control PC
- All Concept machines
- Machines that were converted to ACC
- MOC with Windows XP SP3 or higher (32 / 64 Bit)

Machines with included control PC and programming stations
- PC 1000 Mhz
- Windows XP SP2 or higher (32 / 64 Bit)
- Working memory min. 256 MB RAM
- free hard drive space 400 MB
- Programming station: 1*USB, machine version: 2*USB
- TCP/IP-capable network card for machine version)

Recommended system prerequisites
- PC Dual Core
- Windows 7 or higher
- Working memory 2 GB RAM
- free hard drive space 2 GB

Software installation

- Start Windows XP SP3 or higher
- Start the installation application on the USB stick or your download file.
- Follow the instructions from the installation guide.

For more informations regarding software installation and / or software update please refer to the documentation “short description for WinNC update installation”.

Note:
PC TURN and PC MILL have to be equipped with the conversion kit for ACC in order to operate EMCO WinNC.

Variants of WinNC

You can install EMCO WinNC for the following CNC control types:

- WinNC for SINUMERIK Operate T and M
- WinNC for FANUC 31i T and M
- SINUMERIK 810D/840D T and M
- HEIDENHAIN TNC 426
- FANUC Series 0-TC and 0-MC
- FANUC Series 21 TB and MB
- FAGOR 8055 TC and MC
- CAMConcept T and M
- EMCO EASY CYCLE T and M (except machine licence)

In case there are several control types installed, a menu appears when starting EM Launch from which you can select the desired type.
The following versions can be installed from the WinNC variants:

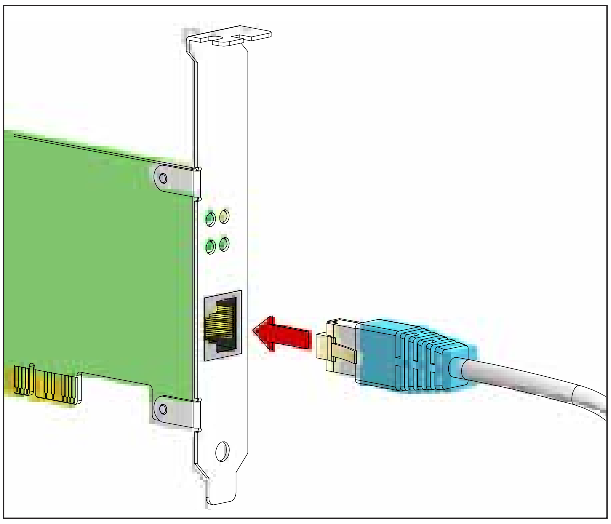
- Demo licence:
  The demo licence is valid for 30 days after the first use. 5 days before the demo licence expires, you can enter another valid licence key (see licence manager)

- Programming station:
  Programming and operation of the appropriate CNC control type is simulated by WinNC on your PC.
  - Single user licence:
    Authorizes to external programming of CNC-controlled machine tools on one PC workstation (machine-independent).
  - Multi-user licence:
    Authorizes to external programming of CNC-controlled machine tools. The multi-user licence can be installed on an unlimited number of PC workstations or in a network within the institute registered by the licensor (machine-independent).
  - Educational licence version:
    Is a time-limited multi-licence especially for schools and educational institutes.

- Machine licence:
  This licence allows to directly operate a PC-controlled machine (PC TURN, Concept TURN, PC MILL, Concept MILL) of WinNC as if it was operated by an ordinary CNC control.

Danger:
Removal and installation of the network card must only be carried by skilled personnel. The computer must be disconnected from the power supply (pull the power plug).

Note:
During a machine installation one network-card is reserved exclusively for the control of the machine.



Connection of the machine to the PC

Network card (ACC)

for:
Concept Turn 55
Concept Mill 55
Concept Turn 105
Concept Mill 105
Concept Turn 60

Only for machines with ACC kit:
PC Turn 50
PC Mill 50
PC Turn 100
PC Mill 120

Network card type: TCP/IP compatible network card

Setting the network card for the local connection to the machine:

IP address: 192.168.10.10
Subnetmask 255.255.255.0

In case of problems observe the instructions of your operating system (Windows help).

Instructions:
If the network connection to the machine could not be established at the start, the above adjustments are to be made.

Starting WinNC

If you choose AUTO START YES during the installation of your machine version, WinNC starts automatically after switching on the PC.

Otherwise proceed as follows:

1 Switch the machine on.

2 Wait for 20 seconds to ensure that the machine operating system is running before the network connection to the PC is established. Otherwise it is possible that no connection can be established.

3 Switch the PC on and start Windows.

4 Click on the start symbol at the bottom.

5 Select program, EMCO and click on WinNC.

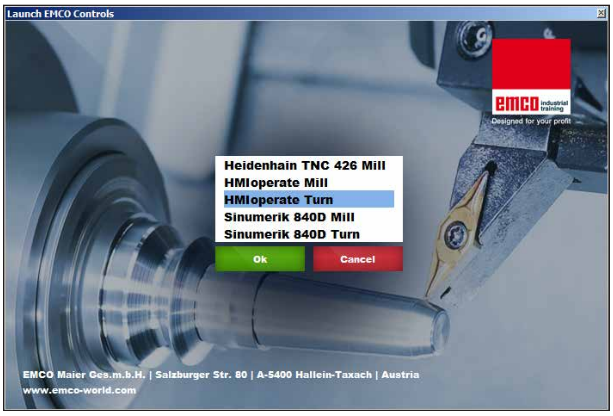
6 The start image will be shown on the screen. The licence holder is registered in the start screen.

7 If you have only installed one CNC control type, it starts immediately.

8 If you have installed several CNC control types, the selection menu appears.

9 Select the desired CNC control type (use cursor buttons or mouse) and press ENTER to start it.

10 If you use the control keyboard, you can select the desired CNC control type with the cursor buttons or mouse and start with the "NC-Start" button.



Selection menu EMLaunch

Note:
EMLaunch displays all WinNC und CAMConcept controls that are installed in the same directory.

Terminating WinNC

1 Switch off auxiliary drive with AUX OFF.
Only for machine places, not for programming stations.

2 By simultaneously pressing these buttons WinNC for Sinumerik Operate will be terminated specifically.
This corresponds to Alt+F4 on the PC keyboard.

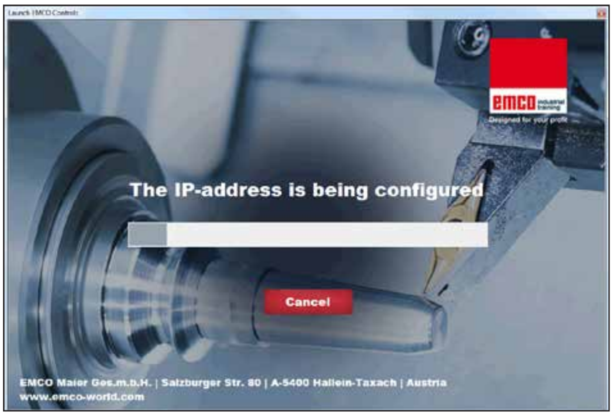
Checks by EmLaunch

In the ACC / ACpn-machine version EmLaunch is checking if a machine is available:

During the network configuration, the IP address is not configured correctly and DHCP for automatic configuration of the IP address is disabled. Connection to the machine is not possible.

DHCP disabled

It is attempt to configure the IP address automatically via DHCP.



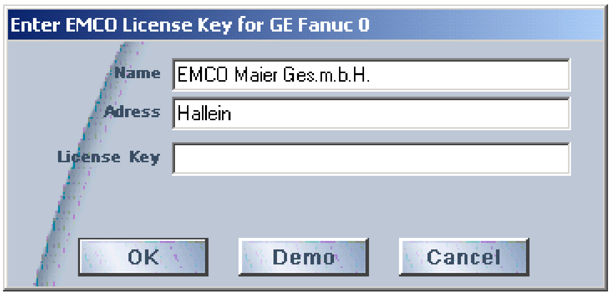
IP-address configuration

The IP configuration is correct and the connection to the machine is checked. Once the machine is available, the selection of the available controls is displayed.

Setup the connection to the machine

Conncection to the machine is OK

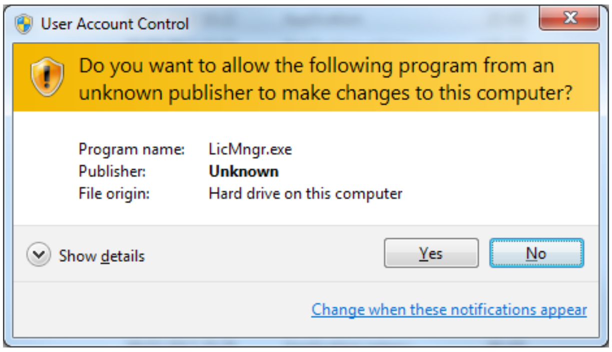
The connection to the machine is completed and the corresponding control can be started.



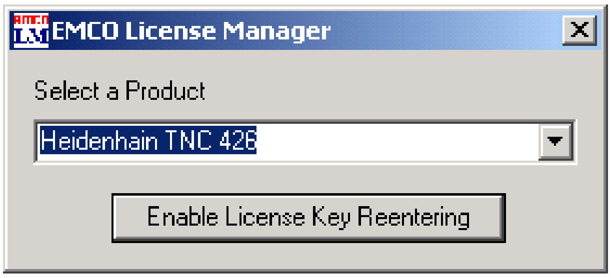
Input window licence key enquiry

Licence input

After the installation of an EMCO software product, an input window appears during initial operation and asks for name, address and licence key. This input window appears for every software product that is installed. In case a demo licence is desired (see page Z1), please select "DEMO". Then the input window reappears only 5 days before the expiry date of the demo licence. A subsequent input of a licence key is also possible via the licence manager (see licence manager below).



Run EMCO licence manager as an administrator



EMCO Licence Manager

Licence manager

The query in the UAC dialog box must be confirmed with Yes in order to start the Licence Manager.

For the release of additional function groups of existing EMCO software products it is necessary to enter a new licence key (exception: demo licence).

The ***EMCO Licence Manager*** (see picture on the bottom on the left) enables the input of further new licence keys. For this purpose select the new product in the selection window and confirm the input.

The next time you start your control software an input window appears and asks you to enter name, address and licence key (see picture on the top left).

Please note that the licence key is asked for each software product individually. The picture on the left shows e.g. the input prompt for the licence key for the software product "Heidenhain TNC 426".

Input licence key:
Start the WinNC with the option “Run as Administrator” right after installing the programm or launching the licence manager.