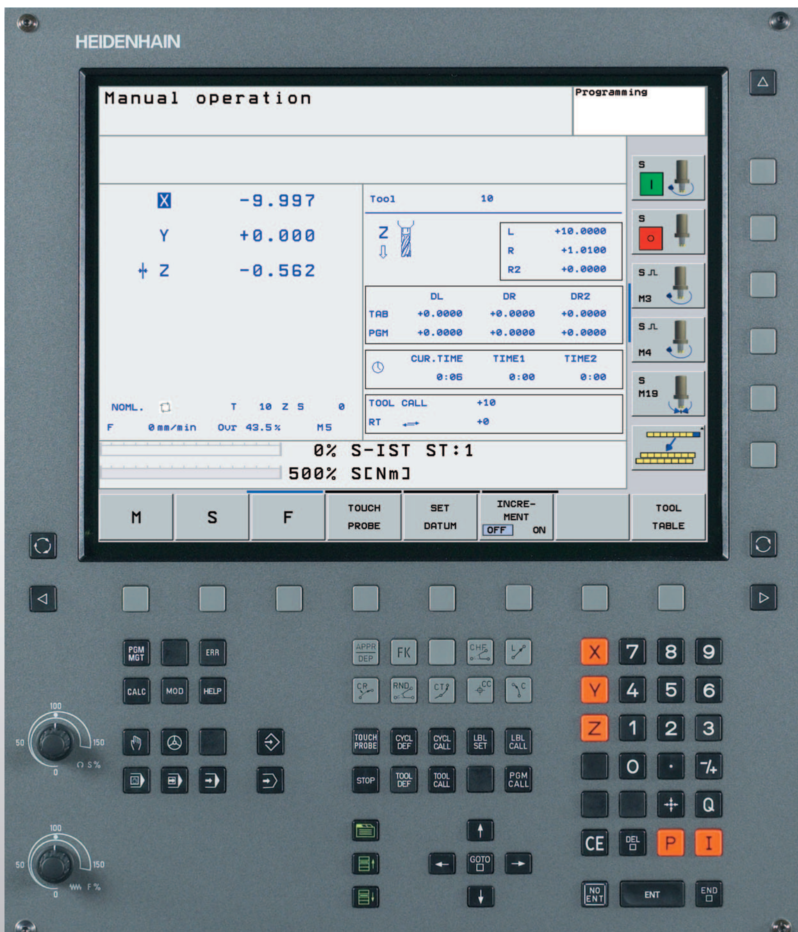
HEIDENHAIN



User's Manual HEIDENHAIN Conversational Programming

TNC 320

NC Software
340 551-01

English (en)
3/2006

Controls on the visual display unit

- Split screen layout
- Switch between machining or programming modes
- Soft keys for selecting functions in screen
- Switch the soft-key rows

Machine operating modes

- Manual Operation
- Electronic Handwheel
- Positioning with Manual Data Input (MDI)
- Program Run, Single Block
- Program Run, Full Sequence

Programming modes

- Programming and Editing
- Test Run

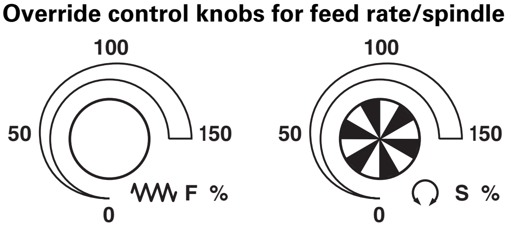
Program/file management, TNC functions

- PGM MGT: Select or delete programs and files; External data transfer
- PGM CALL: Define program call, select datum and point tables
- MOD: MOD Functions
- HELP: Show help texts and illustrations
- ERR: Display all current error messages
- CALC: Pocket calculator

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

- Move highlight
- GOTO: Go directly to blocks, cycles and parameter functions. Open the screen keyboard or a drop-down menu

Override control knobs for feed rate/spindle speed



Programming path movements

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

Tool functions

- TOOL DEF, TOOL CALL: Enter and call tool length and radius

Cycles, subprograms and program section repeats

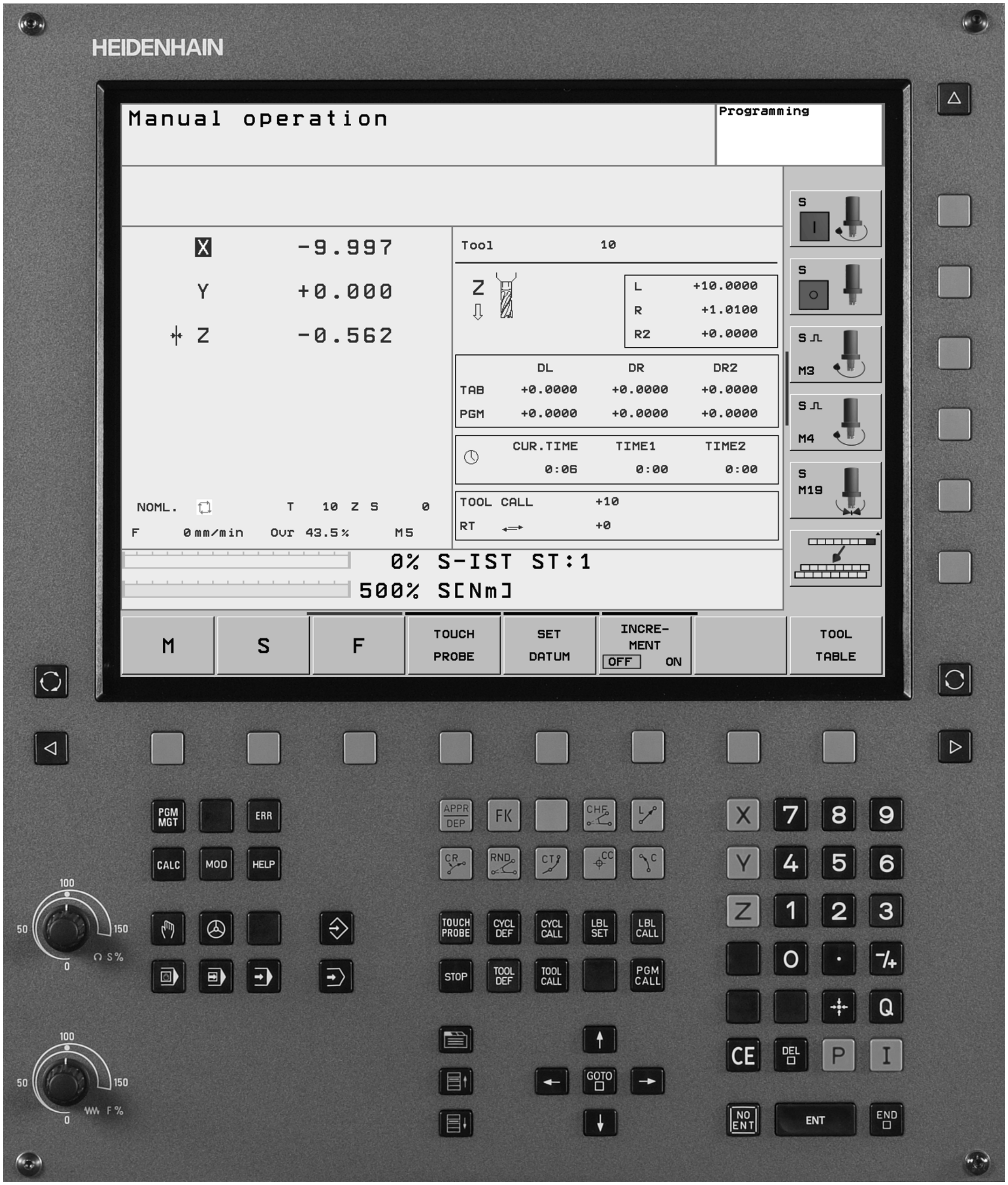
- CYCL DEF, CYCL CALL: Define and call cycles
- LBL SET, LBL CALL: Enter and call labels for subprogramming and program section repeats
- STOP: Program stop in a program
- TOUCH PROBE: Define touch probe cycles

Coordinate axes and numbers: Entering and editing

- X ... V: Select coordinate axes or enter them into the program
- 0 ... 9: Numbers
- . -/+: Decimal point / Reverse algebraic sign
- P I: Polar coordinate input/ Incremental dimensions
- Q: Q parameter programming/Q parameter status
- Assume actual position or values from calculator
- NO ENT: Skip dialog questions, delete words
- ENT: Confirm entry and resume dialog
- END: Conclude block, exit entry
- CE: Clear numerical entry or clear TNC error message
- DEL: Abort dialog, delete program section

Navigation in dialogs

- No function at present
- Up/down one dialog box or button


HEIDENHAIN
Manual operation
Programming
X -9.997
Y +0.000
Z -0.562
Tool 10
L +10.0000
R +1.0100
R2 +0.0000
DL DR DR2
TAB +0.0000 +0.0000 +0.0000
PGM +0.0000 +0.0000 +0.0000
CUR.TIME TIME1 TIME2
0:06 0:00 0:00
TOOL CALL +10
RT +0
NOML.
T 10 Z S 0
F 0mm/min Ovr 43.5% M5
0% S-IST ST:1
500% S[Nm]
M
S
F
TOUCH PROBE
SET DATUM
INCRE-MENT
OFF ON
TOOL TABLE
M3
M4
M19
PGM MGT
ERR
CALC
MOD
HELP
APPR DEP
FK
CHF
CR
RND
CT
CC
TOUCH PROBE
CYCL DEF
CYCL CALL
LBL SET
LBL CALL
STOP
TOOL DEF
TOOL CALL
PGM CALL
GOTO
X
Y
Z
7 8 9
4 5 6
1 2 3
0
Q
CE
DEL
P
I
NO ENT
ENT
END
100
50
150
0
S %
F %

TNC Model, Software and Features

This manual describes functions and features provided by TNCs as of the following NC software numbers.

TNC model	NC software number
TNC 320	340 551-xx

The machine tool builder adapts the usable features of the TNC to his machine by setting machine parameters. Some of the functions described in this manual may not be among the features provided by your machine tool.

TNC functions that may not be available on your machine include:

- Probing function for the 3-D touch probe
- Rigid tapping
- Returning to the contour after an interruption

In addition, the TNC 320 also has software options that can be enabled by your machine tool builder.

Software option
1st additional axis for 4 axes and open-loop spindle
2nd additional axis for 5 axes and open-loop spindle

Please contact your machine tool builder to become familiar with the features of your machine.

Many machine manufacturers, as well as HEIDENHAIN, offer programming courses for the TNCs. We recommend these courses as an effective way of improving your programming skill and sharing information and ideas with other TNC users.

Location of use

The TNC complies with the limits for a Class A device in accordance with the specifications in EN 55022, and is intended for use primarily in industrially-zoned areas.

Contents

2 Manual Operation and Setup 39

5 Programming: Tools 95

1

Introduction

1.1 The TNC 320

HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional machining operations right at the machine in an easy-to-use conversational programming language. The TNC 320 is designed for milling and drilling machine tools with up to 4 axes (optionally 5 axes). Instead of the fourth or fifth axis, you can also change the angular position of the spindle under program control.

Keyboard and screen layout are clearly arranged in such a way that the functions are fast and easy to use.

Programming: HEIDENHAIN conversational format

HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the HEIDENHAIN FK free contour programming does the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining.

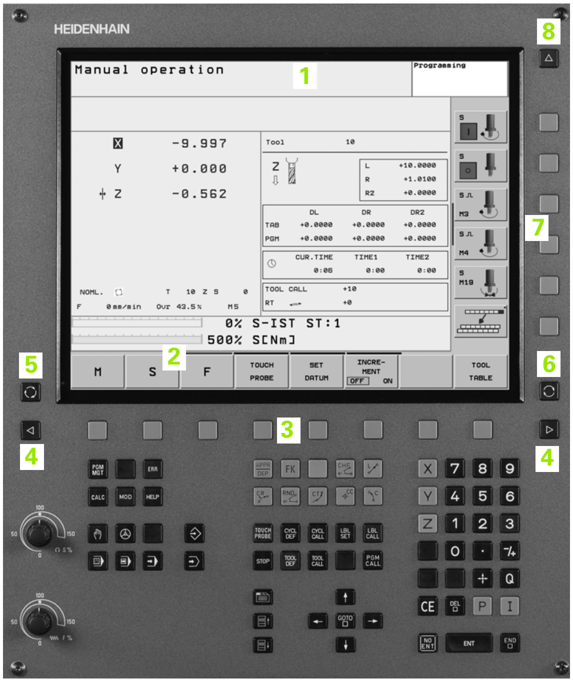
You can also enter and test one program while the control is running another.

Compatibility

The feature content of the TNC 320 is not the same as that of the TNC 4xx series and iTNC 530 controls. Part programs created on the HEIDENHAIN controls TNC 150 B and later can only run on the TNC 320 under a condition. If NC blocks contain invalid elements, the TNC will mark them during download as ERROR blocks.

1.2 Visual Display Unit and Operating Panel

Visual display unit



The TNC is delivered with a 15-inch TFT color flat-panel display (see figure at top right).

1 Header

When the TNC is on, the selected operating modes are shown in the screen header: the machining mode at the left and the programming mode at right. The currently active mode is displayed in the larger box, where the dialog prompts and TNC messages also appear (unless the TNC is showing only graphics).

2 Soft keys

In the footer the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them. The lines immediately above the soft-key row indicate the number of soft-key rows that can be called with the black arrow keys to the right and left. The line representing the active soft-key row is highlighted.

3 Soft-key selection keys

4 Switches the soft-key rows

5 Sets the screen layout

6 Shift key for switchover between machining and programming modes

7 Soft-key selection keys for machine tool builders

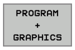
8 Switches soft-key rows for machine tool builders

Screen layout

You select the screen layout yourself: In the programming mode of operation, for example, you can have the TNC show program blocks in the left window while the right window displays programming graphics. You could also display status information in the right window instead of the graphics, or display only program blocks in one large window. The available screen windows depend on the selected operating mode.

To change the screen layout:

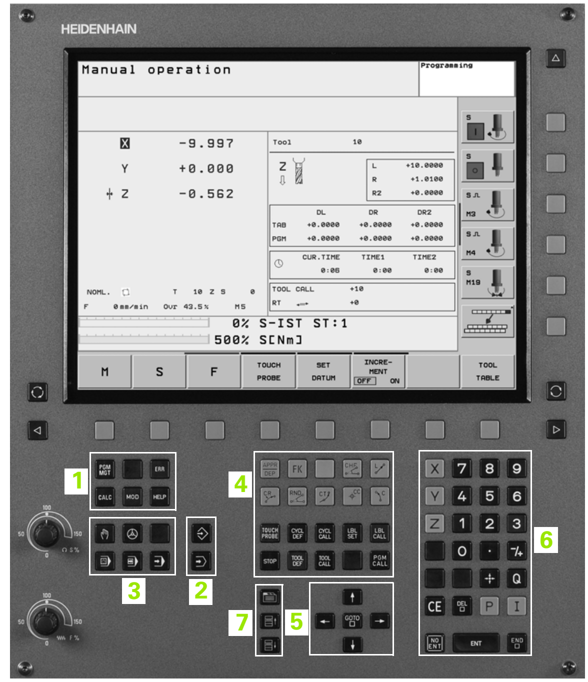
Press the SPLIT SCREEN key: The soft-key row shows the available layout options (see “Modes of Operation”, page 31).



Select the desired screen layout.

Operating panel

The TNC 320 is delivered with an integrated keyboard. The figure at right shows the controls and displays of the keyboard:



1
- File management
- Pocket calculator
- MOD function
- HELP function

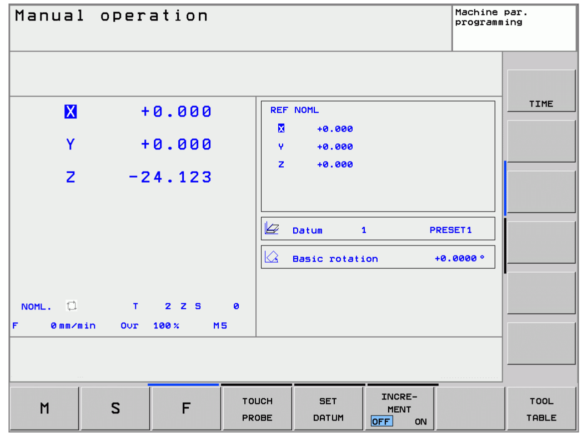
2 Programming modes
3 Machine operating modes
4 Initiation of programming dialog
5 Arrow keys and GOTO jump command
6 Numerical input and axis selection
7 Navigation keys

The functions of the individual keys are described on the inside front cover.

Machine panel buttons, e.g. NC START or NC STOP, are described in the manual for your machine tool.

1.3 Modes of Operation

Manual operation and electronic handwheel



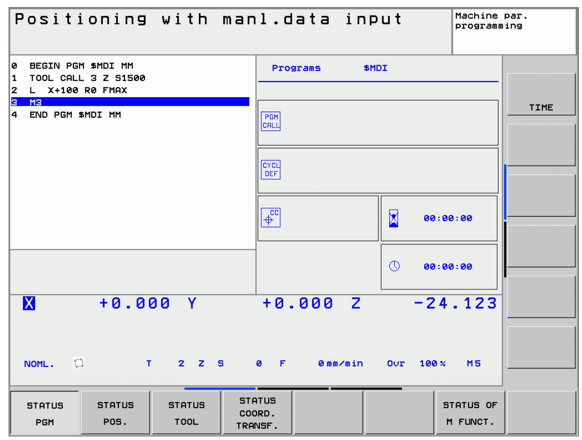
The Manual Operation mode is required for setting up the machine tool. In this operating mode, you can position the machine axes manually or by increments and set the datums.

The Electronic Handwheel mode of operation allows you to move the machine axes manually with the HR electronic handwheel.

Soft keys for selecting the screen layout (select as described previously)

Screen windows	Soft key
Positions	POSITION
Left: positions—Right: status display	POSITION + STATUS

Positioning with Manual Data Input (MDI)

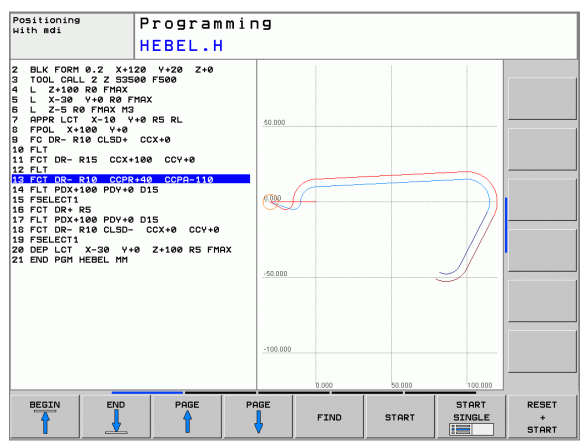


This mode of operation is used for programming simple traversing movements, such as for face milling or pre-positioning.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program blocks—Right: status display	PROGRAM + STATUS

Programming and editing



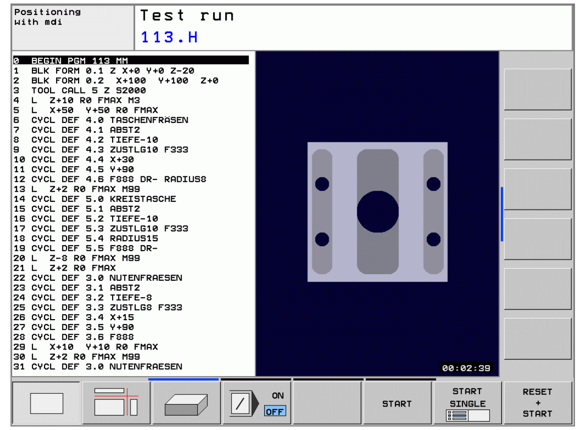
In this mode of operation you can write your part programs. The FK free programming feature, the various cycles and the Q parameter functions help you with programming and add necessary information. If desired, you can have the programming graphics show the individual steps.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program, right: programming graphics	PROGRAM + GRAPHICS

Test Run

In the Test Run mode of operation, the TNC checks programs and program sections for errors, such as geometrical incompatibilities, missing or incorrect data within the program or violations of the work space. This simulation is supported graphically in different display modes.

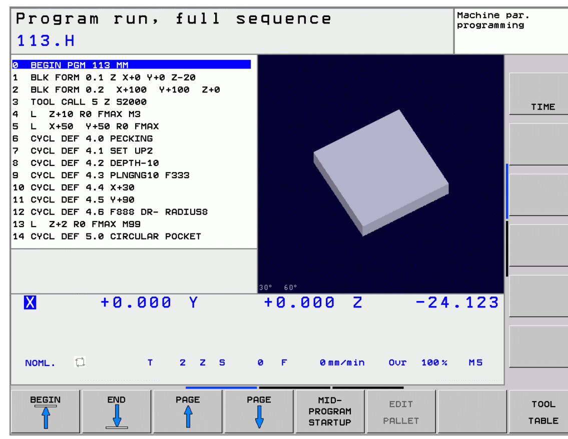


Soft keys for selecting the screen layout: see "Program Run, Full Sequence and Program Run, Single Block", page 32.

Program Run, Full Sequence and Program Run, Single Block

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or to a manual or programmed stop. You can resume program run after an interruption.

In the Program Run, Single Block mode of operation you execute each block separately by pressing the machine START button.



Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PGM
Left: program, right: status	PROGRAM + STATUS
Left: program, right: graphics	PROGRAM + GRAPHICS
Graphics	GRAPHICS

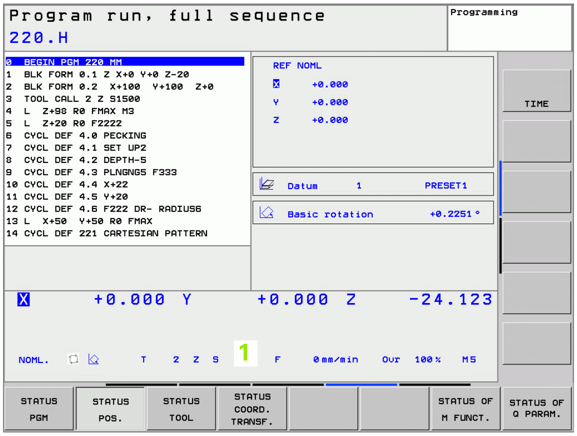
1.4 Status Displays

"General" status display

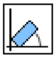
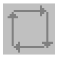
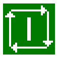
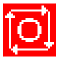
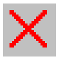
The status display 1 informs you of the current state of the machine tool. It is displayed automatically in the following modes of operation:

- Program Run, Single Block and Program Run, Full Sequence, except if the screen layout is set to display graphics only, and
- Positioning with Manual Data Input (MDI).

In the Manual mode and Electronic Handwheel mode the status display appears in the large window.



Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position.
X Y Z	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information.
T	Tool number T.
F S M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions.
	Axis locked.
Ovr	Override setting in percent
	Axis can be moved with the handwheel.
	Axes are moving under a basic rotation.
	No active program.
	Program run started.
	Program run stopped.
	Program run is being aborted.

Additional status displays

The 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.

To switch on the additional status display:

	Call the soft-key row for screen layout.
PROGRAM + STATUS	Select the layout option for the additional status display.

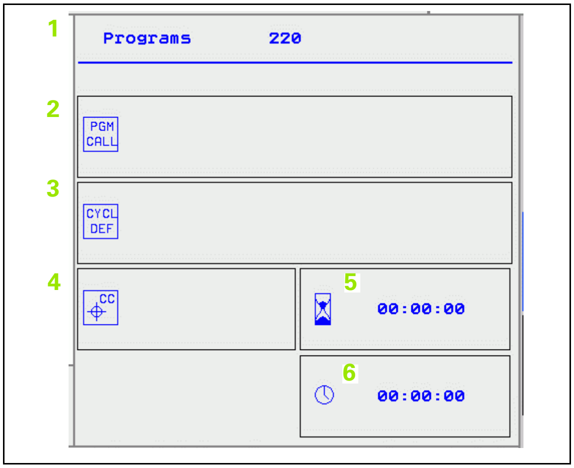
To select an additional status display:

	Shift the soft-key rows until the STATUS soft keys appear.
STATUS PGM	Select the desired additional status display, e.g. general program information.

You can choose between several additional status displays with the following soft keys:

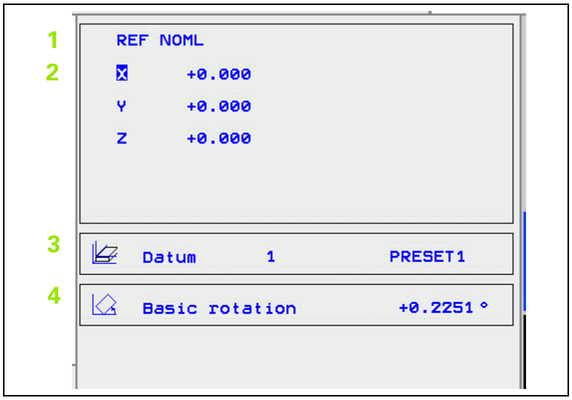
General program information

Soft key	Assignment	Meaning
STATUS PGM	1	Name of the active main program
	2	Active programs
	3	Active machining cycle
	4	Circle center CC (pole)
	5	Machining time
	6	Dwell time counter



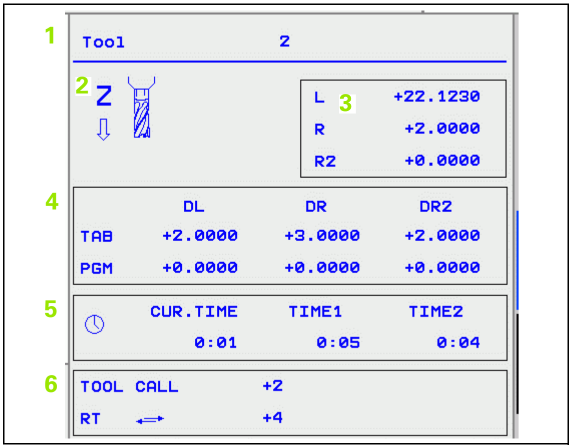
Positions and coordinates

Soft key	Assignment	Meaning
STATUS POS.	1	Type of position display, e.g. actual position
	2	Position display
	3	Number of the active datum from the preset table (function not available on TNC 320).
	4	Angle of a basic rotation



Information on tools

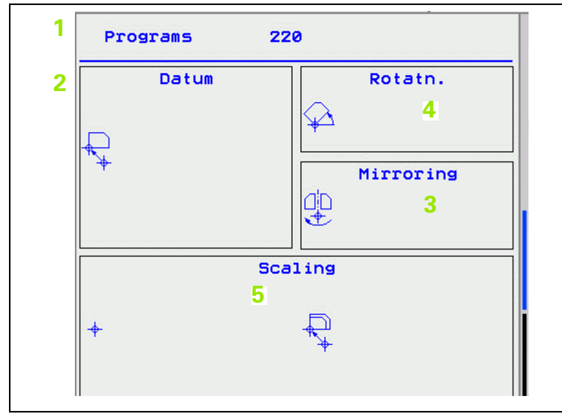
Soft key	Assignment	Meaning
STATUS TOOL	1	■ T: Tool number and name
	2	Tool axis
	3	Tool lengths and radii
	4	Oversizes (delta values) from TOOL CALL (PGM) and the tool table (TAB)
	5	Tool life, maximum tool life (TIME 1) and maximum tool life for TOOL CALL (TIME 2)
	6	Display of the active tool and the (next) replacement tool



Coordinate transformations

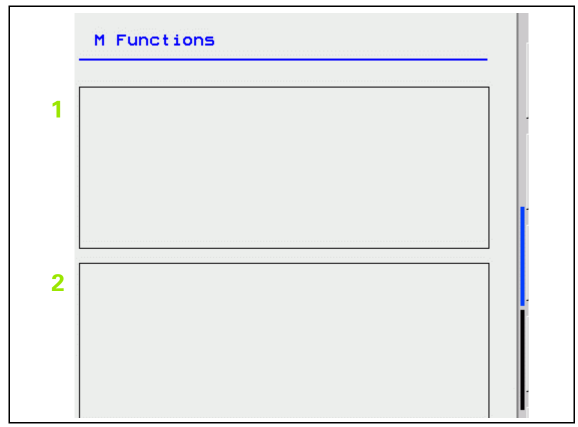
Soft key	Assignment	Meaning
STATUS COORD. TRANSF.	1	Program name
	2	Active datum shift (Cycle 7)
	3	Mirrored axes (Cycle 8)
	4	Active rotation angle (Cycle 10)
	5	Active scaling factor(s) (Cycles 11 / 26)

See “Coordinate Transformation Cycles” on page 281.



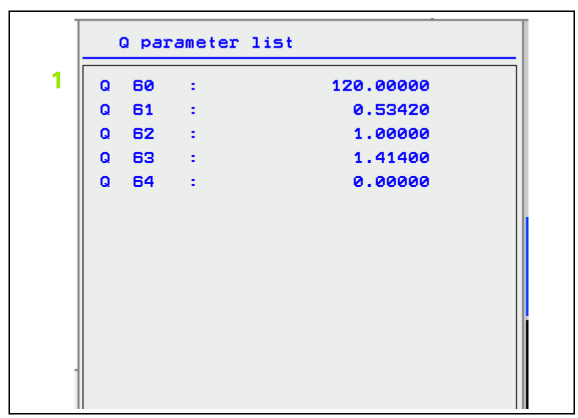
Active miscellaneous functions M

Soft key	Assignment	Meaning
STATUS OF M FUNCT.	1	List of the active M functions with fixed meaning
	2	List of the active M functions that are adapted by your machine manufacturer



Status of Q parameters

Soft key	Assignment	Meaning
STATUS OF Q PARAM.	1	List of Q parameters defined with the Q PARAM LIST soft key



1.5 Accessories: HEIDENHAIN 3-D Touch Probes and Electronic Handwheels

3-D touch probes

With the various HEIDENHAIN 3-D touch probe systems you can:

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run

TS 220, TS 440 and TS 640 touch trigger probes

These touch probes are particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable and may be a more economical alternative.

The TS 440 and TS 640 (see figures at right) feature infrared transmission of the triggering signal to the TNC. This makes them highly convenient for use on machines with automatic tool changers.

Principle of operation: HEIDENHAIN triggering touch probes feature a wear resisting optical switch that generates an electrical signal as soon as the stylus is deflected. This signal is transmitted to the TNC, which stores the current position of the stylus as an actual value.

HR electronic handwheels

Electronic handwheels facilitate moving the axis slides precisely by hand. A wide range of traverses per handwheel revolution is available. Apart from the HR 130 and HR 150 integral handwheels, HEIDENHAIN also offers the HR 410 portable handwheel.

2

Manual Operation and Setup

2.1 Switch-On, Switch-Off

Switch-on

Switch-on and traversing the reference points can vary depending on the machine tool. Refer to your machine manual.

Switch on the power supply for control and machine. The TNC then displays the following dialog:

SYSTEM STARTUP

TNC is started

POWER INTERRUPTED

CE — TNC message that the power was interrupted—clear the message.

CONVERT PLC PROGRAM

The PLC program of the TNC is automatically compiled.

RELAY EXT. DC VOLTAGE MISSING

I — Switch on external dc voltage. The TNC checks the functioning of the EMERGENCY STOP circuit.

MANUAL OPERATION
TRAVERSE REFERENCE POINTS

I — Cross the reference points manually in the displayed sequence: For each axis press the machine START button, or

X Y — Cross the reference points in any sequence: Press and hold the machine axis direction button for each axis until the reference point has been traversed.

If your machine is equipped with absolute encoders, you can leave out traversing the reference mark. In such a case, the TNC is ready for operation immediately after the machine control voltage is switched on.

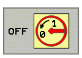
The TNC is now ready for operation in the Manual Operation mode.

> The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the Programming and Editing or Test Run modes of operation immediately after switching on the control voltage.
>
> You can traverse the reference points later by pressing the PASS OVER REFERENCE soft key in the Manual Operation mode.

Switch-off

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

- Select the Manual Operation mode.



- Select the function for shutting down, confirm again with the YES soft key.
- When the TNC displays the message **NOW IT IS SAFE TO TURN POWER OFF** in a superimposed window, you may cut off the power supply to the TNC.

> Inappropriate switch-off of the TNC can lead to data loss.

2.2 Moving the Machine Axes

Note

Traversing with the machine axis direction buttons can vary depending on the machine tool. The machine tool manual provides further information.

To traverse with the machine axis direction buttons:

Select the Manual Operation mode.

X

Press the machine axis direction button and hold it as long as you wish the axis to move, or

X

and

I

Move the axis continuously: Press and hold the machine axis direction button, then press the machine START button.

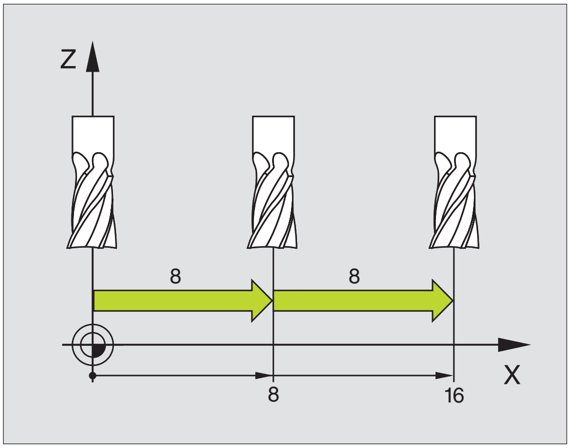
0

To stop the axis, press the machine STOP button.

You can move several axes at a time with these two methods. You can change the feed rate at which the axes are traversed with the F soft key (see “Spindle Speed S, Feed Rate F and Miscellaneous Functions M”, page 45).

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance.



Select the Manual Operation or Electronic Handwheel mode.

INCRE-MENT OFF ON — Select incremental jog positioning: Switch the INCREMENT soft key to ON.

LINEAR AXES:

8 CONFIRM VALUE — Enter the jog increment in mm, e.g. 8 mm, and press the CONFIRM VALUE soft key.

OK — Finish the entry with the OK soft key.

X — Press the machine axis direction button as often as desired

To deactivate the function, press the **Switch off** soft key.

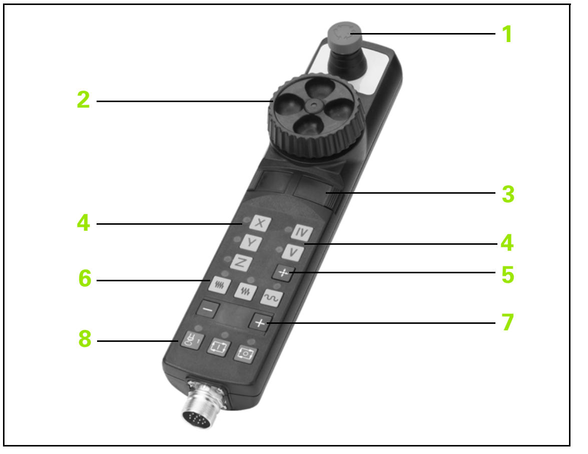
Traversing with the HR 410 electronic handwheel

The portable HR 410 handwheel is equipped with two permissive buttons. The permissive buttons are located below the star grip.

You can only move the machine axes when a permissive button is depressed (machine-dependent function).

The HR 410 handwheel features the following operating elements:

1 EMERGENCY OFF button
2 Handwheel
3 Permissive buttons
4 Axis address keys
5 Actual-position-capture key
6 Keys for defining the feed rate (slow, medium, fast; the feed rates are set by the machine tool builder)
7 Direction in which the TNC moves the selected axis
8 Machine function (set by the machine tool builder)



The red indicator lights show the axis and feed rate you have selected.

It is also possible to move the machine axes with the handwheel during a program run if **M118** is active.

Procedure:

Select the Electronic Handwheel operating mode.

Press and hold a permissive button.

Select the axis.

Select the feed rate.

Move the active axis in the positive or negative direction.

or

2.3 Spindle Speed S, Feed Rate F and Miscellaneous Functions M

Function

In the Manual Operation and Electronic Handwheel operating modes, you can enter the spindle speed S, feed rate F and the miscellaneous functions M with soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous Functions."

The machine tool builder determines which miscellaneous functions M are available on your control and what effects they have.

Entering values

Spindle speed S, miscellaneous function M

S	To enter the spindle speed, press the S soft key.

SPINDLE SPEED S =

1000 I	Enter the desired spindle speed and confirm your entry with the machine START button.

The spindle speed S with the entered rpm is started with a miscellaneous function M. Proceed in the same way to enter a miscellaneous function M.

Feed rate F

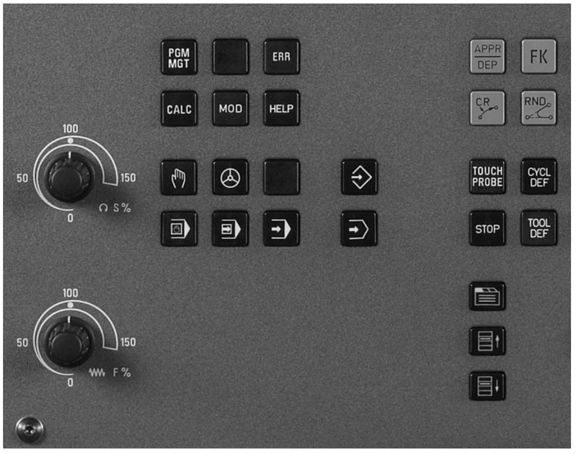
After entering a feed rate F, you must confirm your entry with the OK key instead of the machine START button.

The following is valid for feed rate F:

- If you enter F=0, then the lowest feed rate from the machine parameter **minFeed** is effective
- If the feed rate entered exceeds the value defined in the machine parameter maxFeed, then the parameter value is effective.
- F is not lost during a power interruption

Changing the spindle speed and feed rate

With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value. However, the range can be further limited by the machine parameters **minFeedOverride, maxFeedOverride, minSpindleOverride** and **maxSpindleOverride** (are set by the machine tool builder).



The override dial for spindle speed is only functional on machines with infinitely variable spindle drive.

The minimum and maximum spindle speeds entered as machine parameters are not fallen short of or exceeded, respectively.

If the MP **minSpindleOverride=0%,** then the setting spindle override=0 leads to a spindle stop.

2.4 Datum Setting (Without a 3-D Touch Probe)

Note

For datum setting with a 3-D touch probe, refer to the Touch Probe Cycles Manual.

You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

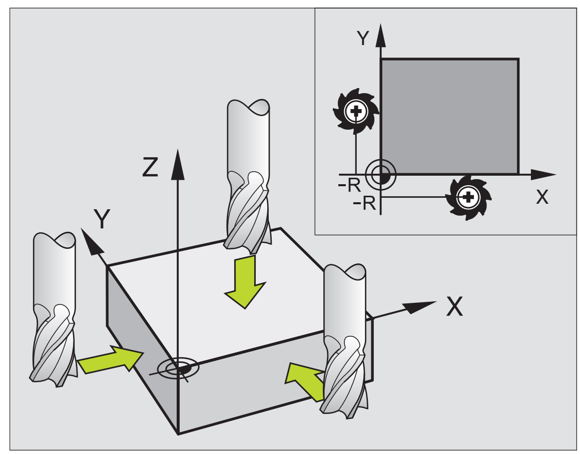
Preparation

- Clamp and align the workpiece.
- Insert the zero tool with known radius into the spindle.
- Ensure that the TNC is showing the actual position values.

Datum setting with axis keys

Fragile workpiece?

If the workpiece surface must not be scratched, you can lay a metal shim of known thickness d on it. Then enter a tool axis datum value that is larger than the desired datum by the value d.



Select the **Manual Operation** mode.

X Y Z

Move the tool slowly until it touches (scratches) the workpiece surface.

X

Select the axis.

DATUM SET Z=

0 ENT

Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness d of the shim. In the tool axis, offset the tool radius.

Repeat the process for the remaining axes.

If you are using a preset tool, set the display of the tool axis to the length L of the tool or enter the sum Z=L+d.

3

Positioning with Manual Data Input (MDI)

3.1 Programming and Executing Simple Machining Operations

The Positioning with Manual Data Input mode of operation is particularly convenient for simple machining operations or pre-positioning of the tool. You can write the a short program in HEIDENHAIN conversational programming and execute it immediately. You can also call TNC cycles. The program is stored in the file $MDI. In the Positioning with MDI operating mode, the additional status displays can also be activated.

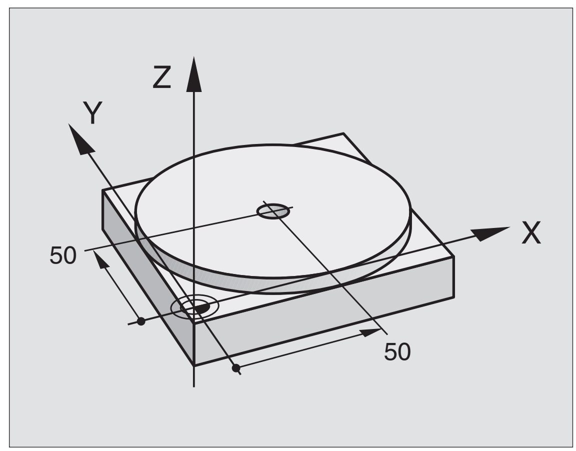
Positioning with Manual Data Input (MDI)

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

To start program run, press the machine START key.

Limitation

FK free contour programming, programming graphics and program run graphics, subprograms, program section repeats, and path compensation cannot be used. The $MDI file must not contain a program call **(PGM CALL).**



Example 1

A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool in L blocks (straight-line blocks) to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle 1 **PECKING.**

`0 BEGIN PGM $MDI MM`	
`1 TOOL DEF 1 L+0 R+5`	Define tool: zero tool, radius 5
`2 TOOL CALL 1 Z S2000`	Call tool: tool axis Z
	Spindle speed 2000 rpm
`3 L Z+200 R0 FMAX`	Retract tool (F MAX = rapid traverse)
`4 L X+50 Y+50 R0 FMAX M3`	Move the tool at F MAX to a position above the hole,
	Spindle on
`6 CYCL DEF 200 DRILLING`	Define DRILLING cycle
`Q200=5 ;SET-UP CLEARANCE`	Set-up clearance of the tool above the hole
`Q201=-15 ;DEPTH`	Total hole depth (algebraic sign=working direction)

Q206=250 ;FEED RATE FOR PLNGNG	Feed rate for pecking
Q202=5 ;PLUNGING DEPTH	Depth of each infeed before retraction
Q210=0 ;DWELL TIME AT TOP	Dwell time after every retraction in seconds
Q203=-10 ;SURFACE COORDINATE	Coordinate of the workpiece surface
Q204=20 ;2ND SET-UP CLEARANCE	Set-up clearance of the tool above the hole
Q211=0.2 ;DWELL TIME AT DEPTH	Dwell time in seconds at the hole bottom
7 CYCL CALL	Call DRILLING cycle
8 L Z+200 R0 FMAX M2	Retract the tool
9 END PGM $MDI MM	End of program

Straight line function L, (see “Straight Line L” on page 125) DRILLING cycle.(see “DRILLING (Cycle 200)” on page 182)

Example 2: Correcting workpiece misalignment on machines with rotary tables

Use the 3-D touch probe to rotate the coordinate system. See “Touch Probe Cycles in the Manual and Electronic Handwheel Operating Modes,” section “Compensating workpiece misalignment,” in the Touch Probe Cycles User’s Manual.

Write down the rotation angle and cancel the Basic Rotation.

Select operating mode: Positioning with MDI.

IV

Select the axis of the rotary table, enter the rotation angle you wrote down previously and set the feed rate. For example: **L C+2.561 F50**

END

Conclude entry.

I

Press the machine START button: The rotation of the table corrects the misalignment.

Protecting and erasing 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.

PGM MGT

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

Move the highlight to the $MDI file.

COPY ABC → XYZ

To select the file copying function, press the COPY soft key.

TARGET FILE =

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

EXECUTE

Copy the file.

END

To close the file manager, press the END soft key.

Erasing the contents of the $MDI file is done in a similar way: Instead of copying the contents, however, you erase them with the DELETE soft key. The next time you select the operating mode Positioning with MDI, the TNC will display an empty $MDI file.

If you wish to delete $MDI, then

- you must not have selected the Positioning with MDI mode (not even in the background).
- you must not have selected the $MDI file in the Programming and Editing mode.
- you must cancel the editing protection of the $MDI file

For further information, see "Copying a single file," page 66.

4

Programming: Fundamentals of NC, File Management, Programming Aids

4.1 Fundamentals

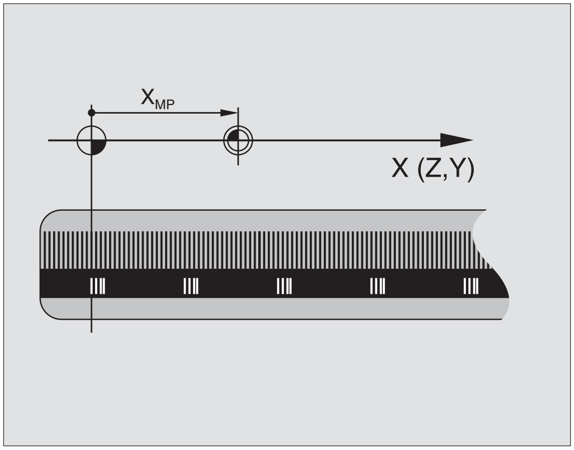
Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. Linear axes are usually equipped with linear encoders, rotary tables and tilting axes with angle encoders.

When a machine axis moves, the corresponding position encoder generates an electrical signal. The TNC evaluates this signal and calculates the precise actual position of the machine axis.

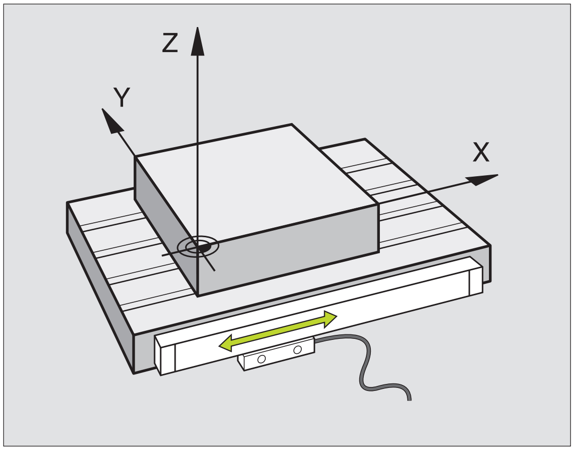
If there is a power interruption, the calculated position will no longer correspond to the actual position of the machine slide. To recover this association, incremental position encoders are provided with reference marks. The scales of the position encoders contain one or more reference marks that transmit a signal to the TNC when they are crossed over. From the signal the TNC can re-establish the assignment of displayed positions to machine positions. For linear encoders with distance-coded reference marks the machine axes need to move by no more than 20 mm, for angle encoders by no more than 20°.

With absolute encoders, an absolute position value is transmitted to the control immediately upon switch-on. In this way the assignment of the actual position to the machine slide position is re-established directly after switch-on.



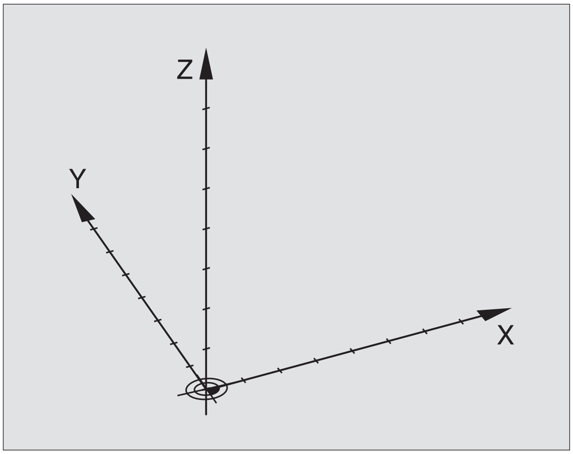
Reference system

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 the three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies 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.

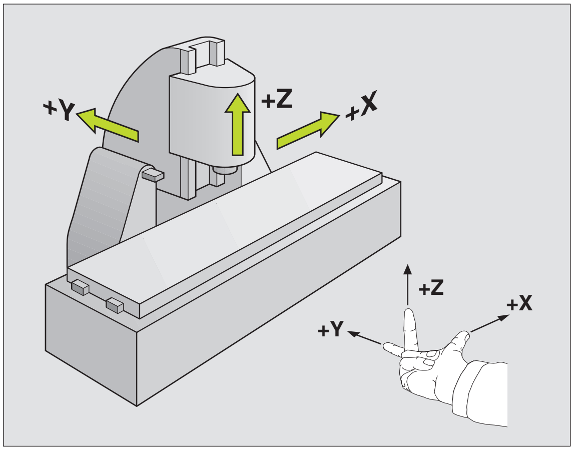
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (reference point) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

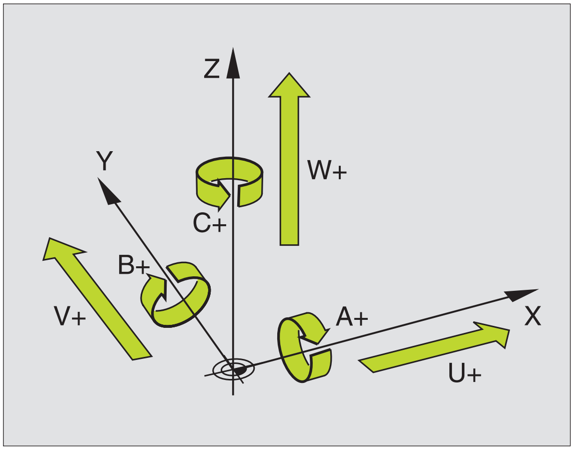


Reference system on milling machines

When using a milling machine, you orient tool movements to the Cartesian coordinate system. The illustration at right shows how the Cartesian coordinate system describes the machine axes. The "right-hand rule" is illustrated for remembering the three axis directions: the middle finger points in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb points in the positive X direction, and the index finger in the positive Y direction.

The TNC 320 can control up to 4 axes (optionally 5). The axes U, V and W (which are not presently supported by the TNC 320) are secondary linear axes parallel to the main axes X, Y and Z, respectively. Rotary axes are designated as A, B and C. The illustration at lower right shows the assignment of secondary axes and rotary axes to the main axes.





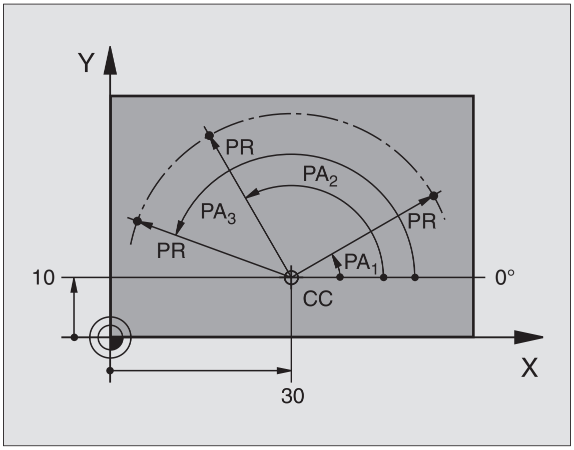
Polar coordinates

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

While the Cartesian coordinates X, Y and Z 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 at a circle center (CC), or pole. A position in a plane can be clearly defined by the:

- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the size of the angle between the reference axis and the line that connects the circle center CC with the position.

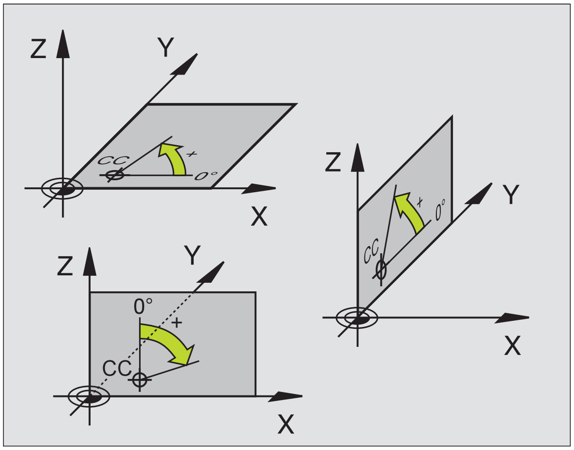
See figure at upper right.



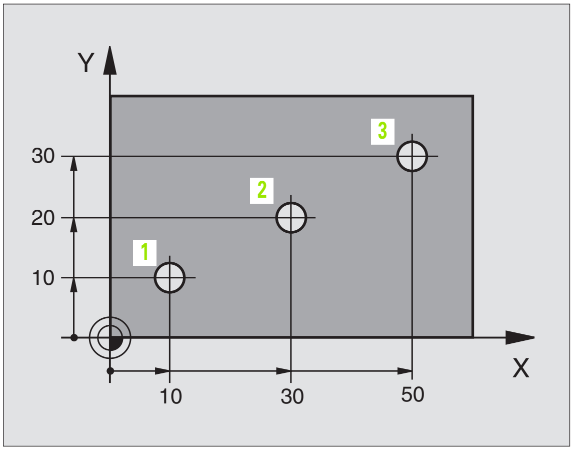
Setting the pole and the angle reference axis

The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the 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



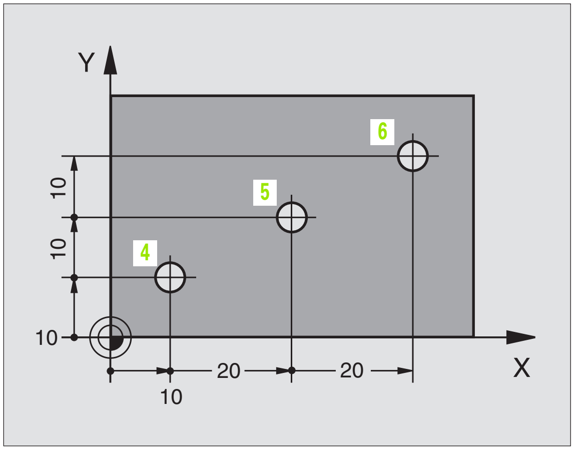
Absolute workpiece positions

Absolute coordinates are position coordinates that are referenced to the datum of the coordinate system (origin). Each position on the workpiece is uniquely 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 you write a part program in incremental coordinates, you thus program the tool to move by the distance between the previous and the subsequent nominal positions. Incremental coordinates are therefore also referred to as chain dimensions.

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

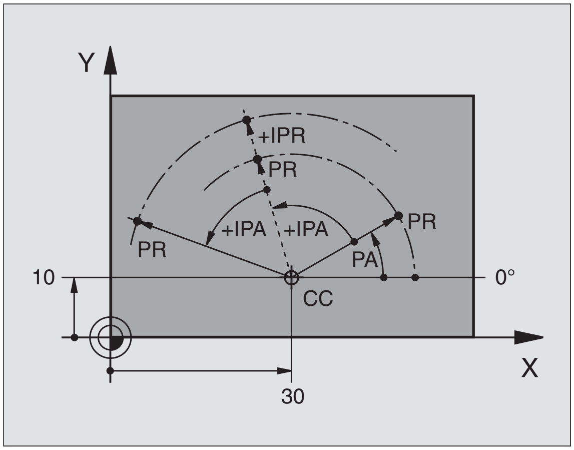
Example 2: Holes dimensioned in incremental coordinates

Absolute coordinates of hole **4**

X = 10 mm
Y = 10 mm

Hole **5**, relative to **4**	Hole **6**, relative to **5**
X = 20 mm	X = 20 mm
Y = 10 mm	Y = 10 mm

Absolute and incremental polar coordinates

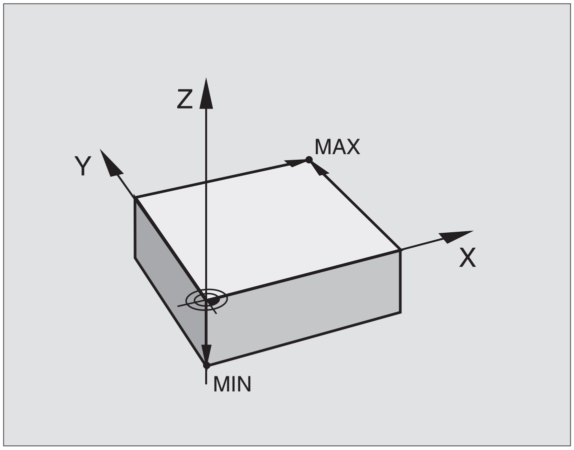


Absolute polar coordinates always refer to the pole and the reference axis.

Incremental polar coordinates always refer to the last programmed nominal position of the tool.

Setting the datum

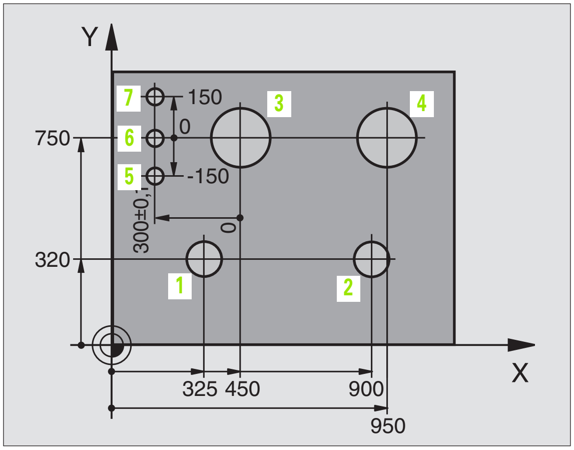
A production drawing identifies a certain form element of the workpiece, usually a corner, as the absolute datum. Before setting the datum, you align the workpiece with the machine axes and move the tool in each axis to a known position relative to the workpiece. You then set the TNC display to either zero or a predetermined position value. This establishes the reference system for the workpiece, which will be used for the TNC display and your part program.



If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles (see "Coordinate Transformation Cycles" on page 281).

If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece which is suitable for deducing the dimensions of the remaining workpiece positions.

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. See "Setting the Datum with a 3-D Touch Probe" in the Touch Probe Cycles User's Manual.



Example

The workpiece drawing at right shows holes (**1** to **4**) whose dimensions are shown with respect to an absolute datum with the coordinates X=0, Y=0. The holes (**5** to **7**) are dimensioned with respect to a relative datum with the absolute coordinates X=450, Y=750. With the **DATUM SHIFT** cycle you can temporarily set the datum to the position X=450, Y=750, to be able to program the holes (**5** to **7**) without further calculations.

4.2 File Management: Fundamentals

Files

Files in the TNC	Type
Programs	
In HEIDENHAIN format	.H
In ISO format	.I
Tables for	
Tools	.T
Tool changers	.TCH
Datums	.D

When you write a part program on the TNC, you must first enter a file name. The TNC saves the program as a file with the same name. The TNC can also save texts and tables as files.

The TNC provides a special file management window in which you can easily find and manage your files. Here you can call, copy, rename and erase files.

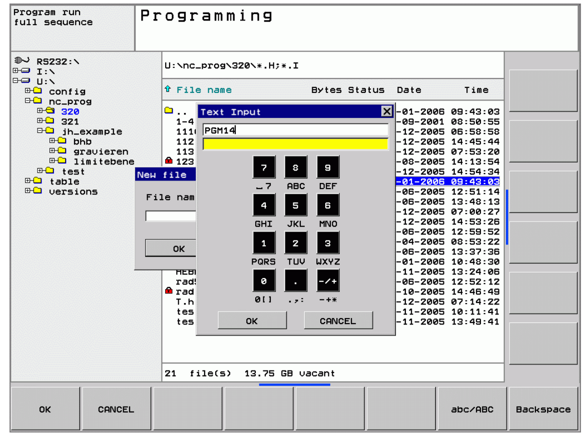
With the TNC you can manage and save files up to a total size of 10 MB.

File names

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

PROG20	.H
File name	File type

Screen keypad



You can enter letters and special characters with the screen keypad or (if available) with a PC keyboard connected over the USB port.

Enter the text with the screen keypad

- Press the GOTO key if you want to enter a text, for example a program name or directory name, using the screen keypad
- The TNC opens a window in which the numeric entry field **1** of the TNC is displayed with the corresponding letters assigned
- You can move the cursor to the desired character by repeatedly pressing the respective key
- Wait until the selected character is transferred to the entry field before you enter the next character
- Use the OK soft key to load the text into the open dialog field

Use the **abc/ABC** soft key to select upper or lower case. If your machine tool builder has defined additional special characters, you can call them with the **SPECIAL CHARACTER** soft key and insert them. To delete individual characters, use the **Backspace** soft key.

Data backup

We recommend saving newly written programs and files on a PC at regular intervals.

HEIDENHAIN provides a backup function for this purpose in the data transfer software TNCremoNT. Your machine tool builder can provide you with a copy of TNCBACK.EXE.

You additionally need a data medium on which all machine-specific data, such as the PLC program, machine parameters, etc., are stored. Please contact your machine tool builder for more information on both the backup program and the floppy disk.

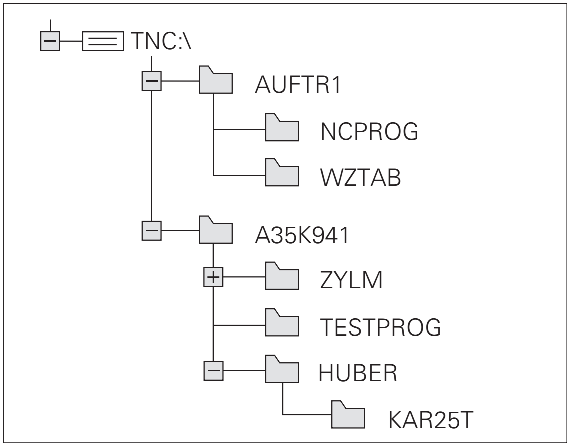
4.3 Working with the File Manager

Directories

If you save many programs in the TNC, we recommend that you save your files in directories (folders) so that you can easily find your data. You can divide a directory into further directories, which are called subdirectories. With the –/+ key or ENT you can show or hide the subdirectories.

Paths

A path indicates the drive and all directories and subdirectories under which a file is saved. The individual names are separated by a backslash "\".



Example

On drive **`TNC:\`** the subdirectory AUFTR1 was created. Then, in the directory **`AUFTR1`** the directory 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`

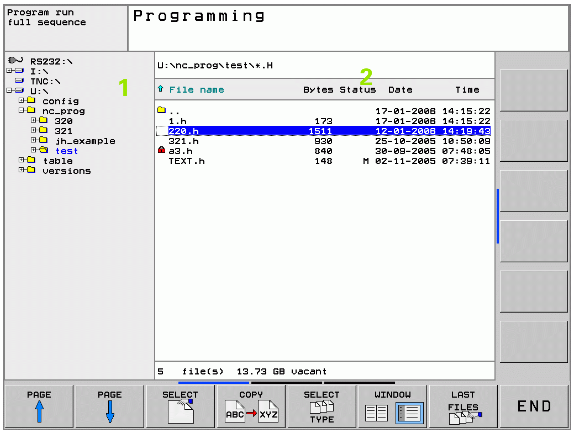
The chart at right illustrates an example of a directory display with different paths.

Overview: Functions of the file manager

Function	Soft key
Copy (and convert) individual files	COPY ABC → XYZ
Display a specific file type	SELECT TYPE
Display the last 10 files that were selected	LAST FILES
Erase a file or directory	DELETE
Mark a file	TAG
Rename a file	RENAME ABC = XYZ
Protect a file against editing and erasure	PROTECT
Cancel file protection	UNPROTECT
Manage network drives	NET
Copy a directory	COPY DIR
Display all the directories of a particular drive	UPDATE TREE
Delete directory with all its subdirectories	DELETE ALL
Sort files by properties	SORT
Create new file	NEW FILE
Select the editor	SELECT EDITOR

Calling the file manager

PGM MGT	Press the PGM MGT key: the TNC displays the file management window (Figure at upper right shows the factory default setting.) If the TNC displays a different screen layout, press the WINDOW soft key.)



The narrow window on the left **1** shows the available drives and directories. Drives designate devices with which data are stored or transferred. One drive is the internal memory of the TNC. Other drives are the RS232, RS422, Ethernet and USB interfaces, which you can used, for example, to connect a personal computer or other storage device. A directory is always identified by a folder symbol to the left and the directory name to the right. The control displays a subdirectory to the right of and below its parent directory. A box with the + symbol in front of the folder symbol indicates that there are further subdirectories, which can be shown with the –/+ key or ENT.

The wide window on the right **2** shows you all files that are stored in the selected directory. Each file is shown with additional information, illustrated in the table below.

Display	Meaning
FILE NAME	Name with an extension, separated by a dot (file type)
BYTE	File size in bytes
STATUS	File properties:
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 a Program Run mode of operation.
	File is protected against editing and erasure.
DATE	Date the file was last changed
TIME	Time the file was last changed

Selecting drives, directories and files

PGM MGT — Call the file manager.

With the arrow keys or the soft keys, you can move the highlight to the desired position on the screen:

← → — Moves the highlight from the left to the right window, and vice versa.

↓ ↑ — Moves the highlight up and down within a window.

PAGE ↑ PAGE ↓ — Moves the highlight one page up or down within a window.

Step 1: Select drive

Move the highlight to the desired drive in the left window:

SELECT or ENT — Select a drive: Press the SELECT soft key or the ENT key.

Step 2: Select a directory

Move the highlight to the desired directory in the left-hand window—the right-hand window automatically shows all files stored in the highlighted directory.

Step 3: Select a file

SELECT TYPE — Press the SELECT TYPE soft key.

SELECT .H — Press the soft key for the desired file type, or

SHOW ALL — Press the SHOW ALL soft key to display all files, or

Move the highlight to the desired file in the right window

SELECT or ENT — The selected file is opened 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

Move the highlight in the left window to the directory in which you want to create a subdirectory.

NEW ENT — Enter the new file name, and confirm with ENT.

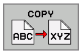
DIRECTORY NAME?

OK — Press the OK soft key to confirm, or

CANCEL — abort with the CANCEL soft key.

Copying a single file

- Move the highlight to the file you wish to copy.



- Press the COPY soft key to select the copy function. The TNC opens a pop-up window

- Enter the name of the destination file and confirm your entry with the ENT key or OK soft key: The TNC copies the file to the active directory or to the corresponding destination directory. The original file is retained

Copying a directory

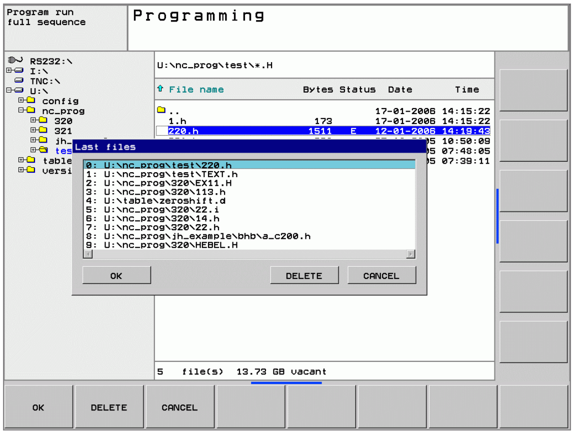
Move the highlight in the left window onto the directory you want to copy. Instead of the COPY soft key, press the COPY DIR soft key. Subdirectories can be copied by the TNC at the same time.

Making a setting in a selection box

In various dialogs, the TNC opens a pop-up window in which you can make settings in selection boxes.

- Move the cursor into the desired selection box and press the GOTO key
- Use the arrow keys to position the cursor to the required setting
- With the OK soft key you confirm the value, and with the CANCEL soft key you discard the selection

Choosing one of the last 10 files selected



PGM MGT — Call the file manager.

LAST FILES — 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 a window.

SELECT or ENT — Select a file: Press the OK soft key or ENT

Deleting a file

- Move the highlight to the file you want to delete

DELETE
- To select the erasing function, press the DELETE soft key
- To confirm, press the OK soft key
- To abort erasure, press the CANCEL soft key

Deleting a directory

- Delete all files and subdirectories stored in the directory that you want to delete
- Move the highlight to the directory you want to delete

DELETE
- To select delete function, press the DELETE ALL soft key. The TNC asks whether you really want to erase the subdirectories and files.
- To confirm, press the OK soft key
- To cancel deletion, press the CANCEL soft key

Marking files

Marking functions	Soft key
Mark a single file	TAG FILE
Mark all files in the directory	TAG ALL FILES
Unmark a single file	UNTAG FILE
Unmark all files	UNTAG ALL FILES

Some functions, such as copying or erasing files, can not only be used for individual files, but also for several files at once. To mark several files, proceed as follows:

Move the highlight to the first file.

TAG

To display the marking functions, press the TAG soft key.

TAG FILE

Mark a file by pressing the TAG FILE soft key.

Move the highlight to the next file you wish to mark:

TAG FILE

To mark more files, press the MARK FILE soft key.

To copy the marked files, with the back soft key, leave the TAG function

COPY ABC → XYZ

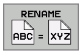
To copy the marked files, select the COPY soft key

DELETE

To delete the marked files, press the back soft key to exit the marking function and then press the DELETE soft key

Renaming a file

- Move the highlight to the file you wish to rename.



- Select the renaming function.
- Enter the new file name; the file type cannot be changed.
- To rename: Press the OK soft key or the ENT key

File sorting

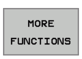
- Select the folder in which you wish to sort the files

- Select the SORT soft key
- Select the soft key with the corresponding display criterion

Additional functions

Protecting a file / Canceling file protection

- Move the highlight to the file you want to protect.



- To select the additional functions, press the MORE FUNCTIONS soft key.

- To enable file protection, press the PROTECT soft key. The file is distinguished by a symbol.
- To cancel file protection, proceed in the same way using the UNPROTECT soft key.

Select the editor

- Move the highlight in the right window onto the file you want to open.

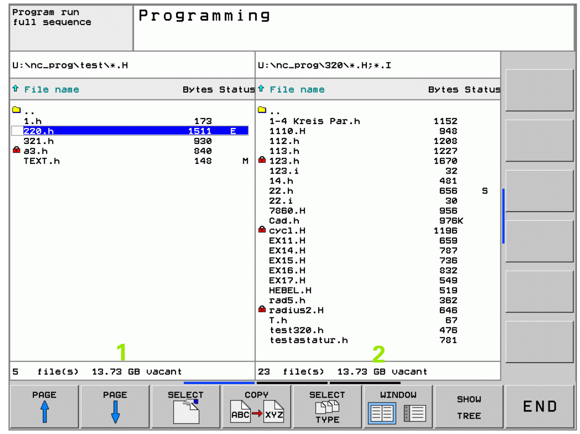
- To select the additional functions, press the MORE FUNCTIONS soft key.

- To select the editor with which to open the selected file, press the SELECT EDITOR soft key
- Mark the desired editor
- Press the OK soft key to open the file

Activate or deactivate USB devices

- To select the additional functions, press the MORE FUNCTIONS soft key.
- Shift the soft-key row.
- Select the soft key for activating or deactivating

Data transfer to or from an external data medium



> You might have to set up the data interface before you can transfer data to an external data medium (see “Setting the Data Interfaces” on page 404).

PGM MGT — Call the file manager.

WINDOW — Select the screen layout for data transfer: press the **WINDOW** soft key. Select the desired directory in both halves of the screen. In the left half of the screen, for example, **1** the TNC shows all files saved on its hard disk. In the right half of the screen **2** it shows all files saved on the external data medium. Use the **SHOW FILES** and **SHOW TREE** soft keys to switch between the folder view and file view.

Use the arrow keys to highlight the file(s) that you want to transfer:

↑ ↓ — Moves the highlight up and down within a window.

← → — Moves the highlight from the left to the right window, and vice versa.

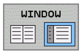
If you wish to copy from the TNC to the external data medium, move the highlight in the left window to the file to be transferred.

To transfer a single file, position the highlight on the desired file.

TAG — To transfer several files: Press the **TAG** soft key (in the second soft-key row, see “Marking files,” page 68) and mark the corresponding files. With the back soft key, exit the **TAG** function again.

Press the COPY soft key

Confirm with the OK soft key or with the ENT key. For long programs, a status window appears on the TNC informing you of the copying progress.



To end data transfer, move the highlight into the left window and then press the WINDOW soft key. The standard file manager window is displayed again.

To select another directory in the split-screen display, press the SHOW TREE soft key. If you press the SHOW FILES soft key, the TNC shows the content of the selected directory!

Copying files into another directory

- Select the screen layout with the two equally sized windows.
- To display directories in both windows, press the SHOW TREE soft key.

In the right window

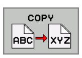
- Move the highlight to the directory to copy the files to and display the files in this directory with the SHOW FILES soft key.

In the left window

- Select the directory with the files to copy and press the SHOW FILES soft key to display them.

- Display the file marking functions.

- Move the highlight to the files to be copied and mark them. You can mark several files in this way, if desired.



- Copy the marked files into the target directory.

Additional marking functions: see “Marking files,” page 68.

If you have marked files in the left and right windows, the TNC copies from the directory in which the highlight is located.

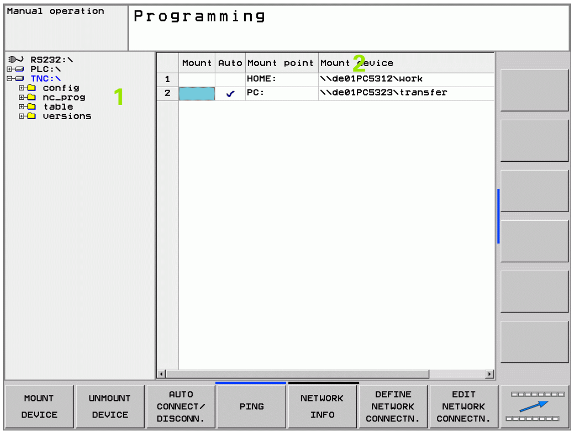
Overwriting files

If you copy files into a directory in which other files are stored under the same name, the TNC will reply with a “protected file” error message. Use the TAG function to overwrite the file anyway:

- To overwrite two or more files, mark them in the "existing files" pop-up window and press the OK soft key
- To leave the files as they are, press the CANCEL soft key

The TNC in a network

> To connect the Ethernet card to your network, see "Ethernet Interface," page 409.
>
> The TNC logs error messages during network operation (see "Ethernet Interface" on page 409).



If the TNC is connected to a network, the TNC displays the connected drives in the directory window **1** (see figure at right). All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been given the corresponding rights.

Connecting and disconnecting a network drive

PGM MGT

- To select the program management: Press the PGM MGT key. If necessary, press the WINDOW soft key to set up the screen as it is shown at the upper right.

NET

- To manage the network drives: Press the NETWORK soft key (second soft-key row). In the right-hand window **2** the TNC shows the network drives available for access. With the soft keys described below you can define the connection for each drive.

Function	Soft key
Establish the network connection. If the connection is active, the TNC marks the **Mnt** column.	MOUNT DEVICE
Delete network connection.	UNMOUNT DEVICE
Automatically establish network connection whenever the TNC is switched on. The TNC marks the **Auto** column if the connection is established automatically	AUTO MOUNT
Use the PING ping function to test your network connection	PING
If you press the NETWORK INFO soft key, the TNC displays the current network settings	NETWORK INFO

USB devices on the TNC

Backing up data from or loading onto the TNC is especially easy with USB devices. The TNC supports the following USB block devices:

- Disk drives with the FAT/VFAT file system
- Memory sticks with the FAT/VFAT file system
- Hard disks with the FAT/VFAT file system
- CD-ROM drives with the Joliet (ISO 9660) file system

The TNC automatically detects these types of USB devices when connected. The TNC does not support USB devices with other file systems (such as NTFS). After connection, the TNC displays an error message.

> The TNC also displays an error message if you connect a USB hub. In this case simply acknowledge the message with the CE key.
>
> In theory, you should be able to connect all USB devices with the file systems mentioned above to the TNC. If problems occur nevertheless, please contact HEIDENHAIN.

The USB devices appear as separate drives in the directory tree, so you can use the file-management functions described in the earlier chapters correspondingly.

In order to remove a USB device, you must proceed as follows:

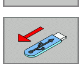
- To call the file manager, press the PGM MGT soft key.

- Select the left window with the arrow key.

- Use the arrow keys to select the USB device to be removed.

- Scroll through the soft-key row.

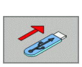
- Select additional functions.



- Select the function for removing USB devices: The TNC removes the USB device from the directory tree

- Exit program management.

In order to re-establish a connection with a USB device that has been removed, press the following soft key:

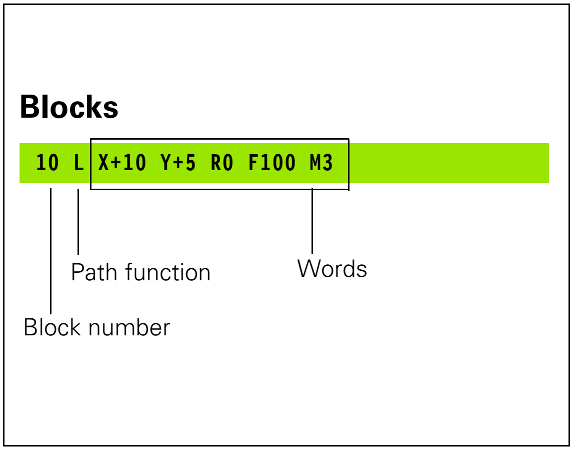


- Select the function for reconnection of USB devices.

4.4 Creating and Writing Programs

Organization of an NC program in HEIDENHAIN conversational format

A part program consists of a series of program blocks. The figure at right illustrates the elements of a block.



The TNC numbers the blocks in ascending sequence.

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

The subsequent blocks contain information on:

- The workpiece blank
- Tool definitions, tool calls
- Feed rates and spindle speeds, as well as
- Path contours, cycles and other functions

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

Defining the blank form – BLK FORM

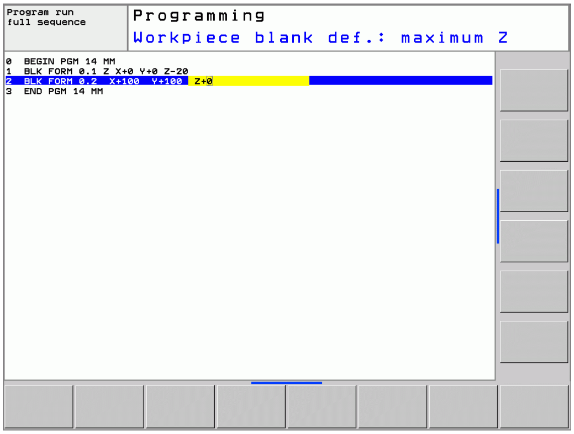
After initiating a new program, you define a cuboid workpiece blank. To define the workpiece blank, press the SPEC FCT soft key and then the BLK FORM soft key. This definition is needed for the TNC's graphic simulation feature. The sides of the workpiece blank lie parallel to the X, Y and Z axes and can be up to 100 000 mm long. The blank form is defined by two of its corner points:

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

You only need to define the blank form if you wish 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. An example of program initiation:



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

PGM MGT

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

Select the directory in which you wish to store the new program:

FILE NAME = 123.H

ENT

Enter the new program name and confirm your entry with the ENT key.

MM

To select the unit of measure, press the MM or INCH soft key. The TNC changes to the program window.

MORE FUNCTIONS

Press the SPECIAL TNC FUNCTIONS soft key

BLK FORM

Press the BLK FORM soft key

WORKING SPINDLE AXIS X/Y/Z ?

Z

Enter the spindle axis.

DEF BLK FORM: MIN-CORNER ?

0 ENT

0 ENT

-40 ENT

Enter in sequence the X, Y and Z coordinates of the MIN point.

DEF BLK FORM: MAX-CORNER ?

100	ENT	Enter in sequence the X, Y and Z coordinates of the MAX point.
100	ENT	
0	ENT	

Example: Display the BLK form in the NC program

`0 BEGIN PGM NEW MM`	Program begin, name, unit of measure
`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 NEW MM`	Program end, name, unit of measure

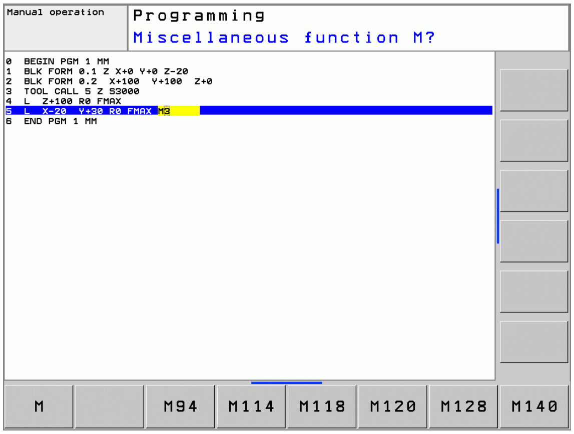
The TNC automatically generates the block numbers as well as the **BEGIN** and **END** blocks.

If you do not wish to define a blank form, cancel the dialog at **Working spindle axis X/Y/Z** by pressing the DEL key!

The TNC can display the graphics only if the shortest side is at least 50 µm long and the longest side is no longer than 99 999.999 mm.

Programming tool movements in conversational format

To program a block, initiate the dialog by pressing a function key. In the screen headline, the TNC then asks you for all the information necessary to program the desired function.



Example of a dialog

Dialog initiation

COORDINATES?

X 10 — Enter the target coordinate for the X axis

Y 20 ENT — Enter the target coordinate for the Y axis, and go to the next question with ENT

RADIUS COMP. RL/RR/NO COMP. ?

ENT — Enter “No radius compensation” and go to the next question with ENT.

FEED RATE F=? / F MAX = ENT

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

MISCELLANEOUS FUNCTION M?

3 ENT — Enter the miscellaneous function **M3** “spindle ON”; pressing the ENT key terminates this dialog.

The program-block window displays the following line:

```
3 L X+10 Y+5 R0 F100 M3
```

Functions for setting the feed rate	Soft key
Rapid traverse	F MAX
Traverse feed rate automatically calculated in **TOOL CALL**	F AUTO
Move at the programmed feed rate (unit of measure is mm/min)	F

Functions for conversational guidance	Key
Ignore the dialog question	NO ENT
End the dialog immediately	END
Abort the dialog and erase the block	DEL

Actual position capture

The TNC enables you to transfer the current tool position into the program, for example during

- Positioning-block programming.
- Cycle programming.
- Define the tools with **TOOL DEF.**

To transfer the correct position values, proceed as follows:

- Place the input box at the position in the block where you want to insert a position value.
- Select the actual-position-capture function: In the soft-key row the TNC displays the axes whose positions can be transferred.
- AXIS Z — Select the axis: The TNC writes the current position of the selected axis into the active input box.

In the working plane the TNC always captures the coordinates of the tool center, even though tool radius compensation is active.

In the tool axis the TNC always captures the coordinates of the tool tip and thus always takes the active tool length compensation into account.

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 key/key
Go to previous page	PAGE ↑
Go to next page	PAGE ↓
Go to beginning of program	BEGIN ↑
Go to end of program	END ↓
Change the position of the current block on the screen: Press this soft key to display additional program blocks that are programmed before the current block.	
Change the position of the current block on the screen: Press this soft key to display additional program blocks that are programmed after the current block.	
Move from one block to the next	↓ ↑
Select individual words in a block	← →
To select a certain block, press the GOTO key, enter the desired block number, and confirm with the ENT key.	GOTO □

Function	Soft key/key
Set the selected word to zero	CE
Erase 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	DEL
Insert the block that was last edited or deleted.	INSERT LAST NC BLOCK

Inserting blocks at any desired location

- Select the block after which you want to insert a new block and initiate the dialog.

Editing and inserting words

- Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.
- To accept the change, press the END key.

If you want to insert a word, press the horizontal arrow key repeatedly until the desired dialog appears. You can then enter the desired value.

Looking for the same words in different blocks

For this function, set the AUTO DRAW soft key to OFF.

→	To select a word in a block, press the arrow keys repeatedly until the highlight is on the desired word.
↓	Select a block with the arrow keys.

The word that is highlighted in the new block is the same as the one you selected previously.

> If you have started a search in a very long program, the TNC shows a progress display window. You then have the option of canceling the search via soft key.
>
> In the tool axis the TNC always captures the coordinates of the tool tip and thus always takes the active tool length compensation into account.

Finding any text

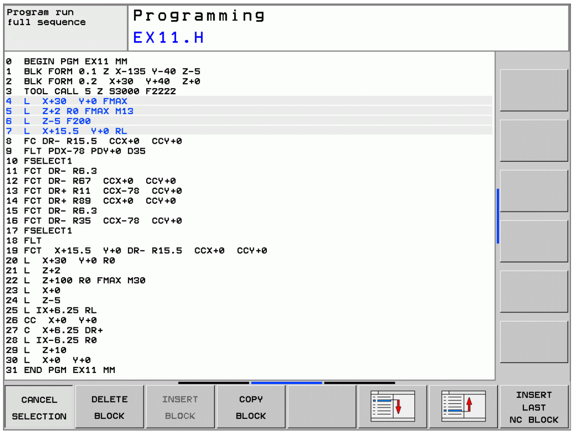
- To select the search function, press the FIND soft key. The TNC displays the dialog prompt **`Find text`**:
- Enter the text that you wish to find.
- To find the text, press the EXECUTE soft key.

Marking, copying, deleting and inserting program sections

The TNC provides certain functions for copying program sections within an NC program or into another NC program—see the table below.

To copy a program section, proceed as follows:

- Select the soft-key row containing the marking functions.
- Select the first (last) block of the section you wish to copy.
- To mark the first (last) block: Press the SELECT BLOCK soft key. The TNC then highlights the first character of the block and superimposes the soft key CANCEL SELECTION.
- Move the highlight to the last (first) block of the program section you wish to copy or delete. The TNC shows the marked blocks in a different color. You can end the marking function at any time by pressing the CANCEL SELECTION soft key.
- To copy the selected program section, press the COPY BLOCK soft key. To delete the selected section, press the DELETE BLOCK soft key. The TNC stores the selected block.
- Using the arrow keys, select the block after which you wish to insert the copied (deleted) program section.



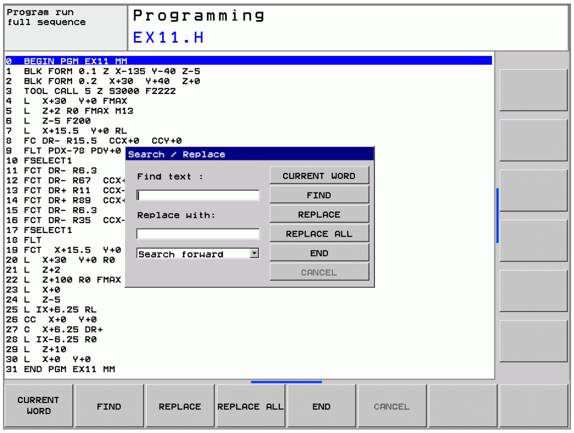
> To insert the section into another program, select the corresponding program using the file manager and then mark the block after which you wish to insert the copied block.

- To insert the block, press the INSERT BLOCK soft key.
- To end the marking function, press the CANCEL SELECTION soft key.

Function	Soft key
Switch marking function on	SELECT BLOCK
Switch marking function off	CANCEL SELECTION
Delete marked block	DELETE BLOCK
Insert block that is stored in the buffer memory	INSERT BLOCK
Copy marked block	COPY BLOCK

The TNC search function

With the search function of the TNC, you can search for any text within a program and replace it by a new text, if required.



Searching for texts

▶ If required, select the block containing the word you wish to find.

FIND

▶ Select the search function: The TNC superimposes the search window and displays the available search functions in the soft-key row (see table of search functions).

X +40

▶ Enter the text to be searched for. Please note that the search is case-sensitive.

CONTINUE

▶ Start the search process: The TNC displays the available search options in the soft-key row (see table of search options on the next page).

EXECUTE

▶ Start the search process: The TNC moves to the next block containing the text you are searching for.

EXECUTE

▶ Repeat the search process: The TNC moves to the next block containing the text you are searching for.

END

▶ End the search function.

Find/Replace any text

> The find/replace function is not possible if
>
> - a program is protected
> - the program is currently being run by the TNC
>
> When using the REPLACE ALL function, ensure that you do not accidentally replace text that you do not want to change. Once replaced, such text cannot be restored.

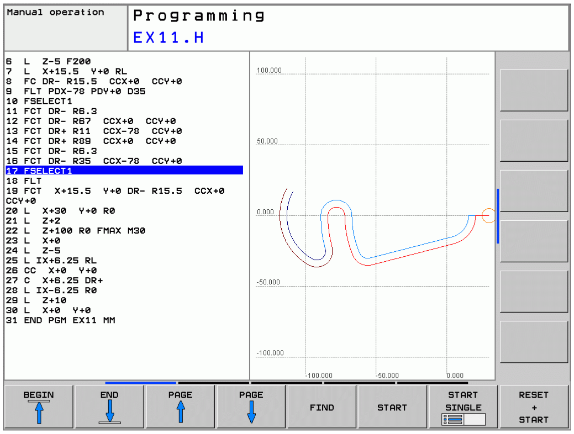
- If required, select the block containing the word you wish to find.

FIND	Select the Search function: The TNC superimposes the search window and displays the available search functions in the soft-key row.
SEARCH + REPLACE	Activate the Replace function: The TNC superimposes a window for entering the text to be inserted.
X	Enter the text to be searched for. Please note that the search is case-sensitive. Then confirm with the ENT key.
Z	Enter the text to be inserted. Please note that the entry is case-sensitive.
CONTINUE	Start the search process: The TNC displays the available search options in the soft-key row (see the table of search options).
WHOLE WORD OFF ON	If required, change the search options.
EXECUTE	Start the search process: The TNC moves to the next occurrence of the text you are searching for.
EXECUTE	To replace the text and then move to the next occurrence of the text, press the REPLACE soft key. To replace all text occurrences, press the REPLACE soft key. To skip the text and move to its next occurrence press the FIND soft key.
END	End the search function.

4.5 Interactive Programming Graphics

Generating / Not generating graphics during programming:

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



- To switch the screen layout to displaying program blocks to the left and graphics to the right, press the SPLIT SCREEN key and PGM + GRAPHICS soft key.

AUTO DRAW OFF ON

- Set the AUTO DRAW soft key to ON. While you are entering the program lines, the TNC generates each path contour you program in the graphics window in the right screen half.

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

Even when AUTO DRAW ON is active, graphics are not generated for program section repeats.

Generating a graphic for an existing program

- Use the arrow keys to select the block up to which you want the graphic to be generated, or press GOTO and enter the desired block number.

RESET + START

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

Additional functions:

Function	Soft key
Generate a complete graphic	RESET + START
Generate interactive graphic blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Stop the programming graphics. This soft key only appears while the TNC is generating the interactive graphics	STOP

Block number display ON/OFF

SHOW OMIT BLOCK NR.

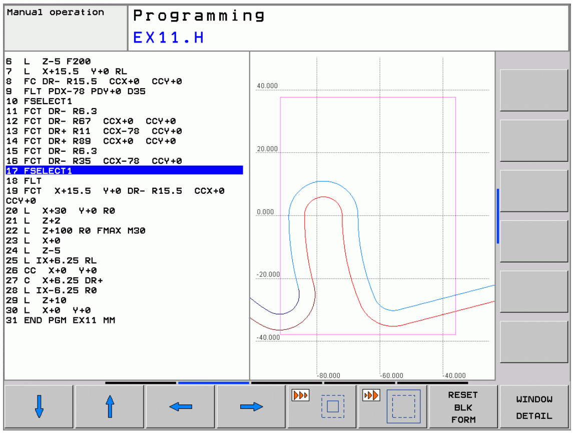
- Shift the soft-key row (see figure at upper right).
- To show block numbers: Set the SHOW OMIT BLOCK NR. soft key to SHOW.
- To omit block numbers: Set the SHOW OMIT BLOCK NR. soft key to OMIT.

Erasing the graphic

CLEAR GRAPHICS

- Shift the soft-key row (see figure at upper right).
- Delete graphic: Press CLEAR GRAPHIC soft key.

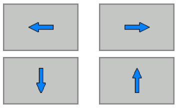
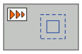
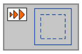
Magnifying or reducing a detail



You can select the graphics display by selecting a detail with the frame overlay. You can now magnify or reduce the selected detail.

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

The following functions are available:

Function	Soft key
Show and move the frame overlay. Press and hold the desired soft key to move the frame overlay.	
Reduce the frame overlay—press and hold the soft key to reduce the detail.	
Enlarge the frame overlay—press and hold the soft key to magnify the detail.	

WINDOW DETAIL

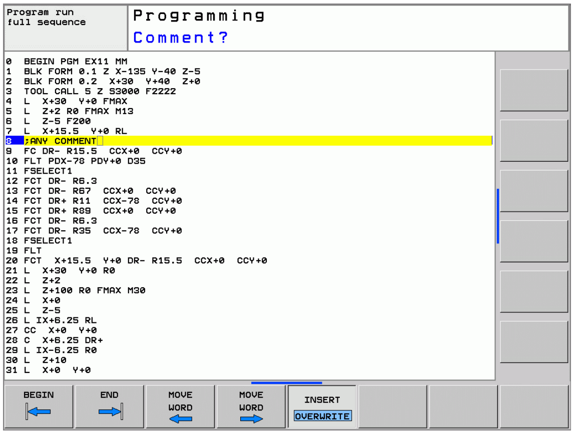
- Confirm the selected area with the WINDOW DETAIL soft key.

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

4.6 Adding Comments

Function

You can add comments to a part program to explain program steps or make general notes.



Adding a comment line

- Select the block after which the comment is to be inserted.
- Select the SPECIAL TNC FUNCTIONS soft key
- Select the COMMENT soft key
- Enter your comment using the screen keyboard (GOTO key) or a USB keyboard, if available, and conclude the block by pressing the END key.

Functions for editing of the comment

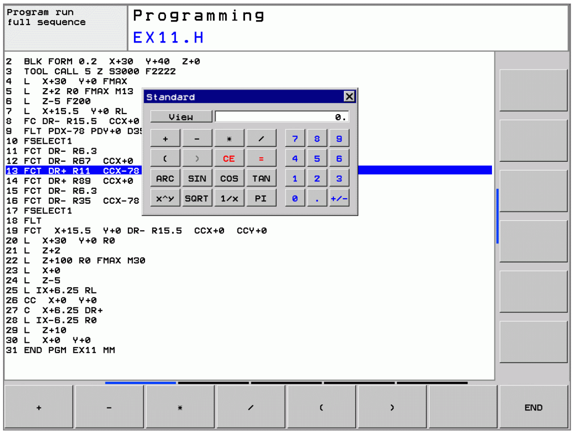
Function	Soft key
Jump to beginning of comment.	BEGIN
Jump to end of comment.	END
Jump to the beginning of a word. Words must be separated by a space.	MOVE WORD
Jump to the end of a word. Words must be separated by a space.	MOVE WORD
Switch between insert mode and overwrite mode.	INSERT OVERWRITE

4.7 Integrated Pocket Calculator

Operation

The TNC features an integrated pocket calculator with the basic mathematical functions.

- Use the CALC key to show and hide the on-line pocket calculator.
- Use soft keys to enter the calculator functions.



Mathematical function	Command (key)
Addition	+
Subtraction	–
Multiplication	*
Division	/
Parenthetic calculations	( )
Arc cosine	ARC
Sine	SIN
Cosine	COS
Tangent	TAN
Powers of values	X^Y
Square root	SQRT
Inversion	1/x
p (3.14159265359)	PI
Add value to buffer memory	M+
Save the value to buffer memory	MS
Recall from buffer memory	MR
Delete buffer memory contents	MC
Natural logarithm	LN
Logarithm	LOG
Exponential function	e^x
Check algebraic sign	SGN
Form the absolute value	ABS
Truncate decimal places	INT

Mathematical function	Command (key)
Truncate integers	FRAC
Modulus operator	MOD
Select view	Display mode
Delete value	DEL

To transfer the calculated value into the program,

- Select the word into which the calculated value is to be transferred by using the arrow keys.
- Superimpose the on-line calculator by using the CALC key and perform the desired calculation.
- Press the actual-position-capture key for the TNC to superimpose a soft-key row.
- Press the CALC soft key for the TNC to transfer the value into the active input box and to close the calculator.

4.8 The Error Messages

Display of errors

The TNC generates error messages when it detects problems such as:

- Incorrect data input
- Logical errors in the program
- Contour elements that are impossible to machine
- Incorrect use of the touch probe system

When an error occurs, it is displayed in red type in the header. Long and multi-line error messages are displayed in abbreviated form. If an error occurs in the background mode, the word "Error" is displayed in red type. Complete information on all pending errors is shown in the error window.

If a rare "processor check error" should occur, the TNC automatically opens the error window. You cannot remove such an error. Shut down the system and restart the TNC.

The error message is displayed in the header until it is cleared or replaced by a higher-priority error.

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block.

Open the error window.

ERR

- Press the ERR key. The TNC opens the error window and displays all accumulated error messages.

Close the error window

END

- Press the END soft key—or

ERR

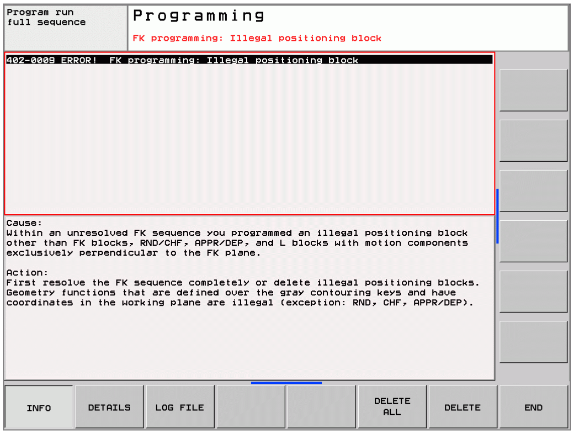
- Press the ERR key. The TNC closes the error window.

Detailed error messages

The TNC displays possible causes of the error and suggestions for solving the problem:

- Open the error window.

- Information on the error cause and corrective action: Position the highlight on the error message and press the INFO soft key. The TNC opens the window with information on the error cause and corrective action
- To leave Info, press the INFO soft key again



DETAILS soft key

The DETAILS soft key supplies information on the error message. This information is only required if servicing is needed.

- Open the error window.

DETAILS

- Detailed information about the error message: Position the highlight on the error message and press the DETAILS soft key. The TNC opens the window with internal information about the error
- To leave Details, press the DETAILS soft key again

Clearing errors

Clearing errors outside of the error window:

CE

- To clear the error/message in the header: Press the CE button.

In some operating modes (such as the Editing mode), the CE button cannot be used to clear the error, since the button is reserved for other functions.

Clearing more than one error:

- Open the error window.

DELETE

- Clear individual errors: Position the highlight on the error message and press the DELETE soft key.

DELETE ALL

- Clear all errors: Press the DELETE ALL soft key.

If the cause of the error has not been removed, the error message cannot be deleted. In this case, the error message remains in the window.

Error log file

The TNC stores errors and important events (e.g. system startup) in en error log file. The capacity of the error log file is limited. If the log file is full, the TNC uses a second log file. If this is also full, the first log file is deleted and written to again, and so on. To view the error history, switch between CURRENT FILE and PREVIOUS FILE.

- Open the error window.

LOG FILE

- Press the LOG FILE soft key

ERROR LOG FILE

- To open the error log file, press the ERROR LOG FILE soft key.

PREVIOUS FILE

- If you need the previous log file, press the PREVIOUS FILE soft key.

CURRENT FILE

- If you need the current log file, press the CURRENT FILE soft key.

The oldest entry is at the beginning of the error log file, and the most recent entry is at the end.

Keystroke log file

The TNC stores keystrokes and important events (e.g. system startup) in the keystroke log file. The capacity of the keystroke log file is limited. If the log file is full, a second log file is created. If this second file becomes full, the first is cleared and written to again, and so on. To view the keystroke history, switch between CURRENT FILE and PREVIOUS FILE.

LOG FILE

- Press the LOG FILE soft key

KEYSTROKE LOG FILE

- To open the keystroke log file, press the KEYSTROKE LOG FILE soft key.

PREVIOUS FILE

- If you need the previous log file, press the PREVIOUS FILE soft key.

CURRENT FILE

- If you need the current log file, press the CURRENT FILE soft key.

The TNC saves each key pressed during operation in the keystroke log file. The oldest entry is at the beginning of the log file, and the most recent entry is at the end.

Overview of the buttons and soft keys for viewing the log files:

Function	Soft key/key
Go to beginning of log file	BEGIN
Go to end of log file	END
Current log file	CURRENT FILE
Previous log file	PREVIOUS FILE
Up/down one line	
Return to main menu	

Informational texts

After a faulty operation, such as pressing a key without function or entering a value outside of the valid range, the TNC displays a (green) text in the header, informing you that the operation was not correct. The TNC clears this informational text upon the next valid input.

Saving service files

If necessary, you can save the "Current status of the TNC," and make it available to a service technician for evaluation. A group of service files is saved (error and keystroke log files, as well as other files that contain information about the current status of the machine and the machining).

If you repeat the "Save service data" function, the previously saved group of service data files is overwritten.

Saving service files:

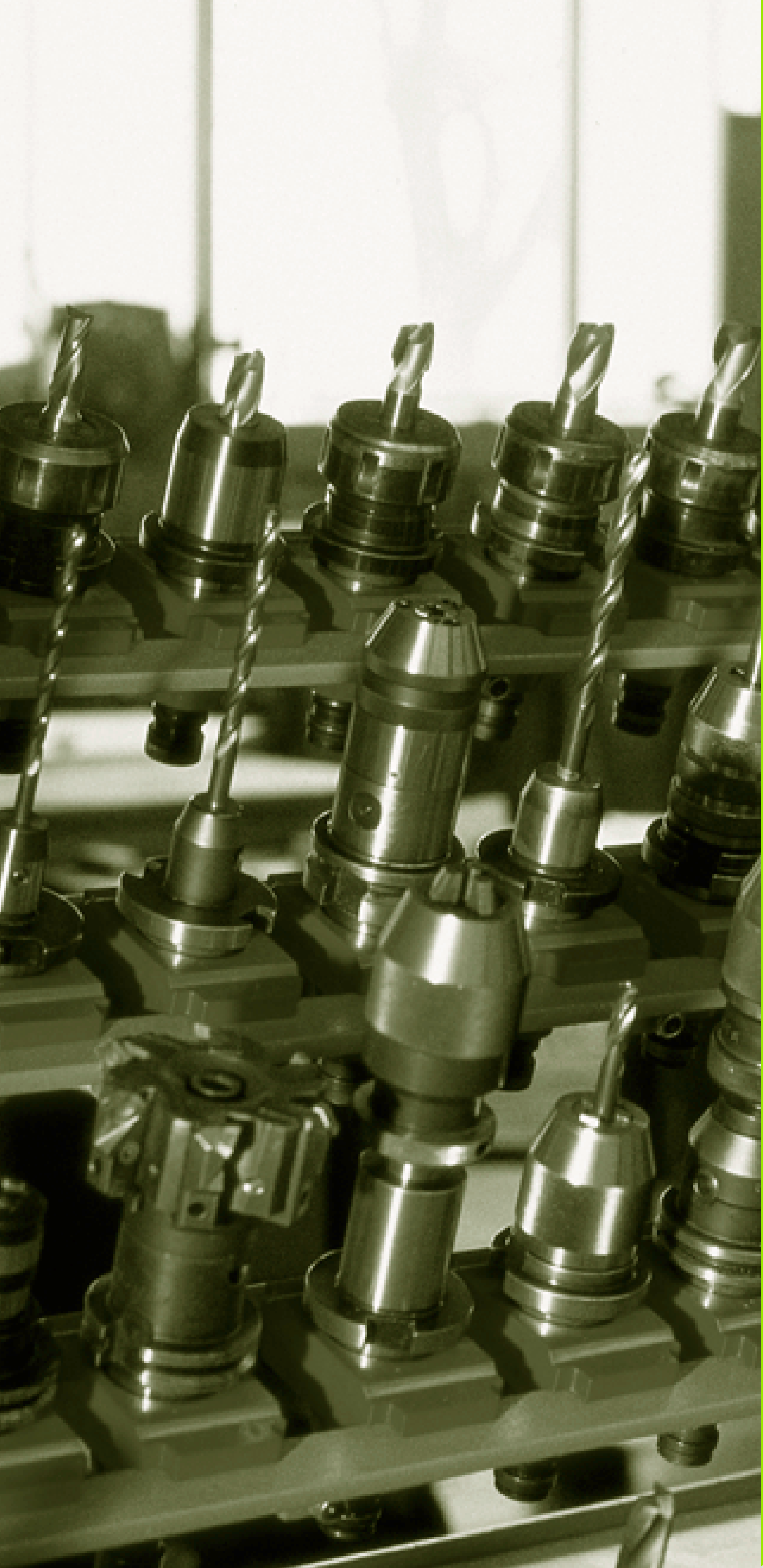
- Open the error window.

LOG FILE

- Press the LOG FILE soft key

SAVE SERVICE FILES

- To save service files, press the SAVE SERVICE FILES soft key.

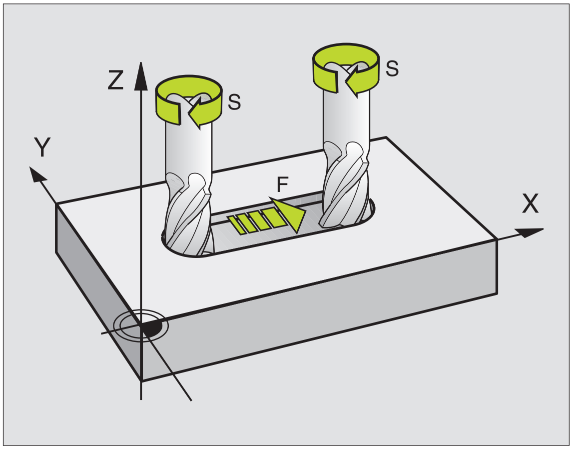


5

Programming: Tools

5.1 Entering Tool-Related Data

Feed rate F



The feed rate **F** is the speed (in millimeters per minute or inches per minute) at which the tool center moves. The maximum feed rates can be different for each machine axis, and are set in 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” on page 117).

Rapid traverse

If you wish to program rapid traverse, enter **F MAX.** To enter **F MAX,** press the ENT key or the F MAX soft key when the dialog question **FEED RATE F = ?** appears on the TNC screen.

> To move your machine at rapid traverse, you can also program the corresponding numerical value, e.g. **F30000.** Unlike **FMAX,** this rapid traverse remains in effect not only in the individual block but in all blocks until you program a new feed rate.

Duration of effect

A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. **F MAX** is only effective in the block in which it is programmed. After the block with **F MAX** is executed, the feed rate will return to the last feed rate entered as a numerical value.

Changing during program run

You can adjust the feed rate during program run 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 the spindle speed only:

TOOL CALL

- To program a tool call, press the TOOL CALL key.
- Ignore the dialog question for **Tool number ?** with the NO ENT key.
- Ignore the dialog question for **Working spindle axis X/Y/Z ?** with the NO ENT key.
- Enter the new spindle speed for the dialog question **Spindle speed S= ?,** and confirm with END.

Changing during program run

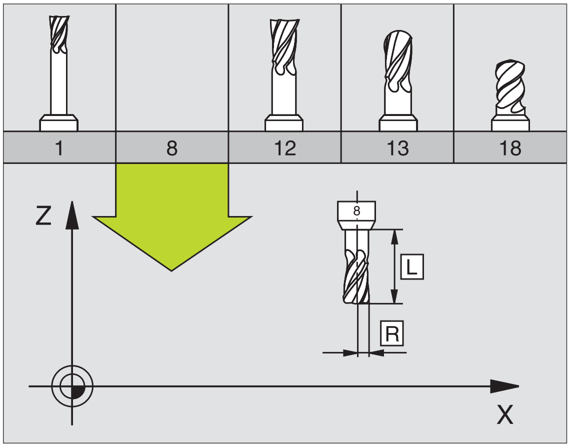
You can adjust the spindle speed during program run with the spindle-speed override knob S.

5.2 Tool Data

Requirements for tool compensation

You usually program the coordinates of path contours as they are dimensioned in the workpiece drawing. To allow the TNC to calculate the tool center path—i.e. the tool compensation—you must also enter the length and radius of each tool you are using.

Tool data can be entered either directly in the part program with TOOL DEF or separately in a tool table. In a tool table, you can also enter additional data for the specific tool. The TNC will consider all the data entered for the tool when executing the part program.



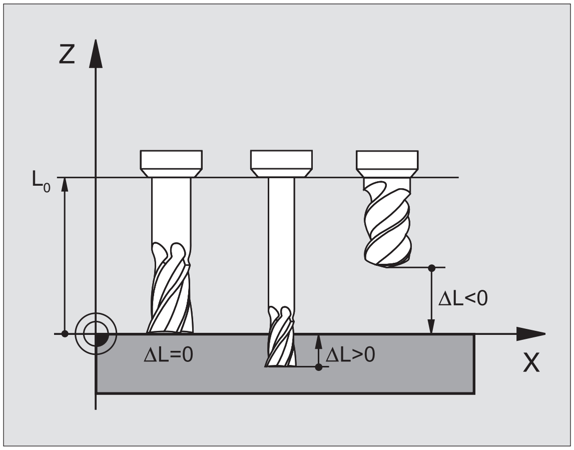
Tool numbers and tool names

Each tool is identified by a number between 0 and 9999. If you are working with tool tables, you can use higher numbers and you can also enter a tool name for each tool. Tool names can have up to 16 characters.

The tool number 0 is automatically defined as the zero tool with the length L=0 and the radius R=0. In tool tables, tool T0 should also be defined with L=0 and R=0.

Tool length L

There are two ways to determine the tool length L:



Determining the difference between the length of the tool and that of a zero tool L0

For the algebraic sign:

L>L0: The tool is longer than the zero tool

L<L0: The tool is shorter than the zero tool

To determine the length:

- Move the zero tool to the reference position in the tool axis (e.g. workpiece surface with Z=0).
- Set the datum in the tool axis to 0 (datum setting).
- Insert the desired tool.
- Move the tool to the same reference position as the zero tool.
- The TNC displays the difference between the current tool and the zero tool.
- Enter the value in the TOOL DEF block or in the tool table by pressing the actual-position-capture key.

Determining the length L with a tool presetter

Enter the determined value directly in the TOOL DEF tool definition block or in the tool table without further calculations.

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii

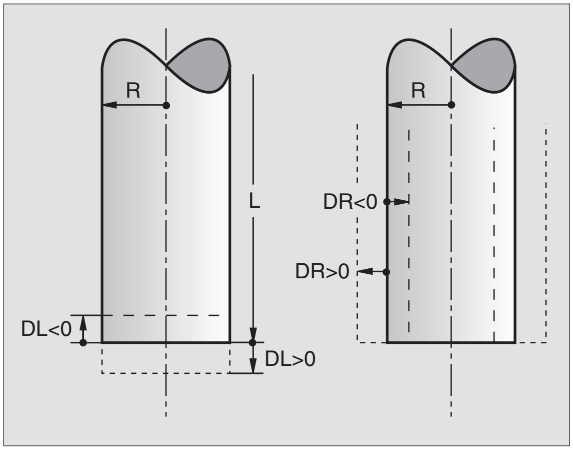
Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (**DL**, **DR**, **DR2**>0). If you are programming 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 usually entered as numerical values. In a **TOOL CALL** block, you can also assign the values to Q parameters.

Input range: You can enter a delta value with up to ± 99.999 mm.



> Delta values from the tool table influence the graphical representation of the **tool.** The representation of the **workpiece** remains the same in the simulation.
>
> Delta values from the TOOL CALL block change the represented size of the **workpiece** during the simulation. The simulated **tool size** remains the same.

Entering tool data into the program

The number, length and radius of a specific tool is defined in the **TOOL DEF** block of the part program.

- To select tool definition, press the TOOL DEF key.

TOOL DEF

- **Tool number:** Each tool is uniquely identified by its tool number.
- **Tool length:** Compensation value for the tool length
- **Tool radius:** Compensation value for the tool radius

> In the programming dialog, you can transfer the value for tool length and tool radius directly into the input line by pressing the desired axis soft key.

Example

```
4 TOOL DEF 5 L+10 R+5
```

Entering tool data in the table

You can define and store up to 9999 tools and their tool data in a tool table. Also see the Editing Functions later in this Chapter. In order to be able to assign various compensation data to a tool (indexing tool number), insert a line and extend the tool number by a point and a number from 1 to 9 (e.g. **T 5.2**).

You must use tool tables if

- you wish to use indexed tools such as stepped drills with more than one length compensation value (Page 102)
- your machine tool has an automatic tool changer
- you want to rough-mill the contour with Cycle 22 (see “ROUGH-OUT (Cycle 22)” on page 262)

Tool table: Standard tool data

Abbr.	Input	Dialog
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 representation of a machining operation with spherical or 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 oversize R2?**
TL	Set tool lock (**TL: T**ool **L**ocked)	**Tool locked?** **Yes = ENT / No = NO ENT**
RT	Number of a replacement tool, if available (**RT**: for **R**eplacement **T**ool; see also **TIME2**	**Replacement tool?**
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information on TIME1.	**Maximum tool age?**
TIME2	Maximum tool life in minutes during **TOOL CALL:** If the current tool age exceeds this value, the TNC changes the tool during the next **TOOL CALL** (see also **CUR.TIME**).	**Maximum tool age for TOOL CALL?**
CUR.TIME	Current age of the tool in minutes: The TNC automatically counts the current tool life (**CUR.TIME**). A starting value can be entered for used tools.	**Current tool life?**

Abbr.	Input	Dialog
`TYPE`	Tool type: Press the SELECT TYPE (3rd soft-key row); the TNC superimposes a window where you can select the type of tool you want. Functions are currently only assigned to the DRILL and MILL tool types.	`Tool type?`
`DOC`	Comment on tool (up to 16 characters)	`Tool description?`
`PLC`	Information on this tool that is to be sent to the PLC	`PLC status?`
`LCUTS`	Tooth length of the tool for Cycle 22	`Tooth length in the tool axis?`
`ANGLE`	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles 22 and 208	`Maximum plunge angle?`
`CUT`	Number of teeth (20 teeth maximum)	`Number of teeth?`
`RTOL`	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	`Wear tolerance: radius?`
`LTOL`	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	`Wear tolerance: length?`
`DIRECT.`	Cutting direction of the tool for measuring the tool during rotation	`Cutting direction (M3 = –)?`
`TT:R-OFFS`	Not supported at present	`Tool offset: radius?`
`TT:L-OFFS`	Not supported at present	`Tool offset: length?`
`LBREAK`	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status **L**). Input range: 0 to 0.9999 mm	`Breakage tolerance: length?`
`RBREAK`	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	`Breakage tolerance: radius?`
`LIFTOFF`	Definition of whether the TNC should retract the tool in the direction of the positive tool axis at an NC stop in order to avoid leaving dwell marks on the contour. If **Y** is defined, the TNC retracts the tool from the contour by 0.1 mm, provided that this function was activated in the NC program with M148 (see "Automatically retract tool from the contour at an NC stop: M148" on page 171).	`Retract tool Y/N ?`

Editing tool tables

The tool table that is active during execution of the part program is designated TOOL.T and must be saved in the “table” directory. The tool table TOOL.T can be edited only in a machine mode of operation.

for archiving or test runs, give the tool tables some other name with the extension .T. By default, for the Test Run and Programming modes the TNC uses the “simtool.t” tool table, which is also stored in the “table” directory. In the Programming and Editing mode, press the TABLE EDITOR soft key to edit it.

To open the tool table TOOL.T:

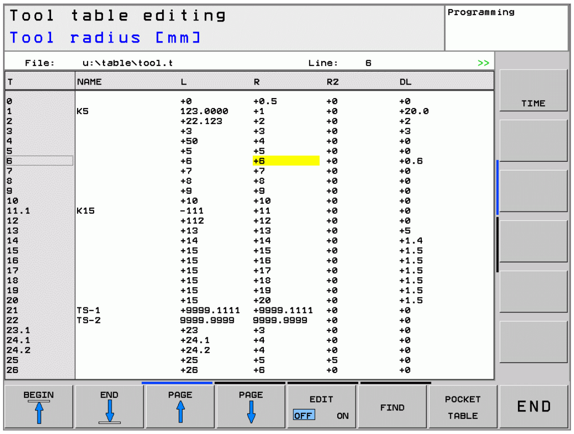
- Select any machine operating mode.

TOOL TABLE

- To select the tool table, press the TOOL TABLE soft key.

EDIT OFF ON

- Set the EDIT soft key to ON.



To open any other tool table

- Select the Programming and Editing mode of operation.

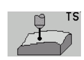
PGM MGT

- 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. Conclude your entry with the ENT key or the SELECT soft key.

When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys. You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table below.

If the TNC cannot show all positions in the tool table in one screen page, the highlight bar at the top of the table will display the symbol “**>>**” or “**<<**”.

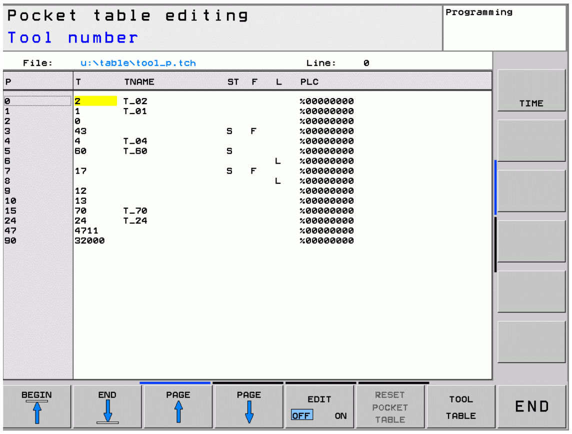
Editing functions for tool tables	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE ↑
Select next page in table	PAGE ↓
Find the text or number	FIND

Editing functions for tool tables	Soft key
Move to beginning of line	BEGIN LINE
Move to end of line	END LINE
Copy highlighted field	COPY FIELD
Insert copied field	PASTE FIELD
Add the entered number of lines (tools) at the end of the table.	APPEND N LINES
Insert a line with definable tool number	INSERT LINE
Delete current line (tool).	DELETE LINE
Sort the tools according to the content of a column	SORT
Show all drills in the tool table	DRILL
Show all touch probes in the tool table	

Leaving the tool table

- Call the file manager and select a file of a different type, such as a part program.

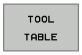
Pocket table for tool changer



The machine tool builder adapts the functional range of the pocket table to the requirements of your machine. The machine tool manual provides further information.

For automatic tool changing you need the pocket table TOOL_P.TCH. The TNC can manage several pocket tables with any file names. To activate a specific pocket table for program run you must select it in the file management of a Program Run mode of operation (status M).

Editing a pocket table in a Program Run operating mode



▶ To select the tool table, press the TOOL TABLE soft key.

▶ To select the pocket table, press the POCKET TABLE soft key.

▶ Set the EDIT soft key to ON.

Selecting a pocket table in the Programming and Editing mode of operation

▶ Call the file manager.

▶ To select the file type, press the SELECT TYPE soft key.

▶ To show files of the type .TCH, press the soft key TCH FILES (second soft-key row).

▶ Select a file or enter a new file name. Conclude your entry with the ENT key or the SELECT soft key.

Abbr.	Input	Dialog
P	Pocket number of the tool in the tool magazine	–
T	Tool number	**Tool number?**
TNAME	Display of the tool name from TOOL.T	–
ST	Special tool (**ST**) with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked in column L (status L).	**Special tool?**
F	Fixed tool number. The tool is always returned to the same pocket in the tool magazine	**Fixed pocket? Yes = ENT / No = NO ENT**
L	Locked pocket (see also column ST)	**Pocket locked Yes = ENT / No = NO ENT**
PLC	Information on this tool pocket that is to be sent to the PLC	**PLC status?**

Editing functions for pocket tables	Soft key
Select beginning of table	BEGIN
Select end of table	END
Select previous page in table	PAGE
Select next page in table	PAGE
Reset pocket table	RESET POCKET TABLE
Reset tool number column T	RESET COLUMN T
Go to beginning of the line	BEGIN LINE
Go to end of the line	END LINE
Simulate a tool change	SIMULATED TOOL CHANGE
Activate a filter	DEFAULT FILTER
Select a tool from the tool table	SELECT
Edit the current field	EDIT CURRENT FIELD
Sort the view	SORT

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 CALL

▶ **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. The TNC automatically places the tool name in quotation marks. The tool name always refers to the entry in the active tool table TOOL.T. If you wish to call a tool with other compensation values, also enter the index you defined in the tool table after the decimal point.

▶ **Working spindle axis X/Y/Z:** Enter the tool axis.

▶ **Spindle speed S:** Spindle speed in rpm

▶ **Feed rate F**: F is effective until you program a new feed rate in a positioning or 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 in the tool axis Z with a spindle speed of 2500 rpm and a feed rate of 350 mm/min. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius 2 with an oversize of 0.05 mm, and the tool radius with an undersize of 1 mm.

```
20 TOOL CALL 5.2 Z S2500 F350 DL+0.2 DR-1 DR2+0.05
```

The character **D** preceding **L** and **R** designates delta values.

Tool preselection with tool tables

If you are working with tool tables, use **TOOL DEF** to preselect the next tool. Simply enter the tool number or a corresponding Q parameter, or type the tool name in quotation marks.

Tool change

The tool change function can vary depending on the individual machine tool. The machine tool manual provides further information.

Tool change position

The tool change position must be approachable without collision. With the miscellaneous functions **M91** and **M92,** you can enter machine-referenced (rather than workpiece-referenced) coordinates for the tool change position. If **TOOL CALL 0** is programmed before the first tool call, the TNC moves the tool spindle in the tool axis to a position that is independent of the tool length.

Manual tool change

To change the tool manually, stop the spindle and move the tool to the tool change position:

- Move to the tool change position under program control.
- Interrupt program run (see "Interrupting machining", page 388).
- Change the tool.
- Resume program run (see "Resuming program run after an interruption", page 389).

Automatic tool change

If your machine tool has automatic tool changing capability, the program run is not interrupted. When the TNC reaches a **TOOL CALL**, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

The function of **M101** can vary depending on the individual machine tool. The machine tool manual provides further information.

The TNC automatically changes the tool if the tool life **TIME2** expires during program run. To use this miscellaneous function, activate **M101** at the beginning of the program. **M101** is reset with **M102.**

The tool is changed automatically

- after the next NC block after expiration of the tool life, or
- at latest one minute after tool life expires (calculation is for a potentiometer setting of 100%)

If the tool life ends during an active M120 (look ahead), the TNC waits to change the tool until after the block in which you canceled the radius compensation with an R0 block.

The TNC automatically changes the tool even if a fixed cycle is being run.

As long as a tool change program is running, the TNC makes no automatic tool change.

Prerequisites for standard NC blocks with radius compensation R0, RR, RL

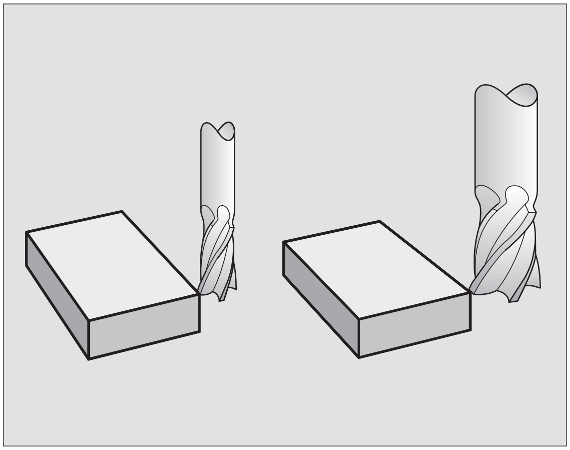
The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool.

5.3 Tool Compensation

Introduction

The TNC adjusts the spindle path in the tool 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 TNC, the tool radius compensation is effective only in the working plane. The TNC accounts for the compensation value in up to five axes including the rotary axes.



Tool length compensation

Length compensation becomes effective automatically as soon as a tool is called and the tool axis moves. To cancel length compensation, call a tool with the length L=0.

> If you cancel a positive length compensation with **TOOL CALL 0,** the distance between tool and workpiece will be reduced.
>
> After **TOOL CALL,** the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

For tool length compensation, the TNC takes the delta values from both the **TOOL CALL** block and the tool table into account:

Compensation value = **L** + $\mathbf{DL}_{\text{TOOL CALL}}$ + $\mathbf{DL}_{\text{TAB}}$ where

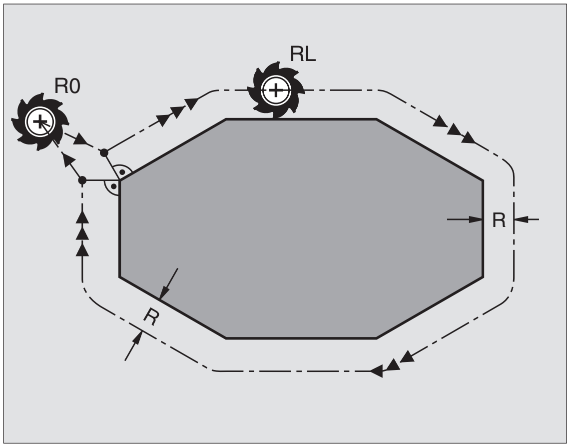
L:	is the tool length **L** from the **TOOL DEF** block or tool table.
DL TOOL CALL	is the oversize for length **DL** in the **TOOL CALL** block (not taken into account by the position display).
DL TAB	is the oversize for length **DL** in the tool table.

Tool radius compensation

The NC block for programming a tool movement contains:

- **RL** or **RR** for radius compensation.
- **R0** if there is no radius compensation.

Radius compensation becomes effective as soon as a tool is called and is moved with a straight line block in the working plane with RL or RR.



> The TNC automatically cancels radius compensation if you:
>
> - program a straight line block with **R0**
> - depart the contour with the **DEP** function
> - program a **PGM CALL**
> - select a new program with PGM MGT.

For tool radius compensation, the TNC takes the delta values from both the **TOOL CALL** block and the tool table into account:

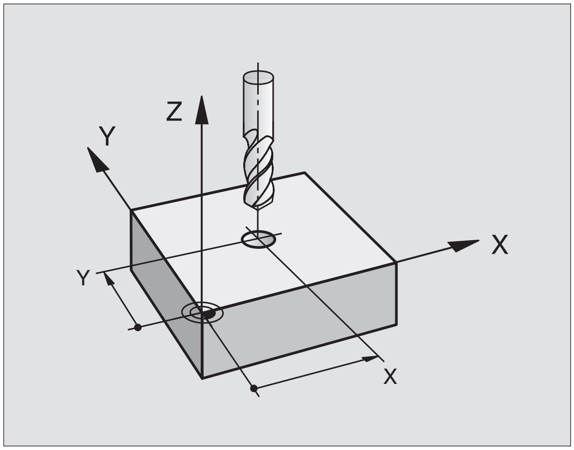
Compensation value = **R** + $\mathbf{DR}_{\text{TOOL CALL}}$ + $\mathbf{DR}_{\text{TAB}}$ where

R	is the tool radius **R** from the **TOOL DEF** block or tool table.
DR TOOL CALL	is the oversize for radius **DR** in the **TOOL CALL** block (not taken into account by the position display).
DR TAB	is the oversize for radius **DR** in the tool table.

Contouring without radius compensation: R0

The tool center moves in the working plane along the programmed path or to the programmed coordinates.

Applications: Drilling and boring, pre-positioning.

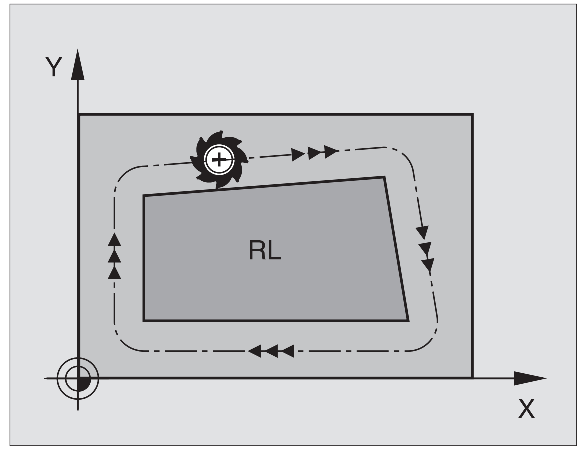


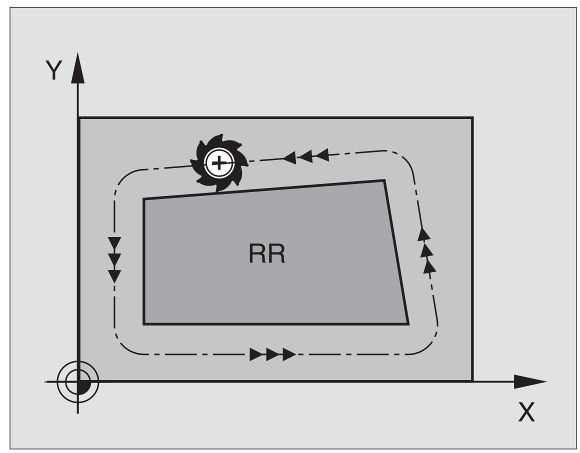
Tool movements with radius compensation: RR and RL

RR The tool moves to the right of the programmed contour

RL The tool moves to the left of the programmed contour

The tool center moves along the contour at a distance equal to the radius. “Right” or “left” are to be understood as based on the direction of tool movement along the workpiece contour. See figures at right.





> 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 does not take effect until the end of the block in which it is first programmed.
>
> Whenever radius compensation is activated with **RR/RL** or canceled with **R0,** the TNC positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.

Entering radius compensation

Program any desired path function, enter the coordinates of the target point and confirm your entry with ENT.

RADIUS COMP.: RL/RR/NO COMP.?

RL — To select tool movement to the left of the contour, press the RL soft key, or

RR — To select tool movement to the right of the contour, press the RR soft key, or

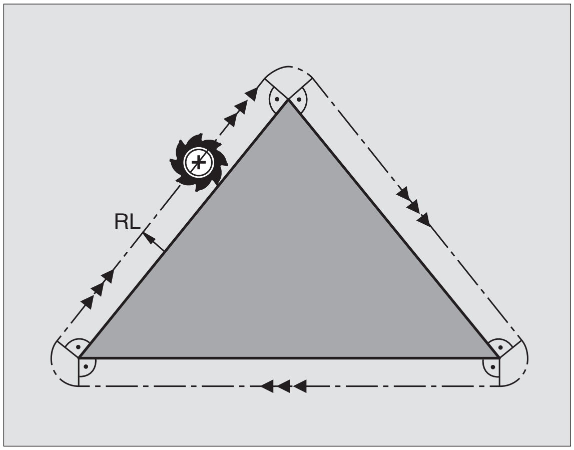
ENT — To select tool movement without radius compensation or to cancel radius compensation, press the ENT key.

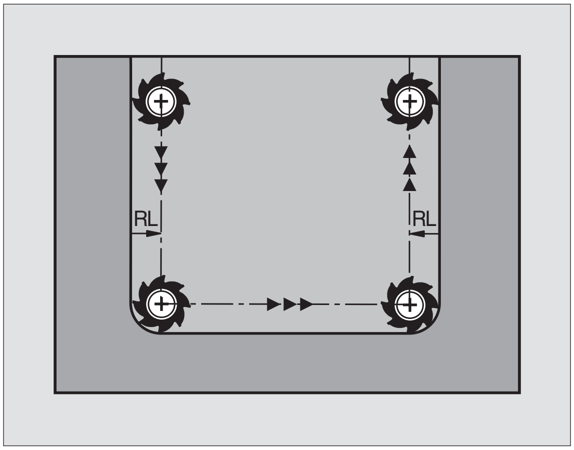
END — To terminate the block, press the END key.

Radius compensation: Machining corners

- Outside corners
  If you program radius compensation, the TNC moves the tool around outside corners on a transitional arc. If necessary, the TNC reduces the feed rate at outside corners to reduce machine stress, for example at very great changes of direction.
- Inside corners
  The TNC calculates the intersection of the tool center paths at inside corners under radius compensation. From this point it then starts the next contour element. This prevents damage to the workpiece. The permissible tool radius, therefore, is limited by the geometry of the programmed contour.

To prevent the tool from damaging the contour, be careful not to program the starting or end position for machining inside corners at a corner of the contour.





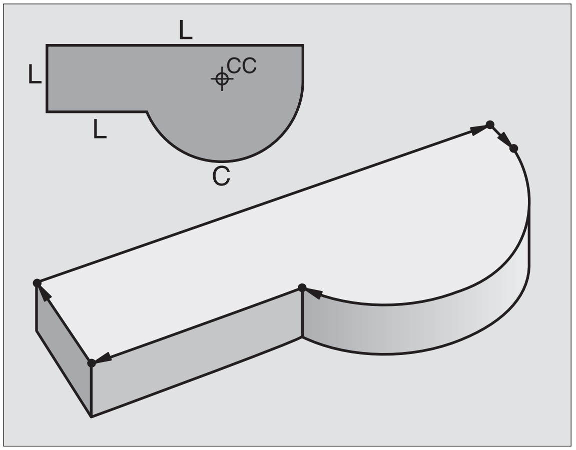
6

Programming:
Programming Contours

6.1 Tool Movements

Path functions

A workpiece contour is usually 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.**

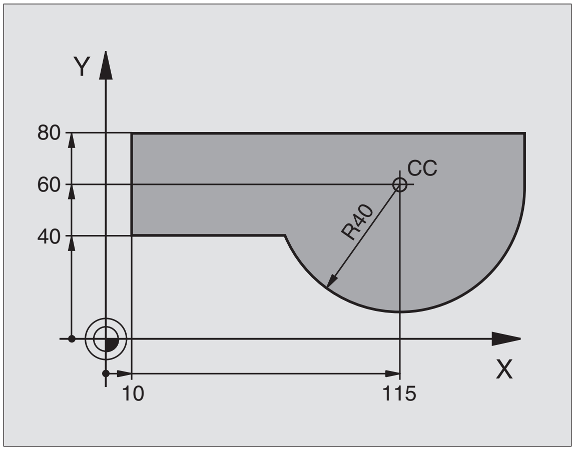


FK Free Contour Programming

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

With FK programming, you also program tool movements for **straight lines** and **circular arcs**.

Miscellaneous functions M



With the TNC’s miscellaneous functions you can affect

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring behavior of the tool

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat. 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 separate program for execution.

Programming with subprograms and program section repeats is described in Chapter 9.

Programming with Q parameters

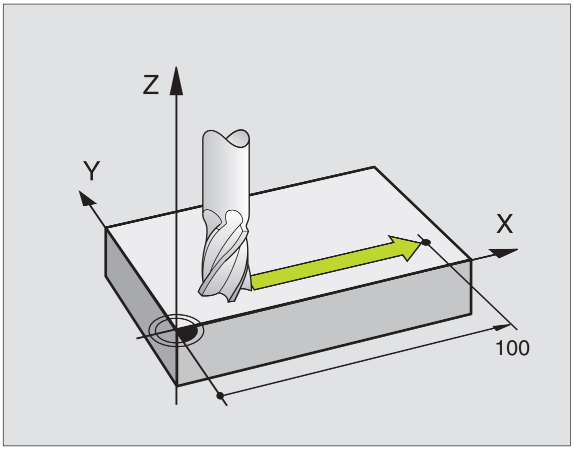
Instead of programming numerical values in a part program, you enter markers called Q parameters. You assign the values to the Q parameters separately with the Q parameter functions. You can use the Q parameters for programming mathematical functions that control program execution or describe a contour.

Programming with Q parameters is described in Chapter 10.

6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

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



The TNC moves all axes programmed in a single block simultaneously.

Movement parallel to the machine axes

The program block contains only one coordinate. The TNC thus moves the tool parallel to the programmed axis.

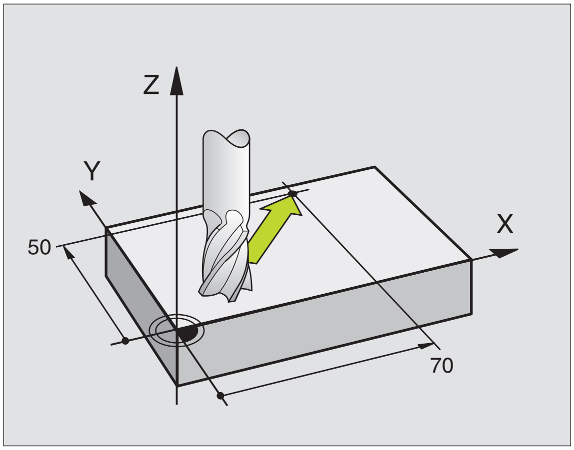
Depending on the machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

```
L X+100
```

L	Path function for a 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 figure at upper right).



Movement in the main planes

The program block contains two coordinates. The TNC thus moves the tool in the programmed plane.

Example:

```
L X+70 Y+50
```

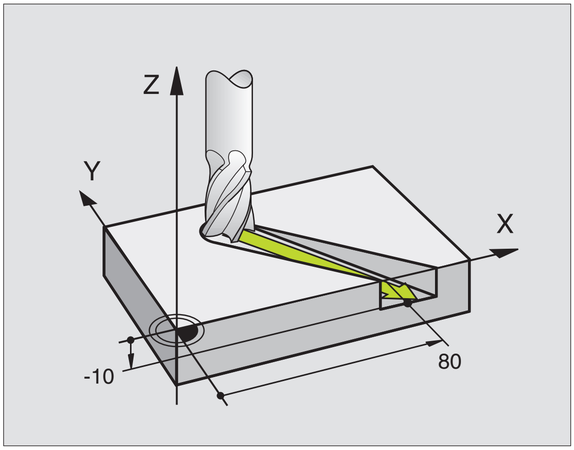
The tool retains the Z coordinate and moves in the XY plane to the X=70, Y=50 position (see figure at center right).

Three-dimensional movement

The program block contains three coordinates. The TNC thus moves the tool in space to the programmed position.

Example:

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

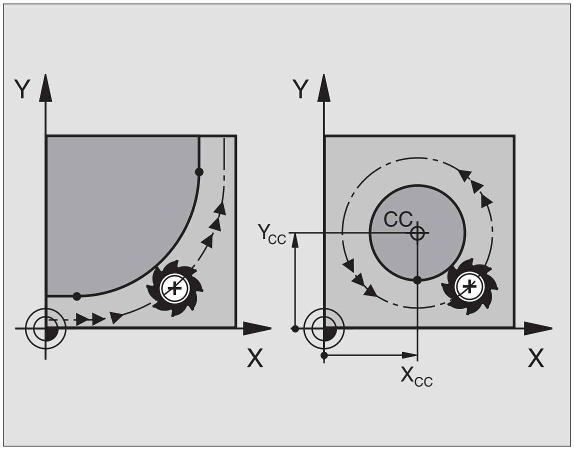


Circles and circular arcs

The TNC moves two axes simultaneously in a circular path relative to the workpiece. You can define a circular movement by entering the circle center CC.

When you program a circle, the control assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a TOOL CALL:

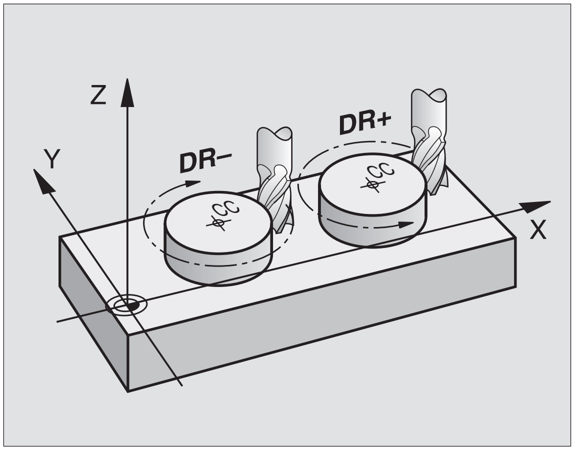
Spindle axis	Main plane
Z	**XY**, also UV, XV, UY
Y	**ZX**, also WU, ZU, WX
X	**YZ**, also VW, YW, VZ



Direction of rotation DR for circular movements

If a circular path has no tangential transition to another contour element, enter the direction of rotation DR:

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 cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line block (see "Path Contours—Cartesian Coordinates," page 125) or approach block (APPR block, see "Contour Approach and Departure," page 119).

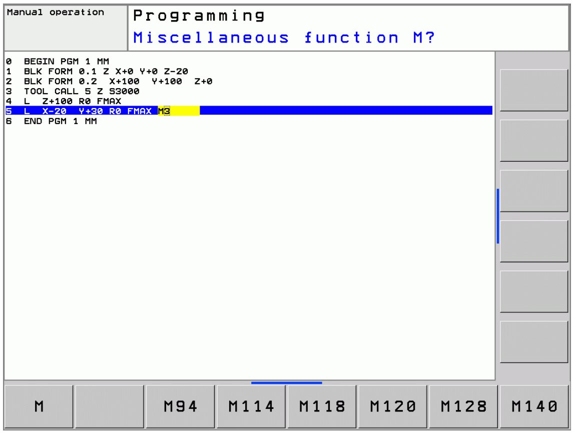
Pre-positioning

Before running a part program, always pre-position the tool to prevent the possibility of damaging it or the workpiece.

Creating the program blocks with the path function keys

The gray path function keys initiate the plain language dialog. The TNC 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 dialog, e.g. for a straight line.

COORDINATES?

X 10

Y 5

ENT

Enter the coordinates of the straight-line end point.

RADIUS COMP.: RL/RR/NO COMP.?

R0

Select the radius compensation (here, press the R0 soft key—the tool moves without compensation).

FEED RATE F=? / F MAX = ENT

100 ENT

Enter the feed rate (here, 100 mm/min), and confirm your entry with ENT. For programming in inches, enter 100 for a feed rate of 10 ipm.

F MAX

Move at rapid traverse: press the FMAX soft key

F AUTO

To traverse with the feed rate defined in the **TOOL CALL** block, press the FAUTO soft key.

MISCELLANEOUS FUNCTION M?

3 ENT Enter a miscellaneous function (here, M3), and terminate the dialog with ENT.

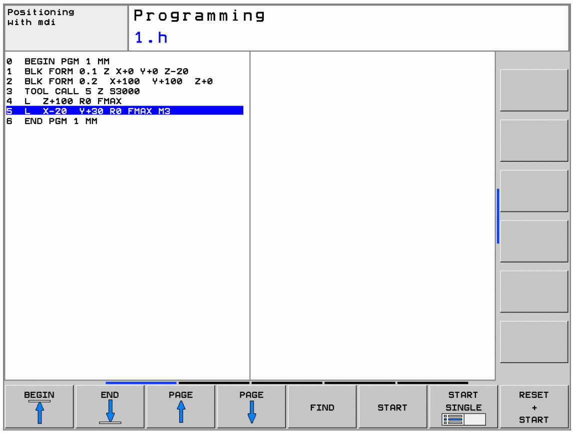
The part program now contains the following line:

```
L X+10 Y+5 RL F100 M3
```

6.3 Contour Approach and Departure

Overview: Types of paths for contour approach and departure

The functions for contour approach APPR and departure DEP are activated with the APPR/DEP key. You can then select the desired path function with the corresponding soft key:

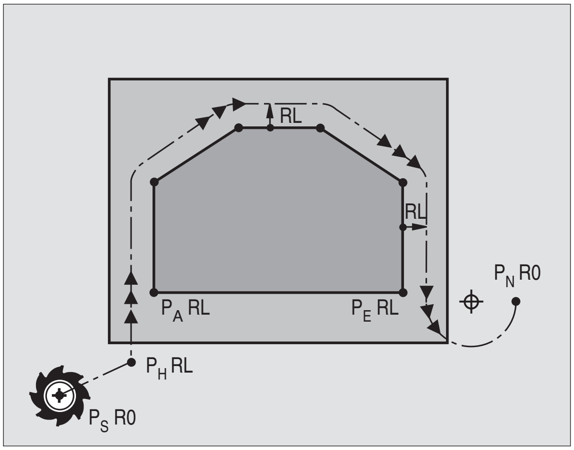


Function	Approach	Departure
Straight line with tangential connection	APPR LT	DEP LT
Straight line perpendicular to a contour point	APPR LN	DEP LN
Circular arc with tangential connection	APPR CT	DEP CT
Circular arc with tangential connection to the contour. Approach and departure to an auxiliary point outside of the contour on a tangentially connecting line.	APPR LCT	DEP LCT

Approaching and departing a helix

The tool approaches and departs a helix on its extension by moving in a circular arc that connects tangentially to the contour. 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 in the block 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 TNC calculates from your input in the APPR or DEP block. The TNC moves from the current position to the auxiliary point P_H at the feed rate last programmed.
- 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 TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.

- End point 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 TNC will first move the tool to P_H in the working plane, and then move it to the entered depth in the tool axis.

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

The TNC does not check whether the programmed contour will be damaged when moving from the actual position to the auxiliary point P_H. Use the test graphics to simulate approach and departure before executing the part program.

With the APPR LT, APPR LN and APPR CT functions, the TNC moves the tool from the actual position to the auxiliary point P_H at the feed rate that was last programmed. With the APPR LCT function, the TNC moves to the auxiliary point P_H at the feed rate programmed with the APPR block. If no feed rate is programmed yet before the approach block, the TNC generates an error message.

Polar coordinates

You can also program the contour points for the following approach/departure functions over polar coordinates:

- APPR LT becomes APPR PLT
- APPR LN becomes APPR PLN
- APPR CT becomes APPR PCT
- APPR LCT becomes APPR PLCT
- DEP LCT becomes DEP PLCT

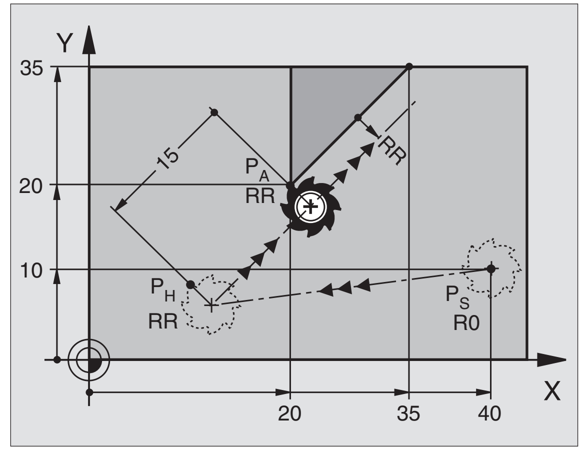
Select by soft key an approach or departure function, then press the orange P key.

Radius compensation

The tool radius compensation is programmed together with the first contour point P_A in the APPR block. The DEP blocks automatically discard the tool radius compensation.

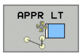
Contour approach without radius compensation: If you program the APPR block with R0, the TNC will calculate the tool path for a tool radius of 0 mm and a radius compensation RR! The radius compensation is necessary to set the direction of contour approach and departure in the APPR/DEP LN and APPR/DEP CT functions.

Approaching on a straight line with tangential connection: APPR LT



The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a straight line that connects tangentially to the contour. The auxiliary point P_H is separated from the first contour point P_A by the distance LEN.

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the APPR/DEP key and APPR LT soft key:

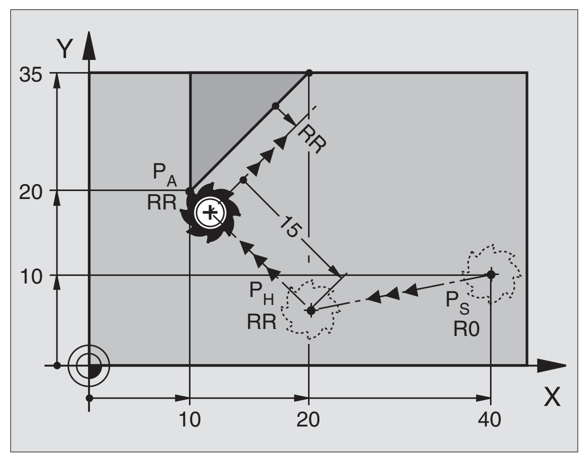


- Coordinates of the first contour point P_A
- LEN: Distance from the auxiliary point P_H to the first contour point P_A
- Radius compensation RR/RL for machining

Example NC blocks

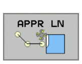
`7 L X+40 Y+10 RO FMAX M3`	Approach P_S without radius compensation
`8 APPR LT X+20 Y+20 Z-10 LEN15 RR F100`	P_A with radius comp. RR, distance P_H to P_A: LEN=15
`9 L Y+35 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

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



The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a straight line perpendicular to the first contour element. The auxiliary point P_H is separated by the distance LEN plus the tool radius from the first contour point P_A.

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the APPR/DEP key and APPR LN soft key:

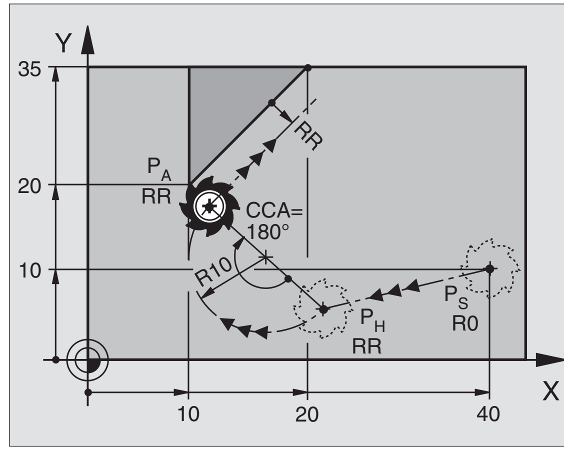


- Coordinates of the first contour point P_A
- Length: Distance to the auxiliary point P_H. Always enter LEN as a positive value!
- Radius compensation RR/RL for machining

Example NC blocks

`7 L X+40 Y+10 RO FMAX M3`	Approach P_S without radius compensation
`8 APPR LN X+10 Y+20 Z-10 LEN15 RR F100`	P_A with radius comp. RR
`9 L X+20 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

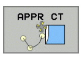
Approaching on a circular path with tangential connection: APPR CT



The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then 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 to 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 dialog with the APPR/DEP key and APPR CT soft key:

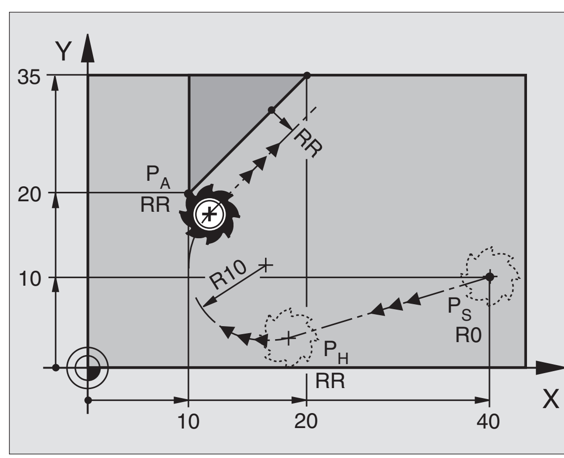


- Coordinates of the first contour point P_A
- Radius R of the circular arc
  - If the tool should approach the workpiece in the direction defined by the radius compensation: Enter R as a positive value.
  - If the tool should approach the workpiece opposite to the radius compensation: Enter R as a negative value.
- Center angle CCA of the arc
  - CCA can be entered only as a positive value.
  - Maximum input value 360°
- Radius compensation RR/RL for machining

Example NC blocks

`7 L X+40 Y+10 R0 FMAX M3`	Approach P_S without radius compensation
`8 APPR CT X+10 Y+20 Z-10 CCA180 R+10 RR F100`	P_A with radius comp. RR, radius R=10
`9 L X+20 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

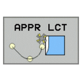
Approaching on a circular arc with tangential connection from a straight line to the contour: APPR LCT



The tool moves on a straight line from the starting point P_S to an auxiliary point P_H. It then moves to the first contour point P_A on a circular arc. The feed rate programmed in the APPR block is in effect.

The arc is connected tangentially both to the line $P_S – P_H$ as well as to the first contour element. Once these lines are known, the radius then suffices to completely define the tool path.

- Use any path function to approach the starting point P_S.
- Initiate the dialog with the APPR/DEP key and APPR LCT soft key:

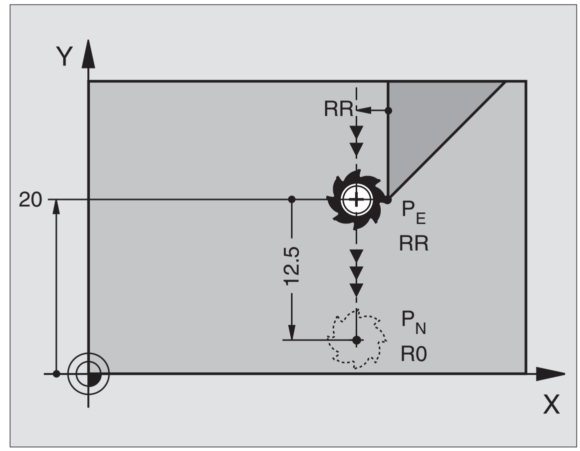


- Coordinates of the first contour point P_A
- Radius R of the circular arc. Enter R as a positive value.
- Radius compensation RR/RL for machining

Example NC blocks

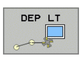
`7 L X+40 Y+10 R0 FMAX M3`	Approach P_S without radius compensation
`8 APPR LCT X+10 Y+20 Z-10 R10 RR F100`	P_A with radius comp. RR, radius R=10
`9 L X+20 Y+35`	End point of the first contour element
`10 L ...`	Next contour element

Departing on a straight line with tangential connection: DEP LT



The tool moves on a straight line from the last contour point P_E to the end point P_N. The line lies on the extension of the last contour element. P_N is separated from P_E by the distance LEN.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and DEP LT soft key:

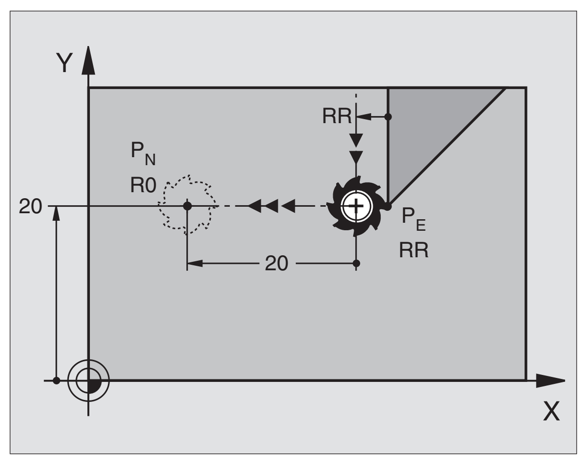


- LEN: Enter the distance from the last contour element P_E to the end point P_N.

Example NC blocks

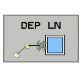
`23 L Y+20 RR F100`	Last contour element: P_E with radius compensation
`24 DEP LT LEN12.5 F100`	Depart contour by LEN=12.5 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

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



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

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and DEP LN soft key:

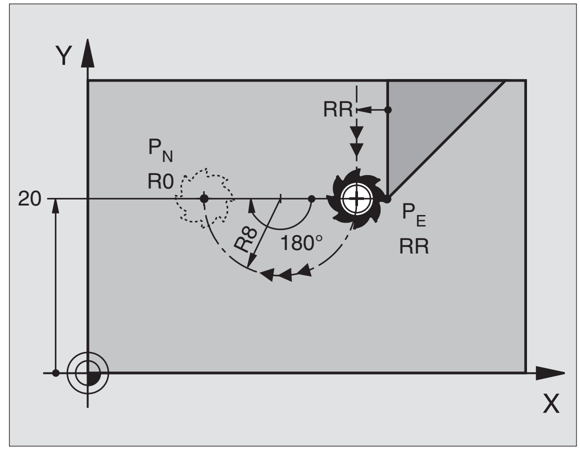


- LEN: Enter the distance from the last contour element to P_N.
  Always enter LEN as a positive value!

Example NC blocks

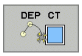
`23 L Y+20 RR F100`	Last contour element: P_E with radius compensation
`24 DEP LN LEN+20 F100`	Depart perpendicular to contour by LEN=20 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

Departure on a circular path with tangential connection: DEP CT



The tool moves on a circular arc from the last contour point P_E to the end point P_N. The 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 dialog with the APPR/DEP key and DEP CT soft key:

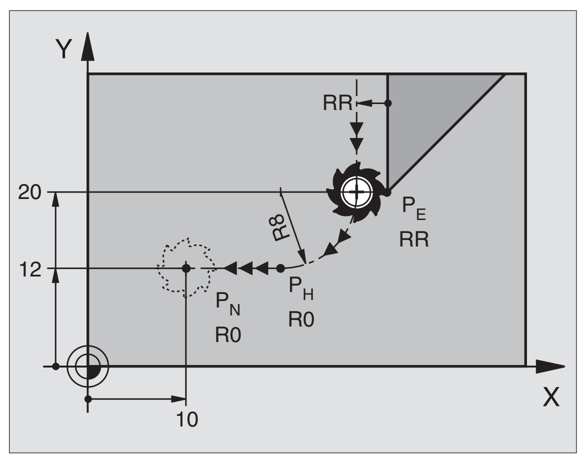


- Center angle CCA of the arc
- Radius R of the circular arc
  - If the tool should depart the workpiece in the direction of the radius compensation (i.e. to the right with RR or to the left with RL): Enter R as a positive value.
  - If the tool should depart the workpiece in the direction **opposite** to the radius compensation: Enter R as a negative value.

Example NC blocks

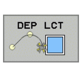
`23 L Y+20 RR F100`	Last contour element: P_E with radius compensation
`24 DEP CT CCA 180 R+8 F100`	Center angle=180°,
	arc radius=8 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

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



The tool moves on a circular arc from the last contour point P_S to an auxiliary point P_H. It then moves on a straight line to the end point P_N. The arc is tangentially connected both to the last contour element and to the line from P_H to P_N. Once these lines are known, the radius R then suffices to completely define the tool path.

- Program the last contour element with the end point P_E and radius compensation.
- Initiate the dialog with the APPR/DEP key and 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 F100`	Last contour element: P_E with radius compensation
`24 DEP LCT X+10 Y+12 R+8 F100`	Coordinates P_N, arc radius=8 mm
`25 L Z+100 FMAX M2`	Retract in Z, return to block 1, end program

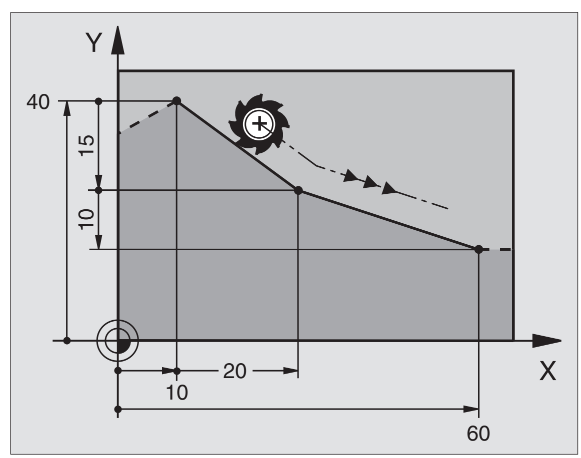
6.4 Path Contours—Cartesian Coordinates

Overview of path functions

Function	Path function key	Tool movement	Required input
Line **L**		Straight line	Coordinates of the end points of the straight line
Chamfer **CHF**		Chamfer between two straight lines	Chamfer side length
Circle Center **CC**		None	Coordinates of the circle center or pole
Circle **C**		Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation
Circular Arc **CR**		Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation
Circular Arc **CT**		Circular arc with tangential connection to the preceding and subsequent contour elements	Coordinates of the arc end point
Corner Rounding **RND**		Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding radius R
FK Free Contour Programming	FK	Straight line or circular path with any connection to the preceding contour element	see "Path Contours—FK Free Contour Programming," page 143

Straight Line L

The TNC moves the tool in 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`**

Example NC blocks

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

Actual position capture

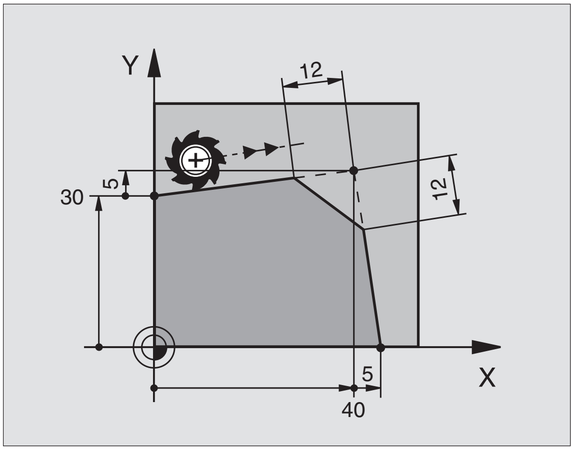
You can also generate a straight-line block (L block) by using the ACTUAL-POSITION-CAPTURE key:

- In the Manual Operation mode, move the tool to the position you wish to capture.
- Switch the screen display to Programming and Editing.
- Select the program block after which you want to insert the L block.
- Press the ACTUAL-POSITION-CAPTURE key: The TNC generates an L block with the actual position coordinates.

Inserting a Chamfer CHF between Two Straight Lines

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

- The blocks before and after the CHF block must be in the same working plane.
- The radius compensation before and after the chamfer block must be the same.
- An inside chamfer must be large enough to accommodate 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
```

You cannot start a contour with a CHF block.

A chamfer is possible only in the working plane.

The corner point is cut off by the chamfer and is not part of the contour.

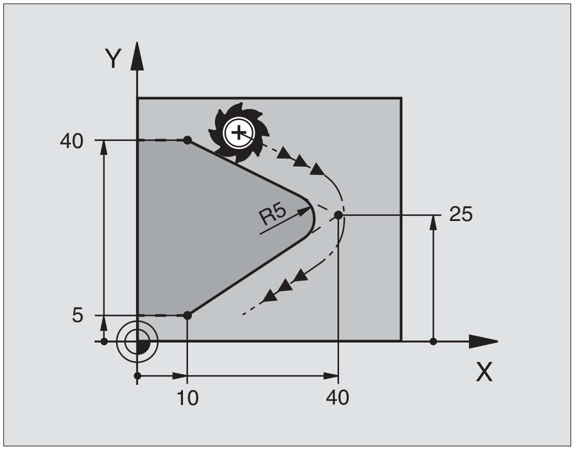
A feed rate programmed in the CHF block is effective only in that block. After the CHF block, the previous feed rate becomes effective again.

Corner Rounding RND

The RND function is used for rounding off corners.

The tool moves on an arc that is tangentially connected to both the preceding and subsequent contour elements.

The rounding arc must be large enough to accommodate the tool.



- **Rounding radius:** Enter the radius

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
```

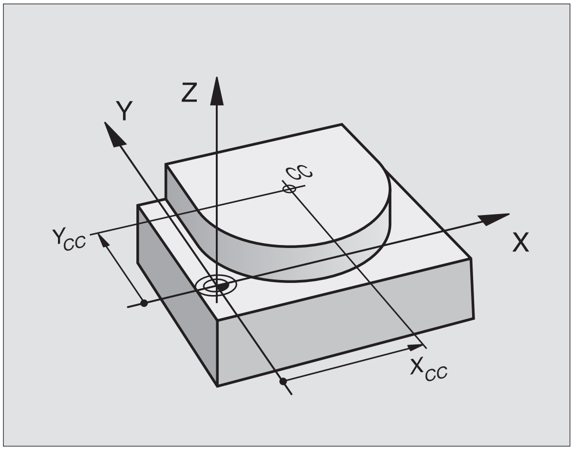
In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

The corner point is cut off by the rounding arc and is not part of the contour.

A feed rate programmed in the RND block is effective only in that block. After the RND block, the previous feed rate becomes effective again.

You can also use an RND block for a tangential contour approach if you do not want to use an APPR function.

Circle center CC



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

- Entering the Cartesian coordinates of the circle center, or
- Using the circle center defined in an earlier block, or
- Capturing 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.

Example NC blocks

```
5 CC X+25 Y+25
```

or

```
10 L X+25 Y+25
11 CC
```

The program blocks 10 and 11 do not refer to the illustration.

Duration of effect

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

Entering the circle center CC incrementally

If you enter the circle center with incremental coordinates, you have programmed it relative to the last programmed position of the tool.

> The only effect of CC is to define a position as circle center: The tool does not move to this position.
>
> The circle center is also the pole for polar coordinates.

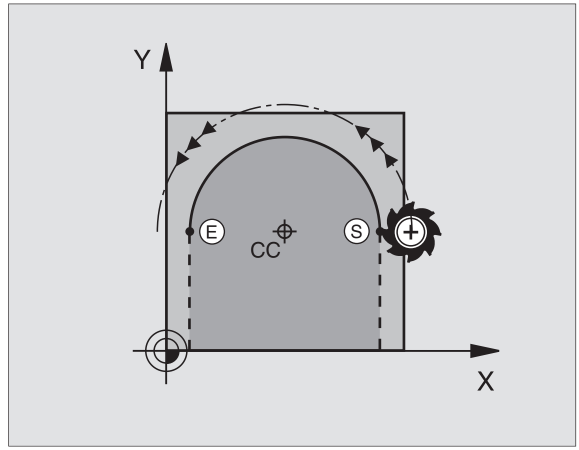
Circular path C around circle center CC

Before programming a circular path C, you must first enter the circle center CC. The last programmed tool position before the C block is used as the circle starting point.

- Move the tool to the circle starting point.

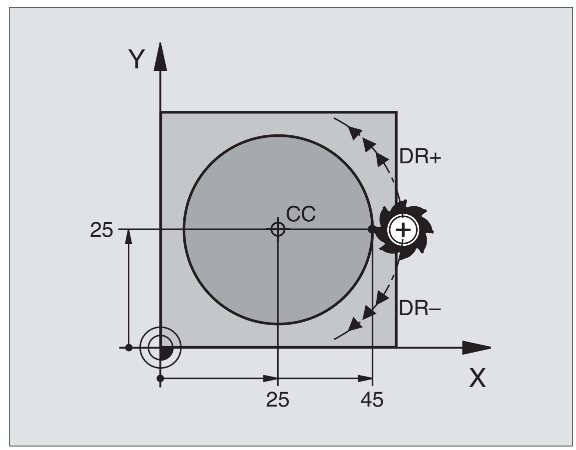
- **Coordinates** of the circle center
- **Coordinates** of the arc end point
- **Direction of rotation DR**

Further entries, if necessary:
- **Feed rate F**
- **Miscellaneous function M**



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+
```



Full circle

For the end point, enter the same point that you used for the starting point.

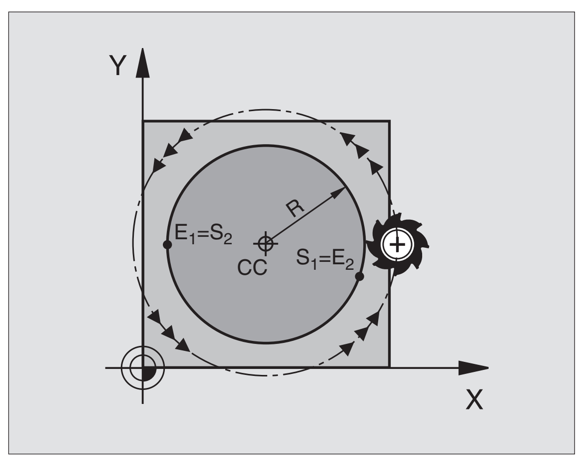
> The starting and end points of the arc must lie on the circle.
>
> Input tolerance: up to 0.016 mm (selected through the "circleDeviation" machine parameter).

Circular path CR with defined radius

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

- **Coordinates** of the arc end point
- **Radius R**
  Note: The algebraic sign determines the size of the 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 arc radius R

The starting and end points on the contour can be connected with four arcs of the same radius:

Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

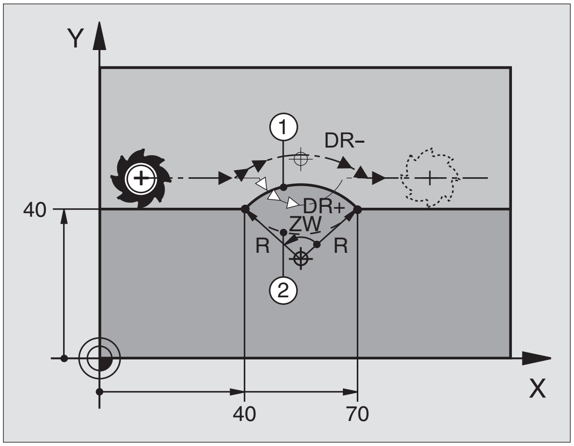
Larger arc: CCA>180°
Enter the radius with a negative sign R<0

The direction of rotation determines whether the arc is curving outward (convex) or curving inward (concave):

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

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

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)
```

or

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

or

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

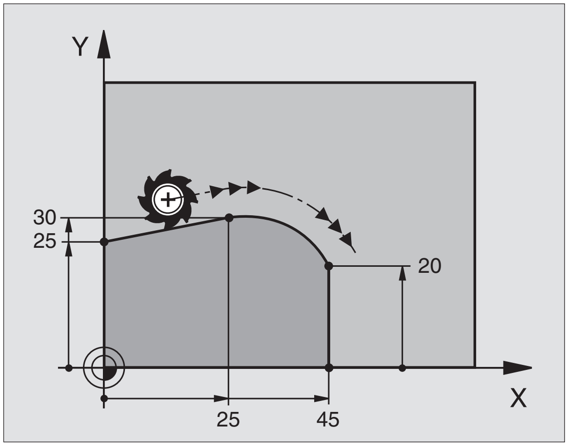
The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.

Circular Path CT with Tangential Connection

The tool moves on an arc that starts tangentially to the previously programmed contour element.

A transition between two contour elements is called tangential when there is no kink or corner at the intersection between the two contours—the transition is smooth.

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



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

Further entries, if necessary:

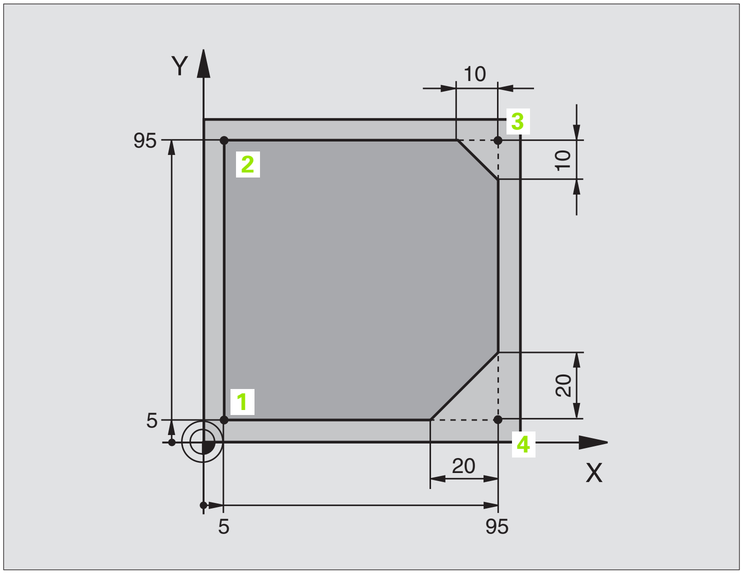
- **`Feed rate F`**
- **`Miscellaneous function M`**

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
```

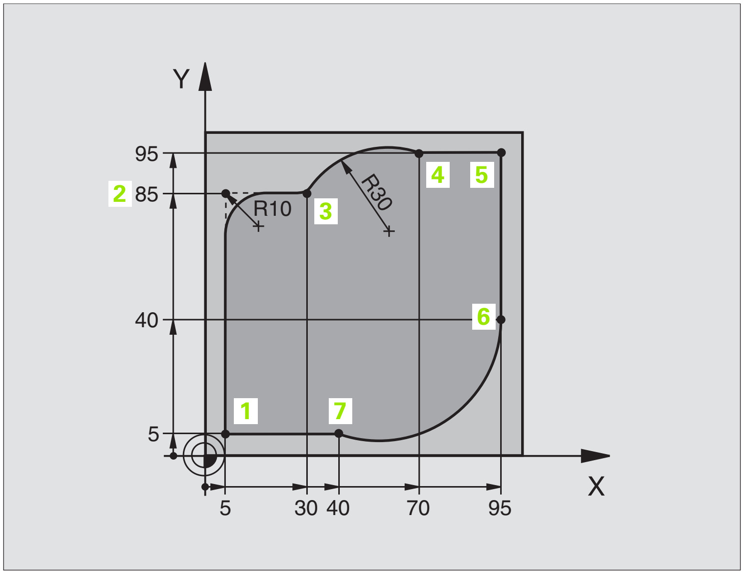
A tangential arc is a two-dimensional operation: the coordinates in the CT block and in the contour element preceding it must be in the same plane as the arc.

Example: Linear movements and chamfers with Cartesian coordinates



0 BEGIN PGM LINEAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+10	Define tool in the program
4 TOOL CALL 1 Z S4000	Call tool in the spindle axis and with the spindle speed S
5 L Z+250 R0 FMAX	Retract tool in the spindle axis at rapid traverse FMAX
6 L X-10 Y-10 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
8 APPR LT X+5 X+5 LEN10 RL F300	Approach the contour at point 1 on a straight line with
	tangential connection
9 L Y+95	Move to point 2
10 L X+95	Point 3: first straight line for corner 3
11 CHF 10	Program chamfer with length 10 mm
12 L Y+5	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
13 CHF 20	Program chamfer with length 20 mm
14 L X+5	Move to last contour point 1, second straight line for corner 4
15 DEP LT LEN10 F1000	Depart the contour on a straight line with tangential connection
16 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
17 END PGM LINEAR MM	

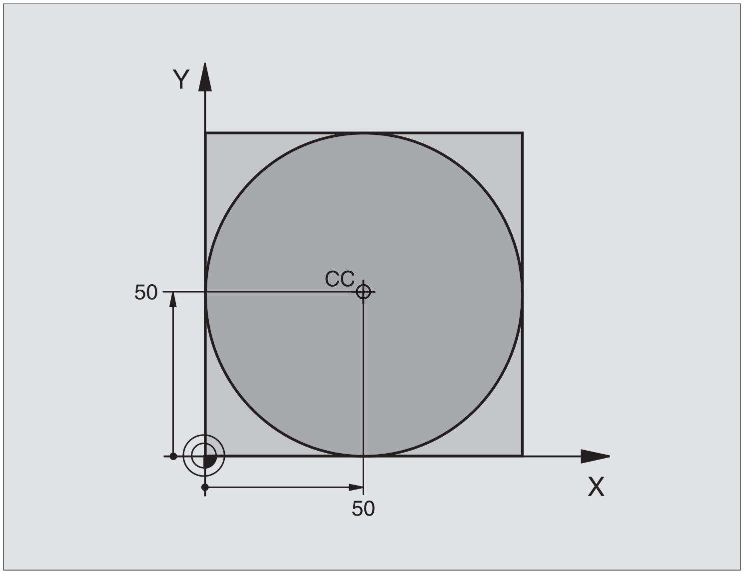
Example: Circular movements with Cartesian coordinates



0 BEGIN PGM CIRCULAR MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+10	Define tool in the program
4 TOOL CALL 1 Z X4000	Call tool in the spindle axis and with the spindle speed S
5 L Z+250 R0 FMAX	Retract tool in the spindle axis at rapid traverse FMAX
6 L X-10 Y-10 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
8 APPR LCT X+5 Y+5 R5 RL F300	Approach the contour at point 1 on a circular arc with
	tangential connection
9 L X+5 Y+85	Point 2: first straight line for corner 2
10 RND R10 F150	Insert radius with R = 10 mm, feed rate: 150 mm/min
11 L X+30 Y+85	Move to point 3: Starting point of the arc with CR
12 CR X+70 Y+95 R+30 DR-	Move to point 4: End point of the arc with CR, radius 30 mm
13 L X+95	Move to point 5
14 L X+95 Y+40	Move to point 6
15 CT X+40 Y+5	Move to point 7: End point of the arc, radius with tangential
	connection to point 6, TNC automatically calculates the radius

16 L X+5	Move to last contour point 1
17 DEP LCT X-20 Y-20 R5 F1000	Depart the contour on a circular arc with tangential connection
18 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
19 END PGM CIRCULAR MM	

Example: Full circle with Cartesian coordinates



`0 BEGIN PGM C-CC MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+12.5`	Define the tool
`4 TOOL CALL 1 Z S3150`	Tool call
`5 CC X+50 Y+50`	Define the circle center
`6 L Z+250 R0 FMAX`	Retract the tool
`7 L X-40 Y+50 R0 FMAX`	Pre-position the tool
`8 L Z-5 R0 F1000 M3`	Move to working depth
`9 APPR LCT X+0 Y+50 R5 RL F300`	Approach the starting point of the circle on a circular arc with
	Connection
`10 C X+0 DR-`	Move to the circle end point (= circle starting point)
`11 DEP LCT X-40 Y+50 R5 F1000`	Depart the contour on a circular arc with tangential
	Connection
`12 L Z+250 R0 FMAX M2`	Retract in the tool axis, end program
`13 END PGM C-CC MM`	

6.5 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 “Fundamentals,” page 143).

Polar coordinates are useful with:

- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

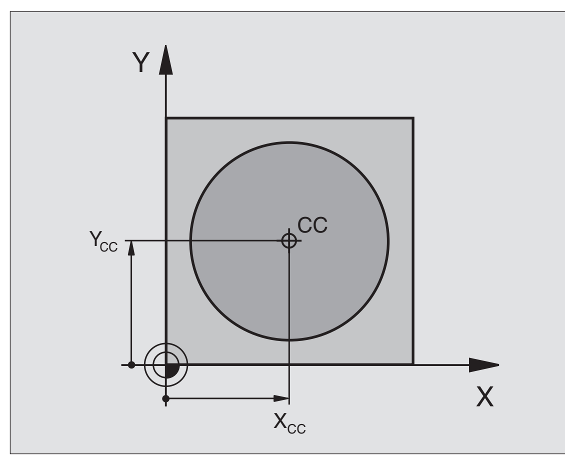
Function	Path function key	Tool movement	Required input
Line **LP**	L + P	Straight line	Polar radius, polar angle of the straight-line end point
Circular Arc **CP**	C + P	Circular path around circle center/ pole CC to arc end point	Polar angle of the arc end point, direction of rotation
Circular Arc **CTP**	CT + P	Circular arc with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helical interpolation	C + 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 blocks containing polar coordinates. Enter the pole in Cartesian coordinates as a circle center in a CC block.

CC

▶ **Coordinates** CC: Enter Cartesian coordinates for the pole, or
If you want to use the last programmed position, do not enter any coordinates. Before programming polar coordinates, define the pole CC. You can only define the pole CC in Cartesian coordinates. The pole CC remains in effect until you define a new pole CC.

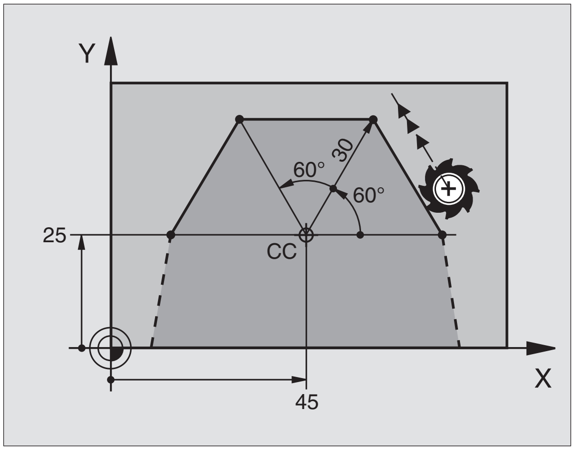


Example NC blocks

```
12 CC X+45 Y+25
```

Straight line LP

The tool moves in 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 coordinates radius PR:`** Enter the distance from the pole CC to the straight-line end point.
- **`Polar coordinates angle PA:`** Angular position of the straight-line end point between –360° and +360°.

The sign of PA depends on the angle reference axis:

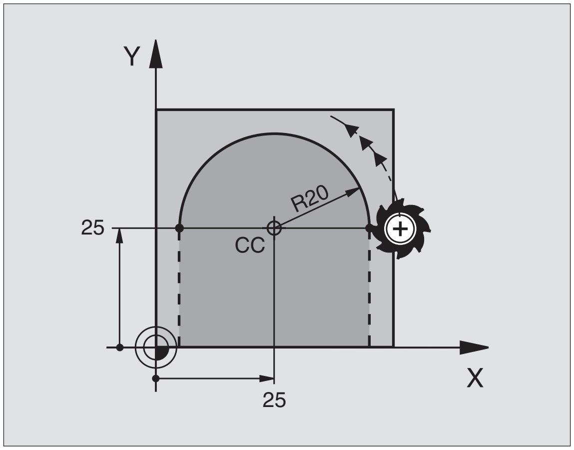
- Angle from angle reference axis to PR is counterclockwise: PA>0
- Angle from angle reference axis to PR is clockwise: PA<0

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 arc. It is defined by the distance from the starting point to the pole CC. The last programmed tool position before the CP block is the starting point of the arc.



- **`Polar-coordinates angle PA:`** Angular position of the arc end point between –5400° and +5400°
- **`Direction of rotation DR`**

Example NC blocks

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

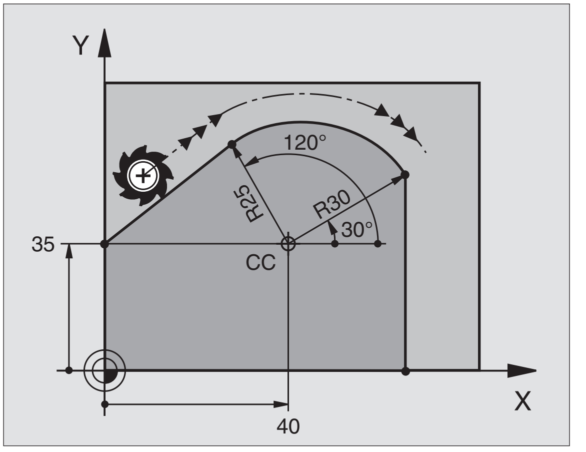
For incremental coordinates, enter the same sign for DR and PA.

Circular Path CTP with Tangential Connection

The tool moves on a circular path, starting tangentially from a preceding contour element.

CT P

- **`Polar coordinates radius PR:`** Distance from the arc end point to the pole CC
- **`Polar coordinates angle PA:`** Angular position of the arc end point



Example NC blocks

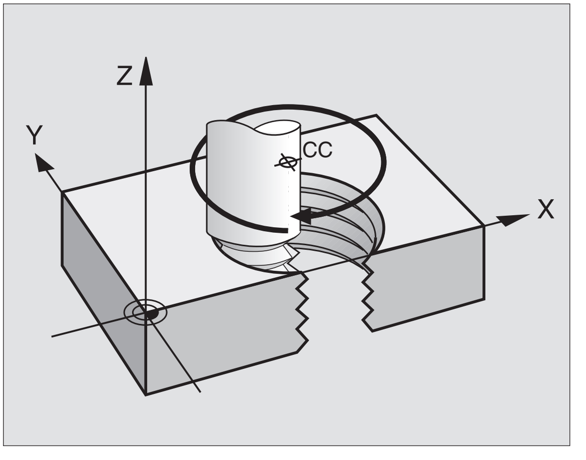
```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```

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

Helical interpolation

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

A helix is programmed only in polar coordinates.



Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, 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 thread beginning and end
Total height h	Thread pitch P times thread revolutions n
Incremental total angle IPA	Number of revolutions times 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Pitch P times (thread revolutions + thread overrun at start of thread)

Shape of the helix

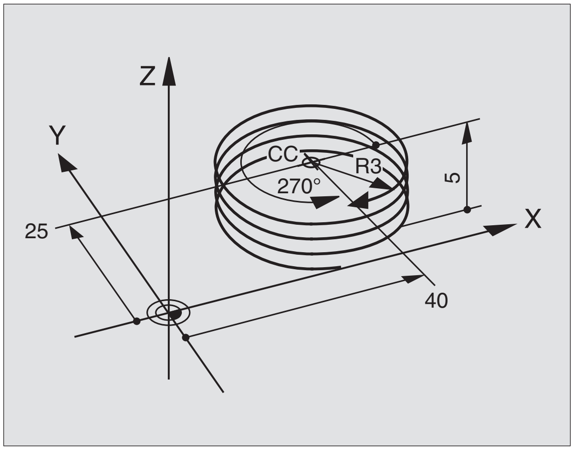
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direction	Direction	Radius comp.
Right-handed Left-handed	Z+ Z+	DR+ DR–	RL RR
Right-handed Left-handed	Z– Z–	DR– DR+	RR RL
External thread			
Right-handed Left-handed	Z+ Z+	DR+ DR–	RR RL
Right-handed Left-handed	Z– Z–	DR– DR+	RL RR

Programming a helix

Always enter the same algebraic sign for the direction of rotation DR and the incremental total angle IPA. The tool may otherwise move in a wrong path and damage the contour.

For the total angle IPA, you can enter a value from –5400° to +5400°. If the thread has more than 15 revolutions, program the helix in a program section repeat (see "Program Section Repeats," page 300).



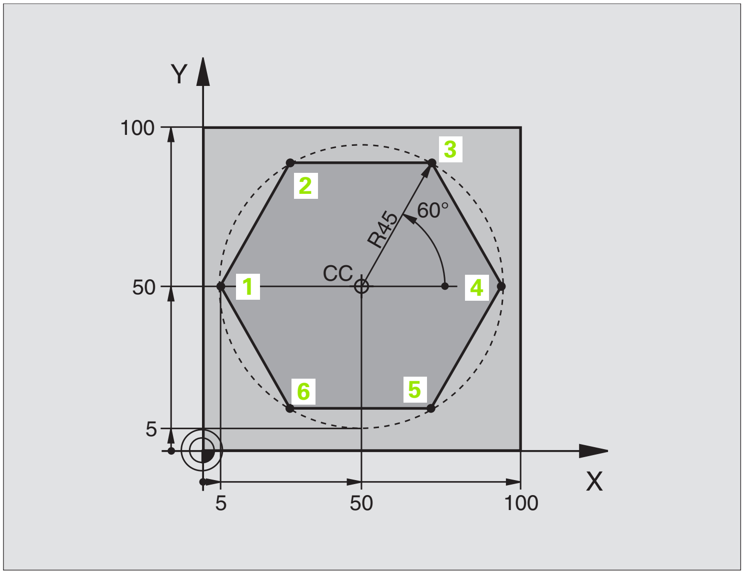
P

- **`Polar coordinates angle:`** Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, specify 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+

Example NC blocks: Thread M6 x 1 mm with 5 revolutions

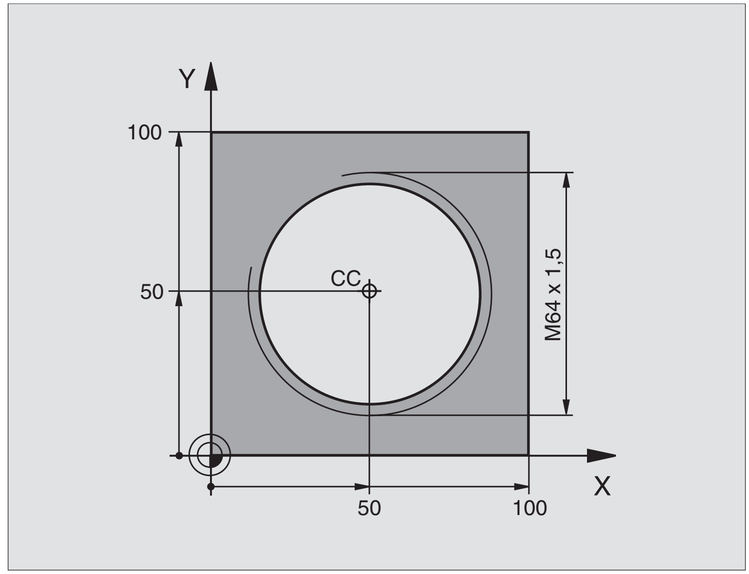
```
12 CC X+40 Y+25
13 L Z+0 F100 M3
14 LP PR+3 PA+270 RL F50
15 CP IPA-1800 IZ+5 DR-
```

Example: Linear movement with polar coordinates



0 BEGIN PGM LINEARPO MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+7.5	Define the tool
4 TOOL CALL 1 Z S4000	Tool call
5 CC X+50 Y+50	Define the datum for polar coordinates
6 L Z+250 R0 FMAX	Retract the tool
7 LP PR+60 PA+180 R0 FMAX	Pre-position the tool
8 L Z-5 R0 F1000 M3	Move to working depth
9 APPR PLCT PR+45 PA+180 R5 RL F250	Approach the contour at point 1 on a circular arc with
	tangential connection
10 LP PA+120	Move to point 2
11 LP PA+60	Move to point 3
12 LP PA+0	Move to point 4
13 LP PA-60	Move to point 5
14 LP PA-120	Move to point 6
15 LP PA+180	Move to point 1
16 DEP PLCT PR+60 PA+180 R5 F1000	Depart the contour on a circular arc with tangential connection
17 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
18 END PGM LINEARPO MM	

Example: Helix



`0 BEGIN PGM HELIX MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+5`	Define the tool
`4 TOOL CALL 1 Z S1400`	Tool call
`5 L Z+250 R0 FMAX`	Retract the tool
`6 L X+50 Y+50 R0 FMAX`	Pre-position the tool
`7 CC`	Transfer the last programmed position as the pole
`8 L Z-12.75 R0 F1000 M3`	Move to working depth
`9 APPR PCT PR+32 PA-182 CCA180 R+2 RL F100`	Approach the contour on a circular arc with tangential connection
`10 CP IPA+3240 IZ+13.5 DR+ F200`	Helical interpolation
`11 DEP CT CCA180 R+2`	Depart the contour on a circular arc with tangential connection
`12 L Z+250 R0 FMAX M2`	Retract in the tool axis, end program
`13 END PGM HELIX MM`	

To cut a thread with more than 16 revolutions

`...`	
`8 L Z-12.75 R0 F1000`	
`9 APPR PCT PR+32 PA-180 CCA180 R+2 RL F100`	
`10 LBL 1`	Identify beginning of program section repeat
`11 CP IPA+360 IZ+1.5 DR+ F200`	Enter the thread pitch as an incremental IZ dimension

`12 CALL LBL 1 REP 24`	Program the number of repeats (thread revolutions)
`13 DEP CT CCA180 R+2`	
`...`	

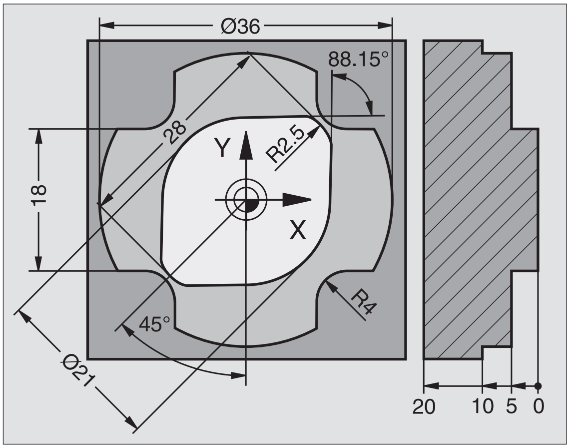
6.6 Path Contours—FK Free Contour Programming

Fundamentals

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

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

You can enter such dimensional data directly by using the FK free contour programming function. The TNC derives the contour from the known coordinate data and supports the programming dialog with the interactive programming graphics. The figure to the upper right shows a workpiece drawing for which FK programming is the most convenient programming method.



The following prerequisites for FK programming must be observed:

The FK free contour programming feature can only be used for programming contour elements that lie in the working plane. The working plane is defined in the first BLK FORM block of the part program.

You must enter all available data for every contour element. Even the data that does not change must be entered in every block—otherwise it will not be recognized.

Q parameters are permissible in all FK elements, except in elements with relative references (e.g. RX or RAN), or in elements that are referenced to other NC blocks.

If both FK blocks and conventional blocks are entered in a program, the FK contour must be fully defined before you can return to conventional programming.

The TNC needs a fixed point from which it can calculate the contour elements. Use the gray path function keys to program a position that contains both coordinates of the working plane immediately before programming the FK contour. Do not enter any Q parameters in this block.

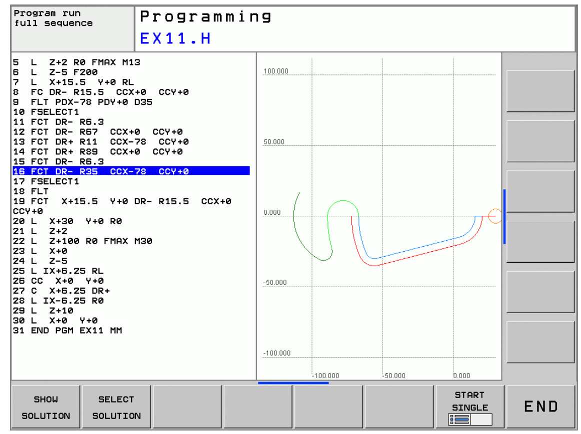
If the first block of an FK contour is an FCT or FLT block, you must program at least two NC blocks with the gray path function keys to fully define the direction of contour approach.

Do not program an FK contour immediately after an LBL label.

Create FK programs for TNC 4xx:

For a TNC 4xx to be able to read-in FK programs created on an TNC 320, the individual FK elements within a block must be in the same sequence as displayed in the soft-key row.

Graphics during FK programming



If you wish to use graphic support during FK programming, select the PROGRAM + GRAPHICS screen layout (see “Programming and editing” on page 31).

Incomplete coordinate data often are not sufficient to fully define a workpiece contour. In this case, the TNC 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 colors:

White The contour element is fully defined.

Green The entered data describe a limited number of possible solutions: select the correct one.

Red The entered data are not sufficient to determine the contour element: enter further data.

If the entered data permit a limited number of possible solutions and the contour element is displayed in green, select the correct contour element as follows:

SHOW SOLUTION

- Press the SHOW SOLUTION soft key repeatedly until the correct contour element is displayed. Use the zoom function (2nd soft-key row) if you cannot distinguish possible solutions in the standard setting.

SELECT SOLUTION

- If the displayed contour element matches the drawing, select the contour element with SELECT SOLUTION.

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

Select the green contour elements as soon as possible with the SELECT SOLUTION soft key. This way you can reduce the ambiguity of subsequent elements.

The machine tool builder may use other colors for the FK graphics.

NC blocks from a program that you called with PGM CALL are displayed in another color.

Show block number in graphic window

To show a block number in the graphic window:

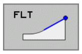
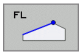
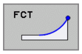
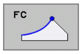
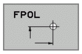
SHOW OMIT BLOCK NR.

- Set the SHOW OMIT BLOCK NR. soft key to SHOW

Initiating the FK dialog

If you press the gray FK button, the TNC displays the soft keys you can use to initiate an FK dialog: See the following table. Press the FK button a second time to deselect the soft keys.

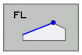
If you initiate the FK dialog with one of these soft keys, the TNC shows additional soft-key rows that you can use for entering known coordinates, directional data and data regarding the course of the contour.

Contour element	Soft key
Straight line with tangential connection	
Straight line without tangential connection	
Circular arc with tangential connection	
Circular arc without tangential connection	
Pole for FK programming	

Free programming of straight lines

Straight line without tangential connection

- To display the soft keys for free contour programming, press the FK key.

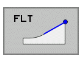


- To initiate the dialog for free programming of straight lines, press the FL soft key. The TNC displays additional soft keys.
- Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green (see "Graphics during FK programming," page 144).

Straight line with tangential connection

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

- To display the soft keys for free contour programming, press the FK key.

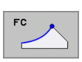


- To initiate the dialog, press the FLT soft key.
- Enter all known data in the block by using the soft keys.

Free programming of circular arcs

Circular arc without tangential connection

- To display the soft keys for free contour programming, press the FK key.

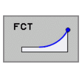


- To initiate the dialog for free programming of circular arcs, press the FC soft key. The TNC displays soft keys with which you can enter direct data on the circular arc or data on the circle center.
- Enter all known data in the block by using these soft keys. The FK graphic displays the programmed contour element in red until sufficient data is entered. If the entered data describes several solutions, the graphic will display the contour element in green (see "Graphics during FK programming," page 144).

Circular arc with tangential connection

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

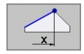
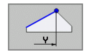
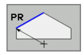
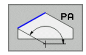
- To display the soft keys for free contour programming, press the FK key.

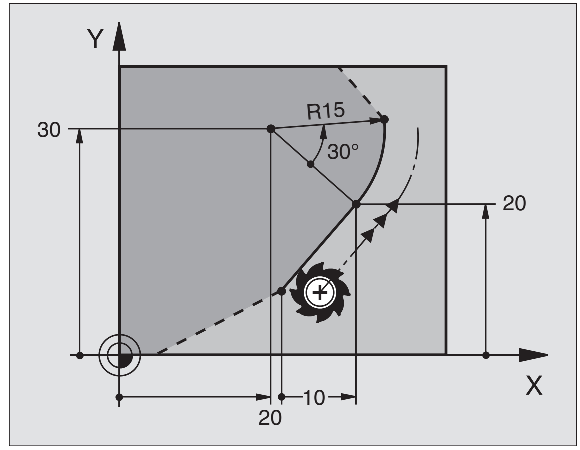


- To initiate the dialog, press the FCT soft key.
- Enter all known data in the block by using the soft keys.

Input possibilities

End point coordinates

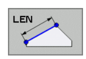
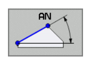
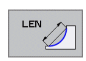
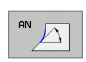
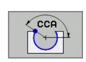
Known data	Soft keys	
Cartesian coordinates X and Y		
Polar coordinates referenced to FPOL		

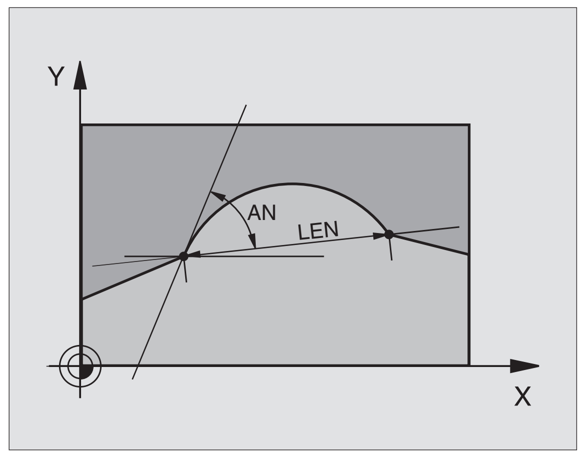


Example NC blocks

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

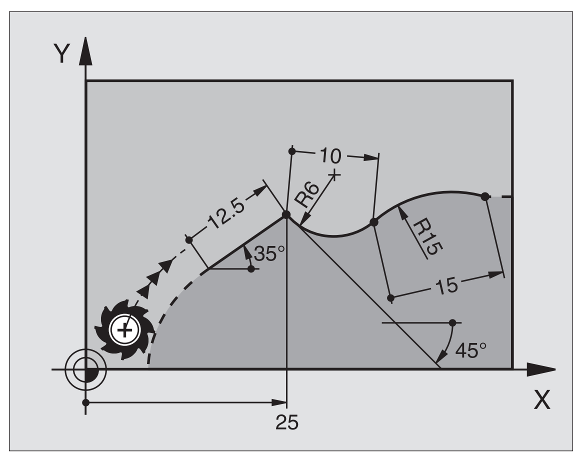
Direction and length of contour elements

Known data	Soft keys
Length of a straight line	
Gradient angle of a straight line	
Chord length LEN of the arc	
Gradient angle AN of the entry tangent	
Center angle of the arc	



Example NC blocks

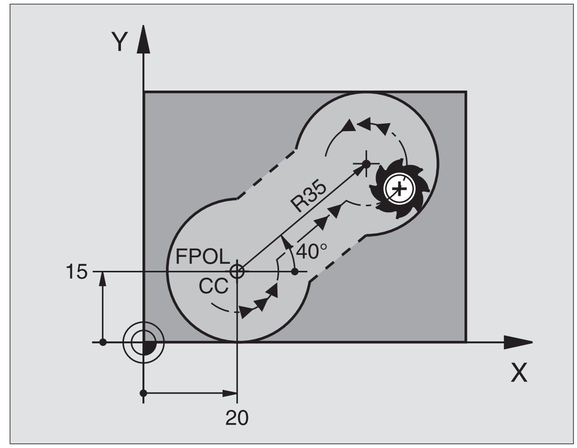
```
27 FLT X+25 LEN 12.5 AN+35 RL F200
28 FC DR+ R6 LEN 10 A-45
29 FCT DR- R15 LEN 15
```



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

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

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



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

Known data	Soft keys	
Circle center in Cartesian coordinates	CCX	CCY
Circle center in polar coordinates	CC PR	CC PA
Rotational direction of the arc	DR− DR+	
Radius of the arc	R	

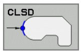
Example NC blocks

```
10 FC CCX+20 CCY+15 DR+ R15
11 FPOL X+20 Y+15
12 FL AN+40
13 FC DR+ R15 CCPR+35 CCPA+40
```

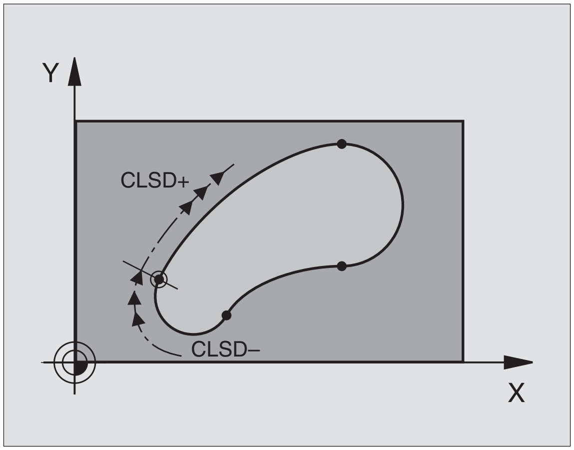
Closed contours

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

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



Beginning of contour:	CLSD+
End of contour:	CLSD–



Example NC blocks

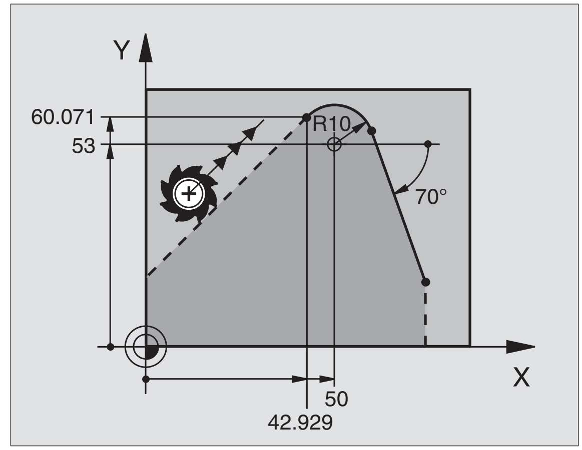
```
12 L X+5 Y+35 RL F500 M3
13 FC DR- R15 CLSD+ CCX+20 CCY+35
...
17 FCT DR- R+15 CLSD-
```

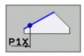
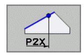
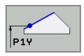
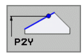
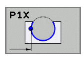
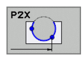
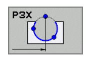
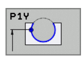
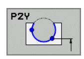
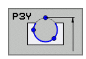
Auxiliary points

You can enter the coordinates of auxiliary points that are located on the contour or in its proximity for both free-programmed straight lines and free-programmed circular arcs.

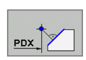
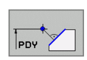
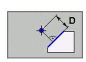
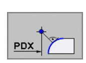
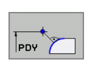
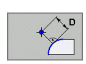
Auxiliary points on a contour

The auxiliary points are located on a straight line or on the extension of a straight line, or on a circular arc.



Known data	Soft keys		
X coordinate of an auxiliary point P1 or P2 of a straight line			
Y coordinate of an auxiliary point P1 or P2 of a straight line			
X coordinate of an auxiliary point P1, P2 or P3 of a circular arc			
Y coordinate of an auxiliary point P1, P2 or P3 of a circular arc			

Auxiliary points near a contour

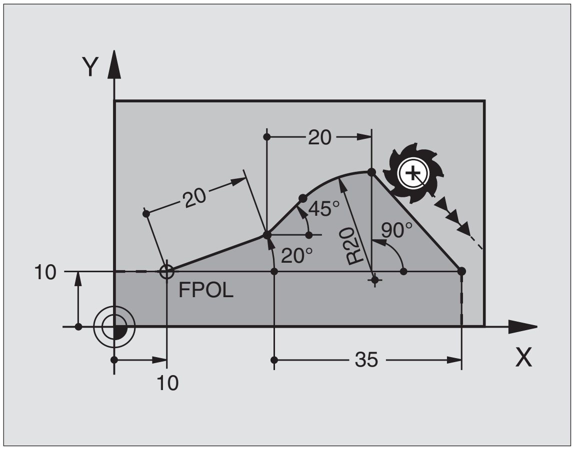
Known data	Soft keys	
X and Y coordinates of an auxiliary point near a straight line		
Distance auxiliary point/straight line		
X and Y coordinates of an auxiliary point near a circular arc		
Distance auxiliary point/circular arc		

Example NC blocks

```
13 FC DR- R10 P1X+42.929 P1Y+60.071
14 FLT AN-70 PDX+50 PDY+53 D10
```

Relative data

Data whose values are based on another contour element are called relative data. The soft keys and program words for entries begin with the letter **"R"** for **R**elative. The figure at right shows the entries that should be programmed as relative data.



> The coordinates and angles for relative data are always programmed in incremental dimensions. You must also enter the block number of the contour element on which the data are based.
>
> The block number of the contour element on which the relative data are based can only be located up to 64 positioning blocks before the block in which you program the reference.
>
> If you delete a block on which relative data are based, the TNC will display an error message. Change the program first before you delete the block.

Data relative to block N: End point coordinates

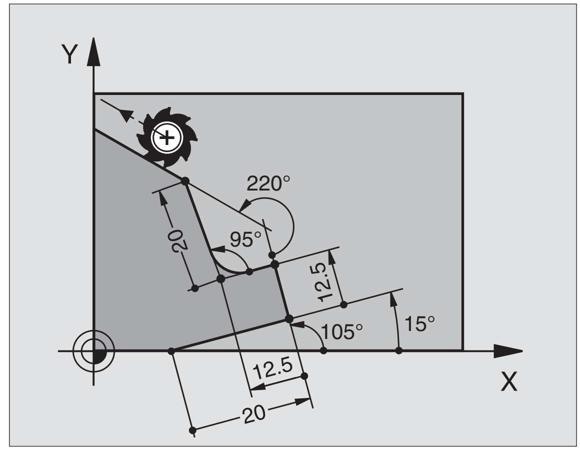
Known data	Soft keys	
Cartesian Coordinates relative to block N	RX N...	RY N...
Polar coordinates relative to block N	RPR N...	RPA N...

Example NC blocks

```
12 FPOL X+10 Y+10
13 FL PR+20 PA+20
14 FL AN+45
15 FCT IX+20 DR- R20 CCA+90 RX 13
16 FL IPR+35 PA+0 RPR 13
```

Data relative to block N: Direction and distance of the contour element

Known data	Soft key
Angle between a straight line and another element or between the entry tangent of the arc and another element	RAN N...
Straight line parallel to another contour element	PAR N...
Distance from a straight line to a parallel contour element	DP

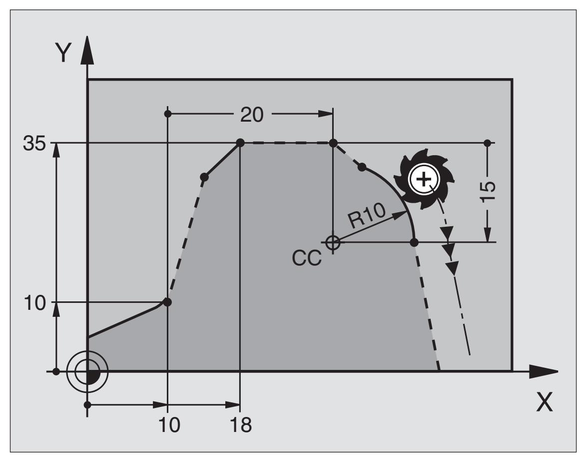


Example NC blocks

```
17 FL LEN 20 AN+15
18 FL AN+105 LEN 12.5
19 FL PAR 17 DP 12.5
20 FSELECT 2
21 FL LEN 20 IAN+95
22 FL IAN+220 RAN 18
```

Data relative to block N: Circle center CC

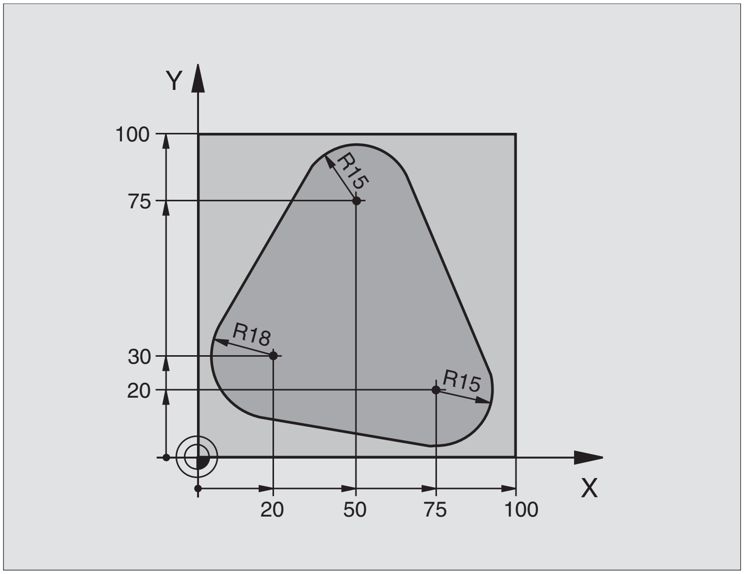
Known data	Soft key	
Cartesian coordinates of the circle center relative to block N	RCCX N...	RCCY N...
Polar coordinates of the circle center relative to block N	RCCPR N...	RCCPA N...



Example NC blocks

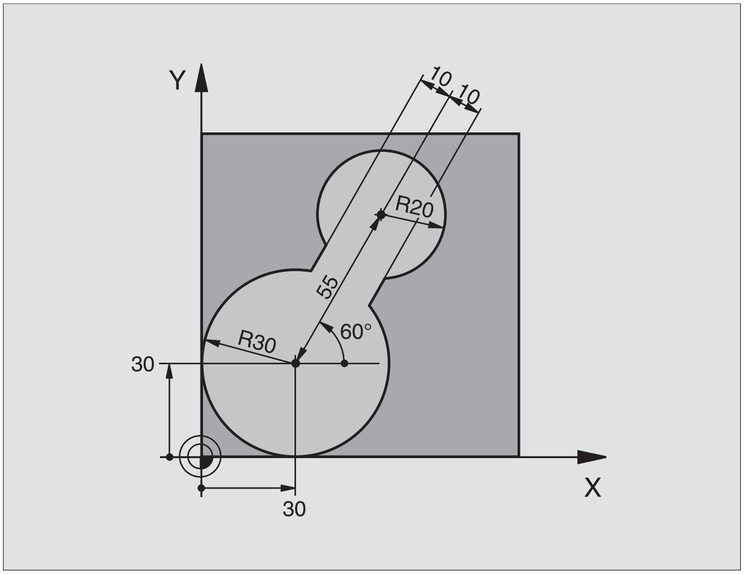
```
12 FL X+10 Y+10 RL
13 FL ...
14 FL X+18 Y+35
15 FL ...
16 FL ...
17 FC DR- R10 CCA+0 ICCX+20 ICCY-15 RCCX12 RCCY14
```

Example: FK programming 1



0 BEGIN PGM FK1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+10	Define the tool
4 TOOL CALL 1 Z S500	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 L X-20 Y+30 R0 FMAX	Pre-position the tool
7 L Z-10 R0 F1000 M3	Move to working depth
8 APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
9 FC DR- R18 CLSD+ CCX+20 CCY+30	FK contour section:
10 FLT	Program all known data for each contour element
11 FCT DR- R15 CCX+50 CCY+75	
12 FLT	
13 FCT DR- R15 CCX+75 CCY+20	
14 FLT	
15 FCT DR- R18 CLSD- CCX+20 CCY+30	
16 DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection
17 L X-30 Y+0 R0 FMAX	
18 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
19 END PGM FK1 MM	

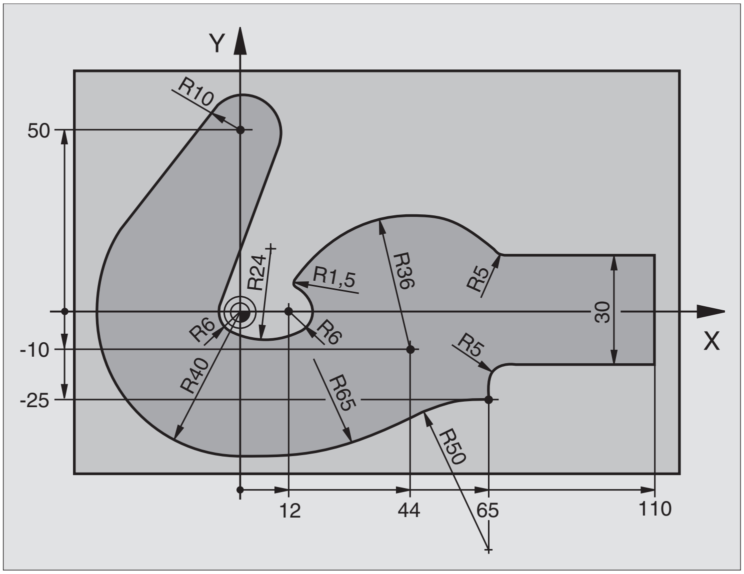
Example: FK programming 2



0 BEGIN PGM FK2 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+2	Define the tool
4 TOOL CALL 1 Z S4000	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 L X+30 Y+30 R0 FMAX	Pre-position the tool
7 L Z+5 R0 FMAX M3	Pre-position the tool in the tool axis
8 L Z-5 R0 F100	Move to working depth

9 APPR LCT X+0 Y+30 R5 RR F350	Approach the contour on a circular arc with tangential connection
10 FPOL X+30 Y+30	FK contour section:
11 FC DR- R30 CCX+30 CCY+30	Program all known data for each contour element
12 FL AN+60 PDX+30 PDY+30 D10	
13 FSELECT 3	
14 FC DR- R20 CCPR+55 CCPA+60	
15 FSELECT 2	
16 FL AN-120 PDX+30 PDY+30 D10	
17 FSELECT 3	
18 FC X+0 DR- R30 CCX+30 CCY+30	
19 FSELECT 2	
20 DEP LCT X+30 Y+30 R5	Depart the contour on a circular arc with tangential connection
21 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
22 END PGM FK2 MM	

Example: FK programming 3



0 BEGIN PGM FK3 MM	
1 BLK FORM 0.1 Z X-45 Y-45 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+120 Y+70 Z+0	
3 TOOL DEF 1 L+0 R+3	Define the tool
4 TOOL CALL 1 Z S4500	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 L X-70 Y+0 R0 FMAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth

8 APPR CT X-40 Y+0 CCA90 R+5 RL F250	Approach the contour on a circular arc with tangential connection
9 FC DR- R40 CCX+0 CCY+0	FK contour section:
10 FLT	Program all known data for each contour element
11 FCT DR- R10 CCX+0 CCY+50	
12 FLT	
13 FCT DR+ R6 CCX+0 CCY+0	
14 FCT DR+ R24	
15 FCT DR+ R6 CCX+12 CCY+0	
16 FSELECT 2	
17 FCT DR- R1.5	
18 FCT DR- R36 CCX+44 CCY-10	
19 FSELECT 2	
20 FCT CT+ R5	
21 FLT X+110 Y+15 AN+0	
22 FL AN-90	
23 FL X+65 AN+180 PAR21 DP30	
24 RND R5	
25 FL X+65 Y-25 AN-90	
26 FC DR+ R50 CCX+65 CCY-75	
27 FCT DR- R65	
28 FSELECT	
29 FCT Y+0 DR- R40 CCX+0 CCY+0	
30 FSELECT 4	
31 DEP CT CCA90 R+5 F1000	Depart the contour on a circular arc with tangential connection
32 L X-70 R0 FMAX	
33 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
34 END PGM FK3 MM	

7

Programming: Miscellaneous Functions

7.1 Entering Miscellaneous Functions M and STOP

Fundamentals

With the TNC's miscellaneous functions—also called M functions—you can affect:

- Program run, e.g., a program interruption
- Machine functions, such as switching spindle rotation and coolant supply on and off
- Contouring behavior of the tool

The machine tool builder may add some M functions that are not described in this User's Manual. Also, the machine tool builder can change the meaning and effect of the M functions described here. Refer to your machine manual.

You can enter up to two M functions at the end of a positioning block or in a separate block. The TNC displays the following dialog question: **`Miscellaneous function M ?`**

You usually enter only the number of the M function in the programming dialog. Some M functions can be programmed with additional parameters. In this case, the dialog is continued for the parameter input.

In the Manual Operation and Electronic Handwheel modes of operation, the M functions are entered with the M soft key.

Please note that some M functions become effective at the start of a positioning block, and others at the end, regardless of their position in the NC block.

M functions come into effect in the block in which they are called.

Some M functions are effective only in the block in which they are programmed. Unless the M function is only effective blockwise, either you must cancel it in a subsequent block with a separate M function, or it is automatically canceled by the TNC at the end of the program.

Entering an M function in a STOP block

If you program a STOP block, the program run or test run is interrupted at the block, for example for tool inspection. You can also enter an M function in a STOP block:

STOP

- To program an interruption of program run, press the STOP key.
- Enter miscellaneous function M.

Example NC blocks

```
87 STOP M6
```

7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

Overview

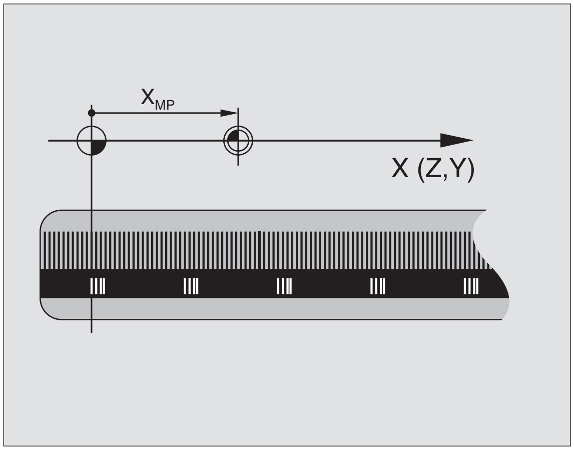
M	Effect	Effective at block	Start	End
M00	Stop program run Spindle STOP Coolant OFF			■
M01	Optional program STOP			■
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (dependent on the **`clearMode`** machine parameter)			■
M03	Spindle ON clockwise		■	
M04	Spindle ON counterclockwise		■	
M05	Spindle STOP			■
M06	Tool change (machine-dependent function) spindle STOP Stop program run			■
M08	Coolant ON		■	
M09	Coolant OFF			■
M13	Spindle ON clockwise Coolant ON		■	
M14	Spindle ON counterclockwise Coolant ON		■	
M30	Same as M02			■

7.3 Programming machine-referenced coordinates: M91/ M92

Programming machine-referenced coordinates: M91/M92

Scale reference point

On the scale, a reference mark indicates the position of the scale reference point.



Machine datum

The machine datum is required for the following tasks:

- Defining the limits of traverse (software limit switches)
- Moving to machine-referenced positions (such as tool change positions)
- Setting the workpiece datum

The distance in each axis from the scale reference point to the machine datum is defined by the machine tool builder in a machine parameter.

Standard behavior

The TNC references coordinates to the workpiece datum, see “Datum Setting (Without a 3-D Touch Probe),” page 47.

Behavior with M91—Machine datum

If you want the coordinates in a positioning block to be referenced to the machine datum, end the block with M91.

> If you program incremental coordinates in an M91 block, enter them with respect to the last programmed M91 position. If no M91 position is programmed in the active NC block, then enter the coordinates with respect to the current tool position.

The coordinate values on the TNC screen are referenced to the machine datum. Switch the display of coordinates in the status display to REF (see “Status Displays,” page 33).

Behavior with M92—Additional machine datum

In addition to the machine datum, the machine tool builder can also define an additional machine-based position as a reference point.

For each axis, the machine tool builder defines the distance between the machine datum and this additional machine datum. Refer to the machine manual for more information.

If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

Radius compensation remains the same in blocks that are programmed with M91 or M92. The tool length, however, is **not** compensated.

Effect

M91 and M92 are effective only in the blocks in which they are programmed.

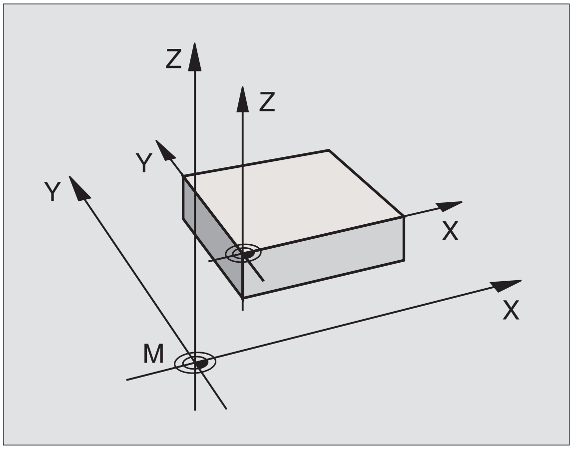
M91 and M92 take effect at the start of block.

Workpiece datum

If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes.

If datum setting is inhibited for all axes, the TNC no longer displays the soft key DATUM SET in the Manual Operation mode.

The figure shows coordinate systems with the machine datum and workpiece datum.



M91/M92 in the Test Run mode

In order to be able to graphically simulate M91/M92 movements, you need to activate working space monitoring and display the workpiece blank referenced to the set datum (see “Showing the Workpiece in the Working Space,” page 383).

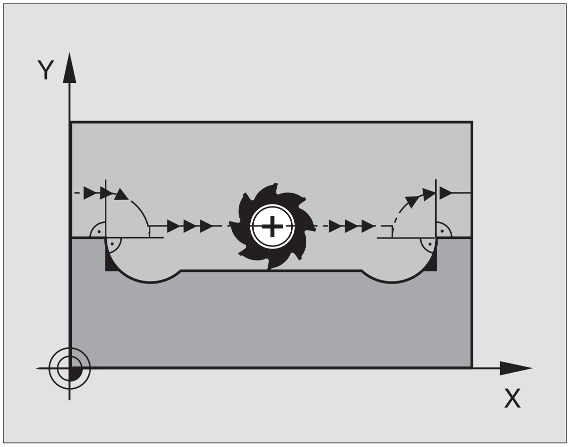
7.4 Miscellaneous Functions for Contouring Behavior

Machining small contour steps: M97

Standard behavior

The TNC inserts a transition arc at outside corners. If the contour steps are very small, however, the tool would damage the contour.

In such cases the TNC interrupts program run and generates the error message "Tool radius too large."



Behavior with M97

The TNC calculates the intersection of the contour elements—as at inside corners—and moves the tool over this point.

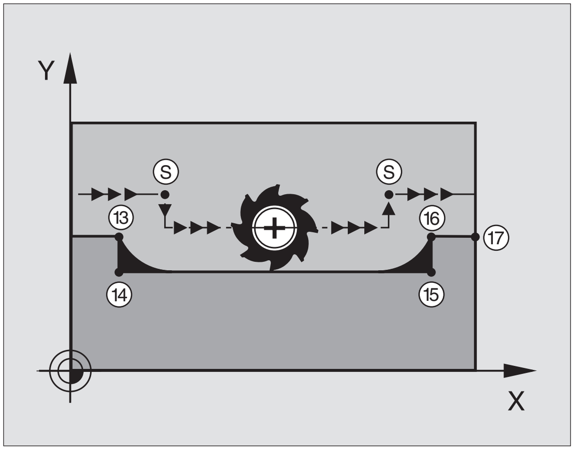
Program M97 in the same block as the outside corner.

> Instead of **M97** you should use the much more powerful function **M120 LA** (see "Behavior with M120" on page 168)!

Effect

M97 is effective only in the blocks in which it is programmed.

> A corner machined with M97 will not be completely finished. You may wish to rework the contour with a smaller tool.



Example NC blocks

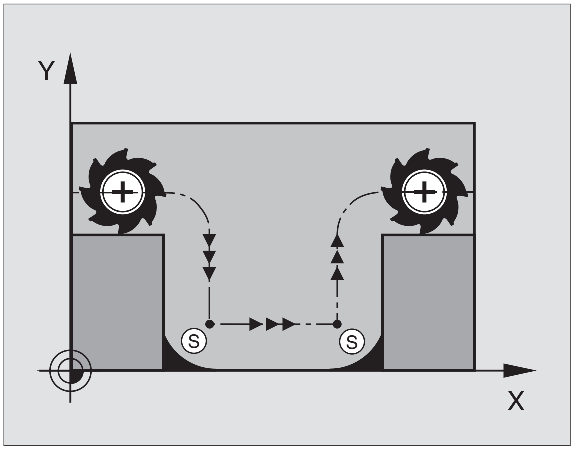
`5 TOOL DEF L ... R+20`	Large tool radius
`...`	
`13 L X... Y... R... F... M97`	Move to contour point 13
`14 L IY-0.5 ... R... F...`	Machine small contour step 13 to 14
`15 L IX+100 ...`	Move to contour point 15
`16 L IY+0.5 ... R... F... M97`	Machine small contour step 15 to 16
`17 L X... Y...`	Move to contour point 17

Machining open contours: M98

Standard behavior

The TNC calculates the intersections of the cutter paths at inside corners and moves the tool in the new direction at those points.

If the contour is open at the corners, however, this will result in incomplete machining.



Behavior with M98

With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined.

Effect

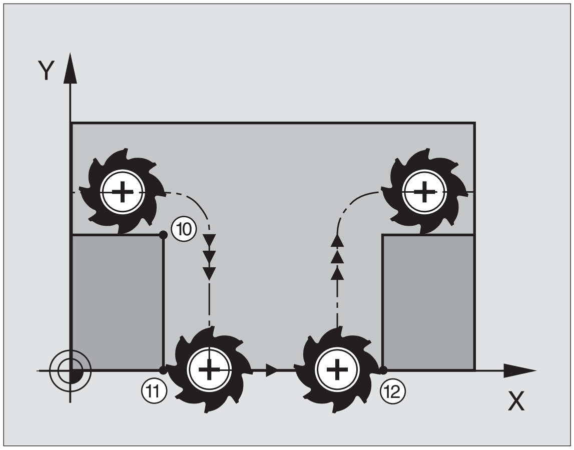
M98 is effective only in the blocks in which it is programmed.

M98 takes effect at the end of block.

Example NC blocks

Move to the contour points 10, 11 and 12 in succession:

```
10 L X... Y... RL F
11 L X... IY... M98
12 L IX+ ...
```



Feed rate for circular arcs: M109/M110/M111

Standard behavior

The TNC applies the programmed feed rate to the path of the tool center.

Behavior at circular arcs with M109

The TNC adjusts the feed rate for circular arcs at inside and outside contours so that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110

The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.

> M110 is also effective for the inside machining of circular arcs using contour cycles. If you define M109 or M110 before calling a machining cycle, the adjusted feed rate is also effective for circular arcs within machining cycles. The initial state is restored after finishing or aborting a machining cycle.

Effect

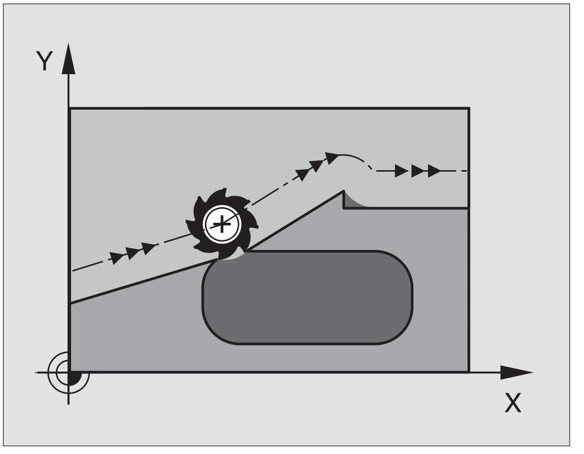
M109 and M110 become effective at the start of block.
To cancel M109 and M110, enter M111.

Calculating the radius-compensated path in advance (LOOK AHEAD): M120

Standard behavior

If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. M97 (see "Machining small contour steps: M97" on page 165) inhibits the error message, but this results in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour.



Behavior with M120

The TNC checks radius-compensated paths for contour undercuts and tool path intersections, and calculates the tool path in advance from the current block. Areas of the contour that might be damaged by the tool are not machined (dark areas in figure at right). You can also use M120 to calculate the radius compensation for digitized data or data created on an external programming system. This means that deviations from the theoretical tool radius can be compensated.

Use LA (**L**ook **A**head) behind M120 to define the number of blocks (maximum: 99) that you want the TNC to calculate in advance. Note that the larger the number of blocks you choose, the higher the block processing time will be.

Input

If you enter M120 in a positioning block, the TNC continues the dialog for this block by asking you the number of blocks LA that are to be calculated in advance.

Effect

M120 must be located in an NC block that also contains radius compensation RL or RR. M120 is then effective from this block until

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA, or
- another program is called with PGM CALL, or

M120 becomes effective at the start of block.

Limitations

- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS. AT N.
- When using the path functions RND and CHF, the blocks before and after them must contain only coordinates in the working plane.
- If you want to approach the contour on a tangential path, you must use the function APPR LCT. The block with APPR LCT must contain only coordinates of the working plane.
- If you want to depart the contour on a tangential path, use the function DEP LCT. The block with DEP LCT must contain only coordinates of the working plane.

Superimposing handwheel positioning during program run: M118

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M118

M118 permits manual corrections by handwheel during program run. Just program M118 and enter an axis-specific value (linear or rotary axis) in millimeters.

Input

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. Use the ENTER key to switch the axis letters.

Effect

Cancel handwheel positioning by programming M118 once again without coordinate input.

M118 becomes effective at the start of block.

Example NC blocks

If you want to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm from the programmed value:

```
L X+0 Y+38.5 RL F125 M118 X1 Y1
```

M118 also functions in the Positioning with MDI mode of operation!

If M118 is active, the MANUAL OPERATION function is not available after a program interruption.

Retraction from the contour in the tool-axis direction: M140

Standard behavior

In the program run modes, the TNC moves the tool as defined in the part program.

Behavior with M104

With M140 MB (move back) you can enter a path in the direction of the tool axis for departure from the contour.

Input

If you enter M140 in a positioning block, the TNC continues the dialog and asks for the desired path of tool departure from the contour. Enter the requested path that the tool should follow when departing the contour, or press the MAX soft key to move to the limit of the traverse range.

In addition, you can program the feed rate at which the tool traverses the entered path. If you do not enter a feed rate, the TNC moves the tool along the entered path at rapid traverse.

Effect

M140 is effective only in the block in which it is programmed.

M140 becomes effective at the start of the block.

Example NC blocks

Block 250: Retract the tool 50 mm from the contour.

Block 251: Move the tool to the limit of the traverse range.

```
250 L X+0 Y+38.5 F125 M140 MB 50 F750
251 L X+0 Y+38.5 F125 M140 MB MAX
```

With **M140 MB MAX** you can only retract in positive direction.

Suppressing touch probe monitoring: M141

Standard behavior

When the stylus is deflected, the TNC outputs an error message as soon as you attempt to move a machine axis.

Behavior with M141

The TNC moves the machine axes even if the touch probe is deflected. This function is required if you wish to write your own measuring cycle in connection with measuring cycle 3 in order to retract the stylus by means of a positioning block after it has been deflected.

If you use M141, make sure that you retract the touch probe in the correct direction.

M141 functions only for movements with straight-line blocks.

Effect

M141 is effective only in the block in which it is programmed.

M141 becomes effective at the start of the block.

Delete basic rotation: M143

Standard behavior

The basic rotation remains in effect until it is reset or is overwritten with a new value.

Behavior with M143

The TNC erases a programmed basic rotation from the NC program.

> The function **M143** is not permitted during a mid-program startup.

Effect

M143 is effective only in the block in which it is programmed.

M143 becomes effective at the start of the block.

Automatically retract tool from the contour at an NC stop: M148

Standard behavior

At an NC stop the TNC stops all traverse movements. The tool stops moving at the point of interruption.

Behavior with M148

> The M148 function must be enabled by the machine tool builder.

The TNC retracts the tool in the direction of the tool axis if, in the **LIFTOFF** column of the tool table, you set the parameter **Y** for the active tool (see "Tool table: Standard tool data" on page 100).

> Remember that, especially on curved surfaces, the surface can be damaged during return to the contour. Back the tool off before returning to the contour!
>
> In the **CfgLiftOff** machine parameter, define the value by which the tool is to be retracted . In the **CfgLiftOff** machine parameter you can also switch off the function.

Effect

M148 remains in effect until deactivated with M149.

M148 becomes effective at the start of block, M149 at the end of block.

7.5 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116

Standard behavior

The TNC interprets the programmed feed rate in a rotary axis in degrees per minute. The contouring feed rate therefore depends on the distance from the tool center to the center of the rotary axis.

The larger this distance becomes, the greater the contouring feed rate.

Feed rate in mm/min on rotary axes with M116

> The machine manufacturer must enter the machine geometry.
>
> Your machine manual provides more detailed information.
>
> M116 works only on rotary tables. M116 cannot be used with swivel heads. If your machine is equipped with a table/head combination, the TNC ignores the swivel-head rotary axes.

The TNC interprets the programmed feed rate in a rotary axis in mm/min. With this miscellaneous function, the TNC calculates the feed rate for each block at the start of the block. With a rotary axis, the feed rate is not changed during execution of the block even if the tool moves toward the center of the rotary axis.

Effect

M116 is effective in the working plane.
With M117 you can reset M116. M116 is also canceled at the end of the program.

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior

The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° is decided by the machine tool builder. They decide whether the TNC should consider the difference between nominal and actual position, or whether the TNC should always (even without M126) choose the shortest path to the programmed position. Examples:

Actual position	Nominal position	Traverse
350°	10°	–340°
10°	340°	+330°

Behavior with M126

With M126, the TNC will move the axis on the shorter path of traverse if you reduce display of a rotary axis to a value less than 360°. Examples:

Actual position	Nominal position	Traverse
350°	10°	+20°
10°	340°	–30°

Effect

M126 becomes effective at the start of block.
To cancel M126, enter M127. At the end of program, M126 is automatically canceled.

Reducing display of a rotary axis to a value less than 360°: M94

Standard behavior

The TNC moves the tool from the current angular value to the programmed angular value.

Example:

Current angular value:	538°
Programmed angular value:	180°
Actual distance of traverse:	–358°

Behavior with M94

At the start of block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. As an alternative you can enter a rotary axis after M94. The TNC then reduces the display only of this axis.

Example NC blocks

To reduce display of all active rotary axes:

```
L M94
```

To reduce display of the C axis only:

```
L M94 C
```

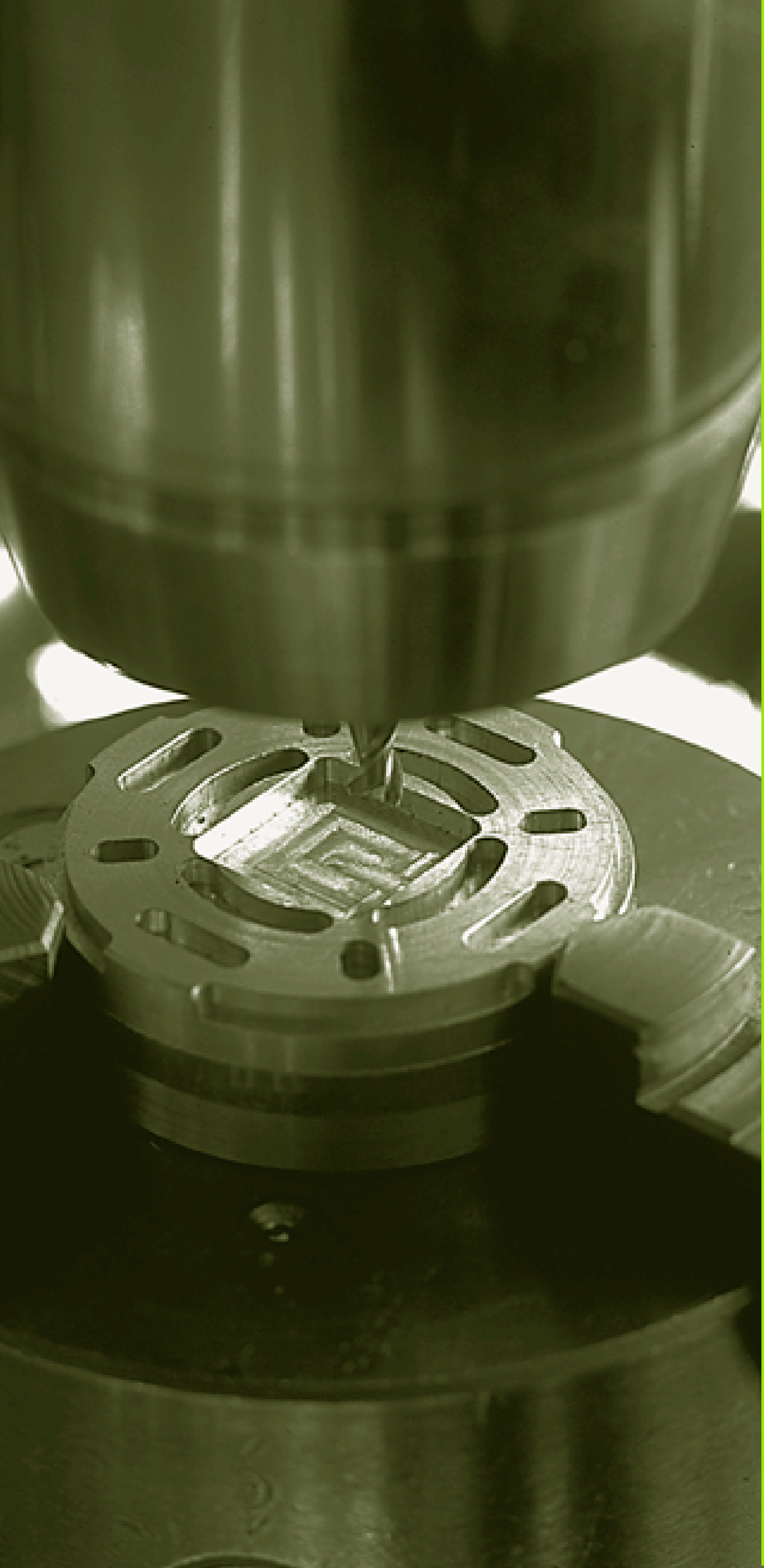
To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

```
L C+180 FMAX M94
```

Effect

M94 is effective only in the block in which it is programmed.

M94 becomes effective at the start of block.



8

Programming: Cycles

8.1 Working with Cycles

Frequently recurring machining cycles that comprise several working steps are stored in the TNC memory as standard cycles. Coordinate transformations and other special cycles are also provided as standard cycles (for an overview: see “”, page 178).

Fixed cycles with numbers 200 and above 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 set-up clearance, Q202 the plunging depth, etc.

Fixed cycles sometimes execute extensive operations. For safety reasons, you should run a graphical program test before machining (see “Test Run” on page 382).

Machine-specific cycles

In addition to the HEIDENHAIN cycles, many machine tool builders offer their own cycles in the TNC. These cycles are available in a separate cycle-number range:

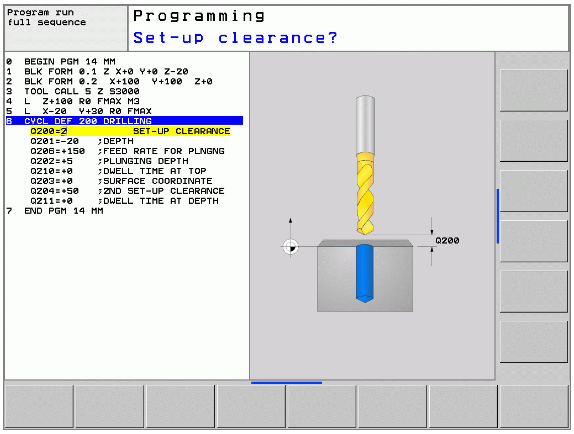
- Cycles 300 to 399
  Machine-specific cycles that are to be defined through the CYCLE DEF key
- Cycles 500 to 599
  Machine-specific cycles that are to be defined through the TOUCH PROBE key

Refer to your machine manual for a description of the specific function.

Sometimes, machine-specific cycles also use transfer parameters, which HEIDENHAIN already used in the standard cycles. The TNC executes DEF-active cycles as soon as they are defined (See also “Calling cycles” on page 179) It executes CALL-active cycles only after they have been called (See also “Calling cycles” on page 179). When DEF-active cycles and CALL-active cycles are used simultaneously, it is important to prevent overwriting of transfer parameters already in use. Use the following procedure:

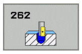
- As a rule, always program DEF-active cycles before CALL-active cycles.
- If you do want to program a DEF-active cycle between the definition and call of a CALL-active cycle, do it only if there is no common use of specific transfer parameters.

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 cycle, for example THREAD MILLING. The TNC initiates a dialog and asks for all 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.

- In the right screen window the TNC shows a graphic of the input parameters. The parameter that is asked for in the dialog prompt is highlighted.
- Enter all parameters requested by the TNC and conclude each entry with the ENT key.
- The TNC ends the dialog when all required data has been entered.

Defining a cycle using the GOTO function

- The soft-key row shows the available groups of cycles.

- The TNC opens a pop-up window
- Enter the cycle number and confirm it with the ENT key. The TNC then initiates the cycle dialog as described above.

Example NC blocks

```
7 CYCL DEF 200 DRILLING
    Q200=2      ;SET-UP CLEARANCE
    Q201=3      ;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 DEPTH
```

Group of cycles	Soft key
Cycles for pecking, reaming, boring, counterboring, tapping and thread milling	DRILLING/ THREAD
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS
Cycles for producing point patterns, such as circular or linear hole patterns	PATTERN
SL (Subcontour List) cycles which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation	SL II
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Special cycles such as dwell time, program call, oriented spindle stop and tolerance	SPECIAL CYCLES

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.

If you define a feed-rate parameter for fixed cycles greater than 200, then instead of entering a numerical value you can use soft keys to assign the feed rate defined in the **TOOL CALL** block (FAUTO soft key), or rapid traverse (FMAX soft key).

If you want to delete a block that is part of a cycle, the TNC asks you whether you want to delete the whole cycle.

Calling cycles

Prerequisites

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 detailed in the descriptions for each 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 point patterns on circles and Cycle 221 for point patterns on lines
- SL Cycle 14 CONTOUR GEOMETRY
- SL Cycle 20 CONTOUR DATA
- Coordinate Transformation Cycles
- Cycle 9 DWELL TIME

You can call all other cycles with the functions described as follows.

Calling a cycle with CYCL CALL

The **CYCL CALL** function calls the fixed cycle that was last defined. The starting point of the cycle is the position that was programmed last before the CYCL CALL block.

CYCL CALL

- To program the cycle call, press the CYCL CALL key.
- Press the CYCL CALL M soft key to enter a cycle call.
- If necessary, enter the miscellaneous function M (for example **M3** to switch the spindle on), or end the dialog by pressing the END key

Calling a cycle with M99/89

The **M99** function, which is active only in the block in which it is programmed, calls the last defined fixed cycle once. You can program **M99** at the end of a positioning block. The TNC moves to this position and then calls the last defined fixed cycle.

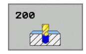
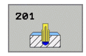
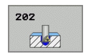
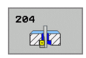
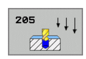
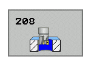
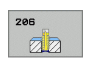
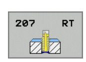
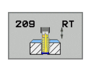
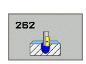
If the ATEK M is to execute the cycle automatically after every positioning block, program the cycle call with **M89.**

To cancel the effect of **M89,** program:

- **M99** in the positioning block in which you move to the last starting point, or
- Define with **CYCL DEF** a new fixed cycle

8.2 Cycles for Drilling, Tapping and Thread Milling

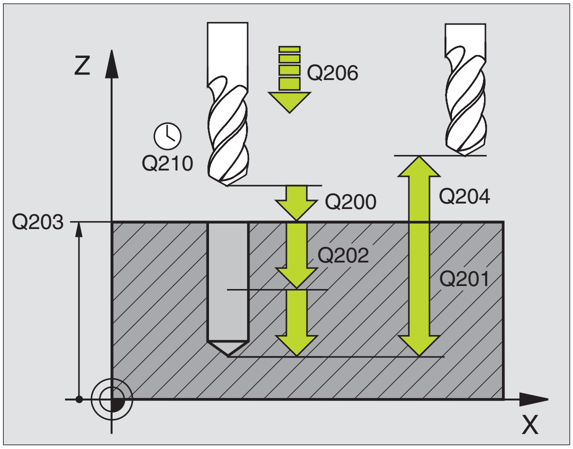
Overview

Cycle	Soft key
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 decrementing	
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	
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 W/ CHIP BRKG 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 MLLNG/CNTSNKG Cycle for milling a thread in pre-drilled material and machining a countersunk chamfer	

Cycle	Soft key
264 THREAD DRILLING/MLLNG Cycle for drilling into the solid material with subsequent milling of the thread with a tool	
265 HEL.THREAD DRLG/MLG Cycle for milling the thread into the solid material	
267 OUTSIDE THREAD MLLNG Cycle for milling an external thread and machining a countersunk chamfer	

DRILLING (Cycle 200)

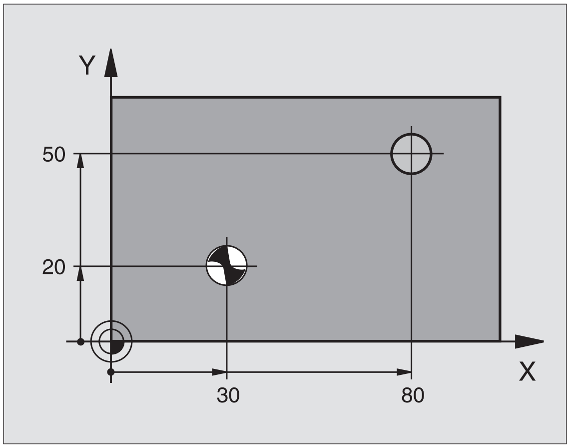
1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the set-up clearance above the workpiece surface.
2. The tool drills to the first plunging depth at the programmed feed rate F.
3. The TNC returns the tool at FMAX to the set-up clearance, dwells there (if a dwell time was entered),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



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. If you program DEPTH = 0, the cycle will not be executed.



Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

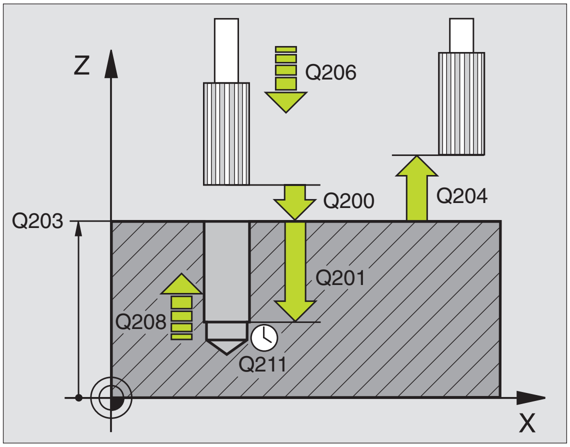
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper).
- **Feed rate for plunging** Q206: Traversing speed of the tool during 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 TNC 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.

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 DEPTH
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
```

REAMING (Cycle 201)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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 for the entered dwell time.
4 The tool then retracts to the set-up clearance at the feed rate F, and from there—if programmed—to the 2nd set-up clearance at FMAX.



Before programming, note the following:

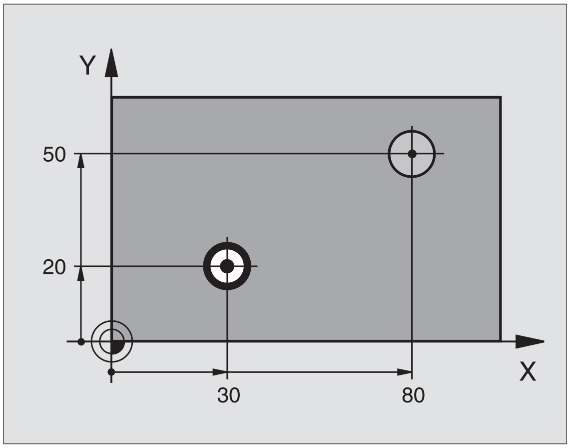
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.

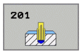
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!





- **`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 reaming 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.

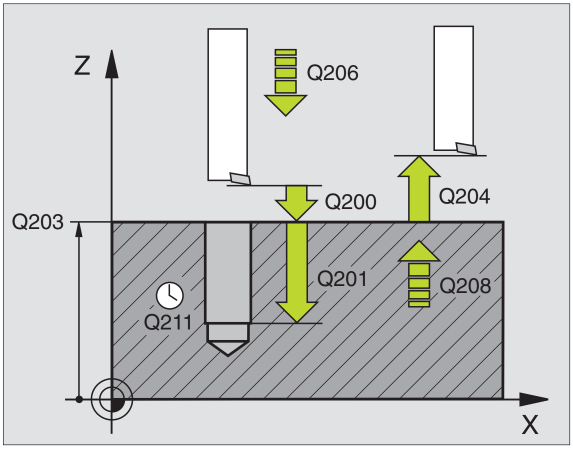
Example: NC blocks

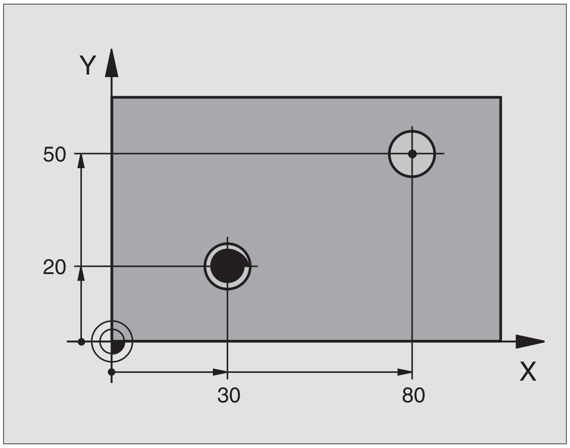
```
10 L Z+100 R0 FMAX
11 CYCL DEF 201 REAMING
     Q200=2      ;SET-UP CLEARANCE
     Q201=-15    ;DEPTH
     Q206=100    ;FEED RATE FOR PLUNGING
     Q211=0.5    ;DWELL TIME AT DEPTH
     Q208=250    ;RETRACTION FEED RATE
     Q203=+20    ;SURFACE COORDINATE
     Q204=100    ;2ND SET-UP CLEARANCE
12 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)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

1 The TNC 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 programmed depth at the feed rate for plunging.
3 If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4 The TNC then orients the spindle to the position that is defined in parameter Q336.
5 If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
6 The TNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at FMAX. If Q214=0, the tool point remains on the wall of the hole.





Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.

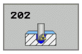
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **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 TNC retracts the tool at the hole bottom (after spindle orientation).

  0 Do not retract tool
  1 Retract tool in the negative ref. axis direction
  2 Retract tool in the neg. secondary axis direction
  3 Retract tool in the positive ref. axis direction
  4 Retract tool in the pos. 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.

During retraction the TNC automatically takes an active rotation of the coordinate system into account.

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

Example: NC blocks

```
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 DEPTH
    Q208=250    ;RETRACTION FEED RATE
    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
```

UNIVERSAL DRILLING (Cycle 203)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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 the set-up clearance, remains there—if programmed—for the entered dwell time, and advances again at FMAX to the set-up 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 TNC 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 the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at FMAX.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.

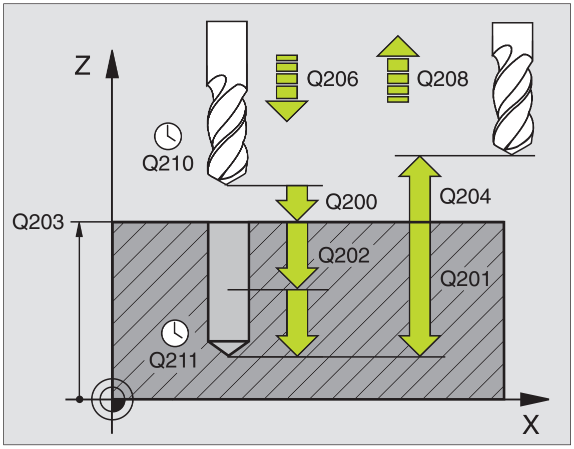
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **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 TNC 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 TNC decreases the plunging depth Q202 after each infeed.
- **No. of breaks before retracting** Q213: Number of chip breaks after which the TNC is to withdraw the tool from the hole for chip release. For chip breaking, the TNC retracts the tool each time by the value in Q256.
- **Minimum plunging depth** Q205 (incremental value): If you have entered a decrement, the TNC 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 TNC retracts the tool at the feed rate in Q206.
- **Retraction rate for chip breaking** Q256 (incremental value): Value by which the TNC retracts the tool during chip breaking.



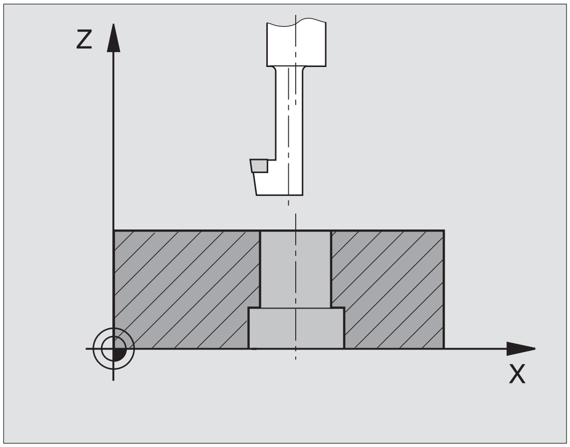
Example: NC blocks

```
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 DEPTH
    Q208=500    ;RETRACTION FEED RATE
    Q256=0.2    ;DIST. FOR CHIP BRKNG
```

BACK BORING (Cycle 204)

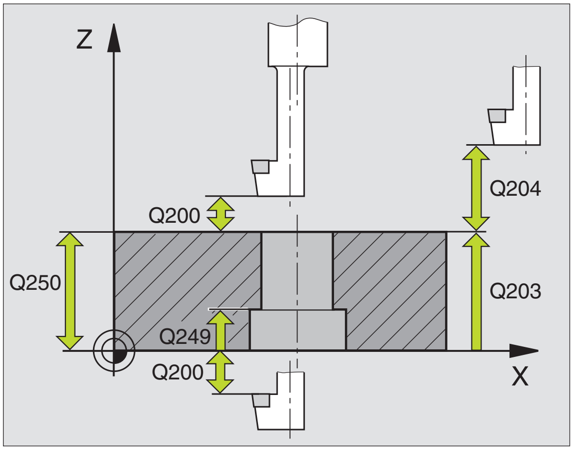
Machine and control must be specially prepared by the machine tool builder for use of this cycle.

Special boring bars for upward cutting are required for this cycle.



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

1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the set-up clearance above the workpiece surface.
2. The TNC 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 the set-up clearance on the underside of the workpiece.
4. The TNC 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. Another oriented spindle stop is carried out and the tool is once again displaced by the off-center distance.
6. The TNC moves the tool at the pre-positioning feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at FMAX.



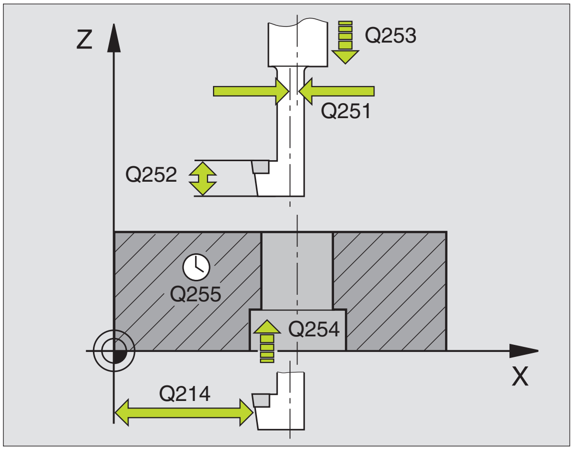
Before programming, note the following:

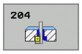
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.

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 TNC considers the tooth length of the boring bar and the thickness of the material.





- **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 countersinking** Q254: Traversing speed of the tool during countersinking 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 TNC displaces the tool by the off-center distance (after spindle orientation).

  1 Retract tool in the negative ref. axis direction
  2 Retract tool in the neg. secondary axis direction
  3 Retract tool in the positive ref. axis direction
  4 Retract tool in the pos. secondary axis direction

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.

- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before it is plunged into or retracted from the bore hole.

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
```

UNIVERSAL PECKING (Cycle 205)

1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.
2. If you enter a deepened starting point, the TNC move at the defined positioning feed rate to the set-up clearance above the deepened starting point.
3. The tool drills to the first plunging depth at the programmed feed rate F.
4. 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 the set-up clearance and then at FMAX to the entered starting position above the first plunging depth.
5. 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.
6. The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.
7. The tool remains at the hole bottom—if programmed—for the entered dwell time to cut free, and then retracts to the set-up clearance at the retraction feed rate. If programmed, the tool moves to the 2nd set-up clearance at FMAX.

Before programming, note the following:

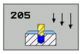
Program a positioning block for the starting point (hole center) in the working plane with radius compensation R0.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

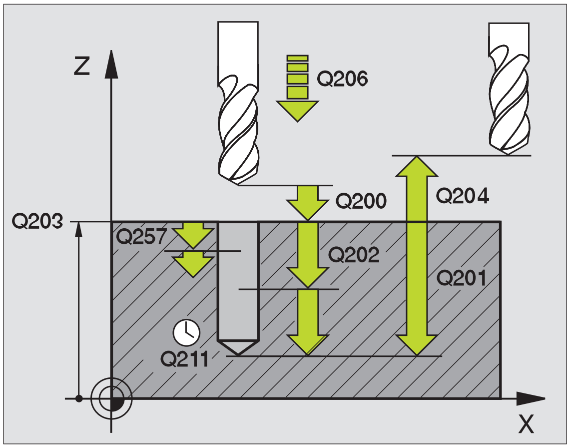
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **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 TNC 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 TNC decreases the plunging depth Q202.
- **Minimum plunging depth** Q205 (incremental value): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205.
- **Upper advanced stop distance** Q258 (incremental value): Set-up clearance for rapid traverse positioning when the TNC 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): Set-up clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth.



If you enter Q258 not equal to Q259, the TNC 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 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 TNC retracts the tool during chip breaking.
- **Dwell time at depth** Q211: Time in seconds that the tool remains at the hole bottom.
- **Deepened starting point** Q379 (incremental with respect to the workpiece surface): Starting position of drilling if a shorter tool has already pilot drilled to a certain depth. The TNC moves at the **feed rate for pre-positioning** from the set-up clearance to the deepened starting point.
- **Feed rate for pre-positioning** Q253: Traversing velocity of the tool during positioning from the set-up clearance to a deepened starting point in mm/min. Effective only if Q379 is entered not equal to 0.

If you use Q379 to enter a deepened starting point, the TNC merely changes the starting point of the infeed movement. Retraction movements are not changed by the TNC, therefore they are calculated with respect to the coordinate of the workpiece surface.

Example: NC blocks

```
11 CYCL DEF 205 UNIVERSAL PECKING
     Q200=2      ;SET-UP CLEARANCE
     Q201=-80    ;DEPTH
     Q206=150    ;FEED RATE FOR PLUNGING
     Q202=15     ;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 DEPTH
     Q379=7.5    ;STARTING POSITION
     Q253=750    ;F PRE-POSITIONING
```

BORE MILLING (Cycle 208)

1. The TNC 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 TNC once again traverses a full circle to remove the material remaining after the initial plunge.
4. The TNC then positions the tool at the center of the hole again.
5. Finally the TNC returns to the set-up clearance at FMAX. If programmed, the tool moves to the 2nd set-up clearance at FMAX.

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. 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 TNC will bore directly to the entered depth without any helical interpolation.

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

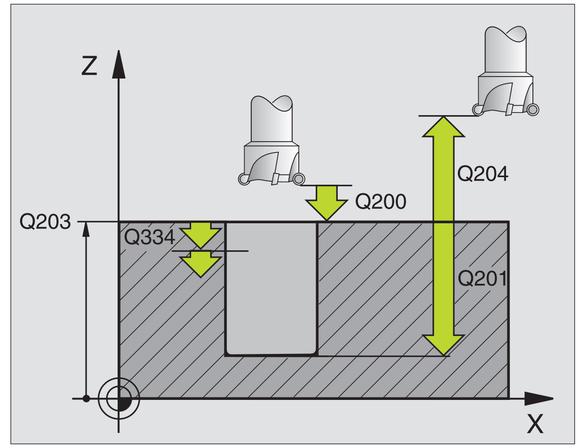
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

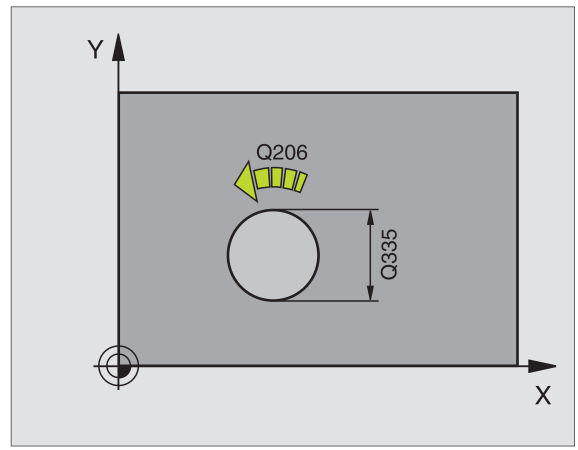
- **Set-up clearance** Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of hole.
- **Feed rate for plunging** Q206: Traversing speed of the tool during helical drilling in mm/min.
- **Infeed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

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

To prevent the infeeds from being too large, enter the maximum plunge angle of the tool in the ANGLE column of the tool table, (see “Tool Data”, page 98). The TNC then automatically calculates the max. infeed permitted and changes your entered value accordingly.

- **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): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC 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 TNC 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.





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
```

TAPPING NEW with floating tap holder (Cycle 206)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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. If programmed, the tool moves to the 2nd set-up clearance at FMAX.
4 At the set-up clearance, the direction of spindle rotation reverses once again.

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. 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.

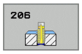
When a cycle is being run, the spindle speed override knob is disabled. The feed-rate override knob is active only within a limited range, which is defined by the machine tool builder (refer to your machine manual).

For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



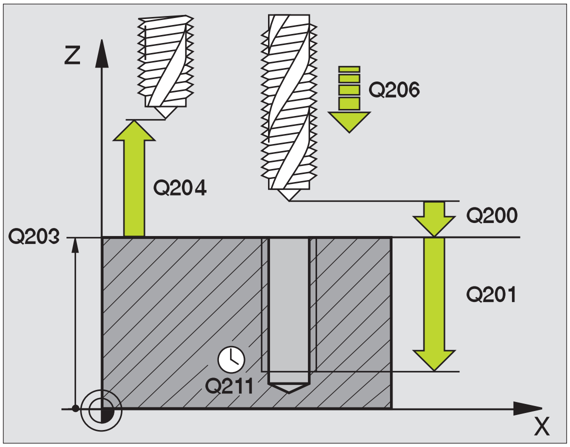
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch.
- **`Total hole depth`** Q201 (thread length, incremental value): Distance between workpiece surface and end of thread.
- **`Feed 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)

Retracting after a program interruption

If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.



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 DEPTH
    Q203=+25    ;SURFACE COORDINATE
    Q204=50     ;2ND SET-UP CLEARANCE
```

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

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

The TNC cuts the thread without a floating tap holder in one or more passes.

1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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. If programmed, the tool moves to the 2nd set-up clearance at FMAX.
4. The TNC stops the spindle turning at set-up clearance.

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 total hole depth parameter determines the working direction.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

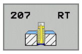
The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

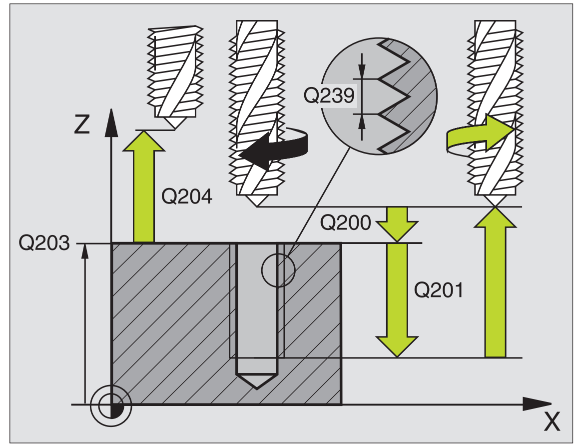
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **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.

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.



Example: NC blocks

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

TAPPING WITH CHIP BREAKING (Cycle 209)

Machine and control must be specially prepared by the machine tool builder for use of this cycle.

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 TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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 TNC repeats this process (2 to 3) until the programmed thread depth is reached.
5. The tool is then retracted to the set-up clearance. If programmed, the tool moves to the 2nd set-up clearance at FMAX.
6. The TNC stops the spindle turning at set-up clearance.

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.

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

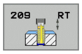
The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

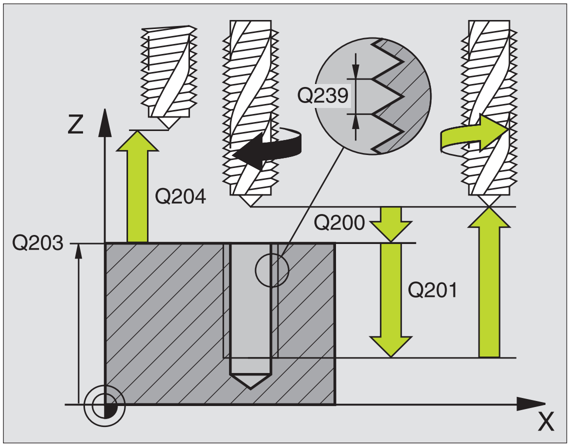
Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



- **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 TNC 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 TNC retracts the tool completely from the hole (to the set-up clearance) for chip release.
- **Angle for spindle orientation** Q336 (absolute value): Angle at which the TNC positions the tool before machining the thread. This allows you to regroove the thread, if required.

Retracting after a program interruption

If you interrupt program run during thread cutting with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.



Example: NC blocks

```
26 CYCL DEF 209 TAPPING W/ CHIP BRKG
    Q200=2      ;SET-UP CLEARANCE
    Q201=-20    ;DEPTH
    Q239=+1     ;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
```

Fundamentals of thread milling

Prerequisites

- Your machine tool should feature internal spindle cooling (cooling lubricant at least 30 bar, compressed air supply at least 6 bar).
- 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+

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. For example, if you only want to repeat the countersinking process of a cycle, enter 0 for the thread depth. The work direction will then be determined from the countersinking depth.

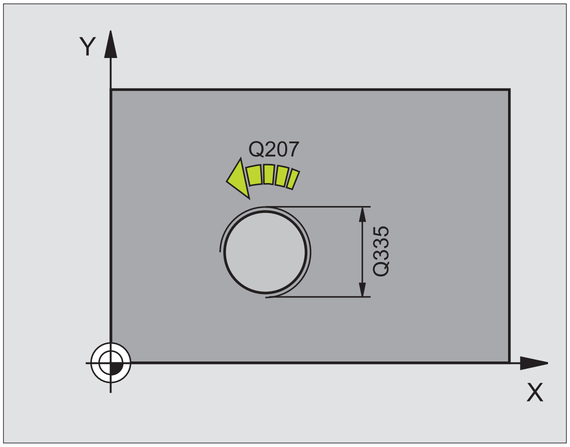
Procedure in case of a tool break

If a tool break occurs during thread cutting, stop the program run, change to the Positioning with 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.

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

The machining direction of the thread changes if you execute a thread milling cycle in connection with Cycle 8 MIRRORING in only one axis.

THREAD MILLING (Cycle 262)



1. The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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. Before the helical approach, a compensating motion of the tool axis is carried out in order to begin at the programmed starting plane for the thread path.
4. Depending on the setting of the parameter for the number of threads, the tool mills the thread in one helical movement, in several offset movements or in one continuous movement.
5. After this, the tool departs the contour tangentially and returns to the starting point in the working plane.
6. At the end of the cycle, the TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

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. 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.

Note that the TNC makes a compensating movement in the tool axis before the approach movement. The length of the compensating movement depends on the thread pitch. Ensure sufficient space in the hole!

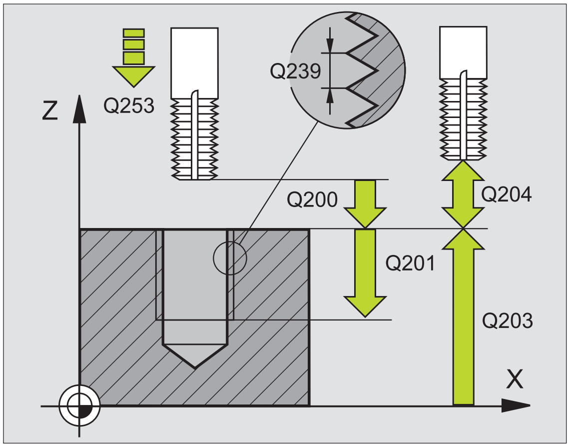
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

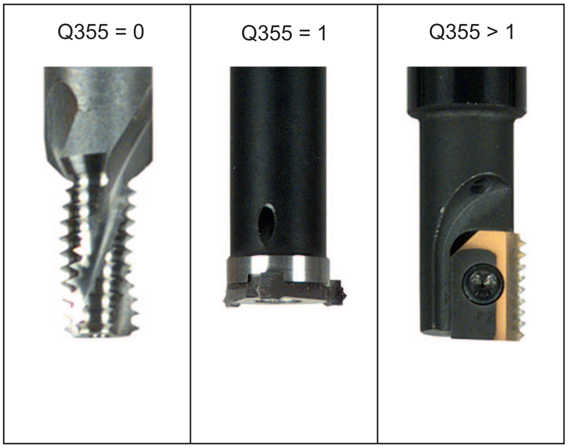
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

262

- **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 TNC 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 with M03.
  +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.





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
```

THREAD MILLING/COUNTERSINKING (Cycle 263)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.

Countersinking

2 The tool moves at the feed rate for pre-positioning to the countersinking depth minus the set-up 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 TNC immediately positions the tool at the feed rate for pre-positioning to the countersinking depth.

4 Then, depending on the available space, the TNC 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 TNC 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 TNC 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 TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

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. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Countersinking depth
3rd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

If you want to countersink with the front of the tool, define the countersinking depth as 0.

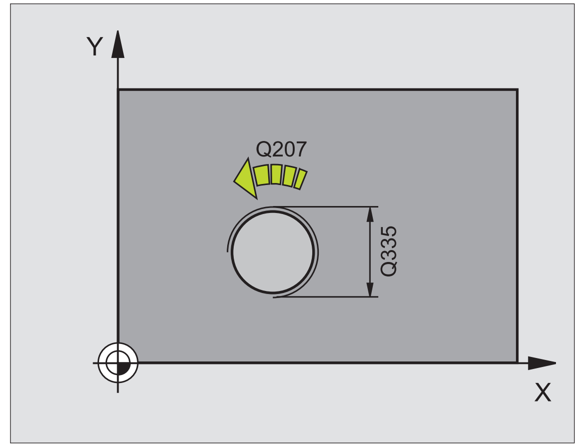
Program the thread depth as a value smaller than the countersinking depth by at least one-third the thread pitch.

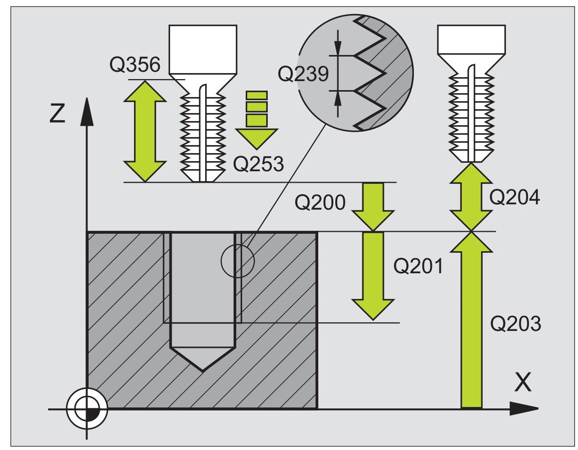
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

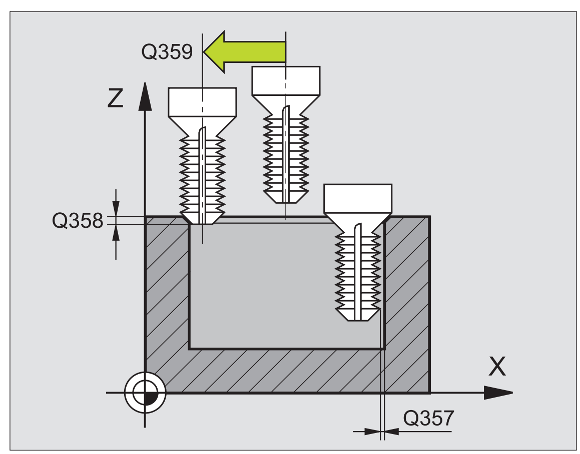
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **`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 with M03.
  +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 tip 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 TNC 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 countersinking** Q254: Traversing speed of the tool during countersinking in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

Example: NC blocks

```
25 CYCL DEF 263 THREAD MLLNG/CNTSNKG
     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 COUNTERBORING
     Q207=500    ;FEED RATE FOR MILLING
```

THREAD DRILLING/MILLING (Cycle 264)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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 the set-up 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 TNC 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 TNC 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 TNC 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 TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

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. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Total hole depth
3rd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

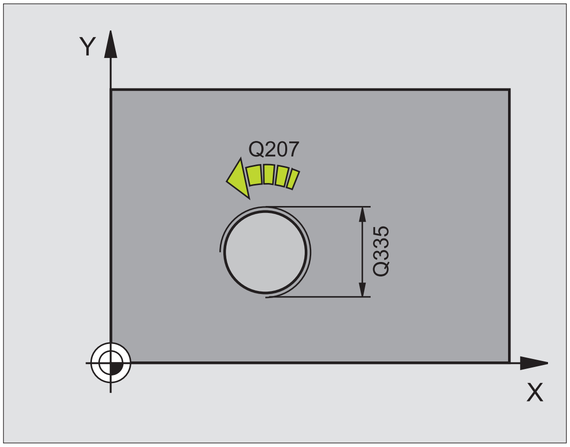
Program the thread depth as a value smaller than the total hole depth by at least one-third the thread pitch.

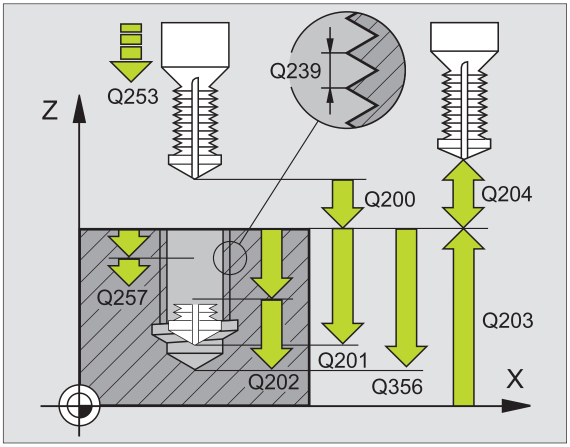
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

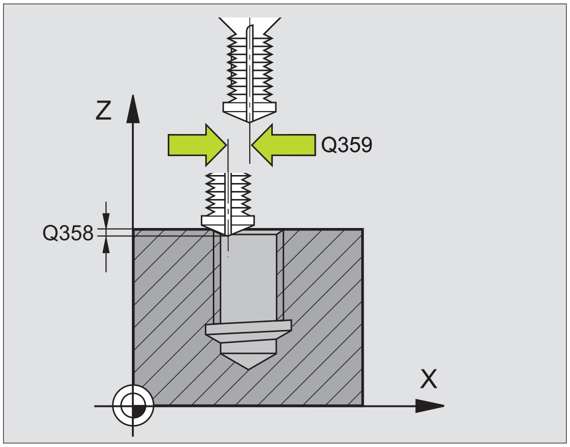
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **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.
- **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 with M03.
  +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 TNC 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): Set-up clearance for rapid traverse positioning when the TNC 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 TNC retracts the tool during chip breaking.
- **Depth at front** Q358 (incremental value): Distance between tool tip 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 TNC 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.

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
```

HELICAL THREAD DRILLING/MILLING (Cycle 265)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up 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 TNC 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 TNC retracts the tool in rapid traverse to set-up clearance, or—if programmed—to the 2nd set-up clearance.

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. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

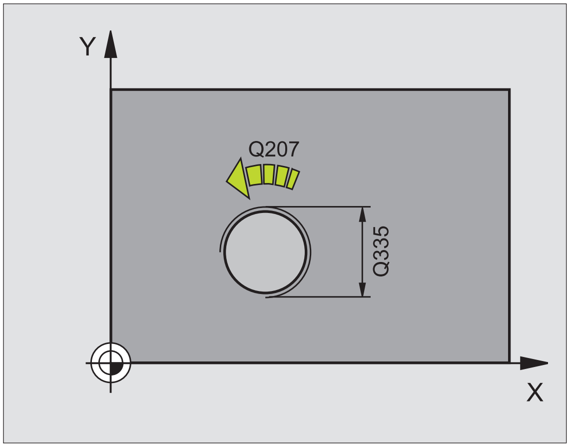
The type of milling (up-cut/climb) is determined by the thread (right-hand/left-hand) and the direction of tool rotation, since it is only possible to work in the direction of the tool.

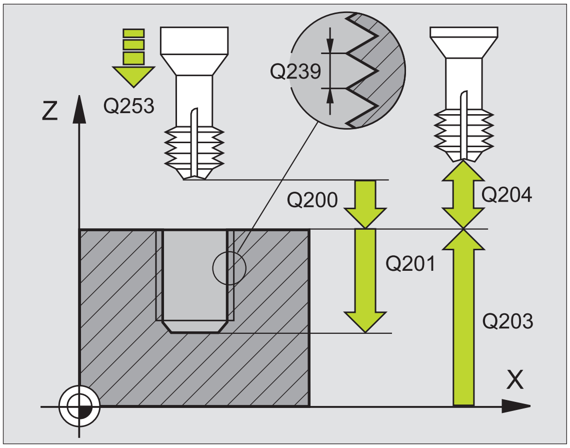
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

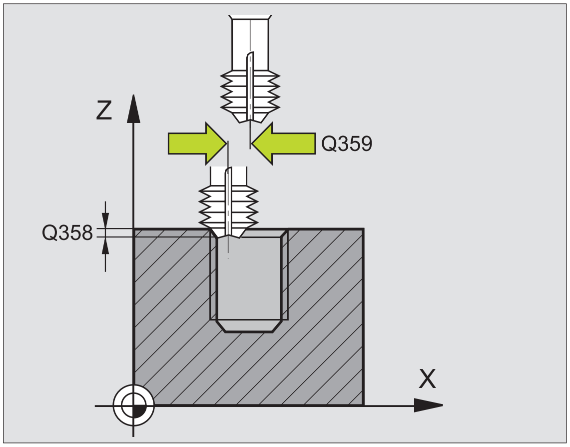
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **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 tip 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 TNC 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 countersinking** Q254: Traversing speed of the tool during countersinking 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
     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 COUNTERBORING
     Q207=500    ;FEED RATE FOR MILLING
```

OUTSIDE THREAD MILLING (Cycle 267)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.

Countersinking at front

2 The TNC moves in 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 TNC 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 on a semicircle to the starting point.

Thread milling

6 The TNC 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 helical movement, in several offset movements or in one continuous 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 TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

Program a positioning block for the starting point (stud center) in the working plane with radius compensation R0.

The offset required before countersinking at the front should be determined ahead of time. You must enter the value from the center of the stud to the center of the tool (uncorrected value).

The algebraic sign of the cycle parameters depth of thread or sinking depth at front determines the working direction. The working direction is defined in the following sequence:
1st: Depth of thread
2nd: Depth at front

If you program a depth parameter to be 0, the TNC does not execute that step.

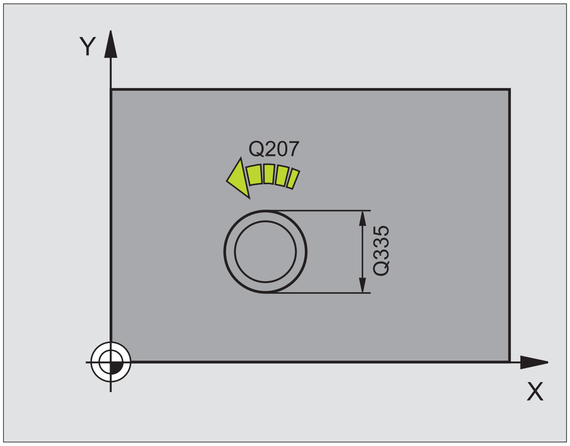
The algebraic sign for the cycle parameter thread depth determines the working direction.

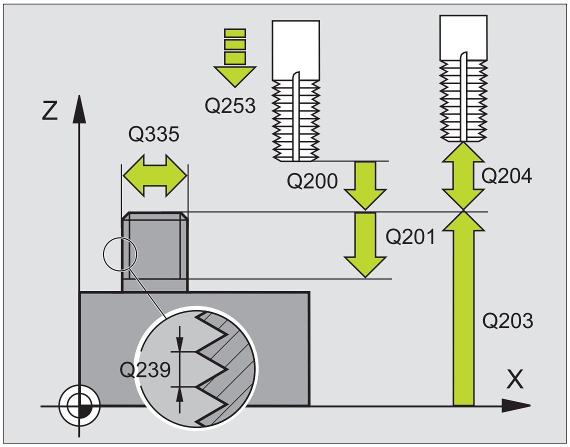
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

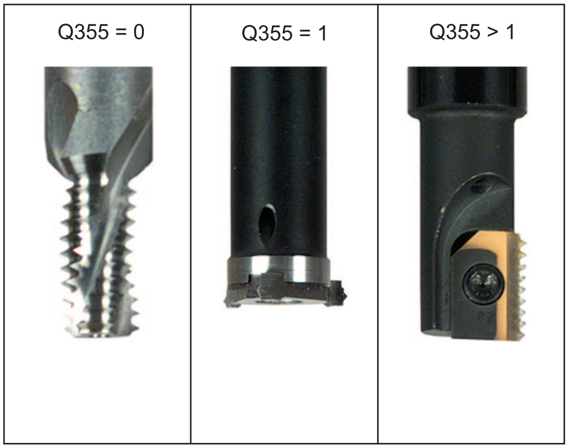
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

- **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 helical line to the thread depth
  1 = continuous helical path over the entire length of the thread
  >1 = several helical paths with approach and departure; between them, the TNC 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 with M03.
  +1 = climb milling
  –1 = up-cut milling





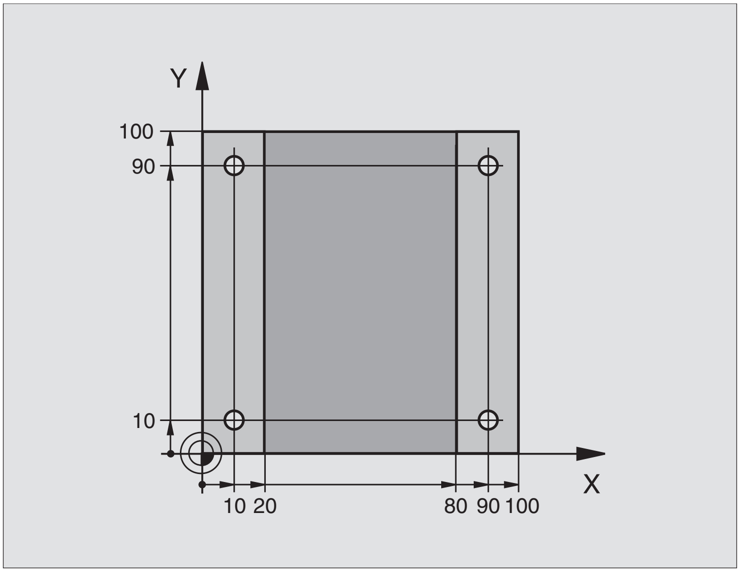


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth at front** Q358 (incremental value): Distance between tool tip and the top surface of the workpiece for countersinking at the front of the tool.
- **Countersinking offset at front** Q359 (incremental value): Distance by which the TNC 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 countersinking** Q254: Traversing speed of the tool during countersinking in mm/min.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.

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 COUNTERBORING
     Q207=500    ;FEED RATE FOR MILLING
```

Example: Drilling cycles

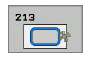
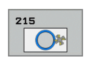
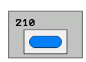
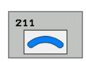


0 BEGIN PGM C200 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+3	Define the tool
4 TOOL CALL 1 Z S4500	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 CYCL DEF 200 DRILLING	Define cycle
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=-10 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q211=0.2 ;DWELL TIME AT DEPTH	

7 L X+10 Y+10 R0 FMAX M3	Approach hole 1, spindle ON
8 CYCL CALL	Call the cycle
9 L Y+90 R0 FMAX M99	Approach hole 2, call cycle
10 L X+90 R0 FMAX M99	Approach hole 3, call cycle
11 L Y+10 R0 FMAX M99	Approach hole 4, call cycle
12 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
13 END PGM C200 MM	

8.3 Cycles for Milling Pockets, Studs and Slots

Overview

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 C. 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	
210 SLOT RECIP. PLNG 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	

POCKET MILLING (Cycle 4)

Cycles 1, 2, 3, 4, 5, 17, 18 are in a group of cycles called special cycles. Here in the second soft-key row, select the OLD CYCLS soft key.

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 TNC retracts the tool to the starting position.

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.

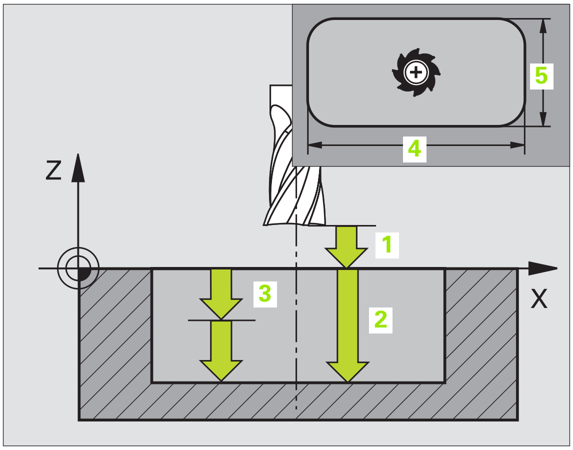
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

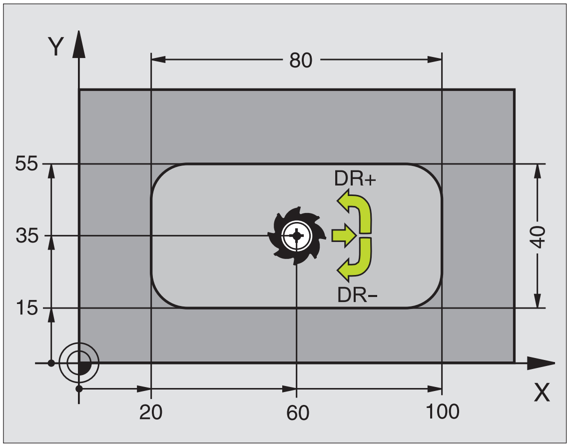
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

The following prerequisite applies for the 2nd side length: 2nd side length greater than [(2 x rounding radius) + stepover factor k].

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!





Example: NC blocks

```
11 L Z+100 R0 FMAX
12 CYCL DEF 4.0 POCKET MILLING
13 CYCL DEF 4.1 SETUP 2
14 CYCL DEF 4.2 DEPTH -10
15 CYCL DEF 4.3 PECKG 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
```

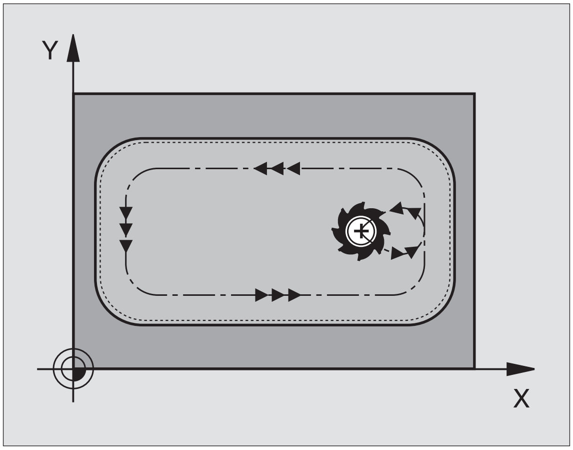
- **Set-up clearance 1** (incremental value): Distance between tool tip (at starting position) and workpiece surface.
- **Depth 2** (incremental value): Distance between workpiece surface and bottom of pocket
- **Plunging depth 3** (incremental value): Infeed per cut
  The TNC will go to depth in one 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 radius:** Radius for the pocket corners.
  If radius = 0 is entered, the pocket corners will be rounded with the radius of the cutter.

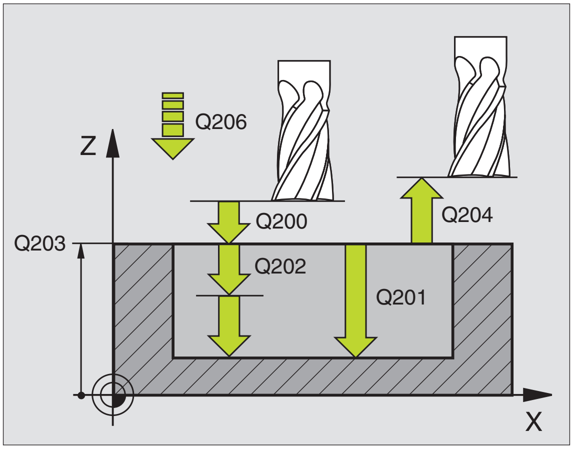
Calculations:
Stepover factor k = K x R

K: Overlap factor, preset in the PocketOverlap machine parameter
R: Cutter radius

POCKET FINISHING (Cycle 212)

1. The TNC M automatically moves the tool in the tool axis to the 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 TNC takes the allowance and tool radius into account for calculating the starting point. If necessary, the TNC penetrates at the pocket center.
3. If the tool is at the 2nd set-up clearance, it moves at rapid traverse FMAX to the 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 TNC retracts the tool at rapid traverse to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).





Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

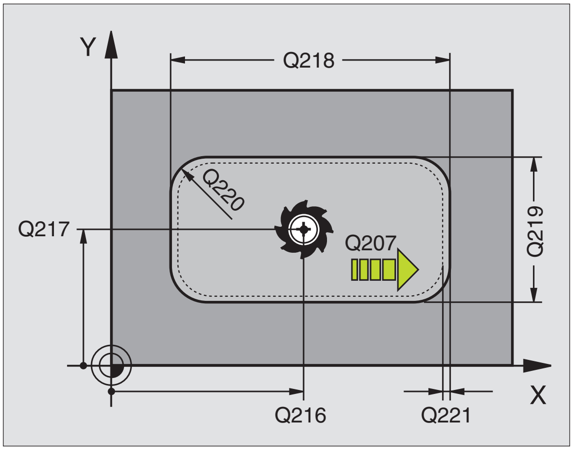
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

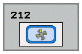
Minimum size of the pocket: 3 times the tool radius.

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



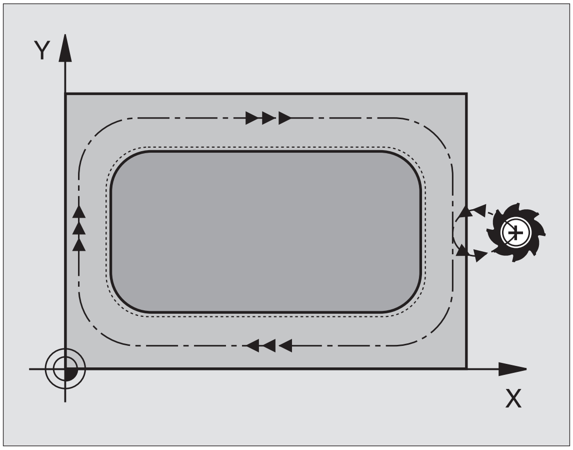


- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min when moving to depth. If you 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 TNC 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 referenced to the length of the pocket.

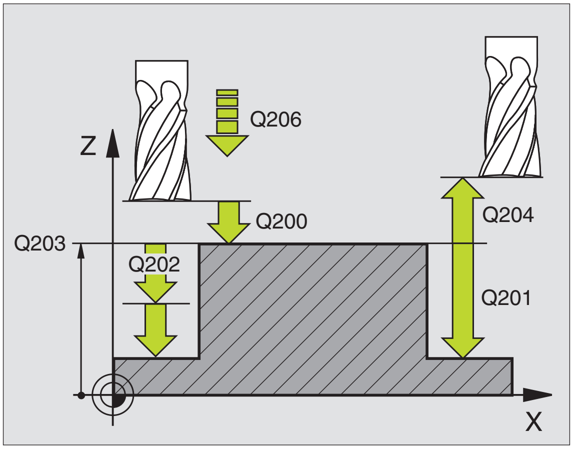
Example: NC blocks

```
354 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      ;OVERSIZE
```

STUD FINISHING (Cycle 213)



1. The TNC moves the tool in the tool axis to the 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 at rapid traverse FMAX to the 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 TNC 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).

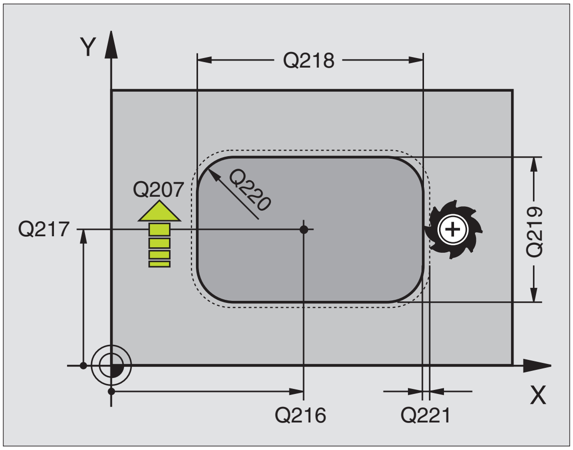


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

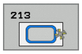
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.



Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



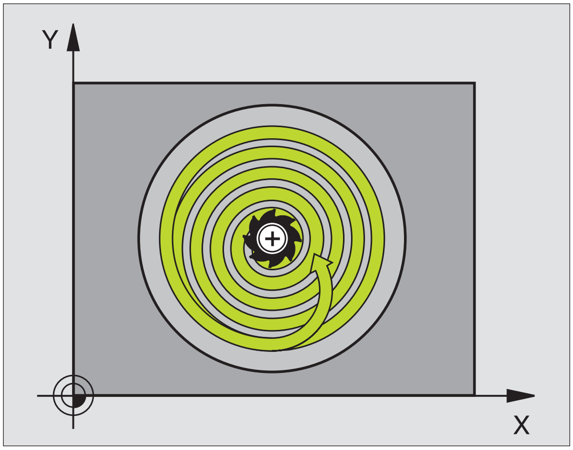
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are 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 referenced to the length of the stud.

Example: NC blocks

```
35 CYCL DEF 213 STUD FINISHING
    Q200=2      ;SET-UP CLEARANCE
    Q291=-20    ;DEPTH
    Q206=150    ;FEED RATE FOR PLUNGING
    Q202=5      ;PLUNGING DEPTH
    Q207=500    ;FEED RATE FOR MILLING
    Q203=+30    ;SURFACE COORDINATE
    Q294=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      ;OVERSIZE
```

CIRCULAR POCKET (Cycle 5)

Cycles 1, 2, 3, 4, 5, 17, 18 are in a group of cycles called special cycles. Here in the second soft-key row, select the OLD CYCLS soft key.



1 The tool penetrates the workpiece at the starting position (pocket center) 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 “POCKET MILLING (Cycle 4)”, page 226.

3 This process is repeated until the depth is reached.

4 At the end of the cycle, the TNC retracts the tool to the starting position.

Before programming, note the following:

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

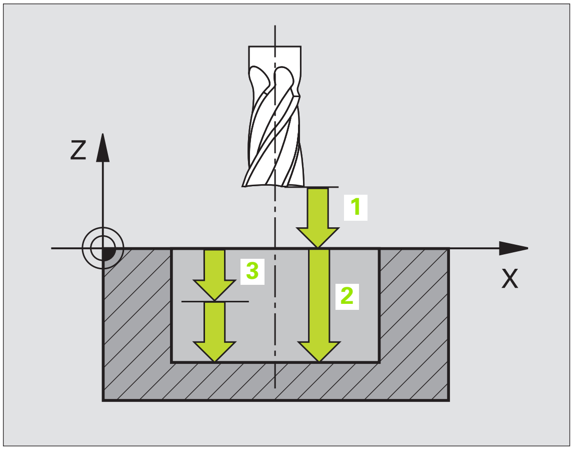
Pre-position over the pocket center with radius compensation R0.

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. If you program DEPTH = 0, the cycle will not be executed.

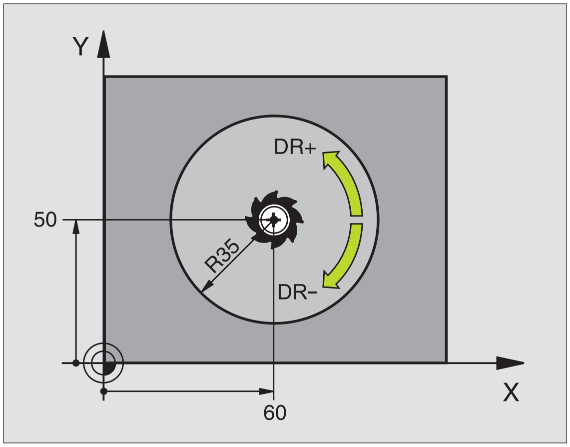
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!



- **`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 TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

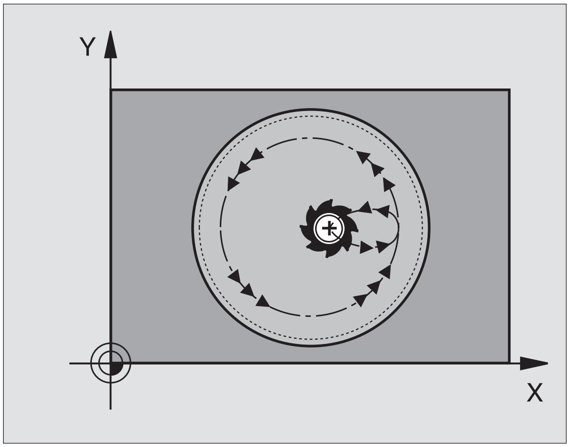
- **Feed rate for plunging:** Traversing speed of the tool during penetration
- **Circular radius:** Radius of the circular pocket
- **Feed rate F:** Traversing speed of the tool in the working plane.
- **Clockwise**
  DR **+:** Climb milling with M3
  DR **–:** Up-cut milling with M3



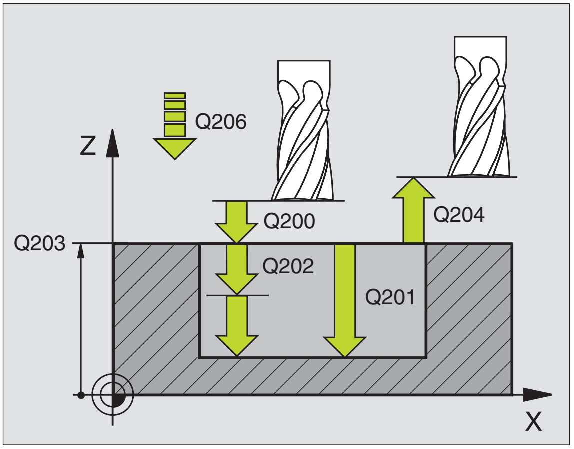
Example: NC blocks

```
16 L Z+100 R0 FMAX
17 CYCL DEF 5.0 CIRCULAR POCKET
18 CYCL DEF 5.1 SETUP 2
19 CYCL DEF 5.2 DEPTH -12
20 CYCL DEF 5.3 PECKG 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 TNC M automatically moves the tool in the tool axis to the 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 TNC 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 TNC plunge-cuts into the pocket center.
3. If the tool is at the 2nd set-up clearance, it moves at rapid traverse FMAX to the 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. After this, the tool departs the contour tangentially 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 TNC retracts the tool at FMAX to the set-up clearance, or, if programmed, to the 2nd set-up clearance and then to the center of the pocket (end position = starting position).

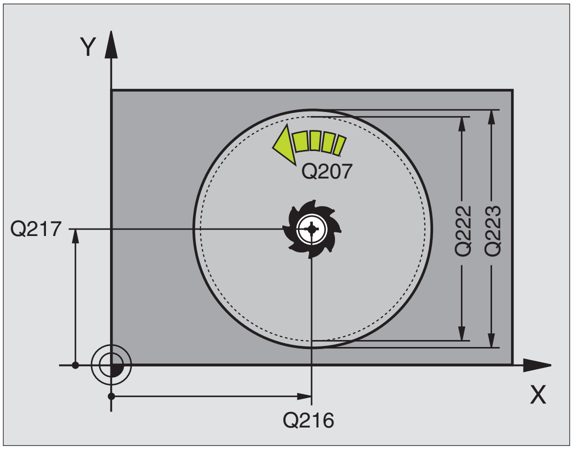


Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

If you 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.



Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

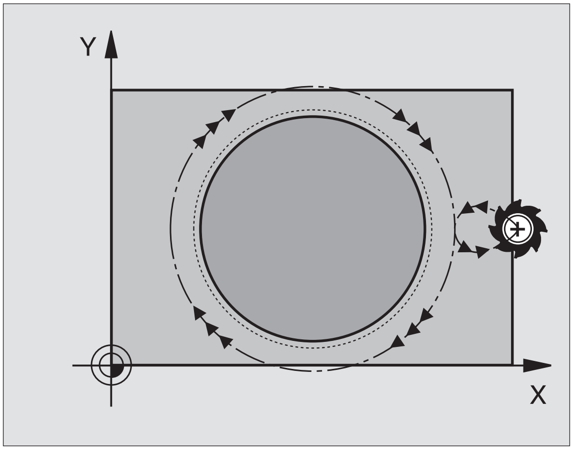
- **Set-up clearance** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **Depth** Q201 (incremental value): Distance between workpiece surface and bottom of pocket.
- **Feed rate for plunging** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting 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 and greater than the tool diameter.

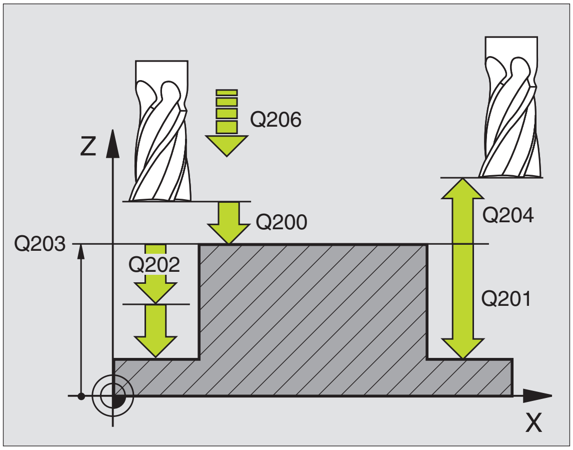
Example: NC blocks

```
42 CYCL DEF 214 C. 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 TNC automatically moves the tool in the tool axis to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and subsequently to the center of the pocket.
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 at a distance of approx. twice the tool radius.
3 If the tool is at the 2nd set-up clearance, it moves at rapid traverse FMAX to the 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 TNC retracts the tool at FMAX to the set-up clearance, or—if programmed—to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).





Before programming, note the following:

The TNC automatically pre-positions the tool in the tool axis and working plane.

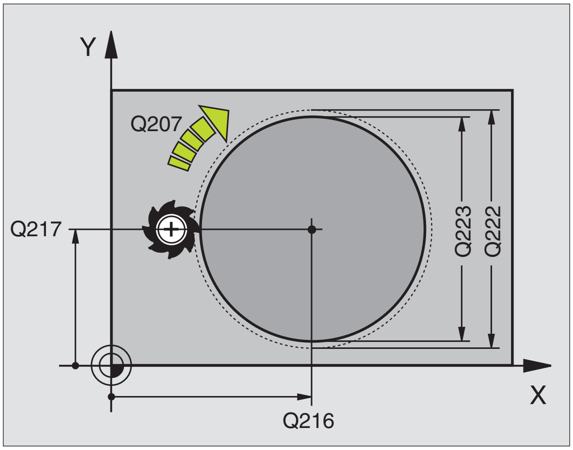
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH = 0, the cycle will not be executed.

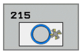
If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low feed rate for plunging.

Danger of collision!

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!



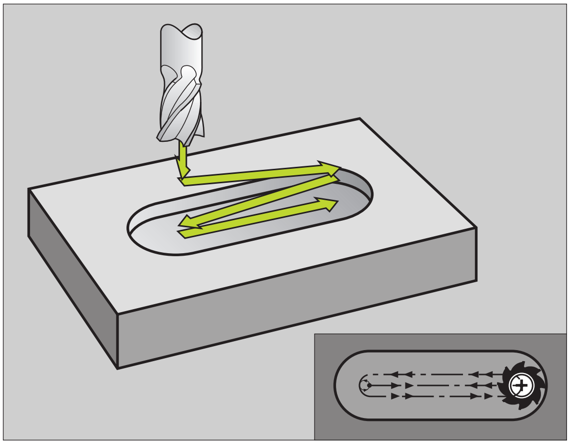


- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of stud.
- **`Feed rate for plunging`** Q206: Traversing speed of the tool in mm/min when moving to depth. If you are 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 (oblong hole) with reciprocating plunge-cut (Cycle 210)



Roughing

1. At rapid traverse, the TNC 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 TNC positions the tool to the 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 TNC 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

5. The TNC positions the tool in the center of the left circle and then moves it tangentially in a semicircle to the left end of the slot. 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 left circle.
7. At the end of the cycle, the tool is retracted at rapid traverse FMAX to the set-up clearance and—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

The TNC 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. If you program DEPTH = 0, the cycle will not be executed.

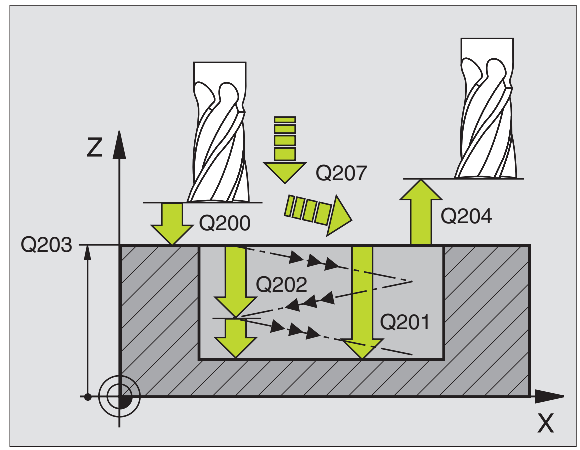
The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

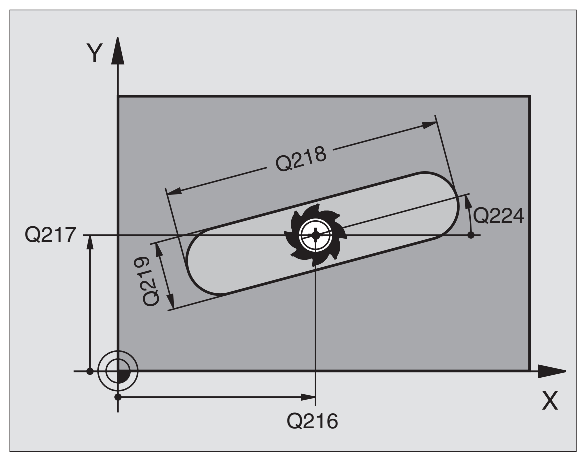
The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.

Danger of collision!

Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!





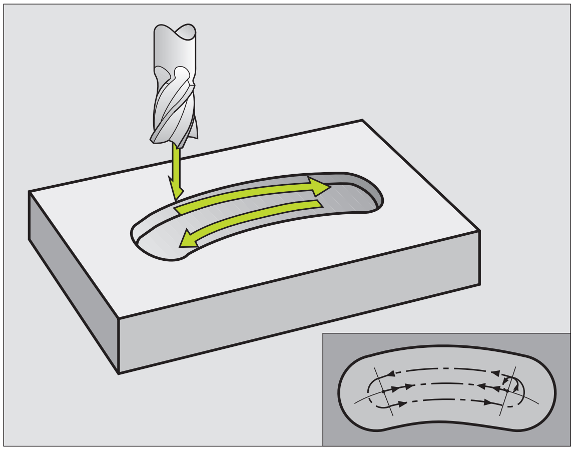
- **`Set-up clearance`** Q200 (incremental value): Distance between tool tip and workpiece surface.
- **`Depth`** Q201 (incremental value): Distance between workpiece surface and bottom of slot.
- **`Feed rate for milling`** Q207: Traversing speed of the tool in mm/min while milling.
- **`Plunging depth`** 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 set-up 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 TNC 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.
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Effective only during finishing if infeed for finishing is entered.

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
     Q206=150    ;FEED RATE FOR PLUNGING
```

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



Roughing

1 At rapid traverse, the TNC 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 (steps 2 to 3) is repeated until the programmed milling depth is reached.
4 For the purpose of face milling, the TNC moves the tool at the milling depth to the other end of the slot.

Finishing

5 The TNC 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 at rapid traverse FMAX to the set-up clearance and—if programmed—to the 2nd set-up clearance.

Before programming, note the following:

The TNC 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. 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 TNC otherwise cannot execute this cycle.

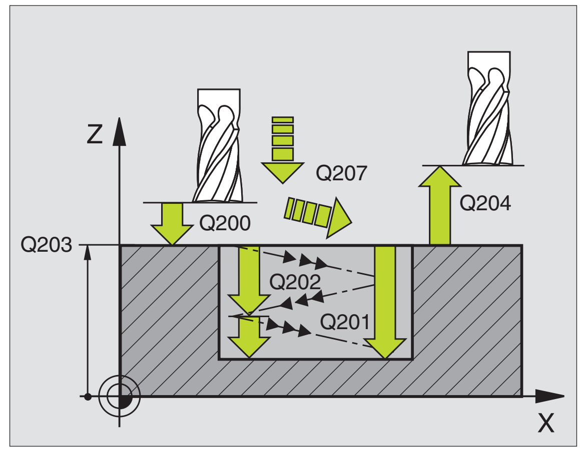
Use the machine parameter suppressDepthErr to define whether, if a positive depth is entered, the TNC should output an error message (on) or not (off).

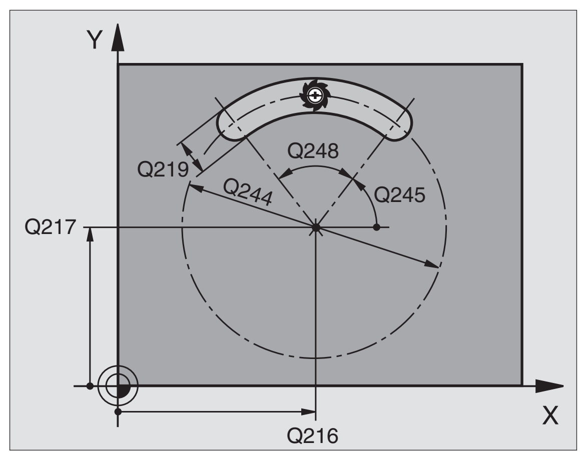
Danger of collision!

Keep in mind that the TNC reverses the calculation for pre-positioning when a **positive depth is entered.** This means that the tool moves at rapid traverse in the tool axis at safety clearance **below** the workpiece surface!

211

- **`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 set-up 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 TNC will carry out the roughing process only (slot milling).
- **`Starting angle`** Q245 (absolute value): Enter the polar angle of the starting point.



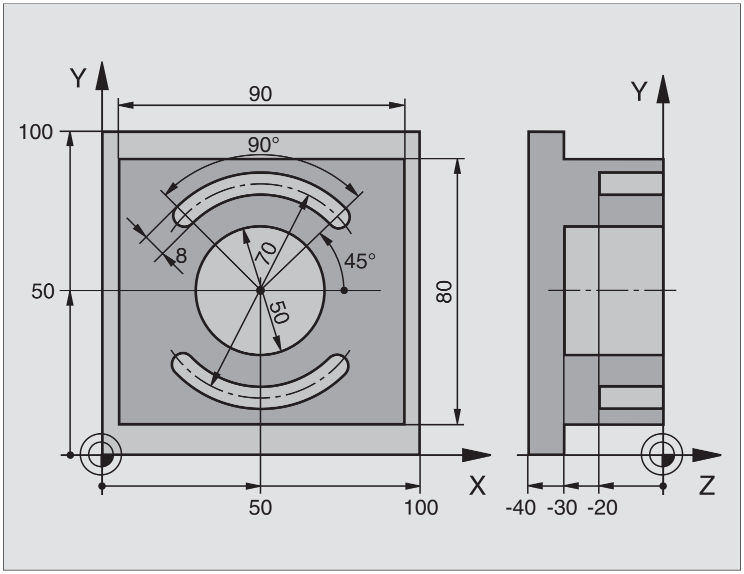


- **Angular length** Q248 (incremental value): Enter the angular length of the slot.
- **Infeed for finishing** Q338 (incremental value): Infeed per cut. Q338=0: Finishing in one infeed.
- **Feed rate for plunging** Q206: Traversing speed of the tool while moving to depth in mm/min. Effective only during finishing if infeed for finishing is entered.

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 DIA.
    Q219=12     ;SECOND SIDE LENGTH
    Q245=+45    ;STARTING ANGLE
    Q248=90     ;ANGULAR LENGTH
    Q338=5      ;INFEED FOR FINISHING
    Q206=150    ;FEED RATE FOR PLUNGING
```

Example: Milling pockets, studs and slots



0 BEGIN PGM C210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+6	Define the tool for roughing/finishing
4 TOOL DEF 2 L+0 R+3	Define slotting mill
5 TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
6 L Z+250 R0 FMAX	Retract the tool

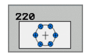
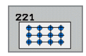
7 CYCL DEF 213 STUD FINISHING	Define cycle for machining the contour outside
Q200=2 ;SET-UP CLEARANCE	
Q201=-30 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q207=250 ;FEED RATE FOR MILLING	
Q203=+0 ;SURFACE COORDINATE	
Q204=20 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q218=90 ;FIRST SIDE LENGTH	
Q219=80 ;SECOND SIDE LENGTH	
Q220=0 ;CORNER RADIUS	
Q221=5 ;OVERSIZE	
8 CYCL CALL M3	Call cycle for machining the contour outside
9 CYCL DEF 5.0 CIRCULAR POCKET	Define CIRCULAR POCKET MILLING cycle
10 CYCL DEF 5.1 SET UP 2	
11 CYCL DEF 5.2 DEPTH -30	
12 CYCL DEF 5.3 PLNGNG 5 F250	
13 CYCL DEF 5.4 RADIUS 25	
14 CYCL DEF 5.5 F400 DR+	
15 L Z+2 R0 F MAX M99	Call CIRCULAR POCKET MILLING cycle
16 L Z+250 R0 F MAX M6	Tool change
17 TOOL CALL 2 Z S5000	Call slotting mill
18 CYCL DEF 211 CIRCULAR SLOT	Cycle definition for slot 1
Q200=2 ;SET-UP CLEARANCE	
Q201=-20 ;DEPTH	
Q207=250 ;FEED RATE FOR MILLING	
Q202=5 ;PECKING DEPTH	
Q215=0 ;MACHINING OPERATION	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q216=+50 ;CENTER IN 1ST AXIS	
Q217=+50 ;CENTER IN 2ND AXIS	
Q244=80 ;PITCH CIRCLE DIA.	
Q219=12 ;SECOND SIDE LENGTH	
Q245=+45 ;STARTING ANGLE	
Q248=90 ;ANGULAR LENGTH	

Q338=5 ;INFEED FOR FINISHING	
Q206=150 ;FEED RATE FOR PLUNGING	
19 CYCL CALL M3	Call cycle for slot 1
20 FN 0: Q245 = +225	New starting angle for slot 2
21 CYCL CALL	Call cycle for slot 2
22 L Z+250 R0 F MAX M2	Retract in the tool axis, end program
23 END PGM C210 MM	

8.4 Cycles for Machining Point Patterns

Overview

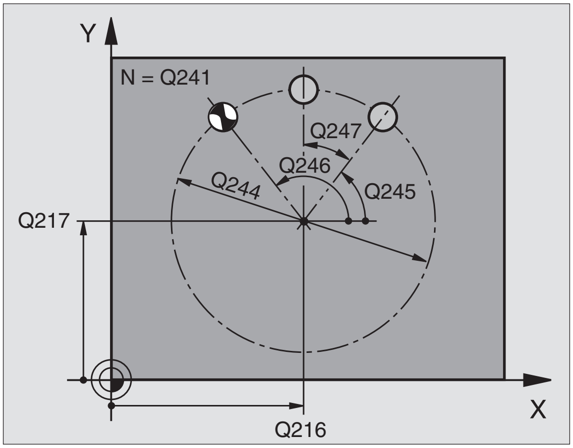
The TNC provides two cycles for machining point patterns directly:

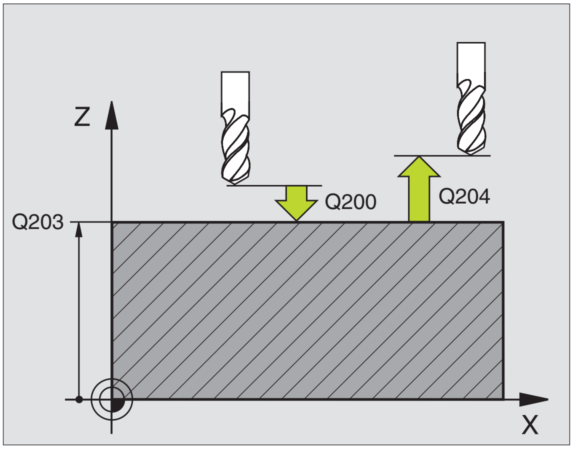
Cycle	Soft key
220 CIRCULAR PATTERN	
221 LINEAR PATTERN	

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

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 TNC moves the tool from its current position to the starting point for the first machining operation.

Sequence:

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

2 From this position the TNC executes the last defined fixed cycle.

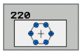
3 The tool then approaches on a straight line or circular arc the starting point for the next machining operation. The tool stops at the set-up clearance (or the 2nd set-up clearance).

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

Before programming, note the following:

Cycle 220 is DEF active, which means that Cycle 220 calls the last defined fixed cycle automatically.

If you combine Cycle 220 with one of the fixed cycles 200 to 209, 212 to 215, 251 to 265 or 267, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle 220 will be effective for the selected fixed cycle.



- **`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 TNC 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 TNC 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.
- **Moving to clearance height** Q301: Definition of how the tool is to move between machining processes.
  0: Move to the set-up clearance between operations.
  1: Move to the 2nd set-up clearance between machining operations.
- **Type of traverse? Line=0/Arc=1** Q365: Definition of the path function with which the tool is to move between machining operations.
  0: Move between operations on a straight line
  1: Move between operations on the pitch circle

Example: NC blocks

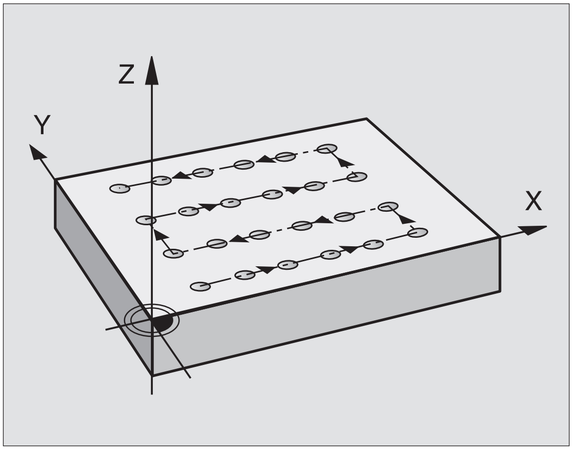
```
53 CYCL DEF 220 POLAR PATTERN
    Q216=+50   ;CENTER IN 1ST AXIS
    Q217=+50   ;CENTER IN 2ND AXIS
    Q244=80    ;PITCH CIRCLE DIA.
    Q245=+0    ;STARTING ANGLE
    Q246=+360  ;STOPPING ANGLE
    Q247=+0    ;STEPPING ANGLE
    Q241=8     ;NR OF REPETITIONS
    Q200=2     ;SET-UP CLEARANCE
    Q203=+30   ;SURFACE COORDINATE
    Q204=50    ;2ND SET-UP CLEARANCE
    Q301=1     ;MOVE TO CLEARANCE
    Q365=0     ;TYPE OF TRAVERSE
```

LINEAR PATTERN (Cycle 221)

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 209, 212 to 215, 265 to 267, 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 TNC automatically moves the tool from its current position to the starting point for the first machining operation.

Sequence:

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

2 From this position the TNC 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 the set-up clearance (or the 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.

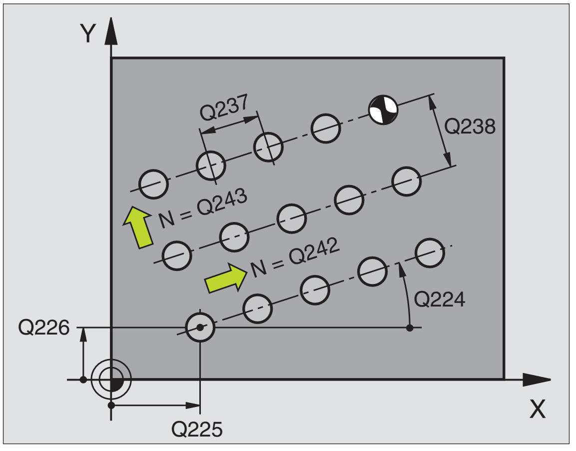
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

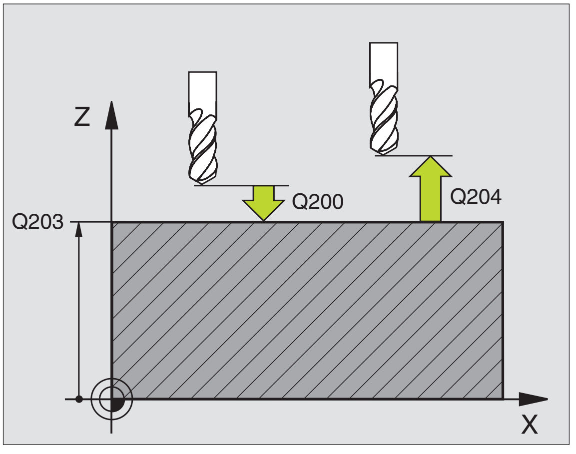
6 From this position the tool approaches the starting point for the next machining operation in the negative reference axis direction.

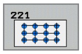
7 This process (6) is repeated until all machining operations in the second line have been executed.

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

9 All subsequent lines are processed in a reciprocating movement.





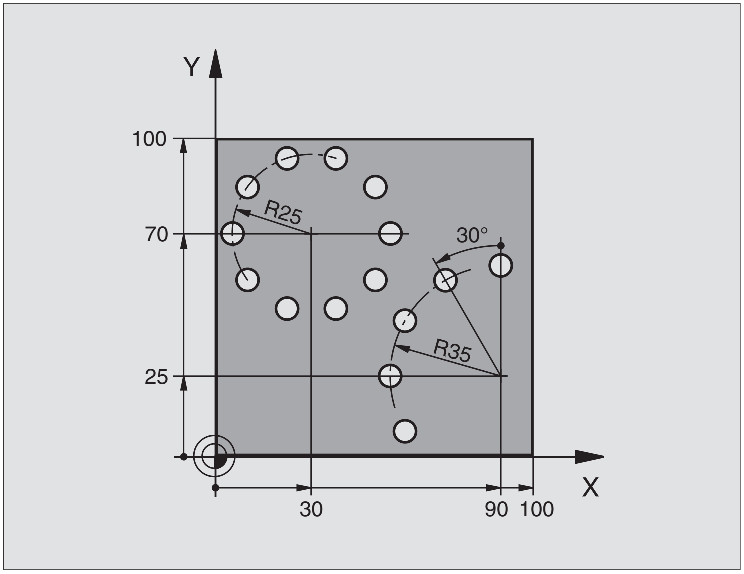


- **Starting point 1st axis** Q225 (absolute value): Coordinate of the starting point in the reference axis of the working plane.
- **Starting point 2nd axis** Q226 (absolute value): Coordinate of the starting point in the minor axis of the working plane.
- **Spacing in 1st axis** Q237 (incremental value): Spacing between each point on a line.
- **Spacing in 2nd axis** Q238 (incremental value): Spacing between each line.
- **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.
- **Moving to clearance height** Q301: Definition of how the tool is to move between machining processes.
  0: Move to the set-up clearance between operations.
  1: Move to the 2nd set-up clearance between the measuring points.

Example: NC blocks

```
54 CYCL DEF 221 CARTESIAN PATTERN
    Q225=+15   ;STARTING PNT 1ST AXIS
    Q226=+15   ;STARTING 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     ;MOVE TO CLEARANCE
```

Example: Circular hole patterns



`0 BEGIN PGM PATTERN MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Define the workpiece blank
`2 BLK FORM 0.2 Y+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+3`	Define the tool
`4 TOOL CALL 1 Z S3500`	Tool call
`5 L Z+250 R0 FMAX M3`	Retract the tool
`6 CYCL DEF 200 DRILLING`	Cycle definition: drilling
`Q200=2 ;SET-UP CLEARANCE`	
`Q201=-15 ;DEPTH`	
`Q206=250 ;FEED RATE FOR PLNGNG`	
`Q202=4 ;PLUNGING DEPTH`	
`Q210=0 ;DWELL TIME`	
`Q203=+0 ;SURFACE COORDINATE`	
`Q204=0 ;2ND SET-UP CLEARANCE`	
`Q211=0.25 ;DWELL TIME AT DEPTH`	

7 CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 1, CYCL 200 is called automatically,
Q216=+30 ;CENTER IN 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+70 ;CENTER IN 2ND AXIS	
Q244=50 ;PITCH CIRCLE DIA.	
Q245=+0 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=+0 ;STEPPING ANGLE	
Q241=10 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=0 ;TYPE OF TRAVERSE	
8 CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 2, CYCL 200 is called automatically,
Q216=+90 ;CENTER IN 1ST AXIS	Q200, Q203 and Q204 are effective as defined in Cycle 220.
Q217=+25 ;CENTER IN 2ND AXIS	
Q244=70 ;PITCH CIRCLE DIA.	
Q245=+90 ;STARTING ANGLE	
Q246=+360 ;STOPPING ANGLE	
Q247=30 ;STEPPING ANGLE	
Q241=5 ;NR OF REPETITIONS	
Q200=2 ;SET-UP CLEARANCE	
Q203=+0 ;SURFACE COORDINATE	
Q204=100 ;2ND SET-UP CLEARANCE	
Q301=1 ;MOVE TO CLEARANCE	
Q365=0 ;TYPE OF TRAVERSE	
9 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
10 END PGM PATTERN MM	

8.5 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 you enter in Cycle 14 CONTOUR GEOMETRY.

> The memory capacity for programming an SL cycle (all contour subprograms) is limited. The number of possible contour elements depends on the TNC's available working memory, the type of contour (inside or outside contour), and the number of subcontours.
>
> SL cycles conduct comprehensive and complex internal calculations as well as the resulting machining operations. For safety reasons, always run a graphical program test before machining! This is a simple way of finding out whether the TNC-calculated program will provide the desired results.

Characteristics of the subprograms

- Coordinate transformations are allowed. 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.
- If you use Q parameters, then only perform the calculations and assignments within the affected contour subprograms.

Example: Program structure: Machining with SL Cycles

```
0 BEGIN PGM SL2 MM
...
12 CYCL DEF 140 CONTOUR GEOMETRY ...
13 CYCL DEF 20 CONTOUR DATA ...
...
16 CYCL DEF 21 PILOT DRILLING ...
17 CYCL CALL
...
18 CYCL DEF 22 ROUGH-OUT ...
19 CYCL CALL
...
22 CYCL DEF 23 FLOOR FINISHING ...
23 CYCL CALL
...
26 CYCL DEF 24 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
```

Characteristics of the fixed cycles

- The TNC automatically positions the tool to the 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 on a tangential arc for side finishing.
- For floor finishing, the tool again approaches the workpiece on 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.

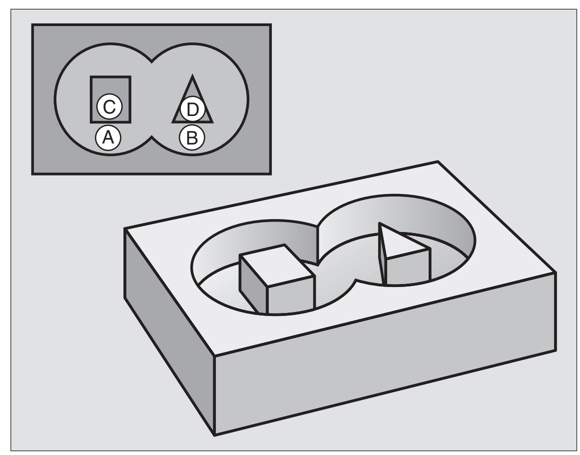
The machining data (such as milling depth, finishing allowance and set-up clearance) are entered as CONTOUR DATA in Cycle 20.

Overview of SL Cycles

Cycle	Soft key	Page
14 CONTOUR GEOMETRY (essential)	14 LBL 1...N	Page 256
20 CONTOUR DATA (essential)	20 CONTOUR DATA	Page 260
21 PILOT DRILLING (optional)	21	Page 261
22 ROUGH-OUT (essential)	22	Page 262
23 FLOOR FINISHING (optional)	23	Page 263
24 SIDE FINISHING (optional)	24	Page 264

CONTOUR (Cycle 14)

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



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.

14 LBL 1...N

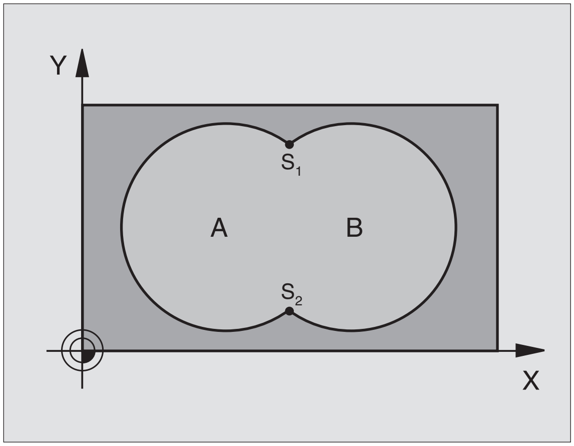
- **`Label numbers for the contour:`** Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.

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

> 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 S_1 and S_2. They do not have to be programmed.

The pockets are programmed as full circles.

Subprogram 1: Pocket A

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

Subprogram 2: Pocket B

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

Example: NC blocks

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

Area of inclusion

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

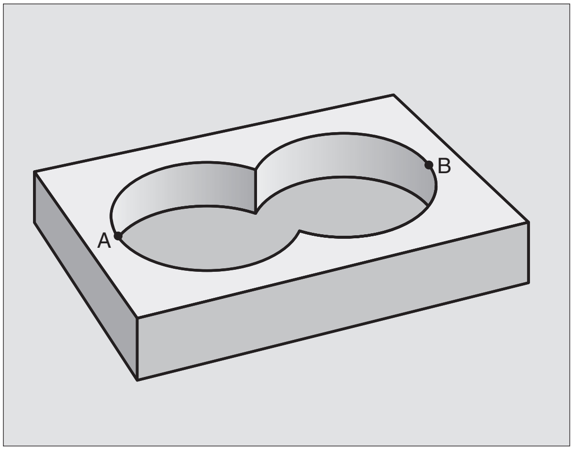
- The surfaces A and B must be pockets
- The first pocket (in Cycle 14) must start outside the second pocket

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:

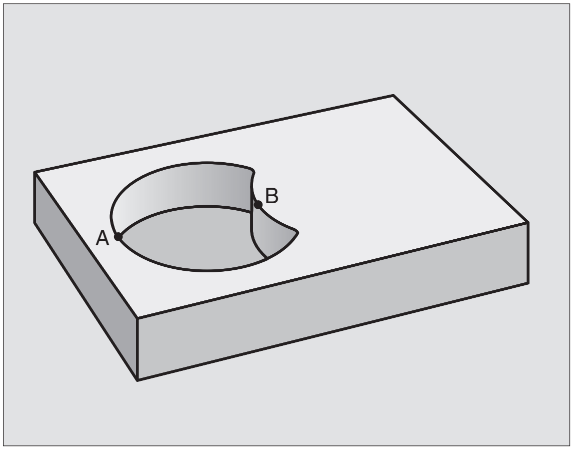
- Surface A must be a pocket and B an island
- A must start outside of B
- B must start inside of A.

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:

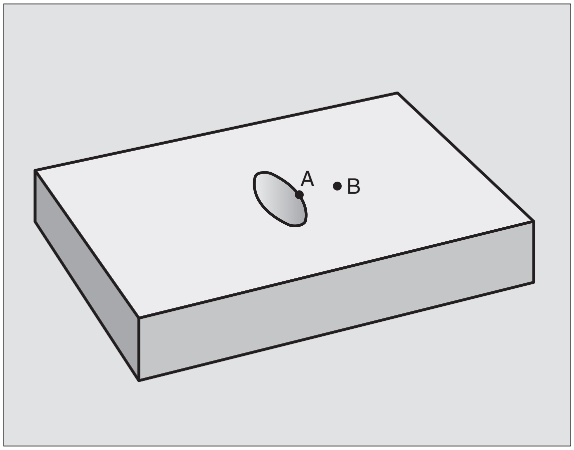
```
56 LBL 2
57 L X+90 Y+50 RL
58 CC X+65 Y+50
59 C X+90 Y+50 DR-
60 LBL 0
```



Area of intersection

Only the area where A and B overlap 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:

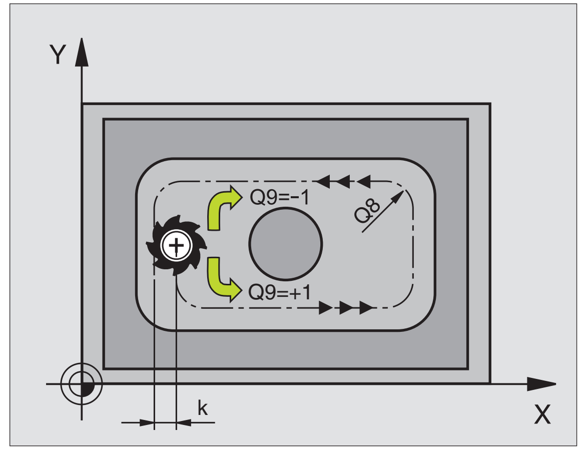
```
51 LBL 1
52 L X+60 Y+50 RR
53 CC X+35 Y+50
54 C X+60 Y+50 DR-
55 LBL 0
```

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.



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.

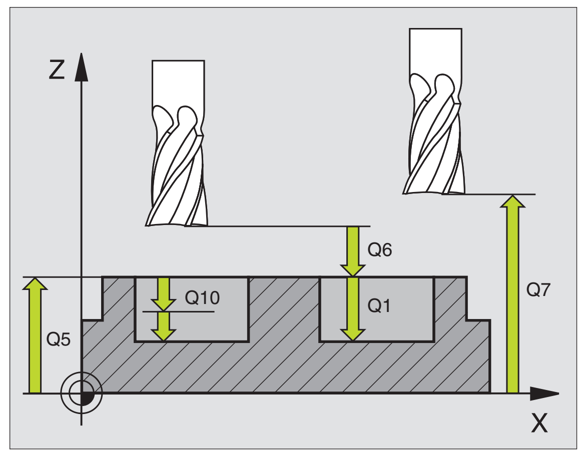
The algebraic sign for the cycle parameter DEPTH determines the working direction. If you program DEPTH=0, the TNC performs the cycle at the depth 0.

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 Q20 cannot be used as program parameters.

20 CONTOUR DATA

- **`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)
  - Counterclockwise (Q9 = +1 climb milling for pocket and island)



Example: NC blocks

```
57 CYCL DEF 20 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
```

PILOT DRILLING (Cycle 21)

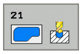
> When calculating the infeed points, the TNC does not account for the delta value **DR** programmed in a **TOOL CALL** block.
>
> In narrow areas, the TNC may not be able to carry out pilot drilling with a tool that is larger than the rough-out tool.

Process

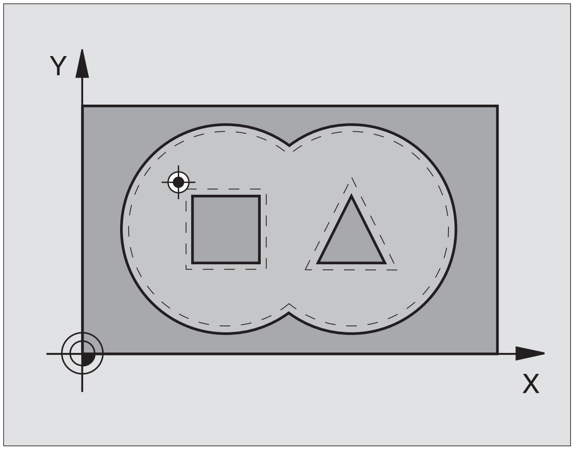
1. The tool drills from the current position to the first plunging depth at the programmed feed rate F.
2. When it reaches the first plunging depth, the tool retracts at rapid traverse FMAX to the starting position and advances again to the first plunging depth minus the advanced stop distance t.
3. 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 TNC repeats this process (1 to 4) until the programmed depth is reached.
6. After a dwell time at the hole bottom, the tool is returned to the starting position at rapid traverse FMAX for chip breaking.

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.

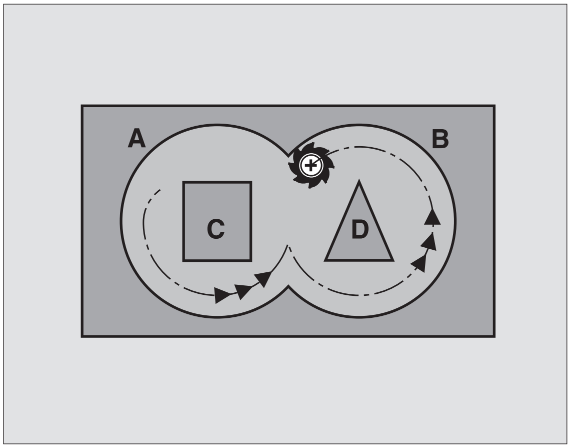


Example: NC blocks

```
58 CYCL DEF 21 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 the 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. In the next step the TNC moves the tool to the next plunging depth and repeats the roughing procedure until the program depth is reached.
5. Finally the TNC retracts the tool to the clearance height.



Before programming, note the following:

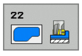
This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle 21.

You define the plunging behavior of Cycle 22 with parameter Q19 and with the tool table in the ANGLE and LCUTS columns:

- If Q19=0 is defined, the TNC always plunges perpendicularly, even if a plunge angle (ANGLE) is defined for the active tool.
- If you define the ANGLE=90°, the TNC plunges perpendicularly. The reciprocation feed rate Q19 is used as plunging feed rate.
- If the reciprocation feed rate Q19 is defined in Cycle 22 and ANGLE is defined between 0.1 and 89.999 in the tool table, the TNC plunges on a zigzag path at the defined ANGLE.
- If the reciprocation feed is defined in Cycle 22 and no ANGLE is in the tool table, the TNC displays an error message.

Example: NC blocks

```
59 CYCL DEF 22 ROUGH-OUT
    Q10=+5     ;PLUNGING DEPTH
    Q11=100    ;FEED RATE FOR PLUNGING
    Q12=350    ;FEED RATE FOR ROUGHING
    Q18=1      ;COARSE ROUGHING TOOL
    Q19=150    ;RECIPROCATION FEED RATE
    Q208=99999 ;RETRACTION 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.

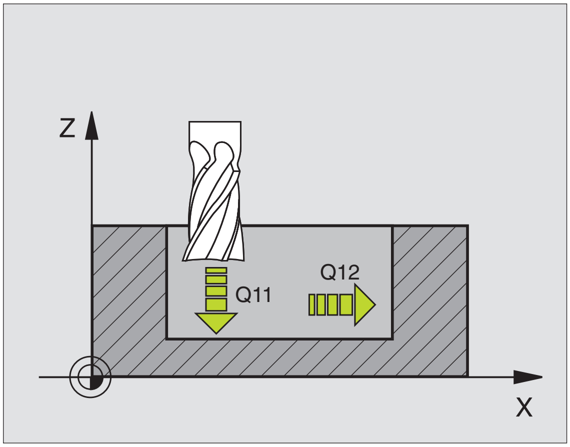
- **Coarse roughing tool number** Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If there was no coarse roughing, enter “0”; if you enter a value other than zero, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool.
  If the portion that is to be roughed cannot be approached from the side, the TNC will plunge-cut as in Q19. For this purpose you must enter the tool length LCUTS in the tool table TOOL.T, (see “Tool Data”, page 98) and define the maximum plunging ANGLE of the tool. The TNC will otherwise generate an error message.
- **Reciprocation feed rate** Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut.
- **Retraction feed rate** Q208: Traversing speed of the tool in mm/min when retracting after machining. If you enter Q208 = 0, the TNC retracts the tool at the feed rate in Q12.

FLOOR FINISHING (Cycle 23)

> 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) if there is sufficient room. If there is not enough room, the TNC moves the tool to depth vertically. The tool then clears the finishing allowance remaining from rough-out.

- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- **Feed rate for milling** Q12: Traversing speed for milling.



Example: NC blocks

```
60 CYCL DEF 23 FLOOR FINISHING
    Q11=100    ;FEED RATE FOR PLUNGING
    Q12=350    ;FEED RATE FOR ROUGHING
```

SIDE FINISHING (Cycle 24)

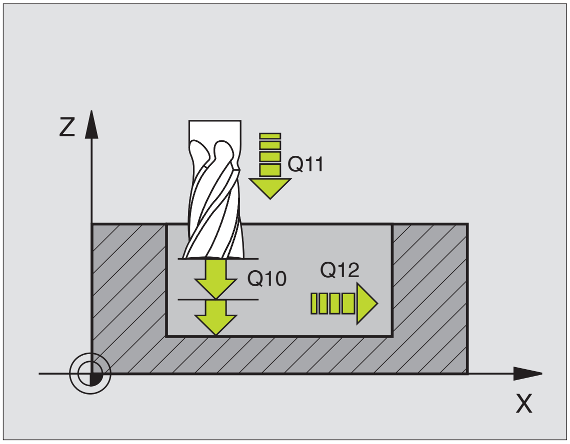
The subcontours are approached and departed on a tangential arc. Each subcontour is finish-milled separately.

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 and the allowance programmed in Cycle 20.

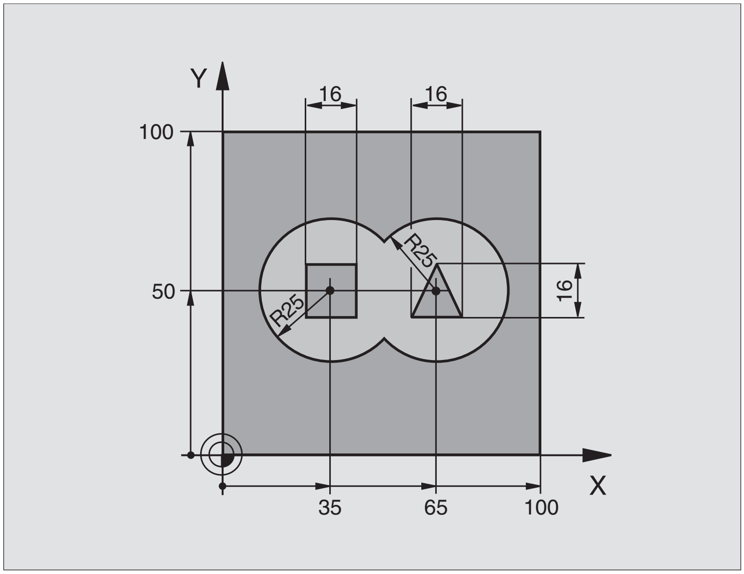


- **Direction of rotation ? Clockwise = -1** Q9: Machining direction:
  +1: Counterclockwise
  -1: Clockwise
- **Plunging depth** Q10 (incremental value): Dimension by which the tool plunges in each infeed.
- **Feed rate for plunging** Q11: Traversing speed of the tool during penetration.
- **Feed rate 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 remaining finishing allowance will be cleared.

Example: NC blocks

```
61 CYCL DEF 24 SIDE FINISHING
    Q9=+1       ;DIRECTION OF ROTATION
    Q10=+5      ;PLUNGING DEPTH
    Q11=100     ;FEED RATE FOR PLUNGING
    Q12=350     ;FEED RATE FOR ROUGHING
    Q14=+0      ;ALLOWANCE FOR SIDE
```

Example: Pilot drilling, roughing-out and finishing overlapping contours



0 BEGIN PGM C21 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the workpiece blank
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+6	Define tool: drill
4 TOOL DEF 2 L+0 R+6	Define the tool for roughing/finishing
5 TOOL CALL 1 Z S2500	Call tool: drill
6 L Z+250 R0 FMAX	Retract the tool
7 CYCL DEF 14.0 CONTOUR GEOMETRY	Define contour subprogram
8 CYCL DEF 14.1 CONTOUR LABEL 1/2/3/4	
9 CYCL DEF 20.0 CONTOUR DATA	Define general machining parameters
Q1=-20 ;MILLING DEPTH	
Q2=1 ;TOOL PATH OVERLAP	
Q3=+0.5 ;ALLOWANCE FOR SIDE	
Q4=+0.5 ;ALLOWANCE FOR FLOOR	
Q5=+0 ;SURFACE COORDINATE	
Q6=2 ;SET-UP CLEARANCE	
Q7=+100 ;CLEARANCE HEIGHT	
Q8=0.1 ;ROUNDING RADIUS	
Q9=-1 ;DIRECTION OF ROTATION	

`10 CYCL DEF 21.0 PILOT DRILLING`	Cycle definition: Pilot drilling
`Q10=5 ;PLUNGING DEPTH`	
`Q11=250 ;FEED RATE FOR PLUNGING`	
`Q13=2 ;ROUGH-OUT TOOL`	
`11 CYCL CALL M3`	Cycle call: Pilot drilling
`12 L Z+250 R0 FMAX M6`	Tool change
`13 TOOL CALL 2 Z S3000`	Call the tool for roughing/finishing
`14 CYCL DEF 22.0 ROUGH-OUT`	Cycle definition: Rough-out
`Q10=5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=350 ;FEED RATE FOR ROUGHING`	
`Q18=0 ;COARSE ROUGHING TOOL`	
`Q19=150 ;RECIPROCATION FEED RATE`	
`Q208=30000 ;RETRACTION FEED RATE`	
`15 CYCL CALL M3`	Cycle call: Rough-out
`16 CYCL DEF 23.0 FLOOR FINISHING`	Cycle definition: Floor finishing
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=200 ;FEED RATE FOR ROUGHING`	
`Q208=30000 ;RETRACTION FEED RATE`	
`17 CYCL CALL`	Cycle call: Floor finishing
`18 CYCL DEF 24.0 SIDE FINISHING`	Cycle definition: Side finishing
`Q9=+1 ;DIRECTION OF ROTATION`	
`Q10=5 ;PLUNGING DEPTH`	
`Q11=100 ;FEED RATE FOR PLUNGING`	
`Q12=400 ;FEED RATE FOR ROUGHING`	
`Q14=+0 ;ALLOWANCE FOR SIDE`	
`19 CYCL CALL`	Cycle call: Side finishing
`20 L Z+250 R0 FMAX M2`	Retract in the tool axis, end program

21 LBL 1	Contour subprogram 1: left pocket
22 CC X+35 Y+50	
23 L X+10 Y+50 RR	
24 C X+10 DR-	
25 LBL 0	
26 LBL 2	Contour subprogram 2: right pocket
27 CC X+65 Y+50	
28 L X+90 Y+50 RR	
29 C X+90 DR-	
30 LBL 0	
31 LBL 3	Contour subprogram 3: square left island
32 L X+27 Y+50 RL	
33 L Y+58	
34 L X+43	
35 L Y+42	
36 L X+27	
37 LBL 0	
38 LBL 4	Contour subprogram 4: triangular right island
39 L X+65 Y+42 RL	
40 L X+57	
41 L X+65 Y+58	
42 L X+73 Y+42	
43 LBL 0	
44 END PGM C21 MM	

8.6 Cycles for Multipass Milling

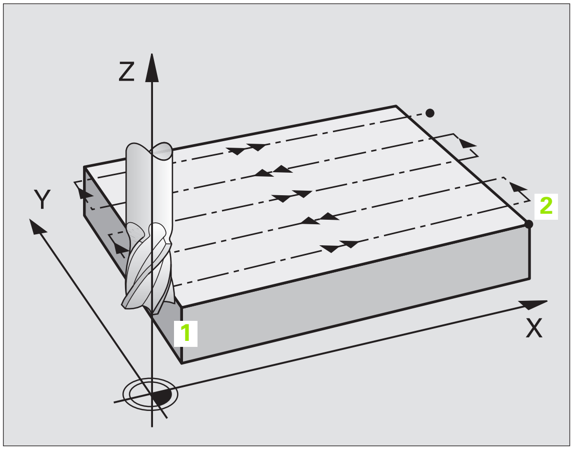
Overview

The TNC offers four cycles for machining surfaces with the following characteristics:

- 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	230
231 RULED SURFACE For oblique, inclined or twisted surfaces	231
232 FACE MILLING For level rectangular surfaces, with indicated oversizes and multiple infeeds	232

MULTIPASS MILLING (Cycle 230)

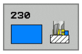


1. From the current position in the working plane, the TNC positions the tool at rapid traverse FMAX to the starting point **1**; the TNC moves the tool by its radius to the left and upward.
2. The tool then moves at FMAX in the tool axis to the set-up clearance. From there it approaches the programmed starting position in the tool axis at the feed rate for plunging.
3. The tool then moves at the programmed feed rate for milling to the end point **2**. The TNC calculates the end point from the programmed starting point, the program length, and the tool radius.
4. The TNC 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 at FMAX to the set-up clearance.

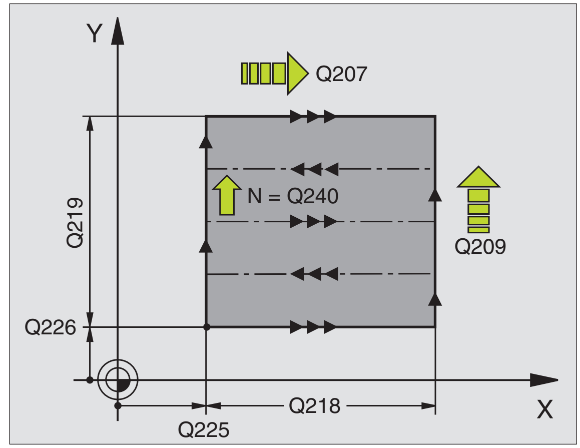
Before programming, note the following:

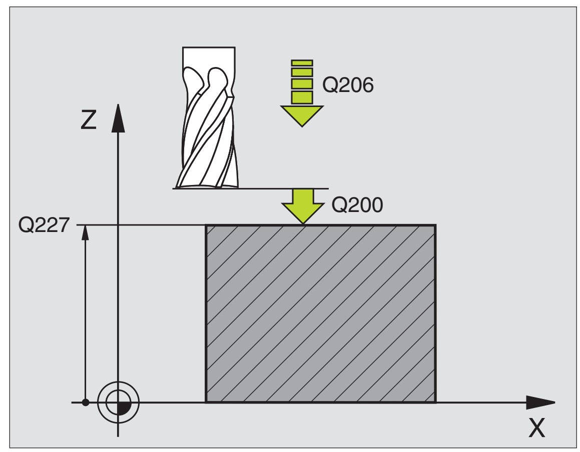
From the current position, the TNC positions the tool at the starting point, first in the working plane and then in the spindle axis.

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 the 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 the 2nd axis.
- **Number of cuts** Q240: Number of passes to be made over the width.
- **Feed rate for plunging** Q206: 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 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.



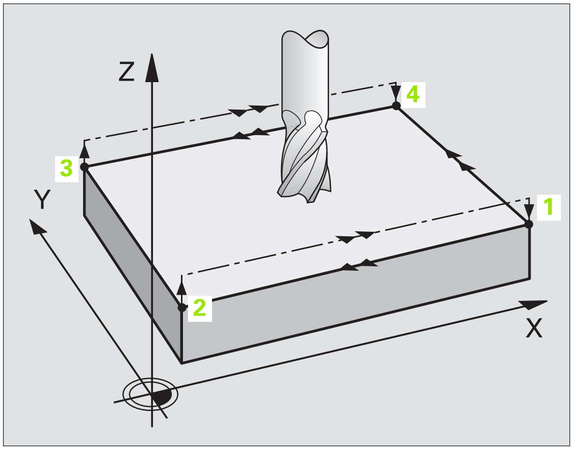


Example: NC blocks

```
71 CYCL DEF 230 MULTIPASS MILLING
    Q225=+10   ;STARTING PNT 1ST AXIS
    Q226=+12   ;STARTING PNT 2ND AXIS
    Q227=+2.5  ;STARTING PNT 3RD 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 TNC 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 at 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 TNC moves the tool back to the last traversed Z value.
5 Then the TNC 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 TNC 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.

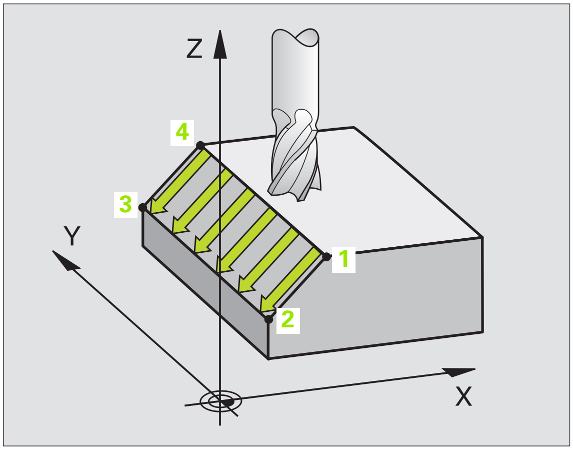


Cutting motion

The starting point, and therefore the milling direction, is selectable because the TNC 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.

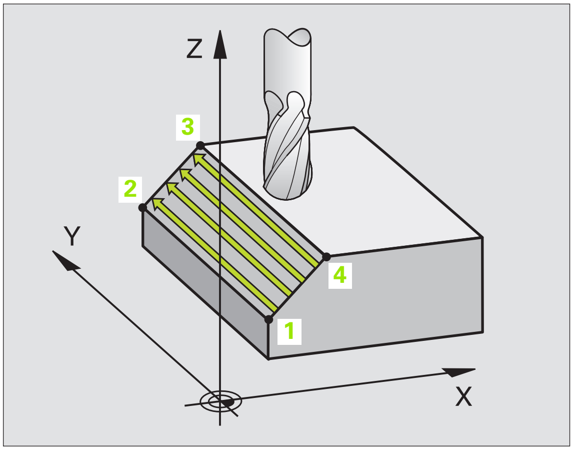
If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways:

- 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.

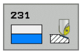


Before programming, note the following:

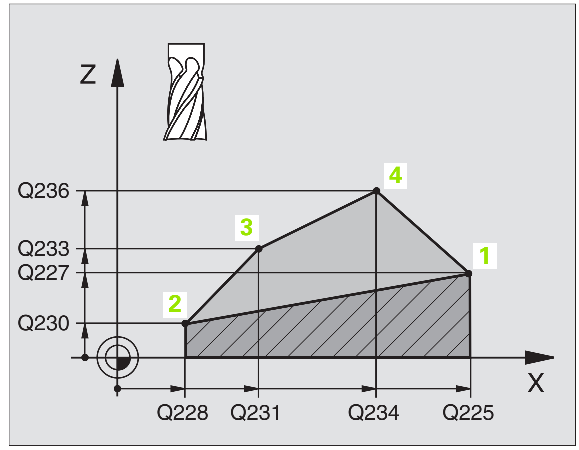
The TNC positions the tool from the current position in a linear 3-D movement to the starting point **1**. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

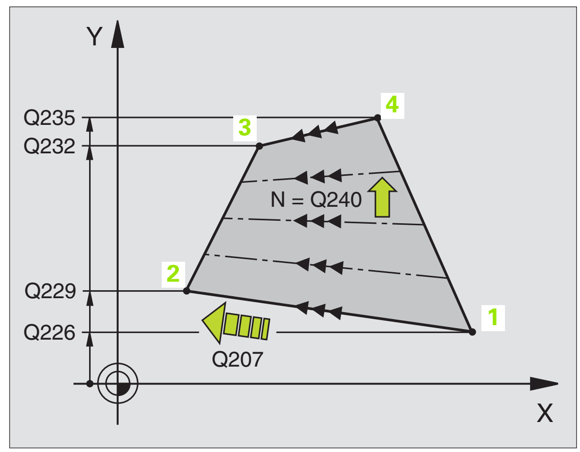
The TNC moves the tool with radius compensation R0 to the programmed positions.

If required, use a center-cut end mill (ISO 1641).



- **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 TNC performs the first step at half the programmed feed rate.

Example: NC blocks

```
72 CYCL DEF 231 RULED SURFACE
    Q225=+0     ;STARTING PNT 1ST AXIS
    Q226=+5     ;STARTING 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
```

FACE MILLING (Cycle 232)

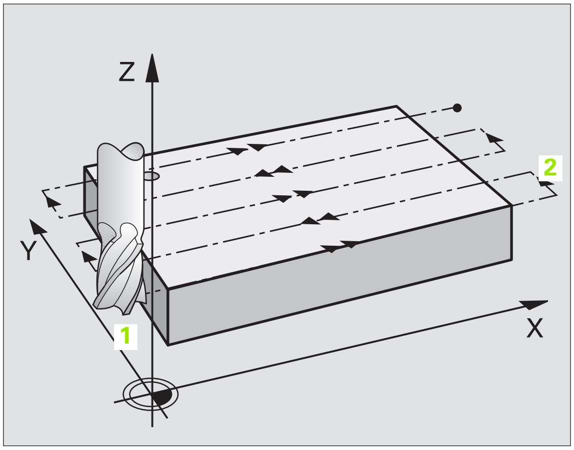
Cycle 232 is used to face mill a level surface in multiple infeeds while taking the finishing allowance into account. Three machining strategies are available:

- **Strategy Q389=0:** Meander machining, stepover outside the surface being machined
- **Strategy Q389=1:** Meander machining, stepover within the surface being machined
- **Strategy Q389=2:** Line-by-line machining, retraction and stepover at the positioning feed rate

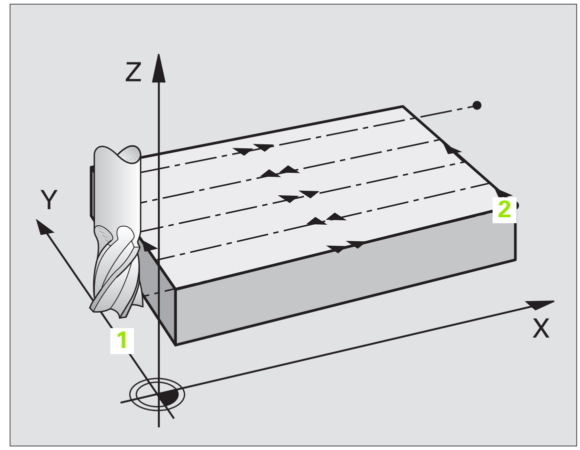
1 From the current position, the TNC positions the tool at rapid traverse (FMAX) to the starting position using positioning logic **1**. If the current position in the spindle axis is greater than the 2nd set-up clearance, the control positions the tool first in the machining plane and then in the spindle axis. Otherwise it first moves to the 2nd set-up clearance and then in the machining plane. The starting point in the machining plane is offset from the edge of the workpiece by the tool radius and the safety clearance to the side.
2 The tool then moves in the spindle axis at the positioning feed rate to the first plunging depth calculated by the control.

Strategy Q389=0

3 The tool then advances to the stopping point **2** at the feed rate for milling. The end point lies **outside** the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.
4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.
5 The tool then moves back in the direction of the starting point **1**.
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the next machining depth is plunged to.
7 In order to avoid non-productive motions, the surface is then machined in reverse direction.
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.
9 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.



Strategy Q389=1



3 The tool then advances to the stopping point **2** at the feed rate for milling. The end point lies **within** the surface. The control calculates the end point from the programmed starting point, the programmed length and the tool radius.

4 The TNC offsets the tool to the starting point in the next pass at the pre-positioning feed rate. The offset is calculated from the programmed width, the tool radius and the maximum path overlap factor.

5 The tool then moves back in the direction of the starting point **1**. The motion to the next line occurs within the workpiece borders.

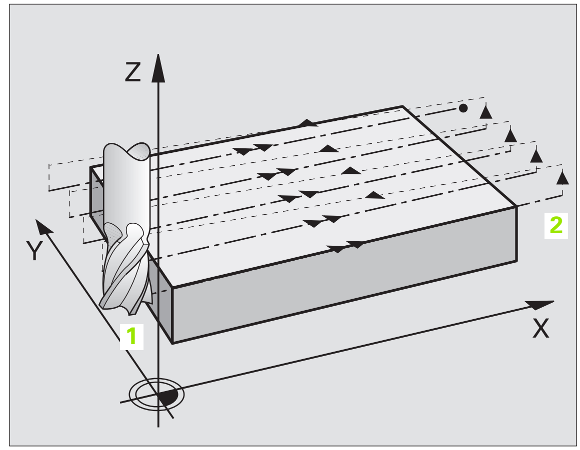
6 The process is repeated until the programmed surface has been completed. At the end of the last pass, the next machining depth is plunged to.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

9 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.

Strategy Q389=2



3 The tool then advances to the stopping point **2** at the feed rate for milling. The end point lies outside the surface. The control calculates the end point from the programmed starting point, the programmed length, the programmed safety clearance to the side and the tool radius.

4 The TNC positions the tool in the spindle axis to the set-up clearance over the current infeed depth, and then moves at the pre-positioning feed rate directly back to the starting point in the next line. The TNC calculates the offset from the programmed width, the tool radius and the maximum path overlap factor.

5 The tool then returns to the current infeed depth and moves in the direction of the next end point **2**.

6 The milling process is repeated until the programmed surface has been completed. At the end of the last pass, the next machining depth is plunged to.

7 In order to avoid non-productive motions, the surface is then machined in reverse direction.

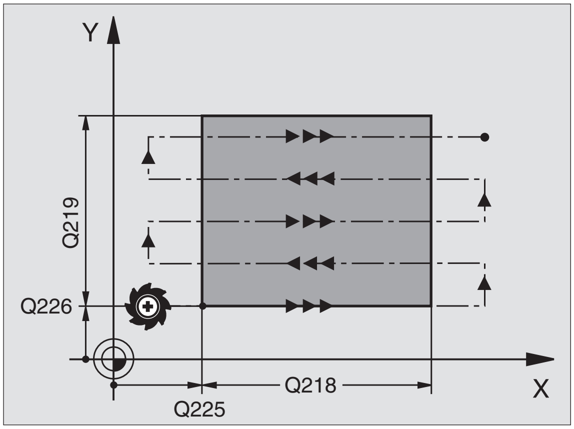
8 The process is repeated until all infeeds have been machined. In the last infeed, simply the finishing allowance entered is milled at the finishing feed rate.

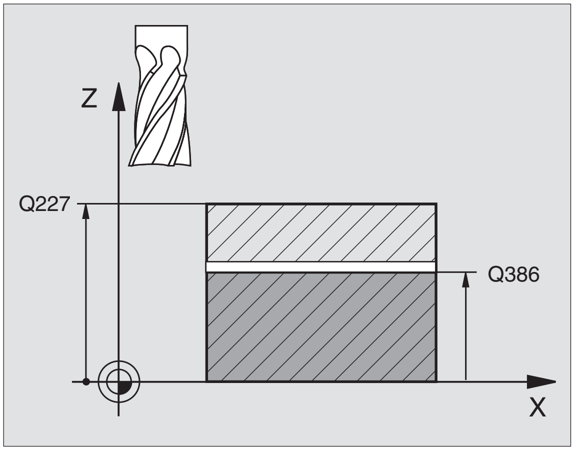
9 At the end of the cycle, the tool is retracted at FMAX to the 2nd set-up clearance.

Before programming, note the following:

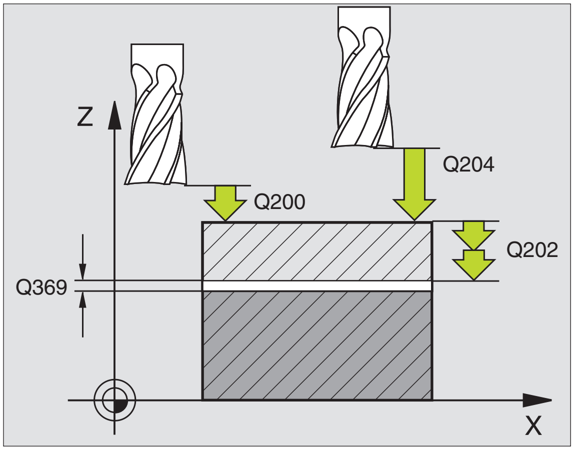
Enter the 2nd set-up clearance in Q204 so that no collision between tool and clamping devices can occur.

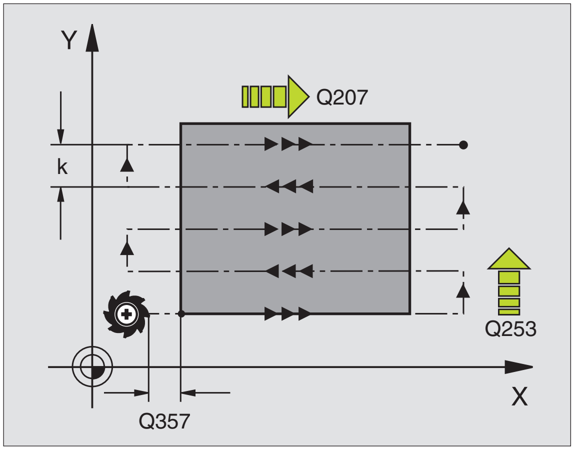
- **Machining strategy (0/1/2)** Q389: Specify how the TNC is to machine the surface:
  0: Meander machining, stepover at positioning feed rate outside the surface to be machined
  1: Meander machining, stepover at feed rate for milling within the surface to be machined
  2: Line-by-line machining, retraction and stepover at the positioning feed rate
- **Starting point in 1st axis** Q225 (absolute value): Starting point coordinate of the surface to be machined 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): Coordinate of the workpiece surface used to calculate the infeeds.
- **End point in 3rd axis** Q386 (absolute value): Coordinate in the spindle axis to which the surface is to be face milled.
- **First side length** Q218 (incremental value): Length of the surface to be machined in the reference axis of the working plane. Use the algebraic sign to specify the direction of the first milling path in reference to the **starting point in the 1st axis.**
- **Second side length** Q219 (incremental value): Length of the surface to be machined in the minor axis of the working plane. Use the algebraic sign to specify the direction of the first stepover in reference to the **starting point in the 2nd axis.**





- **Maximum plunging depth** Q202 (incremental value): **Maximum** amount that the tool is advanced each time. The TNC calculates the actual plunging depth from the difference between the end point and starting point of the tool axis (taking the finishing allowance into account), so that uniform plunging depths are used each time.
- **Allowance for floor** Q369 (incremental value): Distance used for the last infeed.
- **Max. path overlap factor** Q370: **Maximum** stepover factor k. The TNC calculates the actual stepover from the second side length (Q219) and the tool radius so that a constant stepover is used for machining. If you have entered a radius R2 in the tool table (e.g. tooth radius when using a face-milling cutter), the TNC reduces the stepover accordingly.
- **Feed rate for milling** Q207: Traversing speed of the tool in mm/min while milling.
- **Feed rate for finishing** Q385: Traversing speed of the tool in mm/min while milling the last infeed.
- **Feed rate for pre-positioning** Q253: Traversing speed of the tool in mm/min when approaching the starting position and when moving to the next pass. If you are moving the tool transversely to the material (Q389=1), the TNC moves the tool at the feed rate for milling Q207.



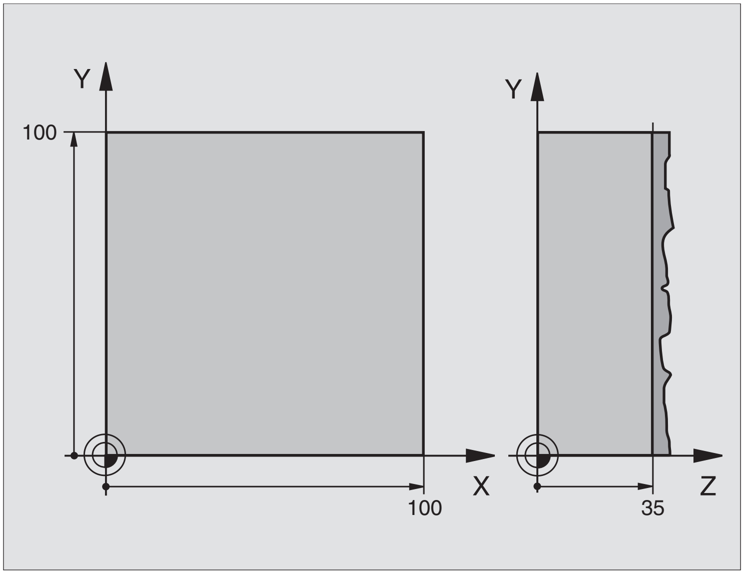


- **Set-up clearance** Q200 (incremental value): Distance between tool tip and the starting position in the tool axis. If you are milling with machining strategy Q389=2, the TNC moves the tool at the set-up clearance over the current plunging depth to the starting point of the next pass.
- **Clearance to side** Q357 (incremental value): Safety clearance to the side of the workpiece when the tool approaches the first plunging depth, and distance at which the stepover occurs if the machining strategy Q389=0 or Q389=2 is used.
- **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

```
71 CYCL DEF 232 FACE MILLING
    Q389=2      ;STRATEGY
    Q225=+10    ;STARTING PNT 1ST AXIS
    Q226=+12    ;STARTING PNT 2ND AXIS
    Q227=+2.5   ;STARTING PNT 3RD AXIS
    Q386=-3     ;END POINT IN 3RD AXIS
    Q218=150    ;FIRST SIDE LENGTH
    Q219=75     ;SECOND SIDE LENGTH
    Q202=2      ;MAX. PLUNGING DEPTH
    Q369=0.5    ;ALLOWANCE FOR FLOOR
    Q370=1      ;MAX. OVERLAP
    Q207=500    ;FEED RATE FOR MILLING
    Q385=800    ;FEED RATE FOR FINISHING
    Q253=2000   ;F PRE-POSITIONING
    Q200=2      ;SET-UP CLEARANCE
    Q357=2      ;CLEARANCE TO SIDE
    Q204=2      ;2ND SET-UP CLEARANCE
```

Example: Multipass milling



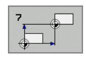
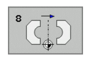
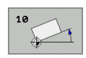
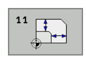
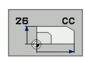
`0 BEGIN PGM C230 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z+0`	Define the workpiece blank
`2 BLK FORM 0.2 X+100 Y+100 Z+40`	
`3 TOOL DEF 1 L+0 R+5`	Define the tool
`4 TOOL CALL 1 Z S3500`	Tool call
`5 L Z+250 R0 FMAX`	Retract the tool
`6 CYCL DEF 230 MULTIPASS MILLING`	Cycle definition: MULTIPASS MILLING
`Q225=+0 ;START IN 1ST AXIS`	
`Q226=+0 ;START IN 2ND AXIS`	
`Q227=+35 ;START IN 3RD AXIS`	
`Q218=100 ;FIRST SIDE LENGTH`	
`Q219=100 ;SECOND SIDE LENGTH`	
`Q240=25 ;NUMBER OF CUTS`	
`Q206=250 ;FEED RATE FOR PLNGNG`	
`Q207=400 ;FEED RATE FOR MILLING`	
`Q209=150 ;STEPOVER FEED RATE`	
`Q200=2 ;SET-UP CLEARANCE`	

7 L X+-25 Y+0 R0 FMAX M3	Pre-position near the starting point
8 CYCL CALL	Call the cycle
9 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
10 END PGM C230 MM	

8.7 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 TNC 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 plane	
11 SCALING FACTOR For increasing or reducing the size of contours	
26 AXIS-SPECIFIC SCALING FACTOR For increasing or reducing the size of contours with scaling factors for each axis	

Effect of coordinate transformations

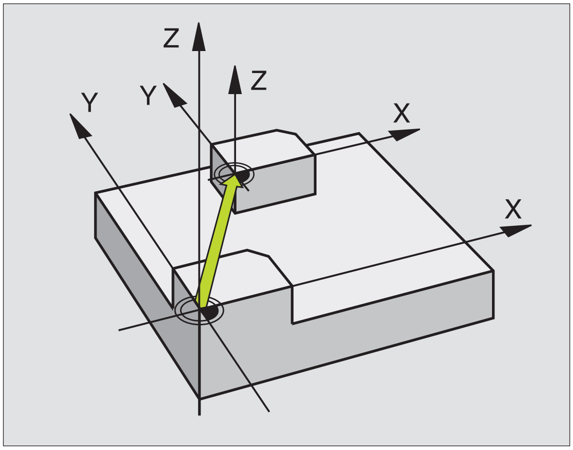
Beginning of effect: A coordinate transformation becomes effective as soon as it is defined—it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

- Define cycles for basic behavior with a new value, such as scaling factor 1.0.
- Execute the miscellaneous function M02 or M30, or an END PGM block (depending on the clearMode machine parameter)
- Select a new program.

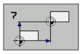
DATUM SHIFT (Cycle 7)

A DATUM SHIFT allows machining operations to be repeated at various locations on the workpiece.



Effect

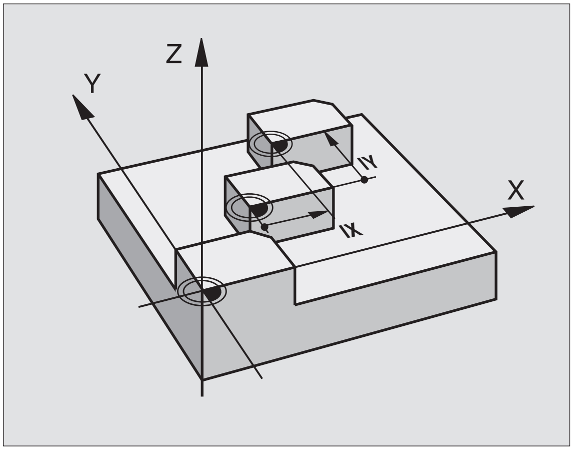
When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display. Input of rotary axes is also permitted.



- **`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)

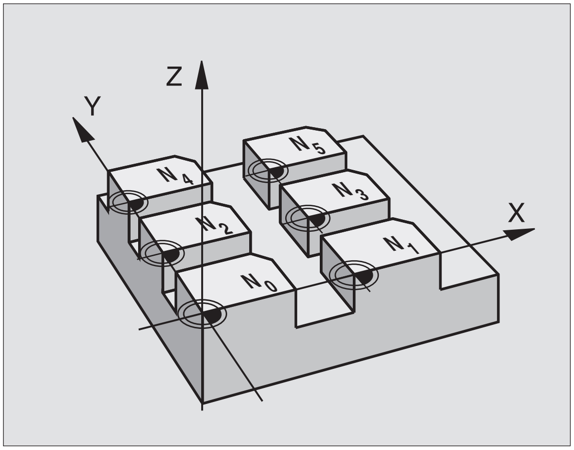
The datum table used depends on the operating mode or is selectable:

- Program Run operating modes: “zeroshift.d” table
- Test-Run operating mode: “simzeroshift.d” table

Datums from a datum table are referenced to the current 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.

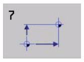


Function

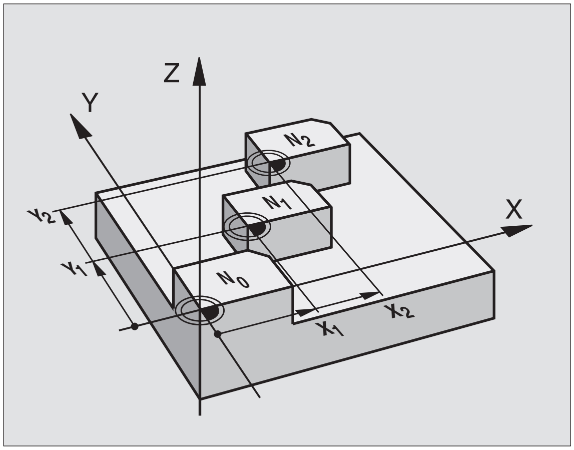
Datum tables are used for

- 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 TNC activates the datum number entered in the Q parameter.



Cancellation

- Call a datum shift to the coordinates X=0; Y=0 etc. from the datum table.
- Execute a datum shift to the coordinates X=0, Y=0 etc. directly with a cycle definition.

Example: NC blocks

```
77 CYCL DEF 7.0 DATUM SHIFT
78 CYCL DEF 7.1 #5
```

Edit the datum table in the Programming and Editing mode of operation.

Select the datum table in the **`Programming and Editing`** mode of operation.

PGM MGT

- To call the file manager, press the PGM MGT key, see “File Management: Fundamentals”, page 59.
- 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
Select end of table	END
Go to previous page	PAGE
Go to next page	PAGE
Insert line (only possible at end of table)	INSERT LINE
Delete line	DELETE LINE
Find	FIND
Go to beginning of line	BEGIN LINE
Go to end of line	END LINE
Copy the present value	COPY FIELD COPY
Insert the copied value	PASTE FIELD PASTE
Add the entered number of lines (reference points) to the end of the table	APPEND N LINES

Configuring the datum table

If you do not wish to define a datum table for an active axis, press the DEL key. Then the TNC clears the numerical value from the corresponding input field?

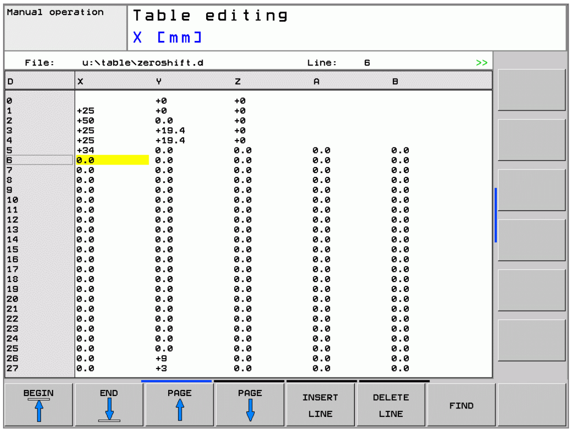
To leave a datum table

Select a different type of file in file management and choose the desired file.

> After you have changed a value in a datum table, you must save the change with the ENT key. Otherwise the change may not be included during program run.

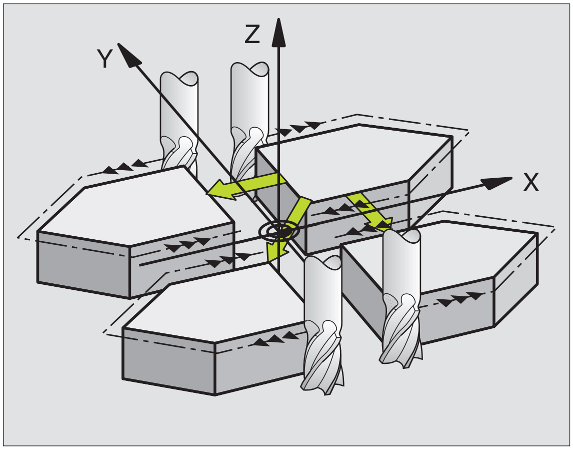
Status displays

The additional status display shows the values of the active datum shift. (see "Coordinate transformations" on page 36):



MIRROR IMAGE (Cycle 8)

The TNC can machine the mirror image of a contour in the working plane.



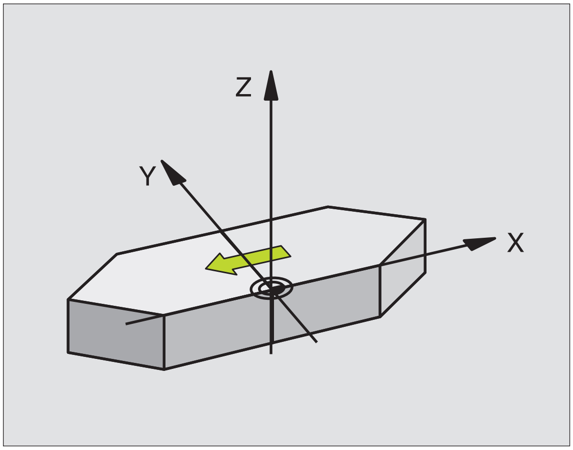
Effect

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.

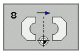
- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

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.



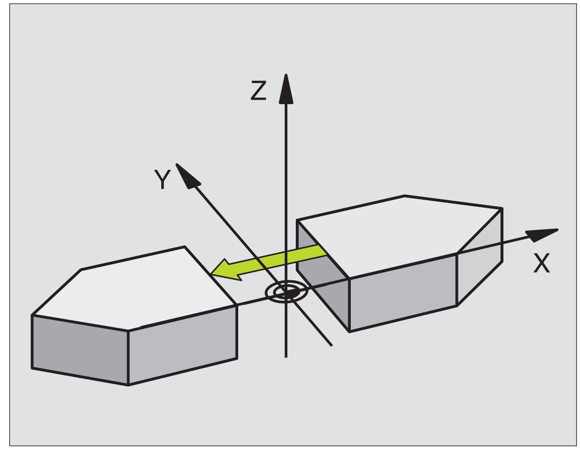
> If you mirror only one axis, the machining direction is reversed for the milling cycles (Cycles 2xx).



- **`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 TNC 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.

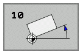
Reference axis for the rotation angle:

- X/Y plane X axis
- Y/Z plane Y axis
- Z/X plane Z axis

Before programming, note the following:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

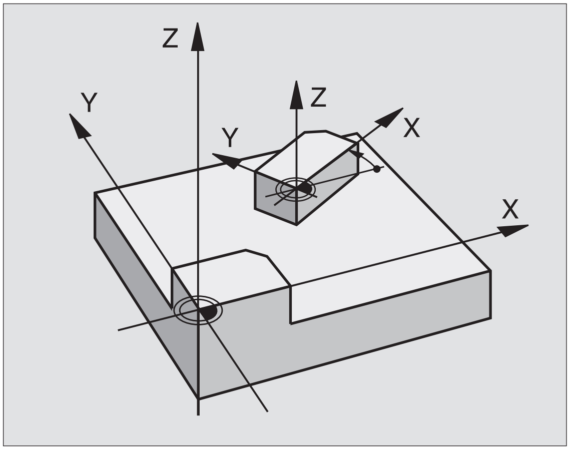
After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.

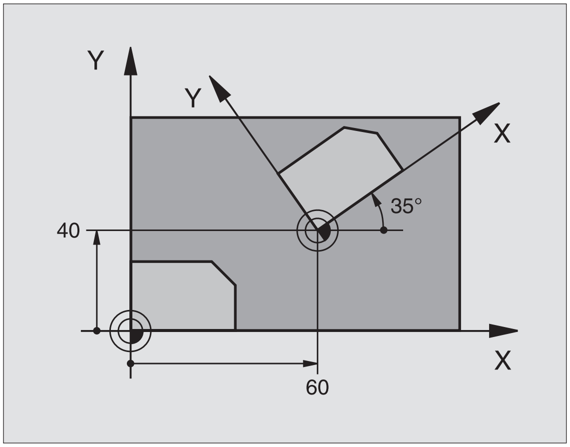


- **Rotation:** Enter the rotation angle in degrees (°). Input range: –360° to +360° (absolute or incremental).

Cancellation

Program the ROTATION cycle once again with a rotation angle of 0°.



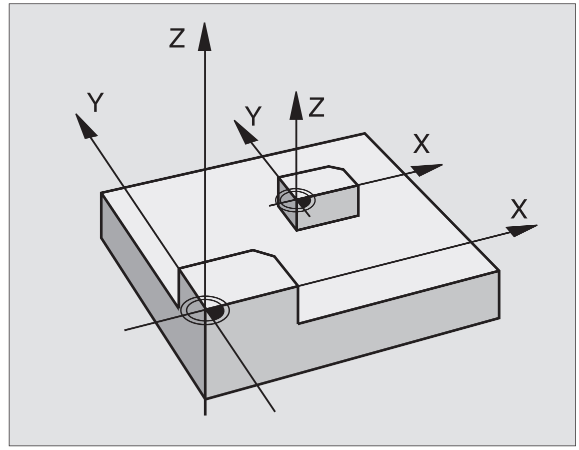


Example: NC blocks

```
12 CALL LBL 1
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 LBL 1
```

SCALING FACTOR (Cycle 11)

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.



Effect

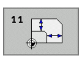
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor has an effect on

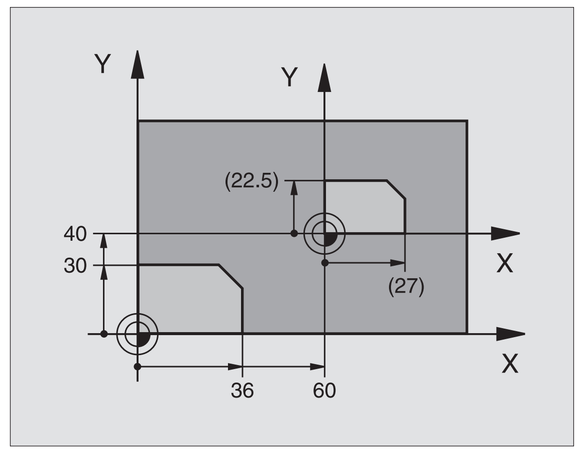
- All three coordinate axes at the same time
- Dimensions in cycles

Prerequisite

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



- **Scaling factor ?:** Enter the scaling factor SCL. The TNC 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.

Example: NC blocks

```
11 CALL LBL 1
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 LBL 1
```

AXIS-SPECIFIC SCALING (Cycle 26)

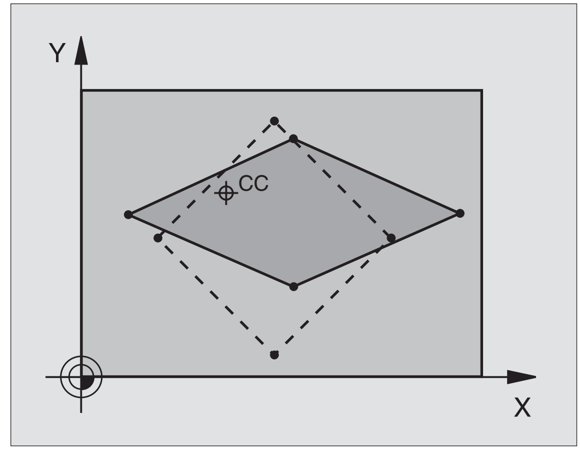
Before programming, note the following:

Coordinate axes sharing coordinates for arcs must be enlarged or reduced by the same factor.

You can program each coordinate axis with its own axis-specific scaling factor.

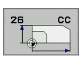
In addition, you can enter the coordinates of a center for all scaling factors.

The size of the contour is enlarged or reduced with reference to the center, and not necessarily (as in Cycle 11 SCALING FACTOR) with reference to the active datum.



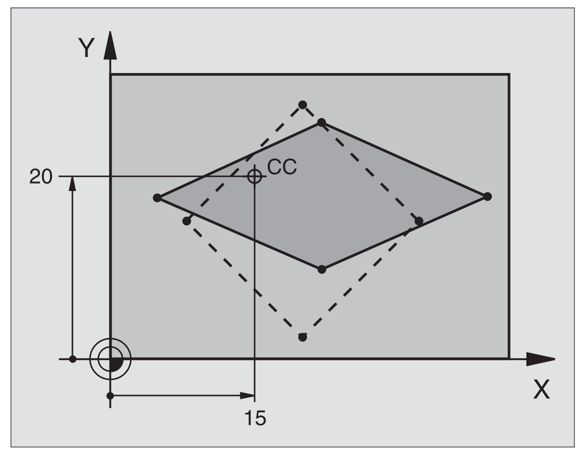
Effect

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.



- **`Axis and scaling factor:`** Enter the coordinate axis/axes as well as the factor(s) involved in enlarging or reducing. Enter a positive value up to 99.999 999.
- **`Center coordinates:`** Enter the center of the axis-specific enlargement or reduction.

The coordinate axes are selected with soft keys.



Cancellation

Program the SCALING FACTOR cycle once again with a scaling factor of 1 for the same axis.

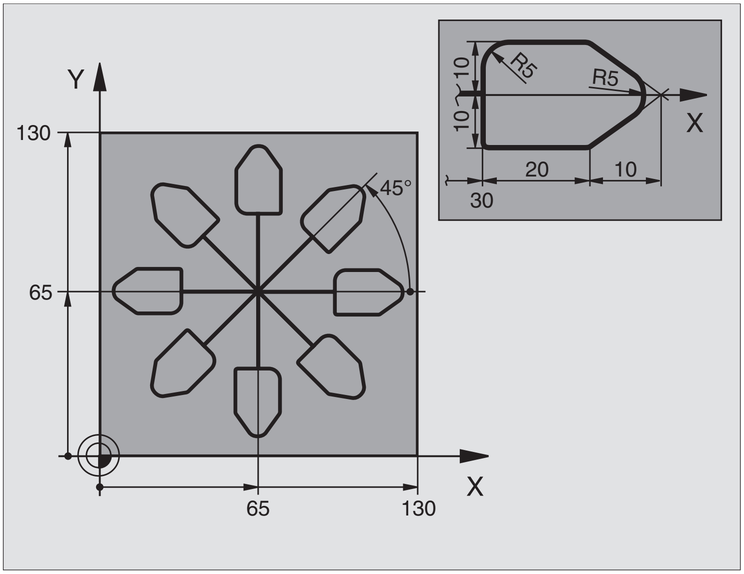
Example: NC blocks

```
25 CALL LBL 1
26 CYCL DEF 26.0 AXIS-SPECIFIC SCALING
27 CYCL DEF 26.1 X 1.4 Y 0.6 CCX+15 CCY+20
28 CALL LBL 1
```

Example: Coordinate transformation cycles

Program sequence

- Program the coordinate transformations in the main program
- For subprograms within a subprogram, see "Subprograms", page 299.



0 BEGIN PGM COTRANS MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the workpiece blank
2 BLK FORM 0.2 X+130 Y+130 Z+0	
3 TOOL DEF 1 L+0 R+1	Define the tool
4 TOOL CALL 1 Z S4500	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center
7 CYCL DEF 7.1 X+65	
8 CYCL DEF 7.2 Y+65	
9 CALL LBL 1	Call milling operation
10 LBL 10	Set label for program section repeat
11 CYCL DEF 10.0 ROTATION	Rotate by 45° (incremental)
12 CYCL DEF 10.1 IROT+45	
13 CALL LBL 1	Call milling operation
14 CALL LBL 10 REP 6/6	Return jump to LBL 10; repeat the milling operation six times
15 CYCL DEF 10.0 ROTATION	Reset the rotation
16 CYCL DEF 10.1 ROT+0	
17 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
18 CYCL DEF 7.1 X+0	
19 CYCL DEF 7.2 Y+0	

20 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
21 LBL 1	Subprogram 1
22 L X+0 Y+0 R0 FMAX	Define milling operation
23 L Z+2 R0 FMAX M3	
24 L Z-5 R0 F200	
25 L X+30 RL	
26 L IY+10	
27 RND R5	
28 L IX+20	
29 L IX+10 IY-10	
30 RND R5	
31 L IX-10 IY-10	
32 L IX-20	
33 L IY+10	
34 L X+0 Y+0 R0 F5000	
35 L Z+20 R0 FMAX	
36 LBL 0	
37 END PGM COTRANS MM	

8.8 Special Cycles

DWELL TIME (Cycle 9)

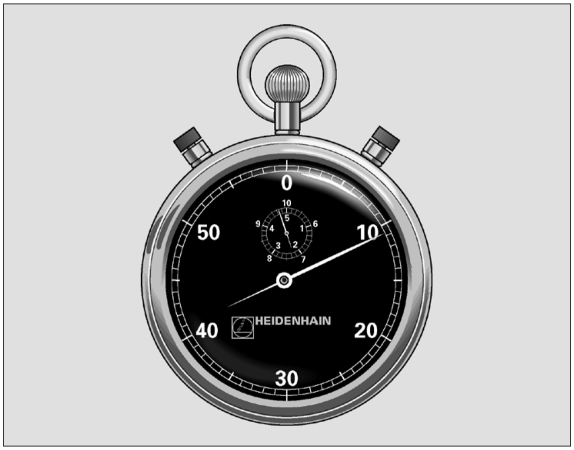
This causes the execution of the next block within a running program to be delayed by the programmed dwell time. A dwell time can be used for such purposes as chip breaking.

Effect

The cycle 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 steps of 0.001 seconds



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.

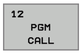
Before programming, note the following:

The program you are calling must be stored on the hard disk of your TNC.

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 only need 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.



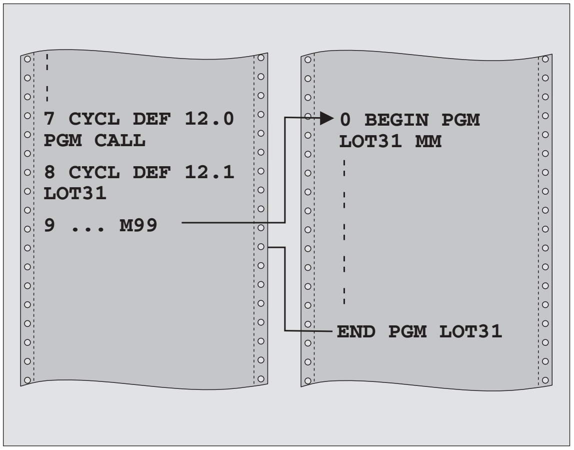
- **Program name:** Enter the name of the program you want to call and, if necessary, the directory it is located in.

Call the program with

- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (executed after every positioning block)

Example: Program call

A callable program 50 is to be called into a program via a cycle call.



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
```

ORIENTED SPINDLE STOP (Cycle 13)

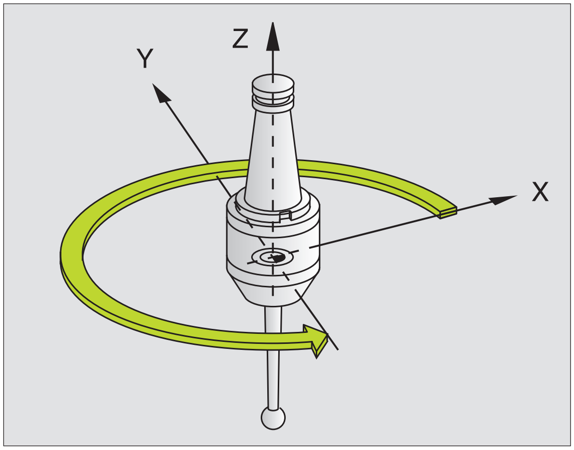
> Machine and control must be specially prepared by the machine tool builder for use of this cycle.

> 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.

Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission



Effect

The angle of orientation defined in the cycle is positioned to by entering M19 or M20 (depending on the machine).

If you program M19 or M20 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set by the machine manufacturer (see your machine manual).

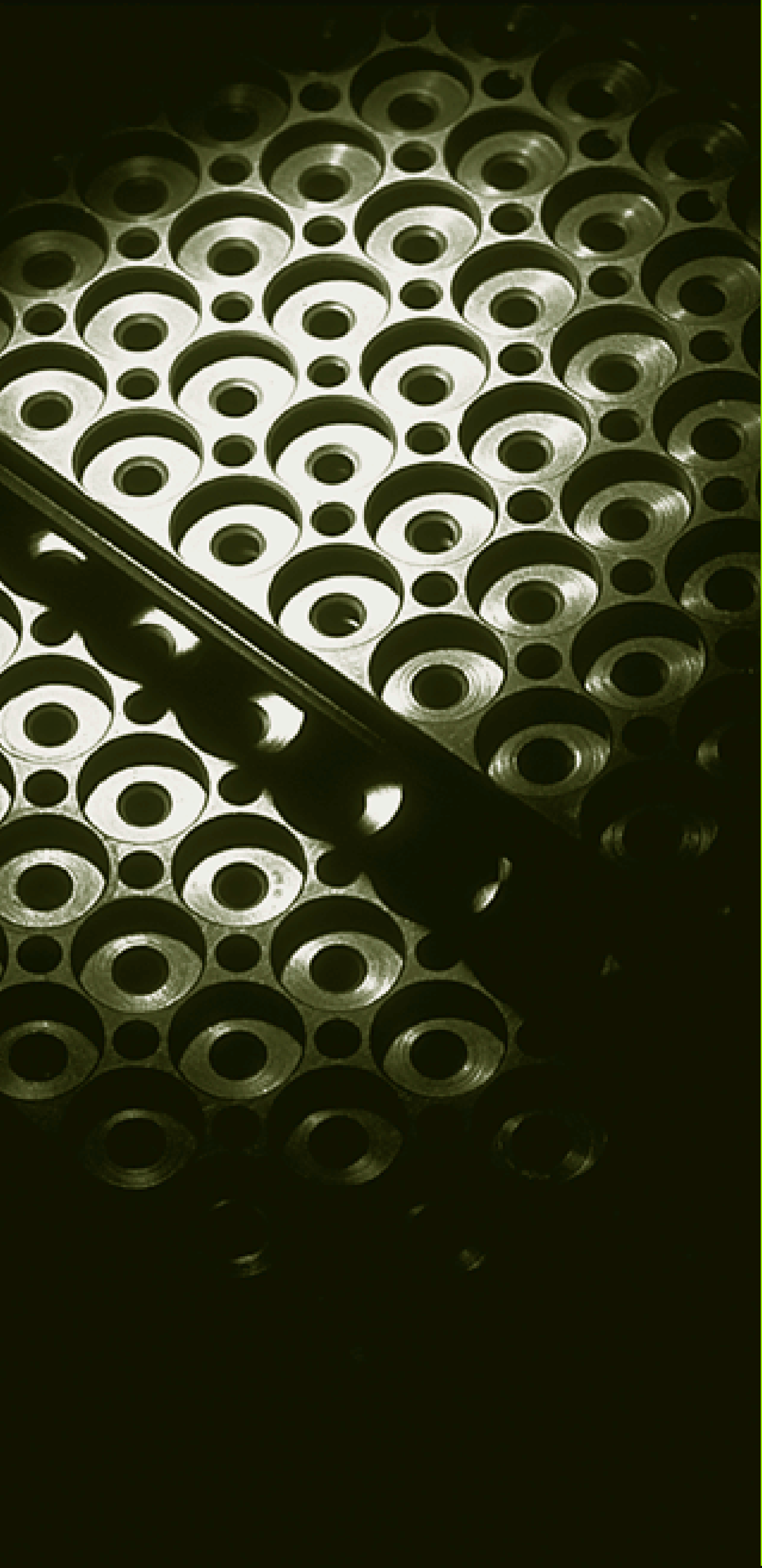
- **Angle of orientation:** Enter the angle according to the reference axis of the working plane.

  Input range: 0 to 360°

  Input resolution: 0.1°

Example: NC blocks

```
93 CYCL DEF 13.0 ORIENTATION
94 CYCL DEF 13.1 ANGLE 180
```



9

Programming: Subprograms and Program Section Repeats

9.1 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 beginnings of subprograms and program section repeats are marked in a part program by labels.

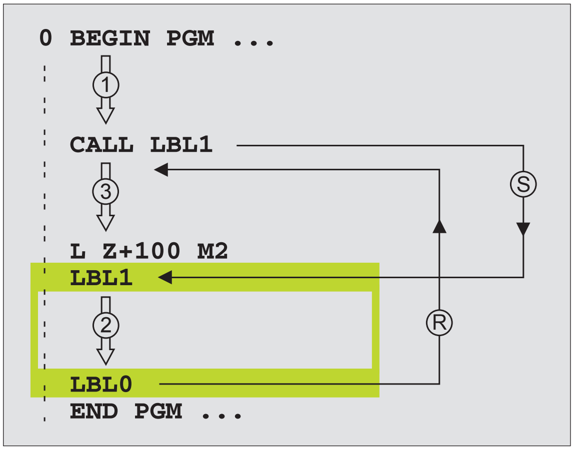
A label is identified by a number between 1 and 65 534 or by a name you define. Each LABEL number or LABEL name can be set only once in the program with LABEL SET. The number of label names you can enter is only limited by the internal memory.

Do not use a label number or label name more than once!

LABEL 0 (LBL 0) is used exclusively to mark the end of a subprogram and can therefore be used as often as desired.

9.2 Subprograms

Operating sequence



1. The TNC executes the part program up to the block in which 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 TNC then resumes the part program from the block after 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 cannot call itself.
- Write subprograms at the end of the main program (behind the block with M02 or M30).
- If subprograms are located before the block with M02 or M30, 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.
- Enter the subprogram number.
- To mark the end, press the LBL SET key and enter the label number "0".

Calling a subprogram

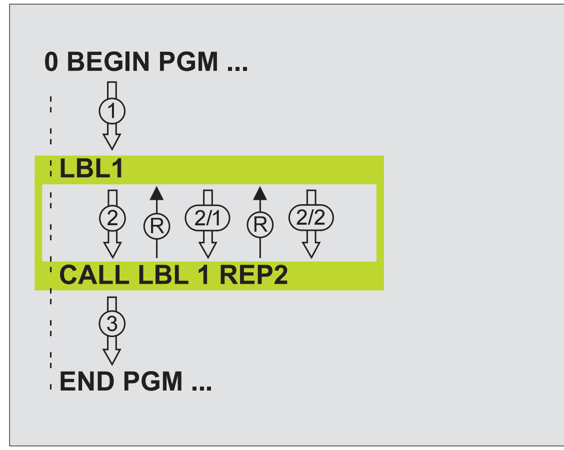
- To call a subprogram, press the LBL CALL key.
- **Label number:** Enter the label number of the subprogram you wish to call. If you want to use a label name, press the key " to switch to text entry.
- **Repeat REP:** Ignore the dialog question with the NO ENT key. Repeat REP is used only for program section repeats.

CALL LBL 0 is not permitted (Label 0 is only used to mark the end of a subprogram).

9.3 Program Section Repeats

Label LBL

The beginning of a program section repeat is marked by the label LBL. The end of a program section repeat is identified by CALL LBL /REP.



Operating sequence

1. The TNC executes the part program up to the end of the program section (CALL LBL /REP).
2. Then the program section between the called LBL and the label call is repeated the number of times entered after REP.
3. The TNC 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 total number of times the program section is executed is always one more than the programmed number of repeats.

Programming a program section repeat

LBL SET

- To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat. If you want to use a label name, press the key " to switch to text entry.
- Enter the program section.

Calling a program section repeat

LBL CALL

- 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 (with Repeat REP).

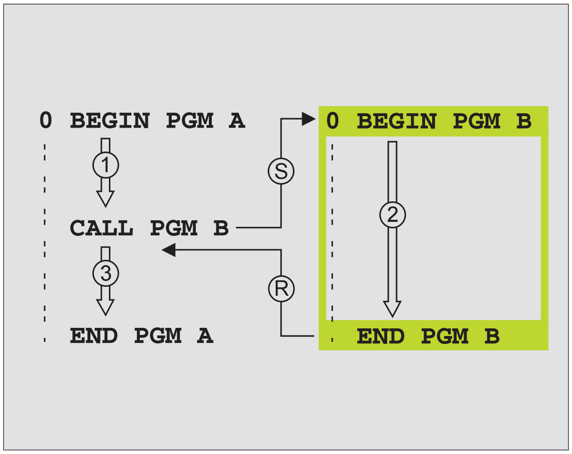
9.4 Separate Program as Subprogram

Operating sequence

1. The TNC executes the part program up to the block in which another program is called with CALL PGM.
2. Then the other program is run from beginning to end.
3. The TNC then resumes the first (calling) part program with the block behind the program call.

Programming notes

- No labels are needed to call any program as a subprogram.
- The called program must not contain the miscellaneous functions M02 or M30.
- The called program must not contain a **CALL PGM** call into the calling program, otherwise an infinite loop will result.



Calling any program as a subprogram

PGM CALL

- To select the functions for program call, press the PGM CALL key.

PROGRAM

- Press the PROGRAM soft key
- Enter the complete path name of the program you want to call and confirm your entry with the END key.

You need only enter the program name if the program you want to call is 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, e.g. **TNC:\ZW35\SCHRUPP\PGM1.H**

If you want to call an ISO program, enter the file type .I after the program name.

You can also call a program with **CYCLE 12 PGM CALL.**

As a rule, Q parameters are effective globally with a **PGM CALL.** So please note that changes to Q parameters in the called program can also influence the calling program.

9.5 Nesting

Types of nesting

- Subprograms within a subprogram
- Program section repeats within a program section repeat
- Subprograms repeated
- Program section repeats within a subprogram

Nesting depth

The nesting depth is the number of successive levels in which program sections or subprograms can call further program sections or subprograms.

- Maximum nesting depth for subprograms: approx. 64 000
- Maximum nesting depth for main program calls: The nesting depth is limited only by the available working memory.
- You can nest program section repeats as often as desired.

Subprogram within a subprogram

Example NC blocks

0 BEGIN PGM UPGMS MM	
...	
17 CALL LBL "SP1"	Call the subprogram marked with LBL SP1
...	
35 L Z+100 R0 FMAX M2	Last program block of the
	main program (with M02)
36 LBL "SP1"	Beginning of subprogram SP2
...	
39 CALL LBL 2	Call the subprogram marked with LBL 2
...	
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 SUBPGMS is executed up to block 17
2. Subprogram 1 is called, and executed up to block 39.
3. Subprogram 2 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 SUBPGMS.
5. Main program SUBPGMS is executed from block 18 up to block 35. Return jump to block 1 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	The program section between this block and LBL 2
...	(block 20) is repeated twice
35 CALL LBL 1 REP 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 27 and block 20 is repeated twice.
3. Main program REPS is executed from block 28 to block 35.
4. Program section between block 35 and block 15 is repeated once (including the program section repeat between 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`	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 subprogram
`...`	
`28 LBL 0`	End of subprogram
`29 END PGM UPGREP MM`	

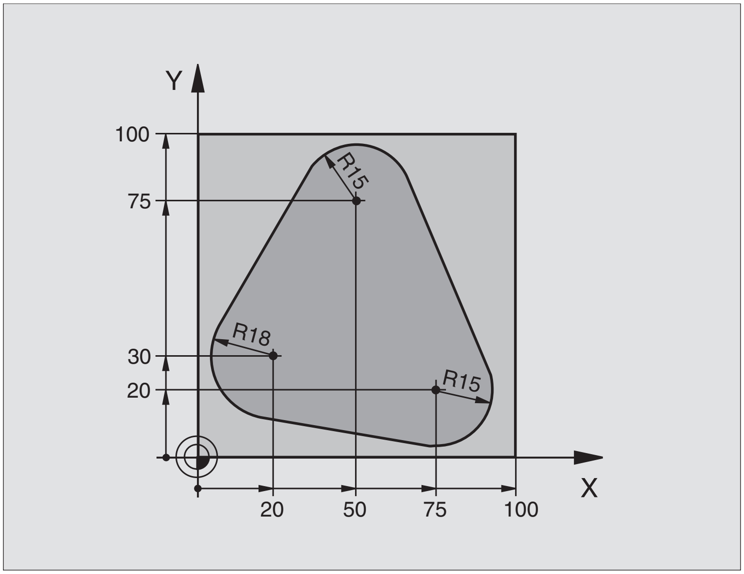
Program execution

1. Main program UPGREP is executed up to block 11.
2. Subprogram 2 is called and executed.
3. Program section between block 12 and block 10 is repeated twice. Subprogram 2 is repeated twice.
4. Main program UPGREP is executed from block 13 to block 19. End of program.

Example: Milling a contour in several infeeds

Program sequence

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Contour milling
- Repeat downfeed and contour-milling



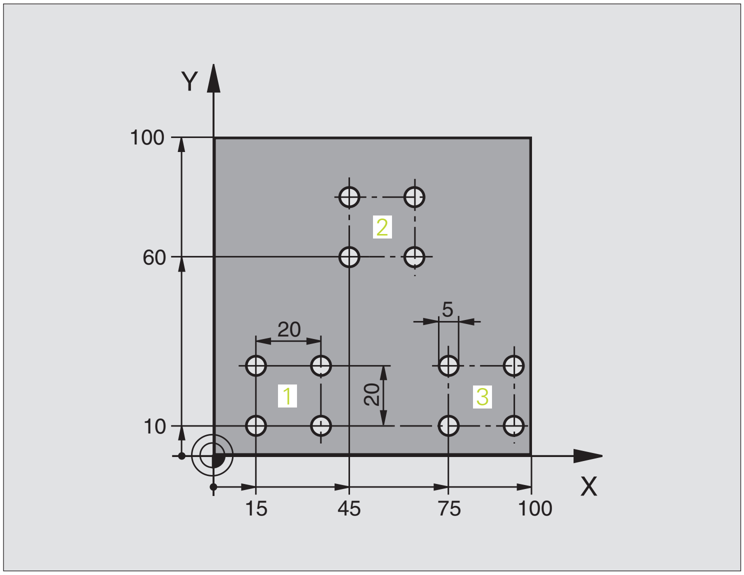
0 BEGIN PGM PGMWDH MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+10	Define the tool
4 TOOL CALL 1 Z S500	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 L X-20 Y+30 R0 FMAX	Pre-position in the working plane
7 L Z+0 R0 FMAX M3	Pre-position to the workpiece surface

8 LBL 1	Set label for program section repeat
9 L IZ-4 R0 FMAX	Infeed depth in incremental values (in space)
10 APPR CT X+2 Y+30 CCA90 R+5 RL F250	Approach to the contour.
11 FC DR- R18 CLSD+ CCX+20 CCY+30	Contour
12 FLT	
13 FCT DR- R15 CCX+50 CCY+75	
14 FLT	
15 FCT DR- R15 CCX+75 CCY+20	
16 FLT	
17 FCT DR- R18 CLSD- CCX+20 CCY+30	
18 DEP CT CCA90 R+5 F1000	Depart the contour
19 L X-20 Y+0 R0 FMAX	Retract tool
20 CALL LBL 1 REP 4	Return jump to LBL 1; section is repeated a total of 4 times.
21 L Z+250 R0 FMAX M2	Retract in the tool axis, end program
22 END PGM PGMWDH MM	

Example: Groups of holes

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1



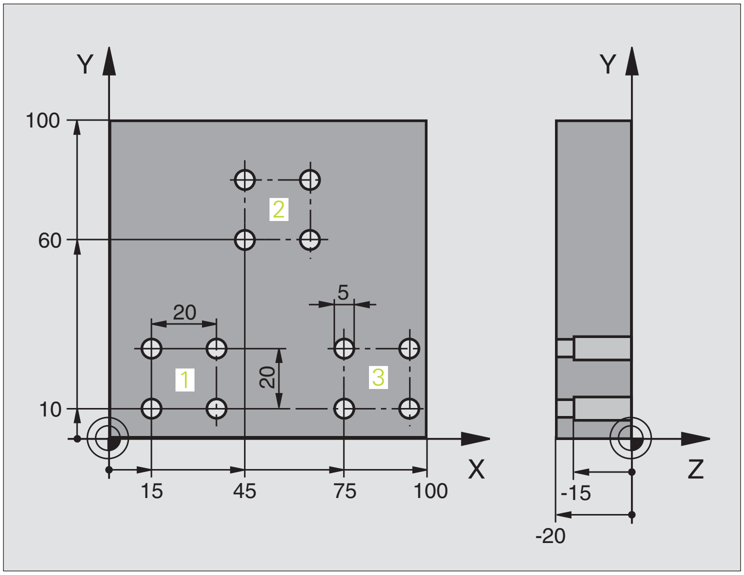
0 BEGIN PGM UP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+2.5	Define the tool
4 TOOL CALL 1 Z S5000	Tool call
5 L Z+250 R0 FMAX	Retract the tool
6 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2 ;SET-UP CLEARANCE	
Q201=-10 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=5 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	

7 L X+15 Y+10 R0 FMAX M3	Move to starting point for group 1
8 CALL LBL 1	Call the subprogram for the group
9 L X+45 Y+60 R0 FMAX	Move to starting point for group 2
10 CALL LBL 1	Call the subprogram for the group
11 L X+75 Y+10 R0 FMAX	Move to starting point for group 3
12 CALL LBL 1	Call the subprogram for the group
13 L Z+250 R0 FMAX M2	End of main program
14 LBL 1	Beginning of subprogram 1: Group of holes
15 CYCL CALL	Hole 1
16 L IX.20 R0 FMAX M99	Move to 2nd hole, call cycle
17 L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
18 L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
19 LBL 0	End of subprogram 1
20 END PGM UP1 MM	

Example: Group of holes with several tools

Program sequence

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2



0 BEGIN PGM UP2 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 DEF 1 L+0 R+4	Define tool: center drill
4 TOOL DEF 2 L+0 R+3	Define tool: drill
5 TOOL DEF 2 L+0 R+3.5	Define tool: reamer
6 TOOL CALL 1 Z S5000	Call tool: center drill
7 L Z+250 R0 FMAX	Retract the tool
8 CYCL DEF 200 DRILLING	Cycle definition: CENTERING
Q200=2 ;SET-UP CLEARANCE	
Q202=-3 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q202=3 ;PLUNGING DEPTH	
Q210=0 ;DWELL TIME AT TOP	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
Q211=0.25 ;DWELL TIME AT DEPTH	
9 CALL LBL 1	Call subprogram 1 for the entire hole pattern

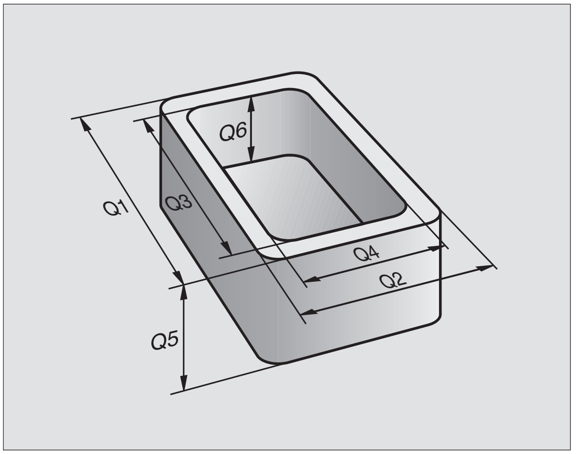
10 L Z+250 R0 FMAX M6	Tool change
11 TOOL CALL 2 Z S4000	Call tool: drill
12 FN 0: Q201 = -25	New depth for drilling
13 FN 0: Q202 = +5	New plunging depth for drilling
14 CALL LBL 1	Call subprogram 1 for the entire hole pattern
15 L Z+250 R0 FMAX M6	Tool change
16 TOOL CALL 3 Z S500	Call tool: reamer
17 CYCL DEF 201 REAMING	Cycle definition: REAMING
Q200=2 ;SET-UP CLEARANCE	
Q201=-15 ;DEPTH	
Q206=250 ;FEED RATE FOR PLNGNG	
Q211=0.5 ;DWELL TIME AT DEPTH	
Q208=400 ;RETRACTION FEED RATE	
Q203=+0 ;SURFACE COORDINATE	
Q204=10 ;2ND SET-UP CLEARANCE	
18 CALL LBL 1	Call subprogram 1 for the entire hole pattern
19 L Z+250 R0 FMAX M2	End of main program
20 LBL 1	Beginning of subprogram 1: Entire hole pattern
21 L X+15 Y+10 R0 FMAX M3	Move to starting point for group 1
22 CALL LBL 2	Call subprogram 2 for the group
23 L X+45 Y+60 R0 FMAX	Move to starting point for group 2
24 CALL LBL 2	Call subprogram 2 for the group
25 L X+75 Y+10 R0 FMAX	Move to starting point for group 3
26 CALL LBL 2	Call subprogram 2 for the group
27 LBL 0	End of subprogram 1
28 LBL 2	Beginning of subprogram 2: Group of holes
29 CYCL CALL	1st hole with active fixed cycle
30 L 9X+20 R0 FMAX M99	Move to 2nd hole, call cycle
31 L IY+20 R0 FMAX M99	Move to 3rd hole, call cycle
32 L IX-20 R0 FMAX M99	Move to 4th hole, call cycle
33 LBL 0	End of subprogram 2
34 END PGM UP2 MM	

10

Programming: Q Parameters

10.1 Principle and Overview

You can program an entire family of parts in a single part program. You do this by entering variables called Q parameters instead of fixed numerical values.



Q parameters can represent information such as:

- Coordinate values
- Feed rates
- Spindle speeds
- Cycle data

Q parameters also enable you to program contours that are defined with mathematical functions. You can also use Q parameters to make the execution of machining steps depend 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 designated by the letter Q and a number between 0 and 1999. They are grouped according to various ranges:

Meaning	Range
Freely applicable parameters, globally effective for all programs stored in the TNC memory	Q1600 to Q1999
Freely applicable parameters, as long as no overlapping with SL cycles can occur, globally effective for the respective program	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, globally effective for all programs that are stored in the TNC memory	Q200 to Q1399
Parameters that are primarily used for call-active OEM cycles, globally effective for all programs that are stored in the TNC memory	Q1400 to Q1499
Parameters that are primarily used for DEF-active OEM cycles, globally effective for all programs that are stored in the TNC memory	Q1500 to Q1599

Programming notes

You can mix Q parameters and fixed numerical values within a program.

> Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius (see "Preassigned Q Parameters," page 360).

Calling Q parameter functions

When you are writing a part program, press the "Q" key (in the numeric keypad for numerical input and axis selection, below the +/– key). The TNC then displays the following soft keys:

Function group	Soft key	Page
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.	Page 317
Trigonometric functions	TRIGO- NOMETRY	Page 319
Function for calculating circles	CIRCLE CALCU- LATION	Page 321
If/then conditions, jumps	JUMP	Page 322
Other functions	DIVERSE FUNCTION	Page 325
Entering formulas directly	FORMULA	Page 356
Formula for string parameters	STRING FORMULA	Page 363

10.2 Part Families—Q Parameters in Place of Numerical Values

The Q parameter function FN0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

`15 FN0: Q10=25`	Assign
`...`	Q10 is assigned the value 25
`25 L X +Q10`	Means L X +25

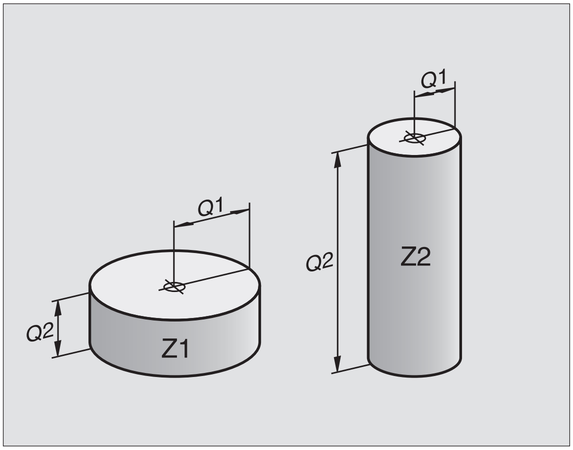
You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example

Cylinder with Q parameters

Cylinder radius	R = Q1
Cylinder height	H = Q2
Cylinder Z1	Q1 = +30 Q2 = +10
Cylinder Z2	Q1 = +10 Q2 = +50



10.3 Describing Contours through Mathematical Operations

Function

The Q parameters listed below enable you to program basic mathematical functions in a part program:

- ▶ Select a Q parameter function: Press the Q key (in the numerical keypad at right). The Q parameter functions are displayed in a soft-key row.
- ▶ To select the mathematical functions, press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Overview

Function	Soft key
FN0: ASSIGN Example: **`FN0: Q5 = +60`** Assigns a numerical value.	FN0 X = Y
FN1: ADDITION Example: **`FN1: Q1 = –Q2 + –5`** Calculates and assigns the sum of two values.	FN1 X + Y
FN2: SUBTRACTION Example: **`FN2: Q1 = +10 – +5`** Calculates and assigns the difference of two values.	FN2 X - Y
FN3: MULTIPLICATION Example: **`FN3: Q2 = +3 * +3`** Calculates and assigns the product of two values.	FN3 X * Y
FN4: DIVISION Example: **`FN4: Q4 = +8 DIV +Q2`** Calculates and assigns the quotient of two values. **Not permitted:** Division by 0	FN4 X / Y
FN5: SQUARE ROOT Example: **`FN5: Q20 = SQRT 4`** Calculates and assigns the square root of a number. **Not permitted:** Square root of a negative number	FN5 SQRT

To 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.

Programming fundamental operations

Example:

Q	Call the Q parameter functions by pressing the Q key.
BASIC ARITHM.	To select the mathematical functions, press the BASIC ARITHMETIC soft key.
FN0 X = Y	To select the Q parameter function ASSIGN, press the FN0 X = Y soft key.

PARAMETER NO. FOR RESULT?

5 ENT	Enter the number of the Q parameter, e.g. 5.

1ST VALUE OR PARAMETER?

10 ENT	Assign the value 10 to Q5.

Q	Call the Q parameter functions by pressing the Q key.
BASIC ARITHM.	To select the mathematical functions, press the BASIC ARITHMETIC soft key.
FN3 X * Y	To select the Q parameter function MULTIPLICATION, press the FN3 X * Y soft key.

PARAMETER NO. FOR RESULT?

12 ENT	Enter the number of the Q parameter, e.g. 12.

1ST VALUE OR PARAMETER?

Q5 ENT	Enter Q5 for the first value.

2ND VALUE OR PARAMETER?

7 ENT	Enter 7 for the second value.

Example: Program blocks in the TNC

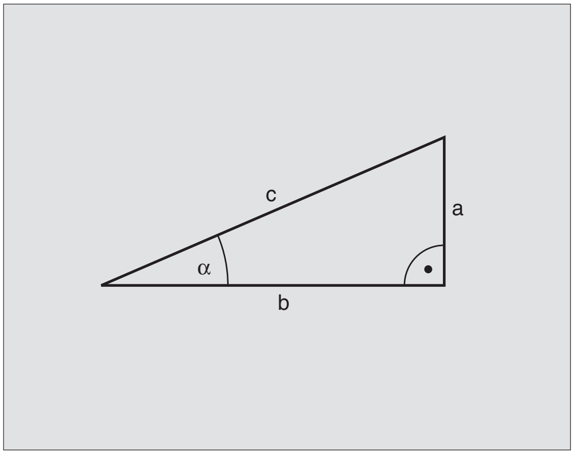
```
16 FN0: Q5 = +10
17 FN3: Q12 = +Q5 * +7
```

10.4 Trigonometric Functions

Definitions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. In this case:

Sine: $\sin \alpha = a / c$
Cosine: $\cos \alpha = b / c$
Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$



where

- c is the side opposite the right angle
- a is the side opposite the angle α
- b is the third side.

The TNC can find the angle from the tangent:

$\alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:

a = 25 mm

b = 50 mm

$\alpha = \arctan (a / b) = \arctan 0.5 = 26.57°$

Furthermore:

$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the ANGLE FUNCTION soft key to call the angle functions. The TNC then displays the following soft keys:

Programming: Compare "Example: Programming fundamental operations."

Function	Soft key
FN6: SINE Example: **`FN6: Q20 = SIN–Q5`** Calculates the sine of an angle in degrees (°) and assigns 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.	
FN8: ROOT SUM OF SQUARES Example: **`FN8: Q10 = +5 LEN +4`** Calculate and assign length from two values.	
FN13: ANGLE Example: **`FN13: Q20 = +25 ANG–Q1`** Calculates the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assigns it to a parameter.	

10.5 Calculating Circles

Function

The TNC can use the functions for calculating circles to calculate the circle center and the circle radius from three or four given points on the circle. The calculation is more accurate if four points are used.

Application: These functions can be used if you wish to determine the location and size of a hole or a pitch circle using the programmable probing function.

Function	Soft key
FN23: Determining the CIRCLE DATA from three points Example: **FN23: Q20 = CDATA Q30**	FN23 3 POINTS OF CIRCLE

The coordinate pairs for three points of the circle must be stored in Parameter Q30 and in the following five parameters, i.e. to Q35.

The TNC then stores the circle center of the reference axis (X with spindle axis Z) in Parameter Q20, the circle center of the minor axis (Y with spindle axis Z) in Parameter Q21 and the circle radius in Parameter Q22.

Function	Soft key
FN24: Determining the CIRCLE DATA from four points Example: **FN24: Q20 = CDATA Q30**	FN24 4 POINTS OF CIRCLE

The coordinate pairs for four points of the circle must be stored in Parameter Q30 and in the following seven parameters, i.e. to Q37.

The TNC then stores the circle center of the reference axis (X with spindle axis Z) in Parameter Q20, the circle center of the minor axis (Y with spindle axis Z) in Parameter Q21 and the circle radius in Parameter Q22.

Note that FN23 and FN24 automatically overwrite the resulting parameter and the two following parameters.

10.6 If-Then Decisions with Q Parameters

Function

The TNC 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 TNC continues the program at the label that is programmed after the condition (for information on labels, see "Labeling Subprograms and Program Section Repeats," page 298). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter PGM CALL after the block with the target label.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

FN9: IF+10 EQU+10 GOTO LBL1

Programming If-Then decisions

Press the JUMP soft key to call the If-Then conditions. The TNC then displays the following soft keys:

Function	Soft key
FN9: IF EQUAL, JUMP Example: **`FN9: IF +Q1 EQU +Q3 GOTO LBL "UPCAN25"`** If the two values or parameters are equal, jump to the given label.	FN9 IF X EQ Y GOTO
FN10: IF NOT EQUAL, JUMP Example: **`FN10: IF +10 NE –Q5 GOTO LBL 10`** If the two values or parameters are not equal, jump to the given label.	FN10 IF X NE Y GOTO
FN11: IF GREATER THAN, JUMP Example: **`FN11: IF+Q1 GT+10 GOTO LBL 5`** If the first parameter or value is greater than the second value or parameter, jump to the given label.	FN11 IF X GT Y GOTO
FN12: IF LESS THAN, JUMP Example: **`FN12: IF+Q5 LT+0 GOTO LBL "ANYNAME"`** If the first value or parameter is less than the second value or parameter, jump to the given label.	FN12 IF X LT Y GOTO

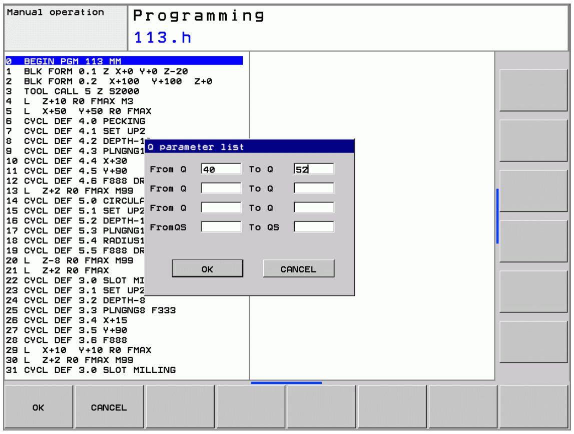
Abbreviations used:

IF	:	If
EQU	:	Equals
NE	:	Not equal
GT	:	Greater than
LT	:	Less than
GOTO	:	Go to

10.7 Checking and Changing Q Parameters

Procedure

You can check Q parameters when writing, testing and running programs in all operating modes and, except in the test run, edit them.



- If you are in a program run, interrupt it if required (for example, by pressing the machine STOP button and the INTERNAL STOP soft key). If you are in a test run, interrupt it.

Q INFO

- To call Q parameter functions: Press the Q INFO soft key in the Programming and Editing mode of operation.
- The TNC opens a pop-up window in which you can enter the desired range for display of the Q parameters or string parameters
- In the Program Run Single Block, Program Run Full Sequence and Test Run modes of operation, select the screen layout Program + Status

STATUS OF Q PARAM.

- Select the Program + Q PARAM soft key

Q PARAMETER LIST

- Select the Q PARAMETER LIST soft key
- The TNC opens a pop-up window in which you can enter the desired range for display of the Q parameters or string parameters

Q PARAMETER REQUEST

- With the Q PARAMETER REQUEST soft key (available only in Manual Operation, Program Run Full Sequence and Program Run Single Block), you can request individual Q parameters. To assign a new value, overwrite the displayed value and confirm with the OK.

10.8 Additional Functions

Overview

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key	Page
FN14:ERROR Output error messages	FN14 ERROR=	Page 326
FN16:F-PRINT Formatted output of texts or Q parameter values	FN16 F-PRINT	Page 328
FN18:SYS-DATUM READ Read system data	FN18 SYS-DATUM READ	Page 331
FN19:PLC Transfer values to the PLC	FN19 PLC=	Page 339
FN20:WAIT FOR Synchronize NC and PLC	FN20 WAIT FOR	Page 340
FN25:PRESET Set datum during program run	FN25 SET DATUM	Page 342
FN29:PLC Transfer up to eight values to the PLC	FN29 PLC	Page 343
FN37:EXPORT Export local Q parameters or QS parameters into a calling program	FN37 EXPORT	Page 344

FN14: ERROR: Displaying error messages

With the function FN14: ERROR you can call messages under program control. The messages were programmed by the machine tool builder or by HEIDENHAIN. Whenever the TNC comes to a block with FN 14 in the Program Run or Test Run mode, it interrupts the program run and displays a message. The program must then be restarted. The error numbers are listed in the table below.

Range of error numbers	Standard dialog text
0 ... 299	FN 14: Error code 0 299
300 ... 999	Machine-dependent dialog
1000 ... 1099	Internal error messages (see table at right)

The machine tool builder can change the **FN14:ERROR** function. Refer to your machine manual.

Example NC block

The TNC is to display the text stored under error number 254.

```
180 FN14: ERROR = 254
```

Error number	Text
1000	Spindle?
1001	Tool axis is missing
1002	Tool radius too small
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRRORING not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Entry value incorrect
1012	Wrong sign programmed
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory entry
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong RPM
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive subprogramming
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too small
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Enter Q218 greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Enter Q222 greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be < 360°
1040	Enter Q223 greater than Q222
1041	Q214: 0 not permitted

Error number	Text
1042	Traverse direction not defined
1043	No datum table active
1044	Position error: center in axis 1
1045	Position error: center in axis 2
1046	Hole diameter too small
1047	Hole diameter too large
1048	Stud diameter too small
1049	Stud diameter too large
1050	Pocket too small: rework axis 1
1051	Pocket too small: rework axis 2
1052	Pocket too large: scrap axis 1
1053	Pocket too large: scrap axis 2
1054	Stud too small: scrap axis 1
1055	Stud too small: scrap axis 2
1056	Stud too large: rework axis 1
1057	Stud too large: rework axis 2
1058	TCHPROBE 425: length exceeds max
1059	TCHPROBE 425: length below min
1060	TCHPROBE 426: length exceeds max
1061	TCHPROBE 426: length below min
1062	TCHPROBE 430: diameter too large
1063	TCHPROBE 430: diameter too small
1064	No measuring axis defined
1065	Tool breakage tolerance exceeded
1066	Enter Q247 unequal 0
1067	Enter Q247 greater than 5
1068	Datum table?
1069	Enter direction Q351 unequal 0
1070	Thread depth too large
1071	Missing calibration data
1072	Tolerance exceeded
1073	Block scan active
1074	ORIENTATION not permitted
1075	3-D ROT not permitted
1076	Activate 3-D ROT
1077	Enter depth as a negative value
1078	Q303 not defined in measuring cycle
1079	Tool axis not allowed
1080	Calculated values incorrect
1081	Contradictory measuring points
1082	Clearance height entered incorrectly
1083	Contradictory type of plunging
1084	Machining cycle not permitted

Error number	Text
1085	Line is write-protected
1086	Oversize greater than depth
1087	No point angle defined
1088	Contradictory data
1089	Slot position 0 not permitted
1090	Enter infeed unequal 0

FN16: F-PRINT: Formatted output of texts or Q parameter values

The function FN16: F-PRINT transfers Q parameter values and texts in a selectable format through the data interface, for example to a printer. If you save the values internally or send them to a computer, the TNC saves the data in the file that you defined in the FN 16 block.

To output the formatted texts and Q parameter values, create a text file with the TNC's text editor. In this file you then define the output format and Q parameters you want to output.

Example of a text file to define the output format:

“TEST RECORD IMPELLER CENTER OF GRAVITY”;

```
"DATE: %2d-%2d-%4d",DAY,MONTH,YEAR4;

"TIME: %2d:%2d:%2d",HOUR,MIN,SEC;"

"________________________"

"NO. OF MEASURED VALUES : = 1";

"*******************************************";#

"X1 = %9.3LF", Q31;

"Y1 = %9.3LF", Q32;

"Z1 = %9.3LF", Q33;

"*******************************************";
```

When you create a text file, use the following formatting functions:

Special character	Function
"..........."	Define the output format for texts and variables between the quotation marks.
%9.3LF	Define format for Q parameter: 9 places in total (with decimal point), three of which are decimal places, long, floating (decimal number)
%S	Format for text variable
,	Separation character between output format and parameter
;	End of block character

The following functions allow you to include the following additional information in the protocol log file:

Code word	Function
CALL_PATH	Gives the path for the NC program where you will find the FN16 function. Example: "Measuring program: %S",CALL_PATH;
M_CLOSE	Closes the file to which you are writing with FN16. Example: M_CLOSE;
L_ENGLISH	Display the text only in English conversational
L_GERMAN	Display the text only in German conversational
L_CZECH	Display text only in Czech conversational
L_FRENCH	Display text only in French conversational
L_ITALIAN	Display text only in Italian conversational
L_SPANISH	Display text only in Spanish conversational
L_SWEDISH	Output text only in Swedish conversational
L_DANISH	Display text only in Danish conversational
L_FINNISH	Display text only in Finnish conversational
L_DUTCH	Display the text only in Dutch conversational
L_POLISH	Display text only in Polish conversational
L_HUNGARIA	Display text only in Hungarian conversational
L_ALL	Display the text independent of the conversational language
HOUR	Number of hours from the real-time clock

Code word	Function
MIN	Number of minutes from the real-time clock
SEC	Number of seconds from the real-time clock
DAY	Day from the real-time clock
MONTH	Month as a number from the real-time clock
STR_MONTH	Month as a string abbreviation from the real-time clock
YEAR2	Two-digit year from the real-time clock
YEAR4	Four-digit year from the real-time clock

In the part program, program FN 16: F-PRINT, to activate the output:

```
96  FN16:  F-PRINT TNC:\MASKE\MASKE1.A/RS232:\PROT1.TXT
```

The TNC then outputs the file PROT1.TXT through the serial interface:

```
CALIBRAT. CHART IMPELLER CENTER GRAVITY

DATE: 27:11:2001

TIME: 8:56:34

NO. OF MEASURED VALUES : = 1

*******************************************

X1 = 149.360

Y1 = 25.509

Z1 = 37.000

*******************************************
```

If you use FN 16 several times in the program, the TNC saves all texts in the file that you have defined with the first FN 16 function. The file is not output until the TNC reads the END PGM block, or you press the NC stop button, or you close the file with M_CLOSE.

In the FN16 block, program the format file and the log file with their respective extensions.

If you enter only the file name for the path of the log file, the TNC saves the log file in the directory in which the NC program with the FN16 function is located.

You can output up to 32 Q parameters per line in the format description file.

FN18: SYS-DATUM READ Read system data

With the function FN 18: SYS-DATUM READ you can read system data and store them in Q parameters. You select the system data through a group number (ID number), and additionally through a number and an index.

Group name, ID No.	Number	Index	Meaning
Program information, 10	3	-	Number of active fixed cycle
	103	Q parameter number	Relevant within NC cycles; for inquiry as to whether the Q parameter given under IDX was explicitly stated in the associated CYCLE DEF.
System jump addresses, 13	1	-	Label jumped to during M2/M30 instead of the value that ends the current program = 0: M2/M30 has the normal effect
	2	-	Label jumped to if FN14: ERROR after the NC CANCEL reaction instead of aborting the program with an error. The error number programmed in the FN14 command can be read under ID992 NR14. Value = 0: FN14 has the normal effect.
	3	-	Label jumped to in the event of an internal server error (SQL, PLC, CFG) instead of aborting the program with an error. Value = 0: Server error has the normal effect.
Machine status, 20	1	-	Active tool number
	2	-	Prepared tool number
	3	-	Active tool axis 0=X, 1=Y, 2=Z, 6=U, 7=V, 8=W
	4	-	Programmed spindle rpm
	5	-	Active spindle status: –1=undefined, 0=M3 active, 1=M4 active, 2=M5 after M3, 3=M5 after M4
	8	-	Coolant status: 0=off, 1=on
	9	-	Active feed rate
	10	-	Index of the prepared tool
	11	-	Index of the active tool
Channel data, 25	1	-	Channel number
Cycle parameter, 30	1	-	Set-up clearance of active fixed cycle
	2	-	Drilling depth / milling depth of active fixed cycle
	3	-	Plunging depth of active fixed cycle
	4	-	Feed rate for pecking in active fixed cycle

Group name, ID No.	Number	Index	Meaning
	5	-	1st side length for rectangular pocket cycle
	6	-	2nd side length for rectangular pocket cycle
	7	-	1st side length for slot cycle
	8	-	2nd side length for slot cycle
	9	-	Radius for circular pocket cycle
	10	-	Feed rate for milling in active fixed cycle
	11	-	Direction of rotation for active fixed cycle
	12	-	Dwell time for active fixed cycle
	13	-	Thread pitch for Cycles 17, 18
	14	-	Milling allowance for active fixed cycle
	15	-	Direction angle for rough out in active fixed cycle
	15	-	Direction angle for rough out in active fixed cycle
	21	-	Probe angle
	22	-	Probe path
	23	-	Probing feed rate
Modal condition, 35	1	-	Dimensioning: 0 = absolute (G90) 1 = incremental (G91)
Data for SQL tables, 40	1	-	Result code for the last SQL command
Data from the tool table, 50	1	Tool no.	Tool length
	2	Tool no.	Tool radius
	3	Tool no.	Tool radius R2
	4	Tool no.	Oversize for tool length DL
	5	Tool no.	Oversize for tool radius DR
	6	Tool no.	Oversize for tool radius DR2
	7	Tool no.	Tool inhibited (0 or 1)
	8	Tool no.	Number of replacement tool
	9	Tool no.	Maximum tool age TIME1
	10	Tool no.	Maximum tool age TIME2
	11	Tool no.	Current tool age CUR. TIME

Group name, ID No.	Number	Index	Meaning
	12	Tool no.	PLC status
	13	Tool no.	Maximum tooth length LCUTS
	14	Tool no.	Maximum plunge angle ANGLE
	15	Tool no.	TT: Number of teeth CUT
	16	Tool no.	TT: Wear tolerance for length LTOL
	17	Tool no.	TT: Wear tolerance for radius RTOL
	18	Tool no.	TT: Rotational direction DIRECT (0=positive/-1=negative)
	19	Tool no.	TT: Offset for radius R-OFFS
	20	Tool no.	TT: Offset for length L-OFFS
	21	Tool no.	TT: Breakage tolerance in length LBREAK
	22	Tool no.	TT: Breakage tolerance in radius RBREAK
	23	Tool no.	PLC value
	24	Tool no.	Center misalignment in reference axis CAL-OF1
	25	Tool no.	Center misalignment in minor axis CAL-OF2
	26	Tool no.	Spindle angle for calibration CAL-ANG
	27	Tool no.	Tool type for pocket table
	28	Tool no.	Maximum rpm NMAX
Pocket table data, 51	1	Pocket number	Tool number
	2	Pocket number	Special tool: 0=no, 1=yes
	3	Pocket number	Fixed pocket: 0=no, 1=yes
	4	Pocket number	Locked pocket: 0=no, 1=yes
	5	Pocket number	PLC status
Pocket number of a tool in the tool-pocket table, 52	1	Tool no.	Pocket number
	2	Tool no.	Tool magazine number
Immediately after TOOL CALL programmed values, 60	1	-	Tool number T
	2	-	Active tool axis 0 = X 6 = U 1 = Y 7 = V 2 = Z 8 = W
	3	-	Spindle speed S

Group name, ID No.	Number	Index	Meaning
	4	-	Oversize for tool length DL
	5	-	Oversize for tool radius DR
	6	-	Automatic TOOL CALL 0 = yes, 1 = no
	7	-	Oversize for tool radius DR2
	8	-	Tool index
	9	-	Active feed rate
Immediately after TOOL DEF programmed values, 61	1	-	Tool number T
	2	-	Length
	3	-	Radius
	4	-	Index
	5	-	Tool data programmed in TOOL DEF 1 = yes, 0 = no
Active tool compensation, 200	1	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active radius
	2	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Active length
	3	1 = without oversize 2 = with oversize 3 = with oversize and oversize from TOOL CALL	Rounding radius R2
Active transformations, 210	1	-	Basic rotation in MANUAL OPERATION mode
	2	-	Programmed rotation with Cycle 10
	3	-	Active mirror axis
			0: mirroring not active

Group name, ID No.	Number	Index	Meaning
			+1: X axis mirrored
			+2: Y axis mirrored
			+4: Z axis mirrored
			+64: U axis mirrored
			+128: V axis mirrored
			+256: W axis mirrored
			Combinations = sum of individual axes
	4	1	Active scaling factor in X axis
	4	2	Active scaling factor in Y axis
	4	3	Active scaling factor in Z axis
	4	7	Active scaling factor in U axis
	4	8	Active scaling factor in V axis
	4	9	Active scaling factor in W axis
	5	1	3-D ROT A axis
	5	2	3-D ROT B axis
	5	3	3-D ROT C axis
	6	-	Tilted working plane active / inactive (–1/0) in a Program Run operating mode
	7	-	Tilted working plane active / inactive (–1/0) in a Manual operating mode
Active datum shift, 220	2	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Traverse range, 230	2	1 to 9	Negative software limit switch in axes 1 to 9

Group name, ID No.	Number	Index	Meaning
	3	1 to 9	Positive software limit switch in axes 1 to 9
	5	-	Software limit switch on or off: 0 = on, 1 = off
Nominal position in the REF system, 240	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
Current position in the active coordinate system, 270	1	1	X axis
		2	Y axis
		3	Z axis
		4	A axis
		5	B axis
		6	C axis
		7	U axis
		8	V axis
		9	W axis
TS triggering touch probe, 350	50	1	Touch probe type
		2	line in the touch probe table
	51	-	Effective length
	52	1	Radius of ring gauge
		2	Rounding radius
	53	1	Center misalignment in ref. axis
		2	Center misalignment in minor axis

Group name, ID No.	Number	Index	Meaning
	54	-	Direction of center misalignment with respect to spindle 0°
		2	Center misalignment in minor axis
	55	1	Rapid traverse
		2	Probe feed rate
	56	1	Maximum measuring range
		2	Set-up clearance
	57	1	Oriented spindle stop possible 0 = no, 1 = yes
		2	Angle of spindle orientation in degrees
Reference point from touch probe cycle, 360	1	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle without probe length compensation but with probe radius compensation (workpiece coordinate system)
	2	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length compensation but with probe radius compensation (machine coordinate system)
	3	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Result of measurement of the touch probe cycles 0 and 1 without touch probe radius and probe length compensation
	4	1 to 9 (X, Y, Z, A, B, C, U, V, W)	Last reference point of a manual touch probe cycle, or last touch point from Cycle 0 without probe length compensation but with probe radius compensation (workpiece coordinate system)
	10	-	Oriented spindle stop
Value from the active datum table in the active coordinate system, 500	Line	Column	Read values
Read data of the current tool, 950	1	-	Tool length L
	2	-	Tool radius R
	3	-	Tool radius R2
	4	-	Oversize for tool length DL
	5	-	Oversize for tool radius DR
	6	-	Oversize for tool radius DR2
	7	-	Tool locked TL: 0 = not locked, 1 = locked

Group name, ID No.	Number	Index	Meaning
	8	-	Number of replacement tool RT
	9	-	Maximum tool age TIME1
	10	-	Maximum tool age TIME2
	11	-	Current tool age CUR. TIME
	12	-	PLC status
	13	-	Maximum tooth length LCUTS
	14	-	Maximum plunge angle ANGLE
	15	-	TT: Number of teeth CUT
	16	-	TT: Wear tolerance for length LTOL
	17	-	TT: Wear tolerance for radius RTOL
	18	-	TT: Rotational direction DIRECT 0 = positive, –1 = negative
	19	-	TT: Offset for radius R-OFFS R = 99999.9999
	20	-	TT: Offset for length L-OFFS
	21	-	TT: Breakage tolerance in length LBREAK
	22	-	TT: Breakage tolerance in radius RBREAK
	23	-	PLC value
	24	-	Tool type TYPE 0 = cutter, 21 = touch probe
Touch probe cycles, 990	1	-	Approach behavior: 0 = standard behavior 1 = effective radius, safety clearance is zero
	2	-	0 = probe monitoring off 1 = probe monitoring on
Execution status, 992	10	-	Block scan active 1 = yes, 0 = no
	11	-	Search phase
	14	-	Number of the last FN14 error
	16	-	Real execution active 1 = execution , 2 = simulation

Example: Assign the value of the active scaling factor for the Z axis to Q25

```
55 FN18: SYSREAD Q25 = ID210 NR4 IDX3
```

FN19: PLC: Transferring values to the PLC

The function FN 19: PLC transfers up to two numerical values or Q parameters to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
56 FN19: PLC=+10/+Q3
```

FN20: WAIT FOR: NC and PLC synchronization

This function may only be used with the permission of your machine tool builder.

With function FN 20: WAIT FOR you can synchronize the NC and PLC with each other during a program run. The NC stops machining until the condition that you have programmed in the FN 20 block is fulfilled. With FN10 the TNC can check the following operands:

PLC operand	Abbreviation	Address range
Marker	M	0 to 4999
Input	I	0 to 31, 128 to 152 64 to 126 (first PL 401 B) 192 to 254 (second PL 401 B)
Output	O	0 to 30 32 to 62 (first PL 401 B) 64 to 94 (second PL 401 B)
Counter	C	48 to 79
Timer	T	0 to 95
Byte	B	0 to 4095
Word	W	0 to 2047
Double word	D	2048 to 4095

Now for the first time with the TNC 320, HEIDENHAIN has equipped a control with an expanded interface for communication between the PLC and NC. This is a new, symbolic Application Programmer Interface (**API**). The familiar previous PLC-NC interface is also available and can be used if desired. The machine tool builder decides whether the new or old TNC API is used. Enter the name of the symbolic operand as string to wait for the defined condition of the symbolic operand.

The following conditions are permitted in the FN 20 block:

Condition	Abbreviation
Equals	==
Less than	<
Greater than	>
Less than or equal	<=
Greater than or equal	>=

Example: Stop program run until the PLC sets marker 4095 to 1

```
32 FN20: WAIT FOR M4095==1
```

Example: Stop program run until the PLC sets the symbolic operand to 1

```
32 FN20: APISPIN[0].NN_SPICONTROLINPOS==1
```

FN 25: PRESET: Setting a new datum

This function can only be programmed if you have entered the code number 555343 (see "Entering Code Numbers," page 399).

With the function FN 25: PRESET, it is possible to set a new datum in an axis of choice during program run.

- Select a Q parameter function: Press the Q key (in the numerical keypad at right). The Q parameter functions are displayed in a soft-key row.
- To select the additional functions, press the DIVERSE FUNCTIONS soft key.
- Select FN25: Switch the soft-key row to the second level, press the FN25 DATUM SET soft key.
- **Axis?:** Enter the axis where you wish to set the new datum and confirm with ENT.
- **Value to be calculated?:** Enter the coordinate for the new datum point in the active coordinate system.
- **New datum?:** Enter the coordinate that the value to be converted should have in the new coordinate system.

Example: Set a new datum at the current coordinate X+100

```
56 FN25: PRESET = X/+100/+0
```

Example: The current coordinate Z+50 will have the value –20 in the new coordinate system

```
56 FN25: PRESET = Z/+50/-20
```

FN29: PLC: Transferring values to the PLC

The function FN 29: PLC transfers up to eight numerical values or Q parameters to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
56 FN29: PLC=+10/+Q3/+Q8/+7/+1/+Q5/+Q2/+15
```

FN37:EXPORT

You need the FN37: EXPORT function if you want to create your own cycles and integrate them in the TNC. The Q parameters 0 to 99 are effective only locally. This means that the Q parameters are effective only in the program in which they were defined. With the FN37: EXPORT function you can export locally effective Q parameters into another (calling) program.

Example: The local Q parameter Q25 is exported

```
56 FN37: EXPORT Q25
```

Example: The local Q parameters Q25 to Q30 are exported

```
56 FN37: EXPORT Q25 - Q30
```

The TNC exports the value that the parameter has at the time of the EXPORT command.

The parameter is exported only to the presently calling program.

10.9 Accessing Tables with SQL Commands

Introduction

Accessing of tables is programmed on the TNC with SQL commands in "transactions." A transaction consists of multiple SQL commands that guarantee an orderly execution of the table entries.

Tables are configured by the machine manufacturer. Names and designations required as parameters for SQL commands are also specified.

The following **terms** are used:

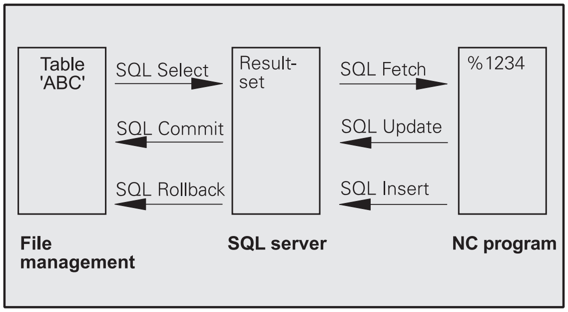
- **Table:** A table consists of x columns and y rows. It is saved as a file in the File Manager of the TNC, and is addressed with the path and file name (=table name). Synonyms can also be used for addressing, as an alternative to the path and file name.
- **Columns:** The number and names of the columns are specified when configuring the table. In some SQL commands the column name is used for addressing.
- **Rows:** The number of rows is variable. You can insert new rows. There are no row numbers or other designators. However, you can select rows based on the contents of a column. Rows can only be deleted in the table editor, not by an NC program.
- **Cell:** The part of a column in a row.
- **Table entry:** Content of a cell.
- **Result set:** During a transaction, the selected columns and rows are managed in the result set. You can view the result set as a sort of "intermediate memory," which temporarily assumes the set of selected columns and rows
- **Synonym:** This term defines a name used for a table instead of its path and file name. Synonyms are specified by the machine manufacturer in the configuration data.

A Transaction

In principle, a transaction consists of the following actions:

- Address table (file), select rows and transfer them to the result set.
- Read rows from the result set, change rows or insert new rows.
- Conclude transaction: If changes/insertions were made, the rows from the result set are placed in the table (file).

Other actions are also necessary so that table entries can be edited in an NC program and to ensure that other changes are not made to copies of the same table rows at the same time. This results in the following **transaction sequence:**



1. A Q parameter is specified for each column to be edited. The Q parameter is assigned to a column—it is "bound" (**SQL BIND...**).
2. Address table (file), select rows and transfer them to the result set. In addition, you define which columns are transferred to the result set (**SQL SELECT...**).

   You can "lock" the selected rows. Other processes can then read these rows, but cannot change the table entries. You should always lock the selected rows when you are going to make changes (**SQL SELECT ... FOR UPDATE**).
3. Read rows from the result set, change rows or insert new rows:
   – Transfer one row of the result set into the Q parameters of your NC program (**SQL FETCH...**).
   – Prepare changes in the Q parameters and transfer one row from the result set (**SQL UPDATE...**).
   – Prepare new table row in the Q parameters and transfer into the result set as a new row (**SQL INSERT...**).
4. Conclude transaction:
   – If changes/insertions were made, the data from the result set is placed in the table (file). The data is now saved in the file. Any locks are canceled, and the result set is released (**SQL COMMIT...**).
   – If table entries were **not** changed or inserted (only read access), any locks are canceled and the result set is released (**SQL ROLLBACK... WITHOUT INDEX**).

Multiple transactions can be edited at the same time.

> You must conclude a transaction, even if it consists solely of read accesses. Only this guarantees that changes/insertions are not lost, that locks are canceled, and that result sets are released.

Result set

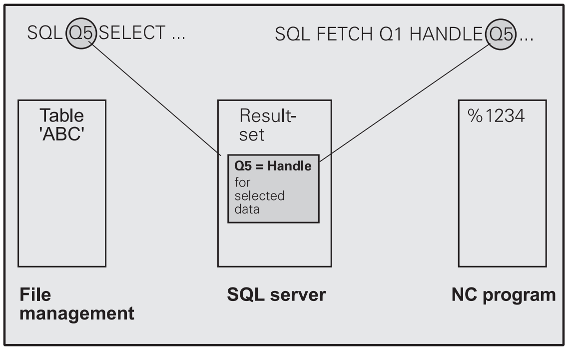
The selected rows are numbered in ascending order within the result set, starting from 0. This numbering is referred to as the **index.** The index is used for read- and write-accesses, enabling a row of the result set to be specifically addressed.

It can often be advantageous to sort the rows in the result set. Do this by specifying the table column containing the sorting criteria. Also select ascending or descending order (**SQL SELECT ... ORDER BY ...**).

The selected rows that were transferred to the result set are addressed with the **HANDLE.** All following SQL commands use the handle to refer to this "set of selected columns and rows."

When concluding a transaction, the handle is released (**SQL COMMIT...** or **SQL ROLLBACK...**). It is then no longer valid.

You can edit more than one result set at the same time. The SQL server assigns a new handle for each "Select" command.



"Binding" Q parameters to columns

The NC program does not have direct access to the table entries in the result set. The data must be transferred in Q parameters. In the other direction, the data is first prepared in the Q parameters and then transferred to the result set.

Specify with **SQL BIND ...** which table columns are mapped to which Q parameters. The Q parameters are "bound" (assigned) to the columns. Columns that are not bound to Q parameters are not included in the read-/write-processes.

If a new table row is generated with **SQL INSERT...,** the columns not bound to Q parameters are filled with default values.

Programming SQL commands

Program SQL commands in the Programming and Editing mode:

SQL

- Call the SQL functions by pressing the SQL soft key.
- Select an SQL command via soft key (see overview) or press the **SQL EXECUTE** soft key and program the SQL command.

Overview of the soft keys

Function	Soft key
SQL EXECUTE Program a "Select" command.	SQL EXECUTE
SQL BIND "Bind" a Q parameter to a table column.	SQL BIND
SQL FETCH Read table rows from the result set and save them in Q parameters.	SQL FETCH
SQL UPDATE Save data from the Q parameters in an existing table row in the result set.	SQL UPDATE
SQL INSERT Save data from the Q parameters in a new table row in the result set.	SQL INSERT
SQL COMMIT Transfer table rows from the result set into the table and conclude the transaction.	SQL COMMIT
SQL ROLLBACK ■ If **INDEX** is not programmed: Discard any changes/insertions and conclude the transaction. ■ If **INDEX** is programmed: The indexed row remains in the result set. All other rows are deleted from the result set. The transaction is **not** concluded.	SQL ROLLBACK

SQL BIND

SQL BIND binds a Q parameter to a table column. The SQL commands "Fetch," "Update" and "Insert" evaluate this binding (assignment) during data transfer between the result set and the NC program.

An **SQL BIND** command without a table or column name cancels the binding. Binding remains effective at the longest until the end of the NC program or subprogram.

- You can program any number of bindings. Read and write processes only take into account the columns that were entered in the "Select" command.
- **SQL BIND...** must be programmed **before** "Fetch," "Update" or "Insert" commands are programmed. You can program a "Select" command without a preceding "Bind" command.
- If in the "Select" command you include columns for which no binding is programmed, an error occurs during read/write processes (program interrupt).

SQL BIND

- **Parameter no. for result:** Q parameter that is bound (assigned) to the table column.
- **Database: Column name:** Enter the table name and column name separated by a period.
  Table name: Synonym or path and file name of this table. The synonym is entered directly, whereas the path and file name are entered in single quotation marks.
  Column designation: Designation of the table column as given in the configuration data.

Example: Bind a Q parameter to a table column

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
```

Example: Cancel binding

```
91 SQL BIND Q881
92 SQL BIND Q882
93 SQL BIND Q883
94 SQL BIND Q884
```

SQL SELECT

SQL SELECT selects table rows and transfers them to the result set.

The SQL server places the data in the result set row-by-row. The rows are numbered in ascending order, starting from 0. This row number, called the **INDEX,** is used in the SQL commands "Fetch" and "Update."

Enter the selection criteria in the **SQL SELECT...WHERE...** option. This lets you restrict the number of rows to be transferred. If you do not use this option, all rows in the table are loaded.

Enter the sorting criteria in the **SQL SELECT...ORDER BY...** option. Enter the column designation and the keyword for ascending/descending order. If you do not use this option, the rows are placed in random order.

Lock out the selected rows for other applications with the **SQL SELECT...FOR UPDATE** option. Other applications can continue to read these rows, but cannot change them. We strongly recommend using this option if you are making changes to the table entries.

Empty result set: If no rows match the selection criteria, the SQL server returns a valid handle but no table entries.

Example: Select all table rows

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
```

Example: Selection of table rows with the WHERE option

```
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE WHERE MEAS_NO<20"
```

Example: Selection of table rows with the WHERE option and Q parameters

```
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE WHERE
MEAS_NO==:'Q11'"
```

Example: Table name defined with path and file name

```
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM 'V:\TABLE\TAB_EXAMPLE' WHERE
MEAS_NO<20"
```

SQL EXECUTE

- **Parameter no. for result:** Q parameter for the handle. The SQL server returns the handle for the group of columns and rows selected with the current select command.
  In case of an error (selection could not be carried out), the SQL server returns the code 1.
  Code 0 identifies an invalid handle.
- **Data bank: SQL command text:** with the following elements:

  SELECT (keyword): Name of the SQL command

  Names of the table columns to be transferred. Separate column names with a comma (see examples). Q parameters must be bound to all columns entered here.

  FROM table name: Synonym or path and file name of this table. The synonym is entered directly, whereas the path and table name are entered in single quotation marks (see examples).

  Optional:
  WHERE selection criteria: A selection criterion consists of a column name, condition (see table) and comparator. Link selection criteria with logical AND or OR.
  Program the comparator directly or with a Q parameter. A Q parameter is introduced with a colon and placed in single quotation marks (see example).

  Optional:
  ORDER BY column name **ASC** to sort in ascending order—or
  ORDER BY column name **DESC** to sort in descending order.
  If neither **ASC** nor **DESC** are programmed, then ascending order is used as the default setting.
  The selected rows are placed in the order determined by the indicated column.

  Optional:
  FOR UPDATE (keyword): The selected rows are locked against write-accesses from other processes.

Condition	Programming
Equal to	= ==
Not equal to	!= <>
Less than	<
Less than or equal to	<=
Greater than	>
Greater than or equal to	>=
Linking multiple conditions:	
Logical AND	AND
Logical OR	OR

SQL FETCH

SQL FETCH reads the row addressed with **INDEX** from the result set, and places the table entries in the bound (assigned) Q parameters. The result set is addressed with the **HANDLE.**

SQL FETCH takes into account all columns entered in the "Select" command.

SQL FETCH

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle or index too large)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).
- **Data bank: Index for SQL result:** Row number within the result set. The table entries of this row are read and are transferred into the bound parameters. If you do not enter an index, the first row is read (n=0). Either enter the row number directly or program the Q parameter containing the index.

Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
```

Example: Row number is programmed directly

```
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX5
```

SQL UPDATE

SQL UPDATE transfers the data prepared in the Q parameters into the row of the result set addressed with **INDEX.** The existing row in the result set is completely overwritten.

SQL UPDATE takes into account all columns entered in the "Select" command.

SQL UPDATE

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle, index too large, value outside of value range or incorrect data format)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).
- **Data bank: Index for SQL result:** Row number within the result set. The table entries prepared in the Q parameters are written to this row. If you do not enter an index, the first row is written to (n=0). Either enter the row number directly or program the Q parameter containing the index.

Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
```

Example: Row number is programmed directly

```
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX5
```

SQL INSERT

SQL INSERT generates a new row in the result set and transfers the data prepared in the Q parameters into the new row.

SQL INSERT takes into account all columns entered in the "Select" command. Table columns not entered in the "Select" command are filled with default values.

SQL INSERT

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle, value outside of value range or incorrect data format)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).

Example: Row number is transferred in a Q parameter

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
40 SQL INSERT Q1 HANDLE Q5
```

SQL COMMIT

SQL COMMIT transfers all rows in the result set back to the table. A lock set with **SELECT...FOR UPDATE** is canceled.

The handle given in the **SQL SELECT** command loses its validity.

SQL COMMIT

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle or equal entries in columns requiring unique entries)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).

Example:

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
40 SQL UPDATE Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL COMMIT Q1 HANDLE Q5
```

SQL ROLLBACK

The execution of **SQL ROLLBACK** depends on whether **INDEX** is programmed:

- If **INDEX** is not programmed: The result set is **not** written back to the table (any changes/insertions are discarded). The transaction is closed and the handle given in the **SQL SELECT** command loses its validity. Typical application: Ending a transaction solely containing read-accesses.
- If **INDEX** is programmed: The indexed row remains. All other rows are deleted from the result set. The transaction is **not** concluded. A lock set with **SELECT...FOR UPDATE** remains for the indexed row. For all other rows it is reset.

SQL ROLLBACK

- **Parameter no. for result:** Q parameter in which the SQL server reports the result:
  0: No error occurred.
  1: Error occurred (incorrect handle)
- **Data bank: SQL access ID:** Q parameter with the **handle** for identifying the result set (also see **SQL SELECT**).
- **Data bank: Index for SQL result:** Row that is to remain in the result set. Either enter the row number directly or program the Q parameter containing the index.

Example:

```
11 SQL BIND Q881 "TAB_EXAMPLE.MEAS_NO"
12 SQL BIND Q882 "TAB_EXAMPLE.MEAS_X"
13 SQL BIND Q883 "TAB_EXAMPLE.MEAS_Y"
14 SQL BIND Q884 "TAB_EXAMPLE.MEAS_Z"
. . .
20 SQL Q5 "SELECT MEAS_NO,MEAS_X,MEAS_Y,
MEAS_Z FROM TAB_EXAMPLE"
. . .
30 SQL FETCH Q1 HANDLE Q5 INDEX+Q2
. . .
50 SQL ROLLBACK Q1 HANDLE Q5
```

10.10 Entering Formulas Directly

Entering formulas

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Press the FORMULA soft key to call the formula functions. The TNC displays the following soft keys in several soft-key rows:

Logic command	Soft key
Addition Example: **Q10 = Q1 + Q5**	+
Subtraction Example: **Q25 = Q7 – Q108**	-
Multiplication Example: **Q12 = 5 * Q5**	*
Division Example: **Q25 = Q1 / Q2**	/
Opening parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	(
Closing parenthesis Example: **Q12 = Q1 * (Q2 + Q3)**	)
Square of a value Example: **Q15 = SQ 5**	SQ
Square root Example: **Q22 = SQRT 25**	SQRT
Sine of an angle Example: **Q44 = SIN 45**	SIN
Cosine of an angle Example: **Q45 = COS 45**	COS
Tangent of an angle Example: **Q46 = TAN 45**	TAN
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: **Q10 = ASIN 0.75**	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: **Q11 = ACOS Q40**	ACOS

Logic command	Soft key
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: **`Q12 = ATAN Q50`**	ATAN
Powers of values Example: **`Q15 = 3^3`**	^
Constant "pi" (3.14159) Example: **`Q15 = PI`**	PI
Natural logarithm (LN) of a number Base 2.7183 Example: **`Q15 = LN Q11`**	LN
Logarithm of a number, base 10 Example: **`Q33 = LOG Q22`**	LOG
Exponential function, 2.7183 to the power of n Example: **`Q1 = EXP Q12`**	EXP
Negate (multiplication by –1) Example: **`Q2 = NEG Q1`**	NEG
Truncate decimal places Form an integer Example: **`Q3 = INT Q42`**	INT
Absolute value of a number Example: **`Q4 = ABS Q22`**	ABS
Truncate places before the decimal point Form a fraction Example: **`Q5 = FRAC Q23`**	FRAC
Check algebraic sign of a number Example: **`Q12 = SGN Q50`** If result for Q12 = 1, then Q50 >= 0 If result for Q12 = –1, then Q50 < 0	SGN
Calculate modulo value Example: **`Q12 = 400 % 360`** Result: Q12 = 40	%

Rules for formulas

Mathematical formulas are programmed according to the following rules:

Higher-level operations are performed first

```
12  Q1 = 5 * 3 + 2 * 10 = 35
```

1st calculation: 5 * 3 = 15

2nd calculation: 2 * 10 = 20

3. Calculation step 15 + 20 = 35

or

```
13  Q2 = SQ 10 - 3^3 = 73
```

1st calculation: 10 squared = 100

2nd calculation: 3 to the power of 3 = 27

3. Calculation step 100 –27 = 73

Distributive law

for calculating with parentheses

a * (b + c) = a * b + a * c

Programming example

Calculate an angle with the arc tangent from the opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q	FORMULA	To select the formula entering function, press the Q key and FORMULA soft key.

PARAMETER NO. FOR RESULT?

ENT	25	Enter the parameter number.
▷	ATAN	Shift the soft-key row and select the arc tangent function.
◁	(	Shift the soft-key row and open the parentheses.
Q	12	Enter Q parameter number 12.
/		Select division.
Q	13	Enter Q parameter number 13.
)	END	Close parentheses and conclude formula entry.

Example NC block

```
37  Q25 = ATAN (Q12/Q13)
```

10.11 Preassigned Q Parameters

The Q parameters Q100 to Q122 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

Values from the PLC: Q100 to Q107

The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Active tool radius: Q108

The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (Tool table or TOOL DEF block)
- Delta value DR from the tool table
- Delta value DR from the TOOL CALL block

Tool axis: Q109

The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = -1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110

The value of Q110 depends on which M function was last programmed for the spindle:

M Function	Parameter value
No spindle status defined	Q110 = -1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M Function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112

The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113

The value of parameter Q113 specifies whether the highest-level NC program (for nesting with PGM CALL) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114

The current value for the tool length is assigned to Q114.

Coordinates after probing during program run

The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe. The coordinates are referenced to the datum that is currently active in the Manual operating mode.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter value
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis Machine-dependent	Q118
Vth axis Machine-dependent	Q119

10.12 String Parameters

Working with string parameters

You need string processing mainly to be able to read value from tables and configuration data.

You can assign a linear sequence of characters (letters, numbers, special characters and spaces) to a string parameter. You can also check and process the assigned or imported values.

Assigning string parameters

You have to assign a string variable before you use it. Use the DECLARE STRING command to do so.

SPECIAL TNC FUNCTIONS

- To select the TNC special functions, press the SPECIAL FUNCTIONS soft key

DECLARE

- Select the DECLARE function

STRING

- Select the STRING soft-key

Example NC block:

```
37  DECLARE STRING QS10 = "TEXT"
```

String processing functions

The STRING FORMULA or FORMULA functions contain various functions for processing the string parameters.

Use the STRING FORMULA function if you want to receive a string parameter (e.g. QS10) as a result.

Q

- Select a Q parameter function: Press the Q key (in the numerical keypad at right). The Q parameter functions are displayed in a soft-key row.
- Shift the soft-key row.

STRING FORMULA

- Select STRING FORMULA function
- Enter the value for the string parameter in which the result is stored
- Press the enter key
- Select the soft key for desired function

ENT

- Press the enter key
- Select the soft key for desired function

The string parameter for a result must also be assigned beforehand. Use the DECLARE STRING function without entering a string.

Use the FORMULA function if you want to receive a numerical value (e.g. Q10) as a result.

Concatenation of string parameters

With the concatenation operator (string parameter || string parameter) you can make a chain of two or more string parameters.

Example: Concatenation of two or more string parameters

```
37  QS10 = QS12 || QS13 || QS14
```

Exporting machine parameters

Because of the organization of the configuration data, access to machine parameters is possible only by designating the key, tag and attribute by using string parameters. Use the CFGREAD function.

Example: Import a machine parameter

```
37  QS20 = CFGREAD( KEY_QS10 TAG_QS11 ATR_QS12 )
```

Converting a numerical value to a string parameter

The TOCHAR function converts a numerical value to a string parameter. The value to be converted can be entered as a number or a Q parameter. Also, you can enter the number of decimal places of the string parameter output.

Example: Convert parameter Q50 as string parameter QS11

```
37  QS11 = TOCHAR( DAT+Q50 DECIMALS4 )
```

Converting a string parameter to a numerical value

The TOCHAR function converts a string parameter to a numerical value. The value to be converted should be only numerical.

Example: Convert string parameter QS11 to a numerical parameter Q82

```
37  Q82 = TONUMB( SRC_QS11 )
```

Reading a substring from a string parameter

With the SUBSTR function you can import a certain range from a string parameter.

Example: A four-character substring (LEN4) is read from the string parameter QS10 beginning with the third character (BEG3).

```
37  QS13 = SUBSTR( SRC_QS10 BEG3 LEN4 )
```

Checking a string parameter

With the INSTR function you can check whether a string parameter is contained in another string parameter.

In SRC_QS, enter the string parameter to be searched. In SEA_QS, enter the string parameter to be found. With the BEG function you can specify a position to begin the search. As the result the TNC returns the first position of appearance. If it does not find such a string parameter, it returns the value 0.

Example: QS10 is checked, starting from the third character, for whether it contains QS13

```
37  Q50 = INSTR( SRC_QS10 SEA_QS13 BEG3 )
```

Reading the length of a string parameter

The STRLEN functions returns the length of a string parameter in the given string variable.

Example: The length of QS15 is requested

```
37  Q52 = STRLEN( SRC_QS15 )
```

Reading the alphabetic order

With the STRCOMP function you can find the alphabetic order of string parameters. If the first string parameter (SRC_QS) is alphabetically before the second (SEA_QS); the TNC returns the result +1. With the reverse sequence the result is –1. If they are alphabetically equal the result is 0.

Example: The alphabetic sequence of QS12 and QS14 is checked

```
37  Q52 = STRCOMP( SRC_QS12 SEA_QS14 )
```

Reading system strings

You can also read string parameters for many system variables (FN 18: SYSREAD). Enter the ID for the system variables plus the value 10000.

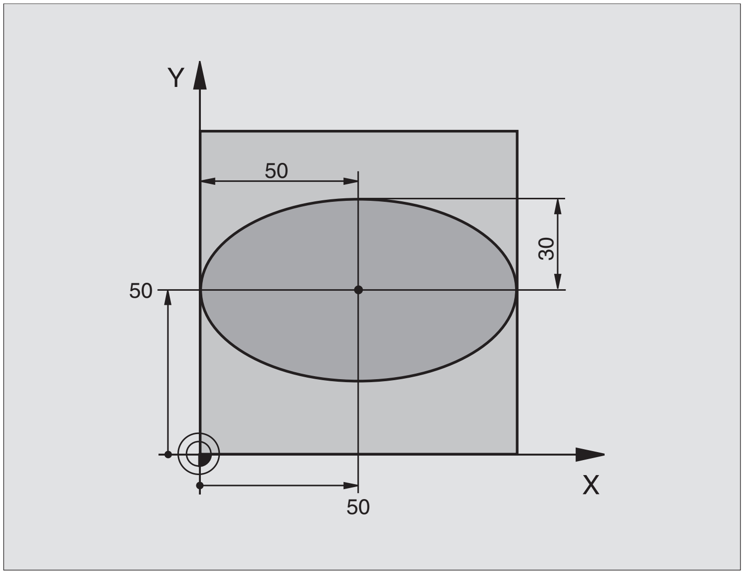
Example: Read the path of the NC program chosen with SEL PGM ".."

```
37  QS14 = SYSSTR( ID10010 NR10 )
```

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculation steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is not taken into account.



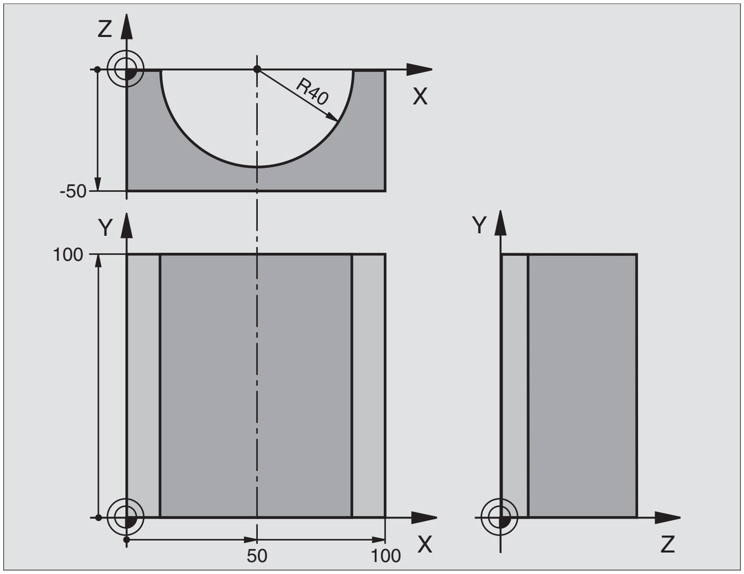
`0 BEGIN PGM ELLIPSE MM`	
`1 FN 0: Q1 = +50`	Center in X axis
`2 FN 0: Q2 = +50`	Center in Y axis
`3 FN 0: Q3 = +50`	Semiaxis in X
`4 FN 0: Q4 = +30`	Semiaxis in Y
`5 FN 0: Q5 = +0`	Starting angle in the plane
`6 FN 0: Q6 = +360`	End angle in the plane
`7 FN 0: Q7 = +40`	Number of calculation steps
`8 FN 0: Q8 = +0`	Rotational position of the ellipse
`9 FN 0: Q9 = +5`	Milling depth
`10 FN 0: Q10 = +100`	Feed rate for plunging
`11 FN 0: Q11 = +350`	Feed rate for milling
`12 FN 0: Q12 = +2`	Set-up clearance for pre-positioning
`13 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the workpiece blank
`14 BLK FORM 0.2 X+100 Y+100 Z+0`	
`15 TOOL DEF 1 L+0 R+2.5`	Define the tool
`16 TOOL CALL 1 Z S4000`	Tool call
`17 L Z+250 R0 FMAX`	Retract the tool
`18 CALL LBL 10`	Call machining operation
`19 L Z+100 R0 FMAX M2`	Retract in the tool axis, end program

20 LBL 10	Subprogram 10: Machining operation
21 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of ellipse
22 CYCL DEF 7.1 X+Q1	
23 CYCL DEF 7.2 Y+Q2	
24 CYCL DEF 10.0 ROTATION	Account for rotational position in the plane
25 CYCL DEF 10.1 ROT+Q8	
26 Q35 = (Q6 - Q5) / Q7	Calculate angle increment
27 Q36 = Q5	Copy starting angle
28 Q37 = 0	Set counter
29 Q21 = Q3 * COS Q36	Calculate X coordinate for starting point
30 Q22 = Q4 * SIN Q36	Calculate Y coordinate for starting point
31 L X+Q21 Y+Q22 R0 FMAX M3	Move to starting point in the plane
32 L Z+Q12 R0 FMAX	Pre-position in tool axis to set-up clearance
33 L Z-Q9 R0 FQ10	Move to working depth
34 LBL 1	
35 Q36 = Q36 + Q35	Update the angle
36 Q37 = Q37 + 1	Update the counter
37 Q21 = Q3 * COS Q36	Calculate the current X coordinate
38 Q22 = Q4 * SIN Q36	Calculate the current Y coordinate
39 L X+Q21 Y+Q22 R0 FQ11	Move to next point
40 FN 12: IF +Q37 LT +Q7 GOTO LBL 1	Unfinished? If not finished, return to LBL 1
41 CYCL DEF 10.0 ROTATION	Reset the rotation
42 CYCL DEF 10.1 ROT+0	
43 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
44 CYCL DEF 7.1 X+0	
45 CYCL DEF 7.2 Y+0	
46 L Z+Q12 F0 FMAX	Move to set-up clearance
47 LBL 0	End of subprogram
48 END PGM ELLIPSE MM	

Example: Concave cylinder machined with spherical cutter

Program sequence

- Program functions only with a spherical cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The machining direction can be altered by changing the entries for the starting and end angles in space:
  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is compensated automatically.



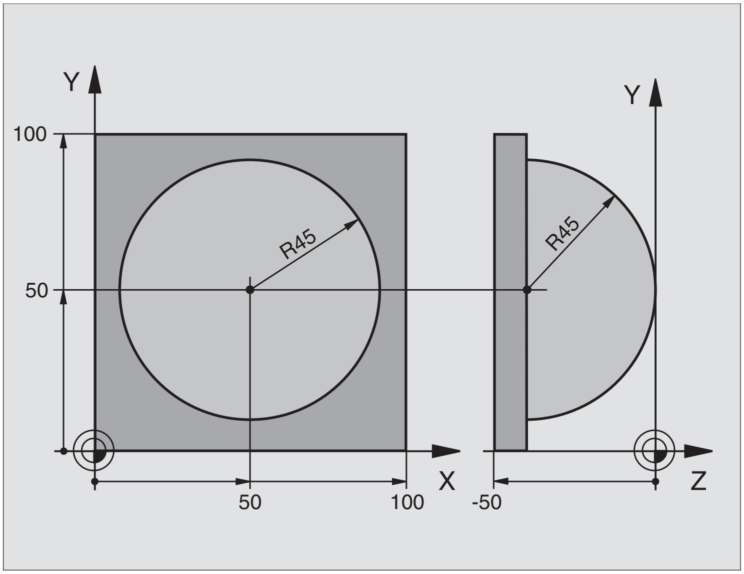
`0 BEGIN PGM CYLIN MM`	
`1 FN 0: Q1 = +50`	Center in X axis
`2 FN 0: Q2 = +0`	Center in Y axis
`3 FN 0: Q3 = +0`	Center in Z axis
`4 FN 0: Q4 = +90`	Starting angle in space (Z/X plane)
`5 FN 0: Q5 = +270`	End angle in space (Z/X plane)
`6 FN 0: Q6 = +40`	Radius of the cylinder
`7 FN 0: Q7 = +100`	Length of the cylinder
`8 FN 0: Q8 = +0`	Rotational position in the X/Y plane
`9 FN 0: Q10 = +5`	Allowance for cylinder radius
`10 FN 0: Q11 = +250`	Feed rate for plunging
`11 FN 0: Q12 = +400`	Feed rate for milling
`12 FN 0: Q13 = +90`	Number of cuts
`13 BLK FORM 0.1 Z X+0 Y+0 Z-50`	Define the workpiece blank
`15 BLK FORM 0.2 X+100 Y+100 Z+0`	
`15 TOOL DEF 1 L+0 R+3`	Define the tool
`16 TOOL CALL 1 Z S4000`	Tool call
`17 L Z+250 R0 FMAX`	Retract the tool
`18 CALL LBL 10`	Call machining operation
`19 FN 0: Q10 = +0`	Reset allowance

`20 CALL LBL 10`	Call machining operation
`21 L Z+100 R0 FMAX M2`	Retract in the tool axis, end program
`22 LBL 10`	Subprogram 10: Machining operation
`23 Q16 = Q6 - Q10 - Q108`	Account for allowance and tool, based on the cylinder radius
`24 FN 0: Q20 = +1`	Set counter
`25 FN 0: Q24 = +Q4`	Copy starting angle in space (Z/X plane)
`26 Q25 = (Q5 - Q4) / Q13`	Calculate angle increment
`27 CYCL DEF 7.0 DATUM SHIFT`	Shift datum to center of cylinder (X axis)
`28 CYCL DEF 7.1 X+Q1`	
`29 CYCL DEF 7.2 Y+Q2`	
`30 CYCL DEF 7.3 Z+Q3`	
`31 CYCL DEF 10.0 ROTATION`	Account for rotational position in the plane
`32 CYCL DEF 10.1 ROT+Q8`	
`33 L X+0 Y+0 R0 FMAX`	Pre-position in the plane to the cylinder center
`34 L Z+5 R0 F1000 M3`	Pre-position in the tool axis
`35 LBL 1`	
`36 CC Z+0 X+0`	Set pole in the Z/X plane
`37 LP PR+Q16 PA+Q24 FQ11`	Move to starting position on cylinder, plunge-cutting obliquely into the material
`38 L Y+Q7 R0 FQ12`	Longitudinal cut in Y+ direction
`39 FN 1: Q20 = +Q20 + +1`	Update the counter
`40 FN 1: Q24 = +Q24 + +Q25`	Update solid angle
`41 FN 11: IF +Q20 GT +Q13 GOTO LBL 99`	Finished? If finished, jump to end
`42 LP PR+Q16 PA+Q24 FQ11`	Move in an approximated “arc” for the next longitudinal cut
`43 L Y+0 R0 FQ12`	Longitudinal cut in Y– direction
`44 FN 1: Q20 = +Q20 + +1`	Update the counter
`45 FN 1: Q24 = +Q24 + +Q25`	Update solid angle
`46 FN 12: IF +Q20 LT +Q13 GOTO LBL 1`	Unfinished? If not finished, return to LBL 1
`47 LBL 99`	
`48 CYCL DEF 10.0 ROTATION`	Reset the rotation
`49 CYCL DEF 10.1 ROT+0`	
`50 CYCL DEF 7.0 DATUM SHIFT`	Reset the datum shift
`51 CYCL DEF 7.1 X+0`	
`52 CYCL DEF 7.2 Y+0`	
`53 CYCL DEF 7.3 Z+0`	
`54 LBL 0`	End of subprogram
`55 END PGM CYLIN`	

Example: Convex sphere machined with end mill

Program sequence

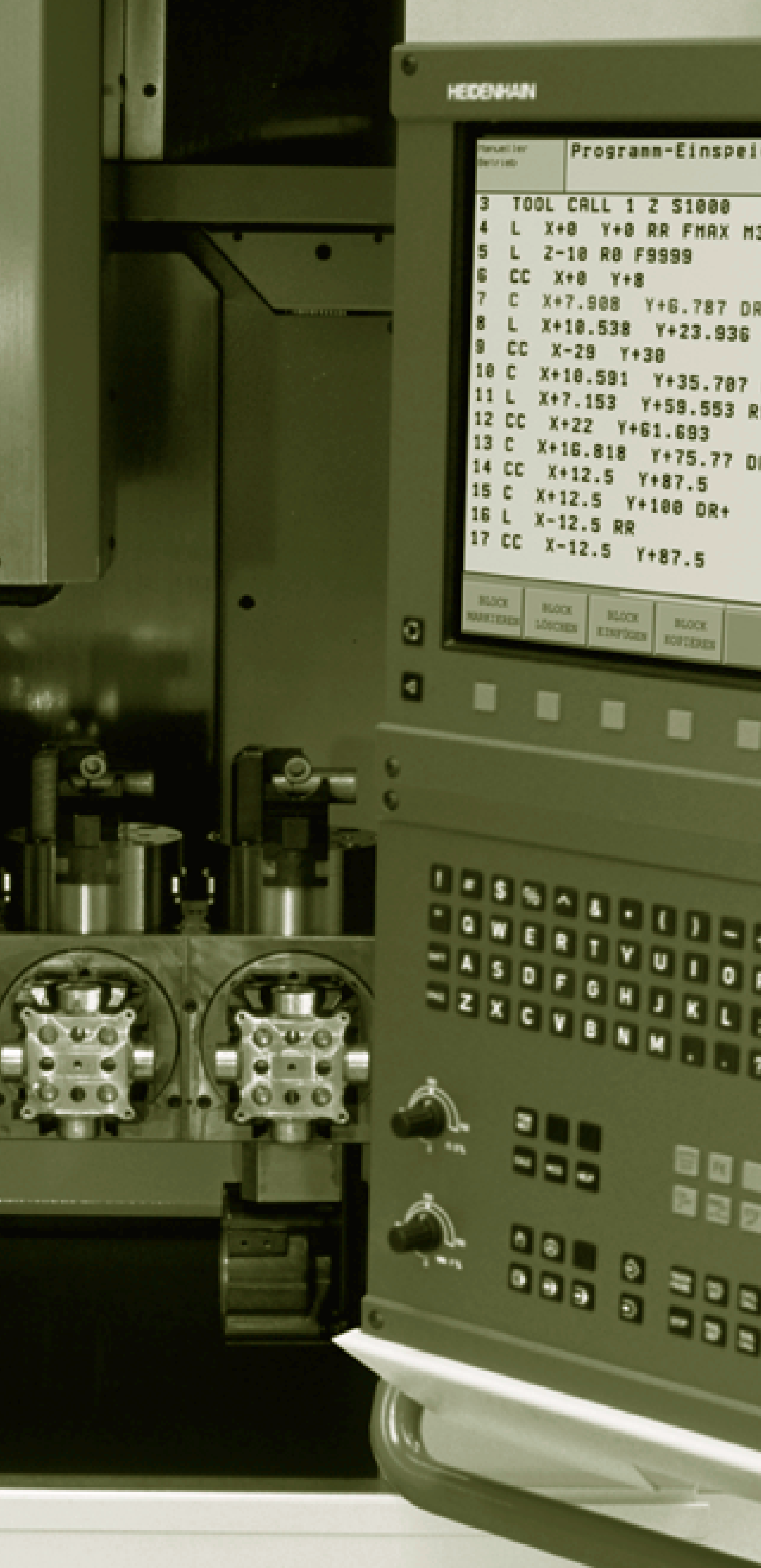
- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined in Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically.



`0 BEGIN PGM SPHERE MM`	
`1 FN 0: Q1 = +50`	Center in X axis
`2 FN 0: Q2 = +50`	Center in Y axis
`3 FN 0: Q4 = +90`	Starting angle in space (Z/X plane)
`4 FN 0: Q5 = +0`	End angle in space (Z/X plane)
`5 FN 0: Q14 = +5`	Angle increment in space
`6 FN 0: Q6 = +45`	Radius of the sphere
`7 FN 0: Q8 = +0`	Starting angle of rotational position in the X/Y plane
`8 FN 0: Q9 = +360`	End angle of rotational position in the X/Y plane
`9 FN 0: Q18 = +10`	Angle increment in the X/Y plane for roughing
`10 FN 0: Q10 = +5`	Allowance in sphere radius for roughing
`11 FN 0: Q11 = +2`	Set-up clearance for pre-positioning in the tool axis
`12 FN 0: Q12 = +350`	Feed rate for milling
`13 BLK FORM 0.1 Z X+0 Y+0 Z-50`	Define the workpiece blank
`14 BLK FORM 0.2 X+100 Y+100 Z+0`	
`15 TOOL DEF 1 L+0 R+7.5`	Define the tool
`16 TOOL CALL 1 Z S4000`	Tool call
`17 L Z+250 R0 FMAX`	Retract the tool

18 CALL LBL 10	Call machining operation
19 FN 0: Q10 = +0	Reset allowance
20 FN 0: Q18 = +5	Angle increment in the X/Y plane for finishing
21 CALL LBL 10	Call machining operation
22 L Z+100 R0 FMAX M2	Retract in the tool axis, end program
23 LBL 10	Subprogram 10: Machining operation
24 FN 1: Q23 = +Q11 + +Q6	Calculate Z coordinate for pre-positioning
25 FN 0: Q24 = +Q4	Copy starting angle in space (Z/X plane)
26 FN 1: Q26 = +Q6 + +Q108	Compensate sphere radius for pre-positioning
27 FN 0: Q28 = +Q8	Copy rotational position in the plane
28 FN 1: Q16 = +Q6 + -Q10	Account for allowance in the sphere radius
29 CYCL DEF 7.0 DATUM SHIFT	Shift datum to center of sphere
30 CYCL DEF 7.1 X+Q1	
31 CYCL DEF 7.2 Y+Q2	
32 CYCL DEF 7.3 Z-Q16	
33 CYCL DEF 10.0 ROTATION	Account for starting angle of rotational position in the plane
34 CYCL DEF 10.1 ROT+Q8	
35 LBL 1	Pre-position in the tool axis
36 CC X+0 Y+0	Set pole in the X/Y plane for pre-positioning
37 LP PR+Q26 PA+Q8 R0 FQ12	Pre-position in the plane
38 CC Z+0 X+Q108	Set pole in the Z/X plane, offset by the tool radius
39 L Y+0 Z+0 FQ12	Move to working depth

40 LBL 2	
41 LP PR+Q6 PA+Q24 R9 FQ12	Move upward in an approximated “arc”
42 FN 2: Q24 = +Q24 - +Q14	Update solid angle
43 FN 11: IF +Q24 GT +Q5 GOTO LBL 2	Inquire whether an arc is finished. If not finished, return to LBL 2.
44 LP PR+Q6 PA+Q5	Move to the end angle in space
45 L Z+Q23 R0 F1000	Retract in the tool axis
46 L X+Q26 R0 FMAX	Pre-position for next arc
47 FN 1: Q28 = +Q28 + +Q18	Update rotational position in the plane
48 FN 0: Q24 = +Q4	Reset solid angle
49 CYCL DEF 10.0 ROTATION	Activate new rotational position
50 CYCL DEF 10.0 ROT+Q28	
51 FN 12: IF +Q28 LT +Q9 GOTO LBL 1	
52 FN 9: IF +Q28 EQU +Q9 GOTO LBL 1	Unfinished? If not finished, return to label 1
53 CYCL DEF 10.0 ROTATION	Reset the rotation
54 CYCL DEF 10.1 ROT+0	
55 CYCL DEF 7.0 DATUM SHIFT	Reset the datum shift
56 CYCL DEF 7.1 X+0	
57 CYCL DEF 7.2 Y+0	
58 CYCL DEF 7.3 Z+0	
59 LBL 0	End of subprogram
60 END PGM SPHERE MM	



11

Test Run and Program Run

11.1 Graphics

Function

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following three display modes: Using soft keys, select whether you desire:

- Plan view
- Projection in 3 planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter. For this purpose, enter R2 = R in the tool table.

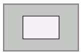
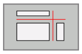
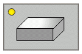
The TNC will not show a graphic if

- the current program has no valid blank form definition
- no program is selected

The graphic simulation is not possible for program sections or programs in which rotary axis movements are defined. In this case, the TNC will display an error message.

Overview of display modes

The control displays the following soft keys in the Program Run and Test Run modes of operation:

Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Limitations during program run

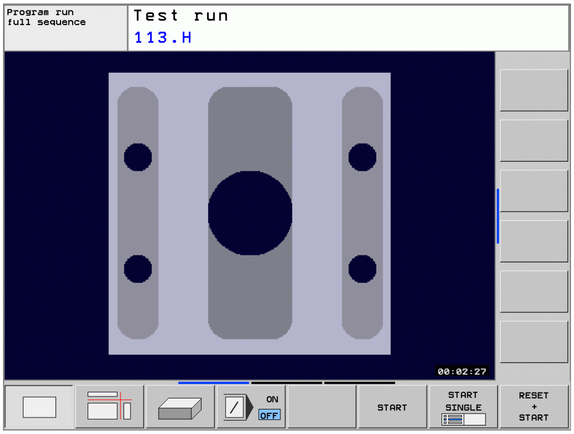
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text **ERROR** in the graphics window. The machining process is continued, however.

Plan view

This is the fastest of the three graphic display modes.

- Press the soft key for plan view.
- Regarding depth display, remember:

The deeper the surface, the darker the shade.

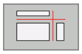


Projection in 3 planes

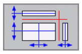
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes.

Details can be isolated in this display mode for magnification (see “Magnifying details,” page 380).

In addition, you can shift the sectional planes with the corresponding soft keys:

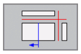
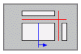
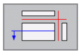
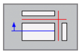
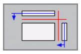
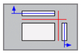


▶ Select the soft key for projection in three planes.



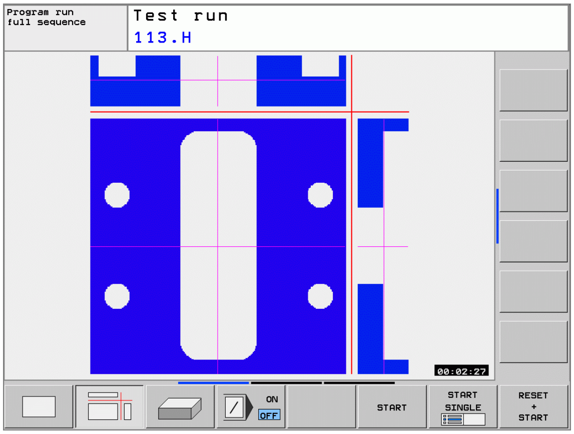
▶ Shift the soft-key row and select the soft key for sectional planes.

▶ The TNC then displays the following soft keys:

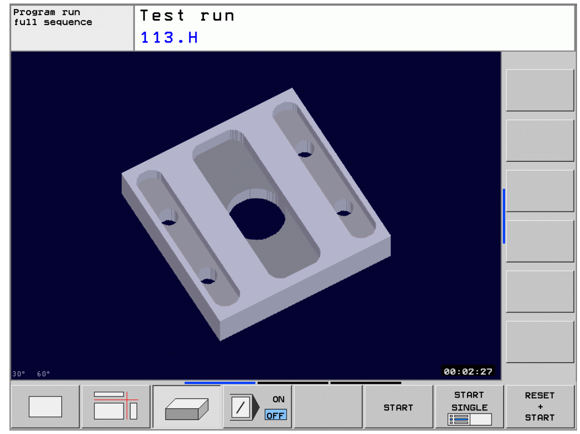
Function	Soft keys	
Shift the vertical sectional plane to the right or left		
Shift the vertical sectional plane forward or backward		
Shift the horizontal sectional plane upwards or downwards		

The positions of the sectional planes are visible during shifting.

The default setting of the sectional plane is selected so that it lies in the working plane and, in the tool axis, on the workpiece center.



3-D view

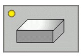


The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

You can rotate the 3-D display about the vertical and horizontal axes. The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation.

The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation.

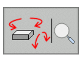
In the Test Run mode of operation you can isolate details for magnification, see "Magnifying details," page 380.



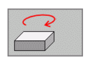
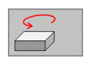
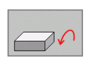
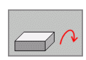
▶ Press the soft key for 3-D view.

Rotating the 3-D view

▶ Shift the soft-key row until the soft-key for the rotation functions appears.



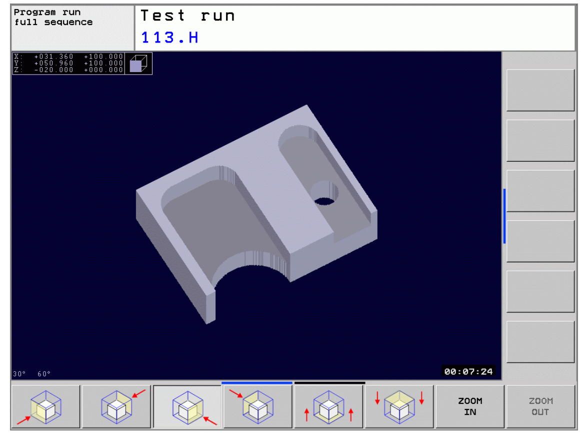
▶ Select the functions for rotation:

Function	Soft keys	
Rotate in 15° steps about the vertical axis		
Rotate in 15° steps about the horizontal axis		

Magnifying details

You can magnify details in the Test Run and a program run operating modes and in the projection in 3 planes and the 3-D display modes.

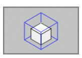
The graphic simulation or the program run, respectively, must first have been stopped. A detail magnification is always effective in all display modes.



Changing the detail magnification

The soft keys are listed in the table.

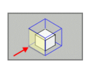
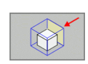
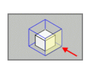
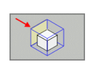
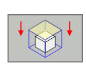
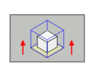
- Interrupt the graphic simulation, if necessary.
- Shift the soft-key row in the Test Run mode, or in a program run mode, respectively, until the soft key for detail enlargement appears.



- Select the functions for section magnification.
- Press the corresponding soft key to select the workpiece surface (see table below).
- To reduce or magnify the blank form, press and hold the ZOOM IN or ZOOM OUT soft keys.
- Shift the soft-key row and select the TRANSFER DETAIL soft key
- Restart the test run or program run by pressing the START soft key (RESET + START returns the workpiece blank to its original state).

Coordinates for magnifying details

The TNC displays the selected workpiece side for each axis and the coordinates of the remaining form during a detail magnification.

Function	Soft keys	
Select the left/right workpiece surface		
Select the front/back workpiece surface		
Select the top/bottom workpiece surface		
Shift the sectional plane to reduce or magnify the blank form	–	+
Select the isolated detail	TRANSFER DETAIL	

After a new workpiece detail magnification is selected, the control "forgets" previously simulated machining operations. The TNC then displays machined areas as unmachined areas.

Repeating graphic simulation

A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.

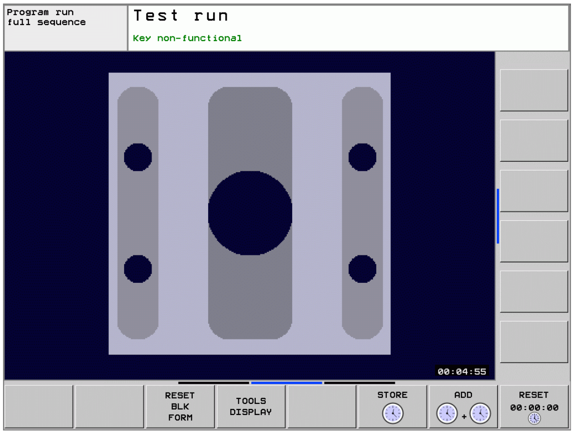
Function	Soft key
Restore workpiece blank to the detail magnification in which it was last shown.	RESET BLK FORM
Reset detail magnification so that the machined workpiece or workpiece blank is displayed as it was programmed with BLK FORM.	WINDOW BLK FORM

With the WINDOW BLANK FORM soft key the TNC returns the graphic of the workpiece blank to its originally programmed dimensions.

Measuring the machining time

Program Run modes of operation

The timer counts and displays the time from program start to program end. The timer stops whenever machining is interrupted.



Test Run

The timer displays the time that the TNC calculates from the duration of tool movements. The time calculated by the TNC can only conditionally be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

Activating the stopwatch function

Shift the soft-key rows until the TNC displays the following soft keys with the stopwatch functions:

Stopwatch functions	Soft key
Store displayed time	STORE
Display the sum of stored time and displayed time	ADD
Clear displayed time	RESET 00:00:00

11.2 Showing the Workpiece in the Working Space

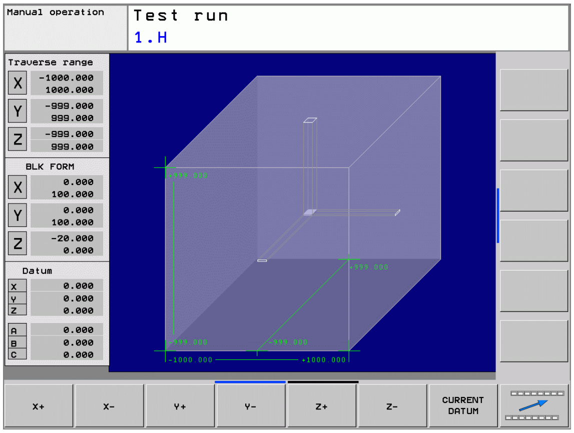
Function

This MOD function enables you to graphically check the position of the workpiece blank or reference point in the machine's working space and to activate work space monitoring in the Test Run mode of operation. This function is activated with the **datum set** soft key.

Another transparent cuboid represents the workpiece blank. Its dimensions are shown in the **BLK FORM** table. The TNC takes the dimensions from the workpiece blank definition of the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the traverse-range cuboid. You can view the position of the active datum within the traverse range by pressing the **CURRENT DATUM** soft key.

For a test run when working-space monitoring is deactivated, it does not matter where the workpiece blank is located within the working space. However, if you activate working-space monitoring, you must graphically shift the workpiece blank so that it lies within the working space. Use the soft keys shown in the table.

You can also activate the current datum for the Test Run operating mode (see the last line of the following table).

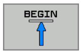
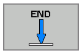


Function	Soft keys	
Shift workpiece blank in positive/negative X direction	X+	X-
Shift workpiece blank in positive/negative Y direction	Y+	Y-
Shift workpiece blank in positive/negative Z direction	Z+	Z-
Show workpiece blank referenced to the set datum	AKT.BZG. PUNKT	

11.3 Functions for Program Display

Overview

In the Program Run modes of operation as well as in the Test Run mode, the TNC provides the following soft keys for displaying a part program in pages:

Function	Soft key
Go back in the program by one screen	
Go forward in the program by one screen	
Go to beginning of program	
Go to end of program	

11.4 Test Run

Function

In the Test Run mode of operation you can simulate programs and program sections to prevent errors from occurring during program run. The TNC checks the programs for the following:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

- Blockwise test run
- Optional block skip
- Functions for graphic simulation
- Measuring the machining time
- Additional status display

Running a program test

If the central tool file is active, a tool table must be active (status S) to run a program test. Select a tool table via the file manager (PGM MGT) in the Test Run mode of operation.

- Select the Test Run operating mode
- Call the file manager with the PGM MGT key and select the file you wish to test, or
- Go to the program beginning: Select line "0" with the GOTO key and confirm your entry with the ENT key.

The TNC then displays the following soft keys:

Function	Soft key
Reset the blank form and test the entire program	RESET + START
Test the entire program	START
Test each program block individually	START SINGLE
Halt program test (soft key only appears once you have started the program test)	STOP

You can interrupt the program test and continue it again at any point—even within a machining cycle. In order to continue the test, the following actions must not be performed:

- Selecting another block with the GOTO key
- Making changes to the program
- Switching the operating mode
- Selecting a new program

11.5 Program Run

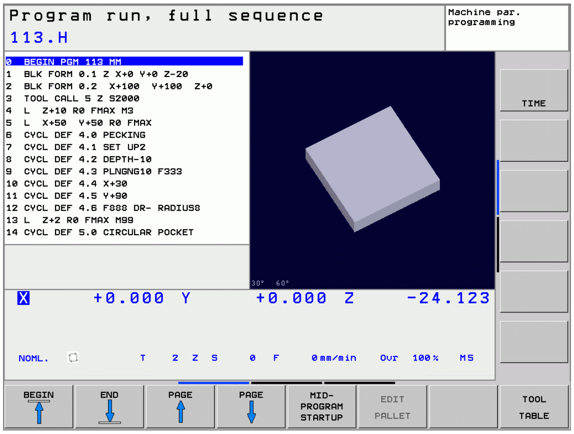
Function

In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or up to a program stop.

In the Program Run, Single Block mode of operation you must start each block separately by pressing the machine START button.

The following TNC functions can be used in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Optional block skip
- Editing the tool table TOOL.T
- Check and change Q parameters
- Superimpose handwheel positioning
- Functions for graphic simulation
- Additional status display



Run a part program

Preparation

1. Clamp the workpiece to the machine table.
2. Set the datum.
3. Select the necessary tables and pallet files (status M).
4. Select the part program (status M).

> You can adjust the feed rate and spindle speed with the override knobs.
>
> It is possible to reduce the rapid traverse speed when starting the NC program using the FMAX soft key. The entered value remains in effect even after the machine has been turned off and on again. In order to re-establish the original rapid traverse speed, you need to re-enter the corresponding value.

Program Run, Full Sequence

- Start the part program with the machine START button.

Program Run, Single Block

- Start each block of the part program individually with the machine START button.

Interrupting machining

There are several ways to interrupt a program run:

- Programmed interruptions
- Pressing the machine STOP button

If the TNC registers an error during program run, it automatically interrupts the machining process.

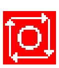
Programmed interruptions

You can program interruptions directly in the part program. The TNC interrupts the program run at a block containing one of the following entries:

- STOP (with and without a miscellaneous function)
- Miscellaneous function M0, M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

Interrupting the machining process with the machine STOP button

- Press the machine STOP button: The block that the TNC is currently executing is not completed. The NC stop signal in the status display blinks (see table).
- If you do not wish to continue the machining process, you can reset the TNC with the INTERNAL STOP soft key. The NC stop signal in the status display goes out. In this case, the program must be restarted from the program beginning.

Symbol	Meaning
	Program run stopped.

Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the Manual Operation mode.

Application example: Retracting the spindle after tool breakage

- Interrupt machining.
- Enable the external direction keys: Press the MANUAL OPERATION soft key.
- Move the axes with the machine axis direction buttons.

On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Refer to your machine manual.

Resuming program run after an interruption

> If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.

If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations (e.g. datum shift, rotation, mirroring)
- The coordinates of the circle center that was last defined

> Note that the stored data remain active until they are reset (e.g. if you select a new program).

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION soft key).

Resuming program run with the START button

You can resume program run by pressing the machine START button if the program was interrupted in one of the following ways:

- The machine STOP button was pressed.
- An interruption was programmed.

Resuming program run after an error

If the error message is not blinking:

- ▶ Remove the cause of the error.
- ▶ To clear the error message from the screen, press the CE key.
- ▶ Restart the program, or resume program run where it was interrupted.

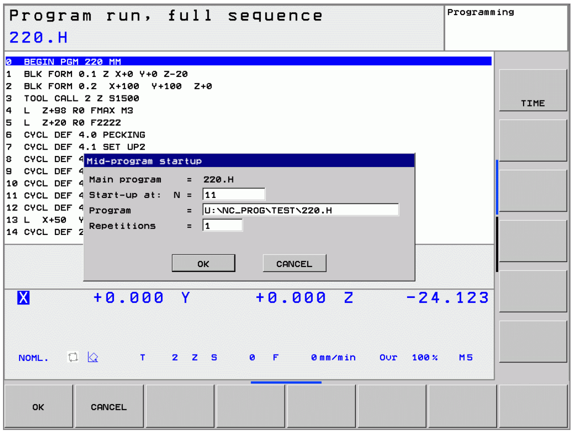
If there is a “processor check error”:

- ▶ Switch to Manual mode,
- ▶ Press the OFF soft key.
- ▶ Remove the cause of the error.
- ▶ Start again.

If you cannot correct the error, write down the error message and contact your repair service agency.

Mid-program startup (block scan)

The RESTORE POS AT feature must be enabled and adapted by the machine tool builder. Refer to your machine manual.



With the RESTORE POS. AT feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.

Mid-program startup must not begin in a subprogram.

All necessary programs, tables and pallet files must be selected in a Program Run mode of operation (status M).

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan.

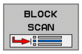
User requests are not possible during mid-program startup.

After a block scan, return the tool to the calculated position with RESTORE POSITION.

Tool length compensation does not take effect until after the tool call and a following positioning block. This applies if you have only changed the tool length.

The TNC skips all touch probe cycles in a mid-program startup. Result parameters that are written to from these cycles might therefore remain empty.

- To go to the first block of the current program to start a block scan, enter GOTO “0”.

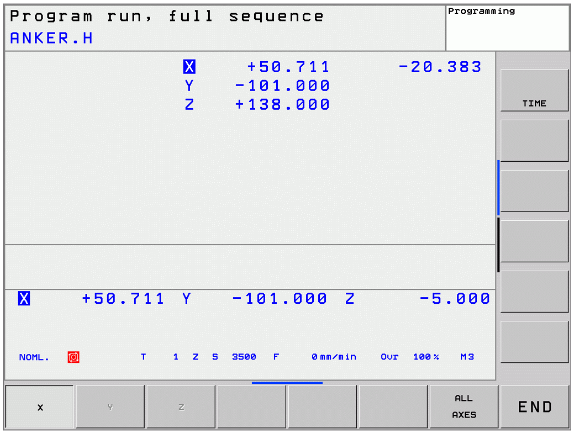


- To select mid-program startup, press the RESTORE POS AT N soft key.
- **`Start-up at N:`** Enter the block number N at which the block scan should end.
- **`Program:`** Enter the name of the program containing block N.
- **`Repetitions:`** If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.
- To start the block scan, press the machine START button.
- Contour approach (see following section).

Returning to the contour

With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.
- Return to the contour after a block scan with RESTORE POS AT, for example after an interruption with INTERNAL STOP.



- To select a return to contour, press the RESTORE POSITION soft key.
- Restore machine status, if required.
- To move the axes in the sequence that the TNC suggests on the screen, press the machine START button.
- To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START key.
- To resume machining, press the machine START key.

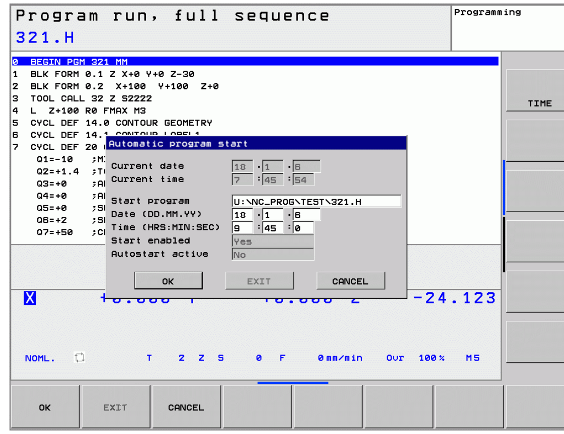
11.6 Automatic Program Start

Function

The TNC must be specially prepared by the machine tool builder for use of the automatic program start function. Refer to your machine manual.

CAUTION—danger to life!

The autostart function must not be used on machines that do not have an enclosed working space.



In a Program Run operating mode, you can use the AUTOSTART soft key (see figure at upper right) to define a specific time at which the program that is currently active in this operating mode is to be started:

AUTOSTART

- Show the window for entering the starting time (see figure at center right).
- **Time (h:min:sec):** Time of day at which the program is to be started.
- **Date (DD.MM.YYYY):** Date at which the program is to be started.
- To activate the start, select OK

11.7 Optional Block Skip

Function

In a test run or program run, the TNC can skip over blocks that begin with a slash "/":

- To run or test the program without the blocks preceded by a slash, set the soft key to ON.

- To run or test the program with the blocks preceded by a slash, set the soft key to OFF.

This function does not work for TOOL DEF blocks.

After a power interruption the control returns to the most recently selected setting.

Inserting the "/" character

- In the **`Programming and Editing`** mode you select the block in which the character is to be inserted.

- Press the SHOW BLOCK soft key

Erasing the "/" character

- In the **`Programming and Editing`** mode you select the block in which the character is to be erased.

- Press the HIDE BLOCK soft key

11.8 Optional Program-Run Interruption

Function

The TNC optionally interrupts the program run or test run at blocks containing M01. If you use M01 in the Program Run mode, the TNC does not switch off the spindle or coolant.

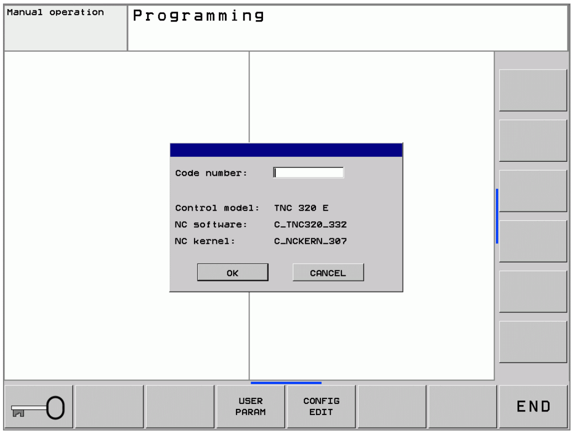
- Do not interrupt Program Run or Test Run at blocks containing M01: Set soft key to OFF.

- Interrupt Program Run or Test Run at blocks containing M01: Set soft key to ON.

12

MOD Functions

12.1 MOD Functions



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

Selecting the MOD functions

Call the operating mode in which you wish to change the MOD functions.

MOD

▶ To select the MOD functions, press the MOD key.

Changing the settings

▶ Select the desired MOD function in the displayed menu with the arrow keys.

There are three possibilities for changing a setting, depending on the function selected:

- Enter only the number
- Change the setting by pressing the ENT key
- Change a setting via a selection window. If more than one possibility is available for a particular setting, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the desired setting directly by pressing the arrow keys and then confirming with ENT. If you don't want to change the setting, close the window again with END.

Exiting the MOD functions

▶ Close the MOD functions with the END key or END soft key.

Overview of MOD functions

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

Programming and Editing:

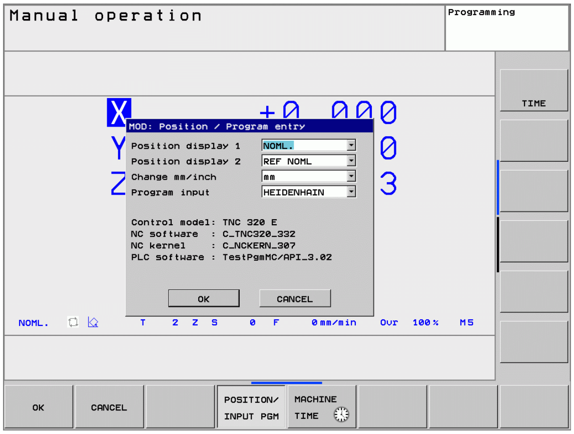
- Display software numbers
- Enter code number
- Machine-specific user parameters (if provided)

Test Run:

- Display software numbers
- Show active tool table in the test run
- Show active datum table in the test run

In all other modes:

- Display software numbers
- Select position display
- Unit of measurement (mm/inches)
- Programming language for MDI
- Select the axes for actual position capture
- Display operating time



12.2 Software Numbers

Function

The following software numbers are displayed on the TNC screen after the MOD functions have been selected:

- **Control model:** Designation of the control (managed by HEIDENHAIN)
- **NC software:** Number of the NC software (managed by HEIDENHAIN)
- **NC kernel:** Number of the NC software (managed by HEIDENHAIN)
- **PLC software:** Number or name of the PLC software (managed by your machine tool builder)

12.3 Entering Code Numbers

Function

The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Enable access to Ethernet configuration	NET123
Enable special functions for Q-parameter programming	555343

12.4 Machine-Specific User Parameters

Function

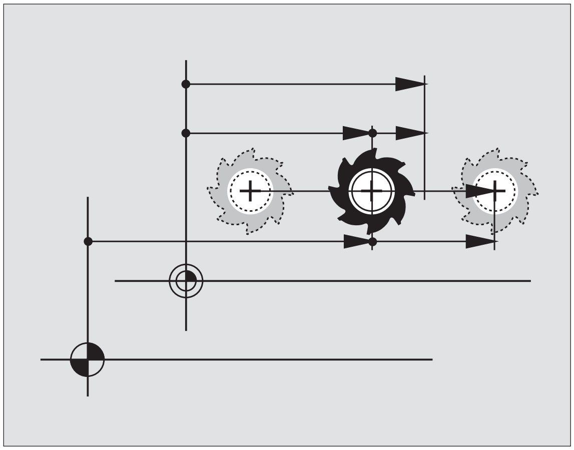
To enable you to set machine-specific functions, your machine tool builder can define which machine parameters are available as user parameters.

Refer to your machine manual.

12.5 Position Display Types

Function

In the Manual Operation mode and in the Program Run modes of operation, you can select the type of coordinates to be displayed.



The figure at right shows the different tool positions:

- Starting position
- Target position of the tool
- Workpiece datum
- Machine datum

The TNC position displays can show the following coordinates:

Function	Display
Nominal position: the value presently commanded by the TNC	NOML.
Actual position; current tool position	ACTL.
Reference position; the actual position relative to the machine datum	REF ACTL
Reference position; the nominal position relative to the machine datum	REF NOML
Servo lag: difference between nominal and actual positions (following error)	LAG
Distance remaining to the programmed position; difference between actual and target positions	DIST.

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

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

12.6 Unit of Measurement

Function

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm) set the Change mm/inches function to mm. The value is displayed to 3 decimal places.
- To select the inch system (e.g. X = 0.6216 inches) set the Change mm/inches function to inches. The value is displayed to 4 decimal places.

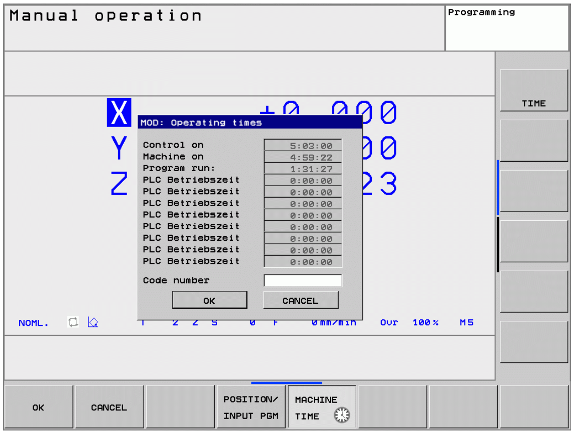
If you would like to activate the inch display, the TNC shows the feed rate in inch/min. In an inch program you must enter the feed rate larger by a factor of 10.

12.7 Display Operating Times

Function

The machine tool builder can provide further operating time displays. The machine tool manual provides further information.

The MACHINE TIME soft key enables you to see various types of operating times:



Operating time	Meaning
Control ON	Operating time of the control since put into service
Machine ON	Operating time of the machine tool since put into service
Program Run	Duration of controlled operation since put into service

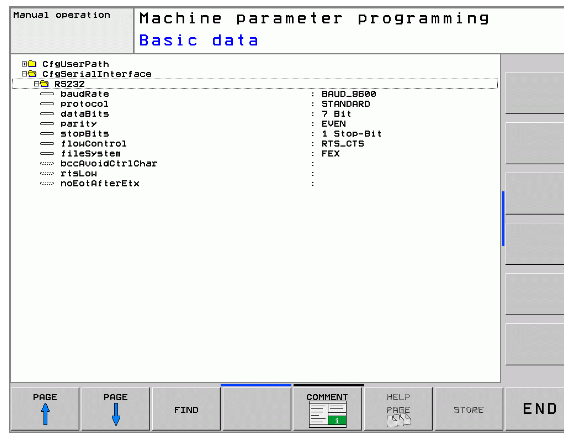
12.8 Setting the Data Interfaces

Serial interface on the TNC 320

The TNC 320 automatically uses the LSV2 transmission protocol for serial data transfer. The LSV2 protocol is permanent and cannot be changed except for setting the baud rate (machine parameter **baudRateLsv2**). You can also specify another type of transmission (interface). The settings described below are therefore effective only for the respectively indicated interface.

Function

To set up a data interface, select the file management (PGM MGT) and press the MOD key. Press the MOD key again and enter the code number123. The TNC shows the user parameter **GfgSerialInterface**, in which you can enter the following settings:



Setting the RS-232 interface

Open the RS232 folder. The TNC then displays the following settings:

Setting the baud rate (baudRate)

You can set the BAUD RATE (data transfer speed) from 110 to 115 200 baud.

Set the protocol (protocol)

The data transmission protocol controls the data flow of a serial transmission (comperable with MP 5030).

Data transmission protocol	Selection
Standard dialog transfer	STANDARD
Blockwise data transfer	BLOCKWISE
Transmission without protocol	RAW_DATA

Set the data bits (dataBits)

By setting the data bits you define whether a character is transmitted with 7 or 8 data bits.

Parity check (parity)

The parity bit helps the receiver to detect transmission errors. The parity bit can be formed in three different ways:

- No parity (NONE): There is no error detection
- Even parity (EVEN): Here there is an error if the receiver finds that it has received an odd number of set bits
- Odd parity (ODD): Here there is an error if the receiver finds that it has received an even number of set bits

Setting the stop bits (stopBits)

The start bit and one or two stop bits enable the receiver to synchronize to every transmitted character during serial data transmission.

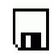
Setting the handshake (flowControl)

With a handshake, two devices are checking the data transmission. A distinction is made between software handshaking and hardware handshaking.

- No dataflow checking (NONE): Handshaking is not active
- Hardware handshaking (RTS_CTS): Transmission stop is active through RTS
- Software handshaking (XON_XOFF): Transmission stop is active through DC3 (XOFF)

Setting the operating mode of the external device (fileSystem)

> The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the FE2 and FEX operating modes.

External device	Operating mode	Symbol
PC with HEIDENHAIN data transfer software TNCremoNT	LSV2	
HEIDENHAIN floppy disk units	FE1	
Non-HEIDENHAIN devices such as punchers, PC without TNCremoNT	FEX	

Software for data transfer

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremoNT data transfer software. With TNCremoNT, data transfer is possible with all HEIDENHAIN controls via the serial interface or the Ethernet interface.

You can download the current version of TNCremoNT free of charge from the HEIDENHAIN Filebase (www.heidenhain.de, <service>, <download area>, <TNCremo NT>).

System requirements for TNCremoNT:

- PC with 486 processor or higher
- Operating system Windows 95, Windows 98, Windows NT 4.0, Windows 2000
- 16 MB RAM
- 5 MB free memory space on your hard disk
- An available serial interface or connection to the TCP/IP network

Installation under Windows

- Start the SETUP.EXE installation program with the File Manager (Explorer).
- Follow the setup program instructions.

Starting TNCremoNT under Windows

- Click <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremoNT>

When you start TNCremoNT for the first time, TNCremoNT automatically tries to set up a connection with the TNC.

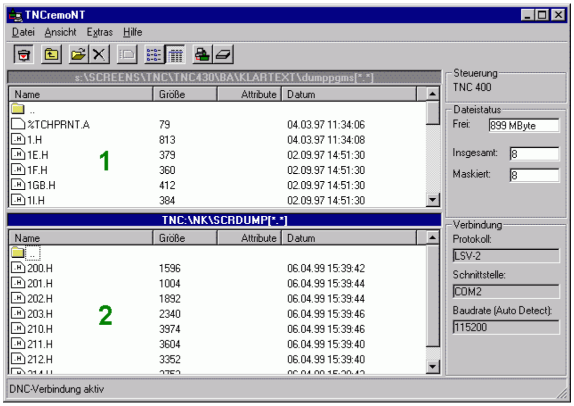
Data transfer between the TNC and TNCremoNT

Check whether the TNC is connected to the correct serial port on your PC or to the network, respectively.

Once you have started TNCremoNT, you will see a list of all files that are stored in the active directory in the upper section of the main window **1**. Using the menu items <File> and <Change directory>, you can change the active directory or select another directory on your PC.

If you want to control data transfer from the PC, establish the connection with your PC in the following manner:

- Select <File>, <Setup connection>. TNCremoNT now receives the file and directory structure from the TNC and displays this at the bottom left of the main window **2**.
- To transfer a file from the TNC to the PC, select the file in the TNC window with a mouse click and drag and drop the highlighted file into the PC window **1**.
- To transfer a file from the PC to the TNC, select the file in the PC window with a mouse click and drag and drop the highlighted file into the TNC window **2**.



If you want to control data transfer from the TNC, establish the connection with your PC in the following way:

- Select <Extras>, <TNCserver>. TNCremoNT is now in server mode. It can receive data from the TNC and send data to the TNC.
- You can now call the file management functions on the TNC by pressing the key PGM MGT (see “Data transfer to or from an external data medium” on page 70) and transfer the desired files.

End TNCremoNT

Select the menu items <File>, <Exit>.

> Refer also to the TNCremoNT context-sensitive help texts where all of the functions are explained in more detail. The help texts must be called with the F1 key.

12.9 Ethernet Interface

Introduction

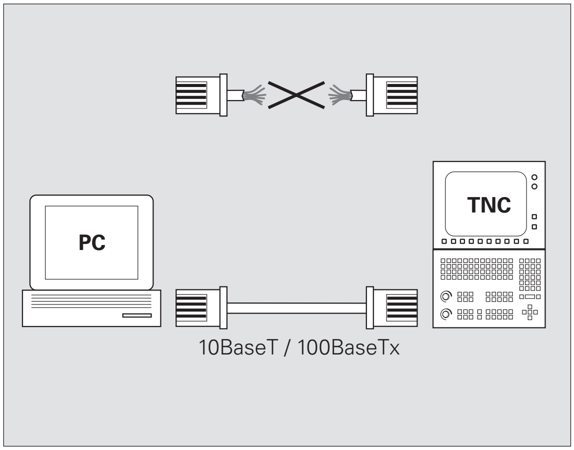
The TNC is shipped with a standard Ethernet card to connect the control as a client in your network. The TNC transmits data via the Ethernet card with

- the **smb** protocol (**s**erver **m**essage **b**lock) for Windows operating systems, or
- the **TCP/IP** protocol family (Transmission Control Protocol/Internet Protocol) and with support from the NFS (Network File System).

Connection possibilities

You can connect the Ethernet card in your TNC to your network through the RJ45 connection (X26, 100BaseTX or 10BaseT), or directly to a PC. The connection is metallically isolated from the control electronics.

For a 100BaseTX or 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.



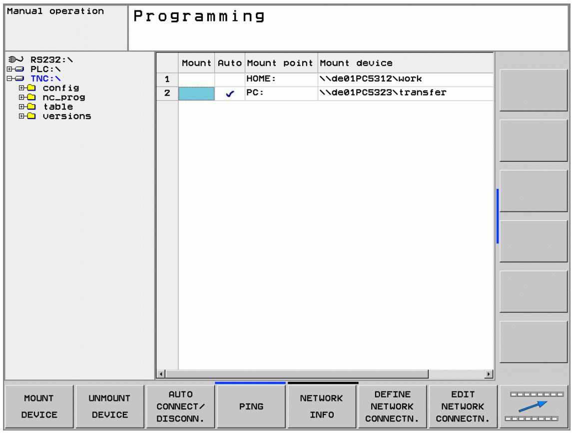
The maximum cable length between TNC and a node depends on the quality grade of the cable, the sheathing and the type of network (100BaseTX or 10BaseT).

No great effort is required to connect the TNC directly to a PC that has an Ethernet card. Simply connect the TNC (port X26) and the PC with an Ethernet crossover cable (trade names: crossed patch cable or STP cable).

Connecting the control to the network

Function overview of network configuration

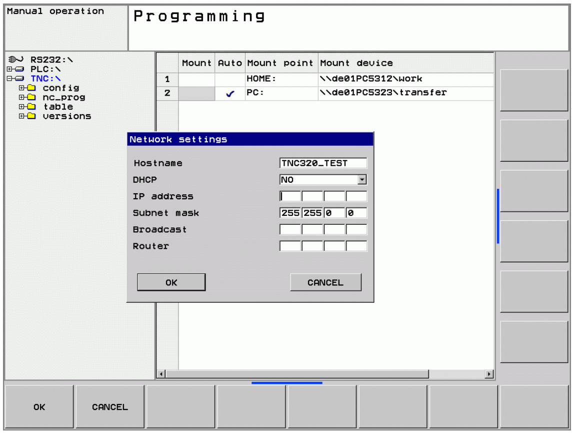
- In the file manager (PGM MGT), select the **Network** Soft key



Function	Soft key
Make a connection to the selected network drive. Successful connection is indicated by a check mark under Mount.	MOUNT DEVICE
Separates the connection to a network drive.	UNMOUNT DEVICE
Activates or deactivates the Automount function (= automatic connection of the network drive during control startup). The status of the function is indicated by a check mark under Auto in the network drive table.	AUTO MOUNT
With the ping function you can check whether a connection is available to a certain participant. The address is entered as four decimal numbers separated by periods (dotted-decimal notation).	PING
The TNC displays an overview window with information on the active network connections.	NETWORK INFO
Configures access to network drives. (Selectable only after entry of the MOD code number NET123.)	DEFINE NETWORK CONNECTN.
Opens the dialog window for editing the data of an existing network connection. (Selectable only after entry of the MOD code number NET123.)	EDIT NETWORK CONNECTN.
Configures the network address of the control. (Selectable only after entry of the MOD code number NET123.)	CONFIGURE NETWORK
Deletes an existing network connection. (Selectable only after entry of the MOD code number NET123.)	DELETE NETWORK CONNECTN.

Configuring the network address of the control.

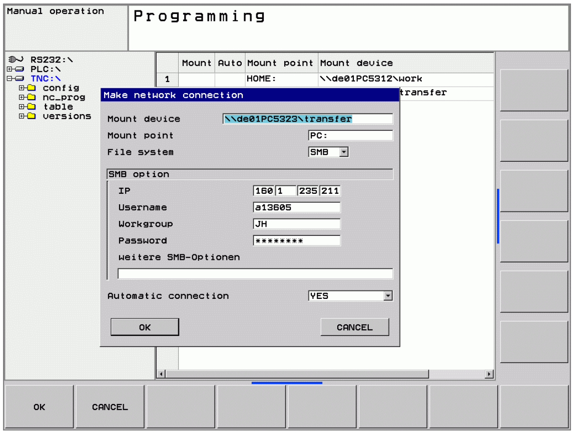
- Connect the TNC (port X26) with a network or a PC
- In the file manager (PGM MGT), select the **Network** soft key.
- Press the MOD key. Enter the code number **NET123**.
- Press the **CONFIGURE NETWORK** soft key to enter the network setting for a specific device (see figure at center right).
- It opens the dialog window for the network configuration



Setting	Meaning
HOSTNAME	Name under which the control logs onto the network. If you use a host name, you must enter the "Fully Qualified Hostname" here. If you do not enter a name here, the control uses the so-called null authentication.
DHCP	DHCP = **D**ynamic **H**ost **C**onfiguration **P**rotocol In the drop-down menu, set **YES.** Then the control automatically draws its network address (IP address), subnet mask, default router and any broadcast address from a DHCP server in the network. The DHCP server identifies the control by its hostname. Your company network must be specially prepared for this function. Contact you network administrator.
IP ADDRESS	Network address of the control: In each of the four adjacent input fields you can enter 3 digits of the IP address. To jump to the next field, press the ENT key. Your network administrator assigns the control's network address.
SUBNET MASK	Serves to differentiate between the network ID and the host ID in the network. Your network administrator assigns the control's subnet mask.
BROADCAST	Broadcast address of the control. It is required only if it differs from the standard setting. The standard setting is formed from the network ID and the host ID, for which all bits are set to 1.
ROUTER	Network address default router: Required only if your network consists of several subnets connected by router.

The entered network configuration does not become effective until the control is rebooted. After the network configuration is concluded with the OK button or soft key, the control asks for confirmation and reboots.

Configuring network access to other devices (mount)



> Make sure that the person configuring your TNC is a network specialist.
>
> The parameters **username, workgroup** and **password** do not need to be entered in all Windows operating systems.

- Connect the TNC (port X26) with a network or a PC
- In the file manager (PGM MGT), select the **Network** soft key.
- Press the MOD key. Enter the code number **NET123**.
- Press the **DEFINE NETWORK CONNECTN.** soft key.
- It opens the dialog window for the network configuration

Setting	Meaning
Mount device	■ Connection over NFS: Directory name to be mounted. This is formed from the network address of the device, a colon, and the name of the directory. Enter the network address as four decimal numbers separated by periods (dotted-decimal notation). Use the correct capitalization when entering the path. ■ To connect individual Windows computers, enter the network name and the share name of the computer, e.g. //PC1791NT/C
Mount point	Device name: The device name entered here is displayed on the control in the program management for the mounted network, e.g. WORLD: (The name must end with a colon!)
File system	File system type: ■ NFS: Network File System ■ SMB: Windows network
NFS option	**rsize:** Packet size in bytes for data reception **wsize:** Packet size for data transmission in bytes **time0=:** Time in tenths of a second, after which the control repeats an unanswered Remote Procedure Call. **soft: YES** repeats the Remote Procedure Call until the NFS server answers. If **NO** is entered, it is not repeated

Setting	Meaning
SMB option	Options that concern the SMB file system type: Options are given without space characters, separated only by commas. Pay attention to capitalization. Options: **ip:** IP address of the Windows PC to which the control is to be connected **username:** User name with which the control is to log on **workgroup:** Work group under which the control is to log on **password:** Password with which the TNC is to log on (up to 80 characters) Further SMB options: Input of further options for the Windows network
Automatic connection	Automount (YES or NO): Here you specify whether the network will be automatically mounted when the control starts up. Devices not automatically mounted can be mounted anytime later in the program management.

You do not need to indicate the protocol with the iTNC 530. It uses the transmission protocol according to RFC 894.

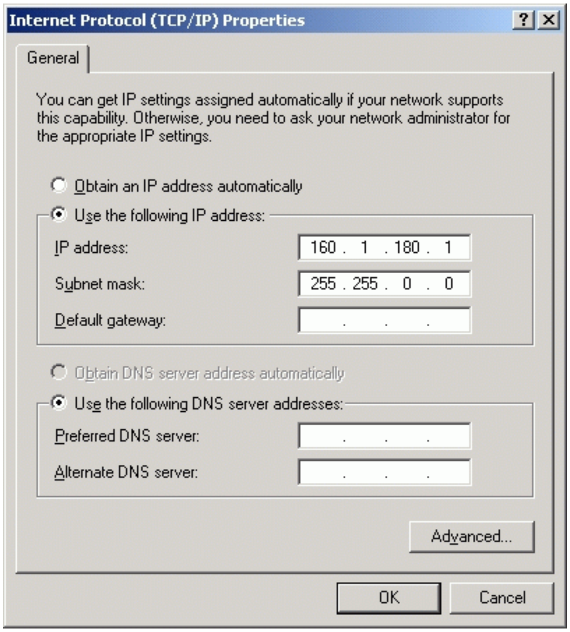
Settings on a PC with Windows 2000

Prerequisite:

The network card must already be installed on the PC and ready for operation.

If the PC that you want to connect the iTNC to is already integrated in your company network, then keep the PC's network address and adapt the iTNC's network address accordingly.

- To open Network Connections, click <Start>, <Control Panel>, <Network and Dial-up Connections>, and then Network Connections.
- Right-click the <LAN connection> symbol, and then <Properties> in the menu that appears.
- Double-click <Internet Protocol (TCP/IP)> to change the IP settings (see figure at top right).
- If it is not yet active, select the <Use the following IP address> option.
- In the <IP address> input field, enter the same IP address that you entered for the PC network settings on the iTNC, e.g. 160.1.180.1
- Enter 255.255.0.0 in the <Subnet mask> input field.
- Confirm the settings with <OK>.
- Save the network configuration with <OK>. You may have to restart Windows now.



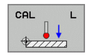
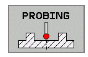
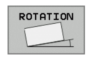
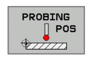
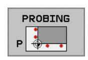
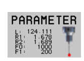
13

Touch Probe Cycles in the Manual and Electronic Handwheel Modes

13.1 Introduction

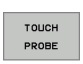
Overview

The following functions are available in the Manual mode:

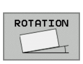
Function	Soft key	Page
Calibrate the effective length		Page 417
Calibrate the effective radius		Page 418
Measure a basic rotation using a line		Page 420
Set the datum in any axis		Page 422
Set a corner as datum		Page 423
Set the circle center as datum		Page 424
Touch probe system data management		Page 424

Selecting probe cycles

- Select the Manual Operation or Electronic Handwheel mode of operation.



- To choose the touch probe functions, press the TOUCH PROBE soft key. The TNC displays additional soft keys—see table above.



- To select the probe cycle, press the appropriate soft key, for example PROBING ROT, and the TNC displays the associated menu.

13.2 Calibrating a Touch Trigger Probe

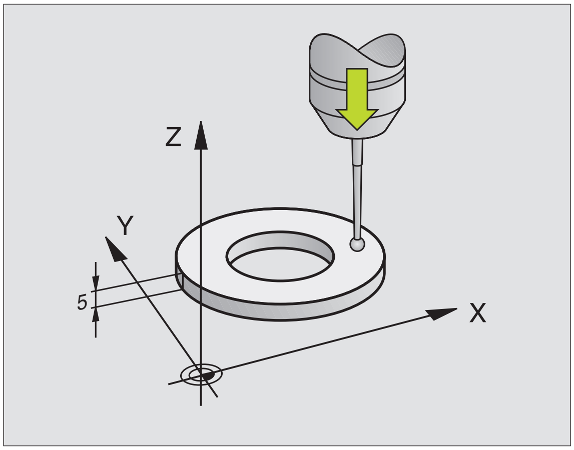
Introduction

The touch probe must be calibrated in the following cases:

- Commissioning
- Stylus breakage
- Stylus exchange
- Change in the probe feed rate
- Irregularities caused, for example, when the machine heats up

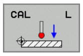
During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the touch probe, clamp a ring gauge of known height and known internal radius to the machine table.

Calibrating the effective length



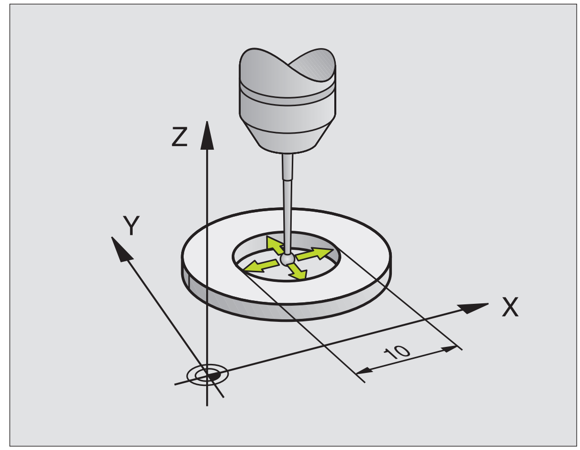
> The effective length of the touch probe is always referenced to the tool datum. The machine tool builder usually defines the spindle tip as the tool datum.

- Set the datum in the spindle axis such that for the machine tool table Z=0.



- To select the calibration function for the touch probe length, press the TOUCH PROBE and CAL. L soft keys. The TNC then displays a menu window with four input boxes.
- Enter the tool axis (with the axis key).
- Datum: Enter the height of the ring gauge.
- The menu items Effective ball radius and Effective length do not require input.
- Move the touch probe to a position just above the ring gauge.
- To change the traverse direction (if necessary), press a soft key or an arrow key.
- To probe the upper surface of the ring gauge, press the machine START button.

Calibrating the effective radius and compensating center misalignment

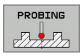


After the touch probe is inserted, it normally needs to be aligned exactly with the spindle axis. The misalignment is measured with this calibration function and compensated electronically.

The TNC rotates the 3-D touch probe by 180° for calibrating the center misalignment.

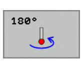
Proceed as follows for manual calibration:

- In the Manual Operation mode, position the ball tip in the bore of the ring gauge.



- To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL. R soft key.
- Enter the radius of the ring gauge.
- To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the effective ball-tip radius.
- If you want to terminate the calibration function at this point, press the END soft key.

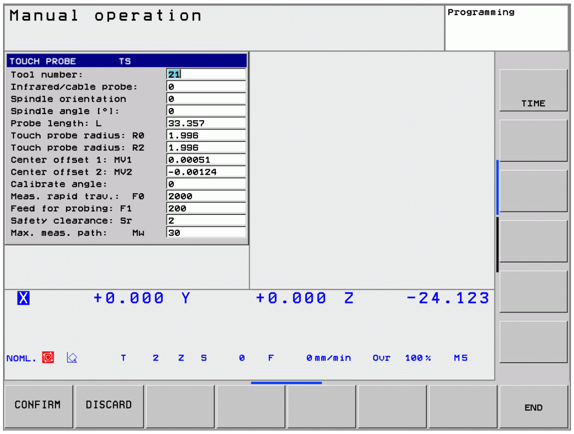
> In order to be able to determine ball-tip center misalignment, the TNC needs to be specially prepared by the machine manufacturer. The machine tool manual provides further information.



- If you want to determine the ball-tip center misalignment, press the180° soft key. The TNC rotates the touch probe by 180°.
- To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the ball-tip center misalignment.

Displaying calibration values

The TNC stores the effective length and radius, as well as the center misalignment, for use when the touch probe is needed again. You can display the values on the screen with the soft keys PARAMETER. The TNC always uses the values from the touch probe management, even it values are also entered in the tool table.



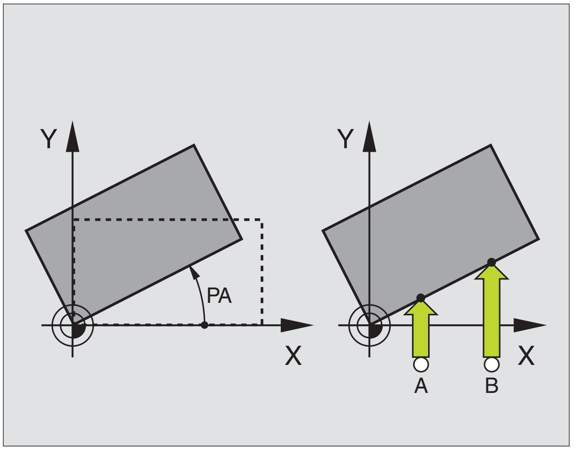
> Make sure that you have activated the correct tool number before using the touch probe, regardless of whether you wish to run the touch probe cycle in automatic mode or manual mode.

13.3 Compensating Workpiece Misalignment

Introduction

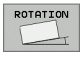
The TNC electronically compensates workpiece misalignment by computing a “basic rotation.”

For this purpose, the TNC sets the rotation angle to the desired angle with respect to the reference axis in the working plane. See figure at right.



> Select the probe direction perpendicular to the angle reference axis when measuring workpiece misalignment.
>
> To ensure that the basic rotation is calculated correctly during program run, program both coordinates of the working plane in the first positioning block.
>
> You can also use a basic rotation in conjunction with the PLANE function. In this case, first activate the basic rotation and then the PLANE function.

Measuring the basic rotation

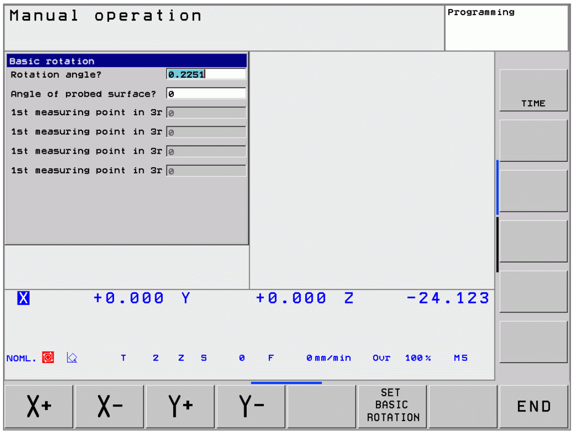


- Select the probe function by pressing the PROBING ROT soft key.
- Position the ball tip at a starting position near the first touch point.
- Select the probe direction perpendicular to the angle reference axis: Select the axis by soft key.
- To probe the workpiece, press the machine START button.
- Position the ball tip at a starting position near the second touch point.
- To probe the workpiece, press the machine START button. The TNC determines the basic rotation and displays the angle after the dialog **Rotation angle =**

Displaying a basic rotation

The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. The TNC also displays the rotation angle in the additional status display (STATUS POS.).

In the status display a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.



To cancel a basic rotation

- Select the probe function by pressing the PROBING ROT soft key.
- Enter a rotation angle of zero and confirm with the ENT key.
- Terminate the probe function by pressing the END key.

13.4 Setting the Datum with a 3-D Touch Probe

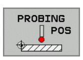
Introduction

The following functions are available for setting the datum on an aligned workpiece:

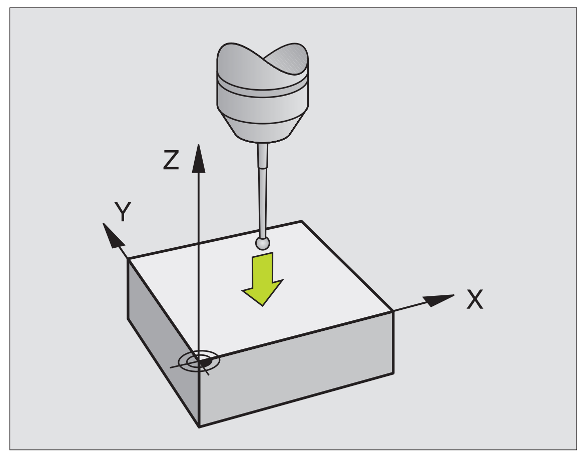
- Datum setting in any axis with PROBING POS
- Defining a corner as datum with PROBING P
- Setting the datum at a circle center with PROBING CC

> Note that during an active datum shift the TNC always bases the probed value on the active preset (or on the datum most recently set in the Manual operating mode), although the datum shift is included in the position display.

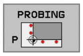
To set the datum in any axis (see figure at right)



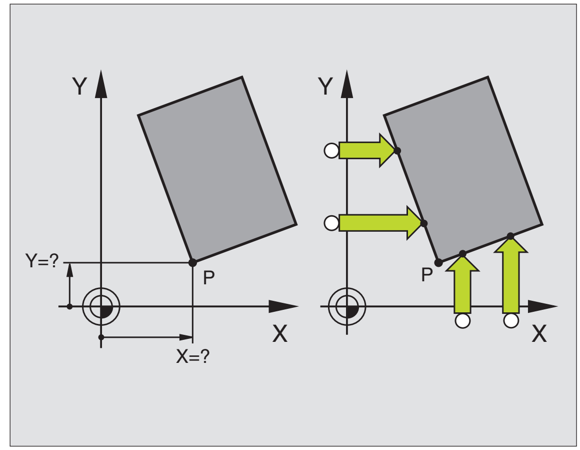
- Select the probe function by pressing the PROBING POS soft key.
- Move the touch probe to a starting position near the touch point.
- Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–. Selection is made via soft keys.
- To probe the workpiece, press the machine START button.
- **Datum:** Enter the nominal coordinate (e.g. 0) and confirm your entry with the SET DATUM soft key.
- To terminate the probe function, press the END key.



Corner as datum—using points already probed for a basic rotation (see figure at right)

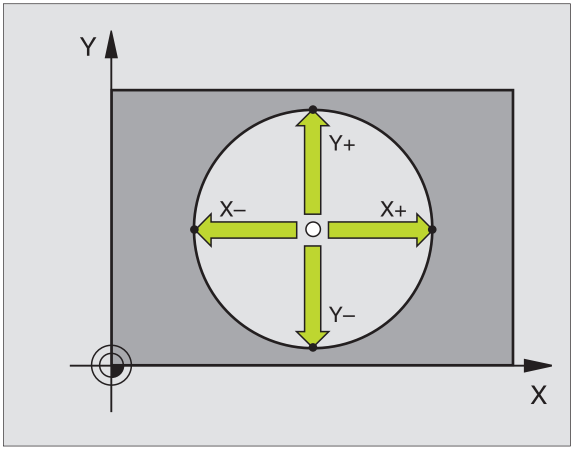


- Select the probe function by pressing the PROBING P soft key.
- Select the probe direction by soft key.
- To probe the workpiece, press the machine START button.
- Probe both workpiece sides twice.
- To probe the workpiece, press the machine START button.
- **Datum:** Enter both datum coordinates into the menu window, and confirm your entry with the SET DATUM soft key.
- To terminate the probe function, press the END key.



Circle center as datum

With this function, you can set the datum at the center of bore holes, circular pockets, cylinders, studs, circular islands, etc.



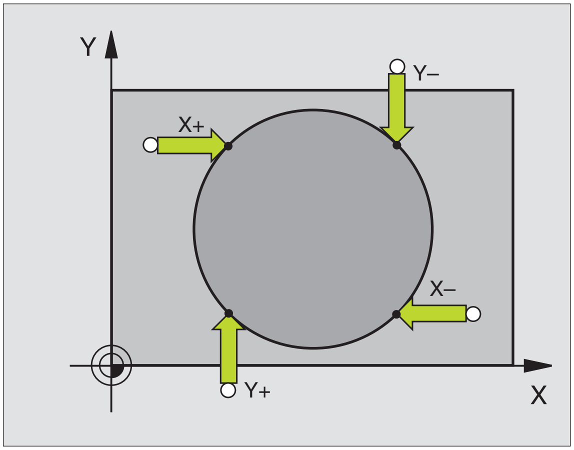
Inside circle

The TNC automatically probes the inside wall in all four coordinate axis directions.

For incomplete circles (circular arcs) you can choose the appropriate probing direction.

- Position the touch probe approximately in the center of the circle.

- Select the probe function by pressing the PROBING CC soft key
- To probe the workpiece, press the machine START button four times. The touch probe touches four points on the inside of the circle.
- If you are probing to find the stylus center (only possible on machines with spindle orientation), press the 180° soft key and probe another four points on the inside of the circle.
- If you are not probing to find the stylus center, press the END key.
- **Datum:** In the menu window, enter both datum coordinates and confirm your entry with the SET DATUM soft key
- To terminate the probe function, press the END key.



Outside circle

- Position the touch probe at the starting position for the first touch point outside of the circle.
- Select the probe direction by soft key.
- To probe the workpiece, press the machine START button.
- Repeat the probing process for the remaining three points. See figure at lower right.
- **Datum:** Enter the coordinates of the datum and confirm your entry with the SET DATUM soft key.
- To terminate the probe function, press the END key.

After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

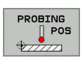
13.5 Measuring Workpieces with a 3-D Touch Probe

Introduction

You can also use the touch probe in the Manual and Electronic Handwheel operating modes to make simple measurements on the workpiece. Numerous programmable probing cycles are available for more complex measuring tasks (see “Automatic Workpiece Measurement” on page 430). With a 3-D touch probe you can determine:

- position coordinates, and from them,
- dimensions and angles on the workpiece.

To find the coordinate of a position on an aligned workpiece:



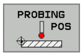
- ▶ Select the probing function by pressing the PROBING POS soft key.
- ▶ Move the touch probe to a starting position near the touch point.
- ▶ Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection.
- ▶ To probe the workpiece, press the machine START button.

The TNC shows the coordinates of the touch point as datum.

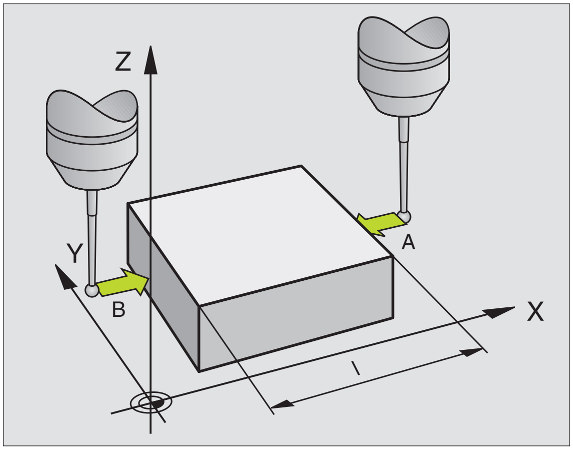
Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point: See “Corner as datum—using points already probed for a basic rotation (see figure at right),” page 423. The TNC displays the coordinates of the probed corner as datum.

To measure workpiece dimensions



- Select the probing function by pressing the PROBING POS soft key.
- Position the touch probe at a starting position near the first touch point A.
- Select the probing direction by soft key.
- To probe the workpiece, press the machine START button.
- If you will need the current datum later, write down the value that appears in the datum display.
- Datum: Enter “0”.
- To terminate the dialog, press the END key.
- Select the probe function by pressing the PROBING POS soft key.
- Position the touch probe at a starting position near the second touch point B
- Select the probe direction with the soft keys: Same axis but from the opposite direction.
- To probe the workpiece, press the machine START button.



The value displayed as datum is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

- Select the probing function by pressing the PROBING POS soft key.
- Probe the first touch point again.
- Set the datum to the value that you wrote down previously.
- To terminate the dialog, press the END key.

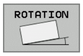
Measuring angles

You can use the 3-D touch probe to measure angles in the working plane. You can measure

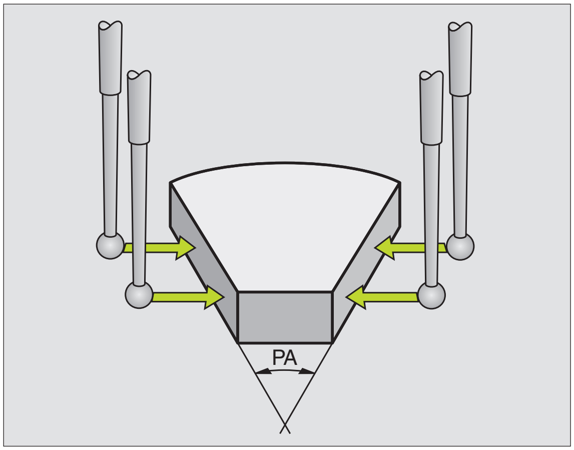
- the angle between the angle reference axis and a workpiece side, or
- the angle between two sides.

The measured angle is displayed as a value of maximum 90°.

To find the angle between the angle reference axis and a side of the workpiece

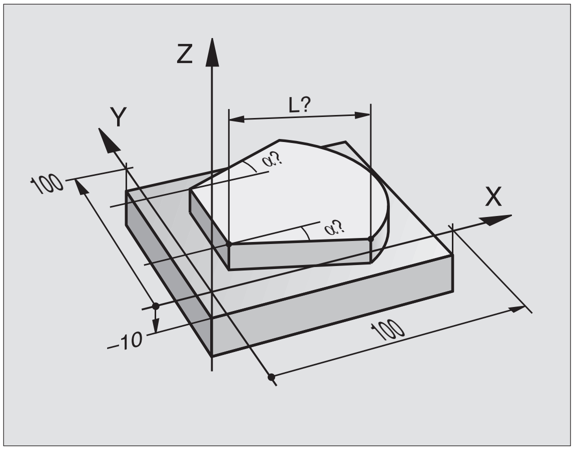


- Select the probe function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation with the side of the workpiece (see "Compensating Workpiece Misalignment" on page 420).
- Press the PROBING ROT soft key to display the angle between the angle reference axis and the side of the workpiece as the rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation.
- This is done by setting the rotation angle to the value that you wrote down previously.



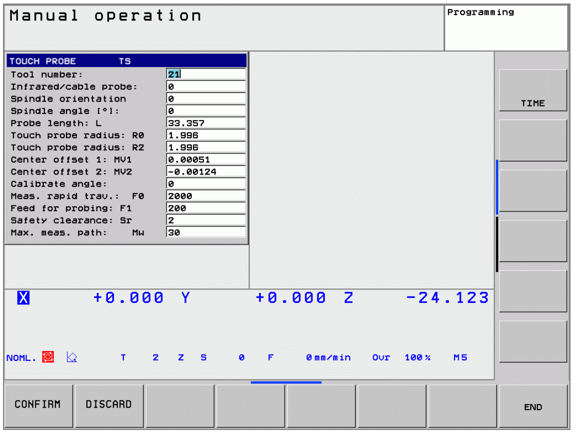
To measure the angle between two workpiece sides:

- Select the probe function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation with the side of the workpiece (see "Compensating Workpiece Misalignment" on page 420).
- Probe the second side as for a basic rotation, but do not set the rotation angle to zero!
- Press the PROBING ROT soft key to display the angle PA between the two sides as the rotation angle.
- Cancel the basic rotation, or restore the previous basic rotation by setting the rotation angle to the value that you wrote down previously.



13.6 Touch Probe Data Management

Introduction



To make it possible to cover the widest possible range of applications, the touch probe management enable offers several settings to enable you to determine the behavior common to all touch probe cycles: The TNC always uses the values from the touch probe management, even it values are also entered in the tool table. Press the PARAMETER soft key to open the touch probe management window.

Tool number

Number by which the touch probe is registered in the tool table

Infrared/cable probe

0: Touch probe with cable
1: Infrared touch probe (machine-dependent function **180° rotation** allowed)

Oriented spindle stop

0: No oriented spindle stop
1: Spindle orientation (the touch probe is oriented so that it is always probed by the same point on the touch probe stylus tip)

Spindle angle

Enter the angle of the touch probe at its home position. This value is used for spindle orientation during calibration of the ball-tip radius and for internal calculations (machine-dependent function).

Probe length

Length (ascertained by calibration) by which the TNC offsets the touch probe dimension

Touch probe radius R

Radius (ascertained by calibration) by which the TNC offsets the touch probe dimension

Touch probe radius R2

Ball-radius (ascertained by calibration) by which the TNC offsets the touch probe dimension

Center offset 1

Offset of the touch probe axis to the spindle axis for the reference axis

Center offset 2

Offset of the touch probe axis to the spindle axis for the minor axis

Calibrate angle

Here the TNC enters the orientation angle with which the touch probe was calibrated

Rapid traverse for measuring

Feed rate at which the touch probe pre-positions, or is positioned between the measuring points

Feed for probing

Feed rate at which the TNC is to probe the workpiece.

Set-up clearance

In the setup clearance you define how far from the defined (or calculated) touch point the probe is to be pre-positioned. The smaller the value you enter, the more exactly must you define the touch point position.

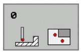
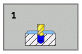
Maximum measuring range

If the stylus is not deflected within the defined path, the TNC outputs an error message.

13.7 Automatic Workpiece Measurement

Overview

The TNC offers three cycles for measuring workpieces and setting the datum automatically. To define the cycles, press the TOUCH PROBE key in the Programming and Editing or Positioning with MDI operating mode.

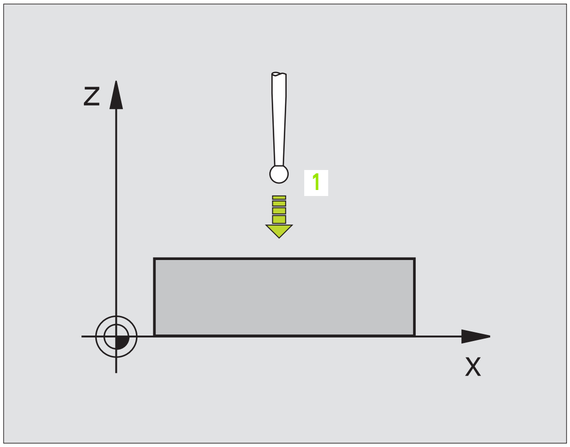
Cycle	Soft key
0 REFERENCE PLANE Measuring a coordinate in a selectable axis	
1 POLAR DATUM PLANE Measuring a point in a probing direction	
3 MEASURE Measuring the position and diameter of a hole	

Reference system for measurement results

The TNC transfers all the measurement results to the result parameters and the protocol file in the active coordinate system, or as the case may be, the displaced coordinate system.

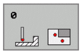
DATUM PLANE touch probe cycle 0

1 The touch probe moves at rapid traverse to the starting position **1** programmed in the cycle.
2 Then the touch probe approaches the workpiece at the assigned feed rate. The probing direction is to be defined in the cycle.
3 After the TNC has saved the position, the probe retracts to the starting point and saves the measured coordinate in a Q parameter. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in the parameters Q115 to Q119. For the values in these parameters the TNC does not account for the stylus length and radius.



Before programming, note the following:

Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.



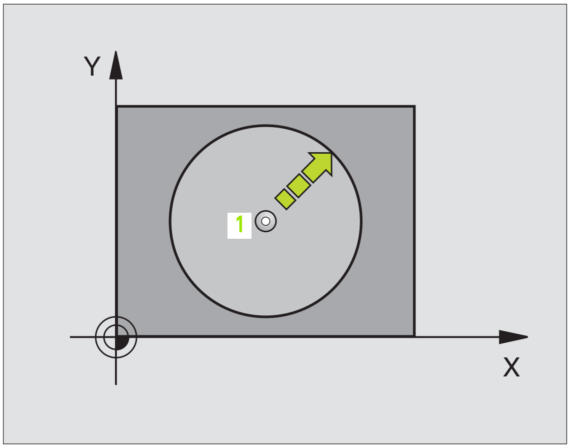
- **`Parameter number for result:`** Enter the number of the Q parameter to which you want to assign the coordinate.
- **`Probing axis/Probing direction:`** Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Confirm your entry with the ENT key.
- **`Position value:`** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe.
- To conclude the input, press the ENT key.

Example: NC blocks

```
67 TCH PROBE 0.0 REF. PLANE Q5 X-
68 TCH PROBE 0.1 X+5 Y+0 Z-5
```

DATUM PLANE touch probe cycle 1

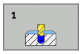
Touch probe cycle 1 measures any position on the workpiece in any direction.



1 The touch probe moves at rapid traverse to the starting position **1** programmed in the cycle.
2 Then the touch probe approaches the workpiece at the assigned feed rate. During probing the TNC moves simultaneously in 2 axes (depending on the probing angle). The scanning direction is defined by the polar angle entered in the cycle.
3 After the TNC has saved the position, the probe returns to the starting point. The TNC also stores the coordinates of the touch probe position at the time of the triggering signal in parameters Q115 to Q119.

Before programming, note the following:

Pre-position the touch probe in order to avoid a collision when the programmed pre-positioning point is approached.



- **`Probing axis:`** Enter the probing axis with the axis selection keys or ASCII keyboard. Confirm your entry with the ENT key.
- **`Probing angle:`** Angle, measured from the probing axis, at which the touch probe is to move.
- **`Position value:`** Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe.
- To conclude the input, press the ENT key.

Example: NC blocks

```
67 TCH PROBE 1.0 POLAR DATUM PLANE
68 TCH PROBE 1.1 X ANGLE: +30
69 TCH PROBE 1.2 X+5 Y+0 Z-5
```

MEASURING (touch probe cycle 3)

Touch probe cycle 3 measures any position on the workpiece in a selectable direction. Unlike other measuring cycles, Cycle 3 enables you to enter the measuring path and feed rate directly. Also, the touch probe retracts by a definable value after determining the measured value.

1 The touch probe moves from the current position at the entered feed rate in the defined probing direction. The probing direction must be defined in the cycle as a polar angle.
2 After the TNC has saved the position, the touch probe stops. The TNC saves the X, Y, Z coordinates of the probe-tip center in three successive Q parameters. You define the number of the first parameter in the cycle.
3 Finally, the TNC moves the touch probe back by that value against the probing direction that you defined in the parameter **MB.**

Before programming, note the following:

Enter the maximum retraction path **MB** to be just large enough to prevent a collision.

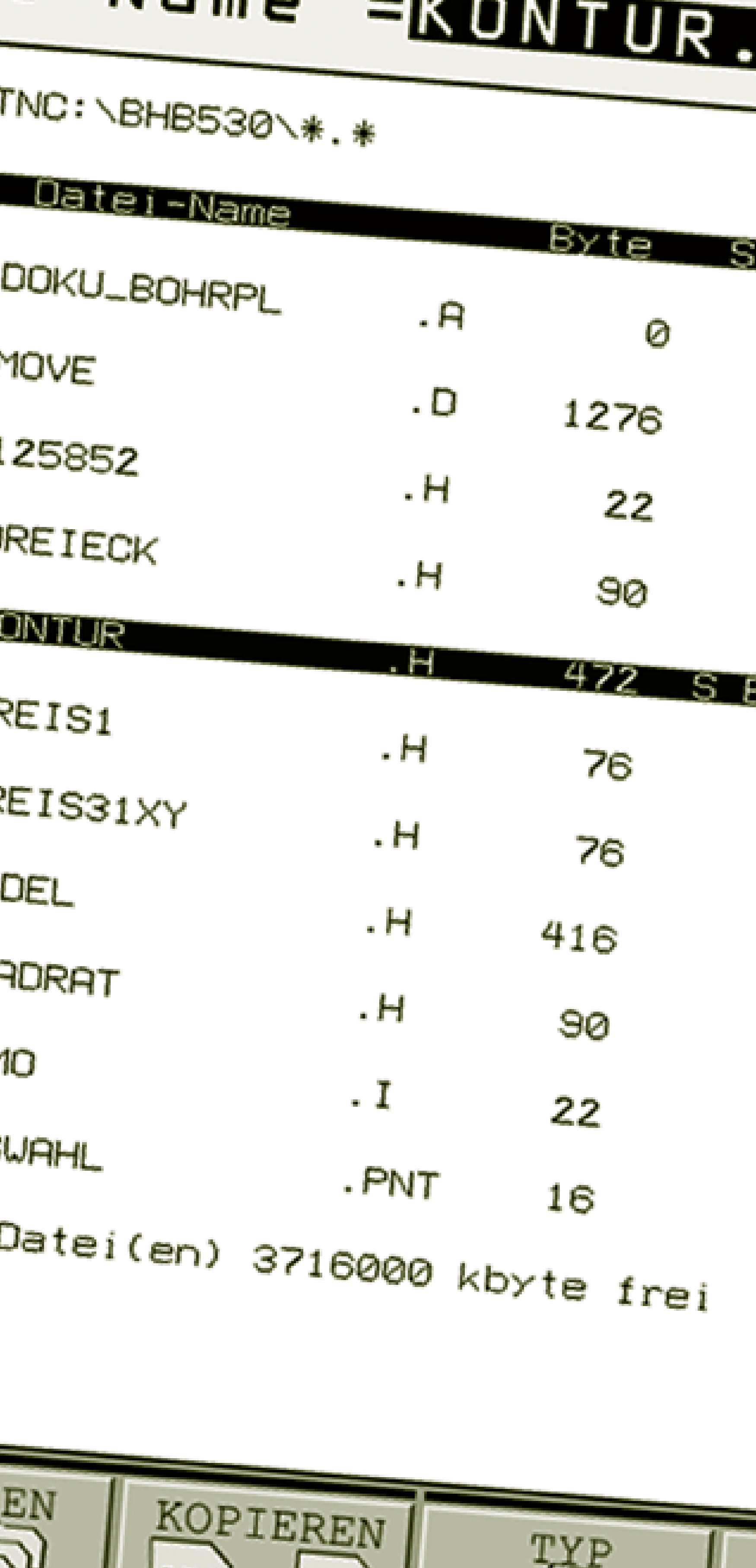
If the TNC could not determine a valid touch point, the fourth result parameter will have the value –1.

3

- **Parameter number for result:** Enter the number of the Q parameter to which you want the TNC to assign the first coordinate (X).
- **Probe axis:** Enter the reference axis of the working plane (X for tool axis Z, Z for tool axis Y, and Y for tool axis X), and confirm with ENT.
- **Probing angle:** Angle, measured from the probing axis, at which the touch probe is to move. Confirm with ENT.
- **Maximum measuring path:** Enter the maximum distance from the starting point by which the touch probe may move. Confirm with ENT.
- **Feed rate:** Enter the measuring feed rate in mm/min.
- **Maximum retraction path:** Traverse path in the direction opposite the probing direction, after the stylus was deflected.
- **REFERENCE SYSTEM (0=ACT/1=REF):** Specify whether the result of measurement is to be saved in the actual coordinate system (ACT), or with respect to the machine coordinate system (REF).
- To conclude the input, press the ENT key.

Example: NC blocks

```
5 TCH PROBE 3.0 MEASURING
6 TCH PROBE 3.1 Q1
7 TCH PROBE 3.2 X ANGLE: +15
8 TCH PROBE
3.3 DIST +10 F100 MB:1 REFERENCE SYSTEM:0
```



14

Tables and Overviews

14.1 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 interface for HEIDENHAIN devices

The interface complies with the requirements of EN 50 178 for "low voltage electrical separation."

When using the 25-pin adapter block:

TNC		Connecting cable 365 725-xx			Adapter block 310 085-01		Connecting cable 274 545-xx		
Male	Assignment	Female	Color	Female	Male	Female	Male	Color	Female
1	Do not assign	1		1	1	1	1	WH/BN	1
2	RXD	2	Yellow	3	3	3	3	Yellow	2
3	TXD	3	Green	2	2	2	2	Green	3
4	DTR	4	Brown	20	20	20	20	Brown	8
5	Signal GND	5	Red	7	7	7	7	Red	7
6	DSR	6	Blue	6	6	6	6		6
7	RTS	7	Gray	4	4	4	4	Gray	5
8	CTR	8	Pink	5	5	5	5	Pink	4
9	Do not assign	9					8	Violet	20
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

When using the 9-pin adapter block:

TNC		Connecting cable 355 484-xx			Adapter block 363 987-02		Connecting cable 366 964-xx		
Male	Assignment	Female	Color	Male	Female	Male	Female	Color	Female
1	Do not assign	1	Red	1	1	1	1	Red	1
2	RXD	2	Yellow	2	2	2	2	Yellow	3
3	TXD	3	White	3	3	3	3	White	2
4	DTR	4	Brown	4	4	4	4	Brown	6
5	Signal GND	5	Black	5	5	5	5	Black	5
6	DSR	6	Violet	6	6	6	6	Violet	4
7	RTS	7	Gray	7	7	7	7	Gray	8
8	CTR	8	WH/GN	8	8	8	8	WH/GN	7
9	Do not assign	9	Green	9	9	9	9	Green	9
Hsg.	Ext. shield	Hsg.	Ext. shield	Hsg.	Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Non-HEIDENHAIN devices

The connector pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device.

This often depends on the unit and type of data transfer. The table below shows the connector pin layout on the adapter block.

Adapter block 363 987-02		Connecting cable 366 964-xx		
Female	**Male**	**Female**	**Color**	**Female**
1	1	1	Red	1
2	2	2	Yellow	3
3	3	3	White	2
4	4	4	Brown	6
5	5	5	Black	5
6	6	6	Violet	4
7	7	7	Gray	8
8	8	8	WH/GN	7
9	9	9	Green	9
Hsg.	Hsg.	Hsg.	Ext. shield	Hsg.

Ethernet interface RJ45 socket

Maximum cable length:

- Unshielded: 100 m
- Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant	
5	Vacant	
6	REC–	Receive Data
7	Vacant	
8	Vacant	

14.2 Technical Information

Explanation of symbols

■ Standard

● Axis option

User functions	
Description	■ Basic version: 3 axes plus spindle ● 1st additional axis for 4 axes and open-loop or closed-loop spindle ● 2nd additional axis for 5 axes and open-loop spindle
Programming	HEIDENHAIN conversational
Position entry	■ Nominal positions for line segments and arcs in Cartesian or polar coordinates ■ Absolute or incremental dimensions ■ Display and entry in mm or inches
Tool Compensations	■ Tool radius in the working plane and tool length ■ Calculating the radius-compensated contour up to 99 blocks in advance (M120)
Tool tables	Multiple tool tables with any number of tools
Constant cutting speed	■ With respect to the path of the tool center ■ With respect to the cutting edge
Background programming	Create one program with graphical support while another program is running.
Contour elements	■ Straight line ■ Chamfer ■ Circular path ■ Circle center ■ Circle radius ■ Tangentially connecting circle ■ Corner rounding
Contour approach and departure	■ Via straight line: tangential or perpendicular ■ Via circular arc
FK free contour programming	■ FK free contour programming in HEIDENHAIN conversational format with graphic support for workpiece drawings not dimensioned for NC
Program jumps	■ Subprograms ■ Program section repeat ■ Program as subprogram

User functions	
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping ■ Cycles for milling internal and external threads ■ Milling and finishing rectangular and circular pockets ■ Cycles for multipass milling of flat and twisted surfaces ■ Cycles for milling linear and circular slots ■ Linear and circular point patterns ■ Contour-parallel contour pocket ■ OEM cycles (special cycles developed by the machine tool builder) can also be integrated
Coordinate transformation	■ Datum shift, rotation, mirroring, scaling (axis-specific)
Q parameters Programming with variables	■ Mathematic functions =, +, –, *, /, sin α , cos α $\sqrt{a^2 + b^2}$ $\sqrt{a}$ ■ Logical comparisons (=, =/, <, >) ■ Calculating with parentheses ■ tan α , arcus sin, arcus cos, arcus tan, a^n, e^n, In, log, absolute value of a number, the constant π , negation, truncation of digits before or after the decimal point ■ Functions for calculating circles
Programming support	■ Pocket calculator ■ Complete list of all current error messages ■ Context-sensitive help function for error messages ■ Graphical support during programming of cycles ■ Comment blocks in the NC program
Actual position capture	■ Actual positions can be transferred directly into the NC program
Test Run graphics Display modes	■ Graphic simulation before a program run, even while another program is being run ■ Plan view / projection in 3 planes / 3-D view ■ Magnification of details
Interactive programming graphics	■ In the Programming and Editing mode, the contour of the NC blocks is drawn on screen while they are being entered (2-D pencil-trace graphics), even while another program is running
Program Run graphics Display modes	■ Graphic simulation of real-time machining in plan view / projection in 3 planes / 3-D view
Machining time	■ Calculating the machining time in the Test Run mode of operation ■ Display of the current machining time in the Program Run modes
Returning to the contour	■ Mid-program startup in any block in the program, returning the tool to the calculated nominal position to continue machining ■ Program interruption, contour departure and reapproach
Datum tables	■ Multiple datum tables, for storing workpiece-related datums

User functions	
Touch Probe Cycles	■ Calibrating a touch probe ■ Compensation of workpiece misalignment, manual or automatic ■ Datum setting, manual or automatic ■ Automatic workpiece measurement ■ Cycles for automatic tool measurement

Specifications	
Components	■ Main computer with TNC keyboard and integrated 15.1-inch TFT color flat-panel display with soft keys
Program memory	■ 10 MB (on compact flash memory card CFR)
Input resolution and display step	■ To 0.1 µm for linear axes ■ To 0.0001° for angular axes
Input range	■ Maximum 999 999 999 mm or 999 999 999°
Interpolation	■ Line in 4 axes ■ Arc in 2 axes ■ Helix: combination of circular and linear motion
Block processing time 3-D straight line without radius compensation	□ 6 ms (3-D straight line without radius compensation)
Axis control	■ Position loop resolution: Signal period of the position encoder/1024 ■ Cycle time of position controller: 3 ms ■ Cycle time of speed controller: 600 µs
Traverse range	■ Maximum 100 m
Spindle speed	■ Maximum 100 000 rpm (analog speed command signal)
Error compensation	■ Linear and nonlinear axis error, backlash, reversal spikes during circular movements, thermal expansion ■ Stick-slip friction
Data interfaces	■ One each RS-232-C /V.24 max. 115 kilobaud ■ Expanded data interface with LSV-2 protocol for remote operation of the TNC through the data interface with the HEIDENHAIN software TNCremo ■ Ethernet interface 100 Base T approx. 2 to 5 megabaud (depending on file type and network load) ■ 2 x USB 1.1
Ambient temperature	■ Operation: 0 °C to +45 °C (32 °F to 113 °F) ■ Storage: –30 °C to +70 °C (–22 °F to 158 °F)

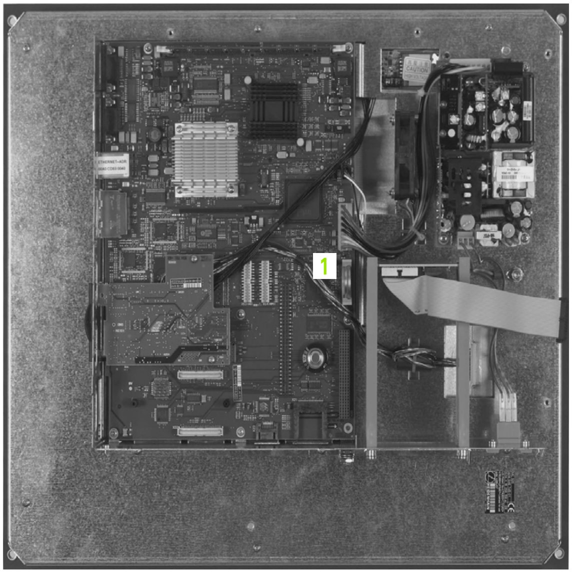
Accessories	
Electronic handwheels	One **HR 410** portable handwheel or One **HR 130** panel-mounted handwheel or Up to three **HR 150** panel-mounted handwheels via HRA 110 handwheel adapter
Touch Probes	**TS 220:** 3-D touch trigger probe with cable connection, or **TS 440:** 3-D touch trigger probe with infrared transmission **TS 640:** 3-D touch trigger probe with infrared transmission

Input format and unit of TNC functions	
Positions, coordinates, circle radii, chamfer lengths	–99 999.9999 to +99 999.9999 (5.4: places before decimal point, places after decimal point) [mm]
Tool numbers	0 to 32 767.9 (5.1)
Tool names	16 characters, enclosed by quotation marks with TOOL CALL. Permitted special characters: #, $, %, &, -
Delta values for tool compensation	–99.9999 to +99.9999 (2.4) [mm]
Spindle speeds	0 to 99 999.999 (5.3) [rpm]
Feed rates	0 to 99 999.999 (5.3) [mm/min] or [mm/tooth] or [mm/rev]
Dwell time in Cycle 9	0 to 3600.000 (4.3) [s]
Thread pitch in various cycles	–99.9999 to +99.9999 (2.4) [mm]
Angle of spindle orientation	0 to 360.0000 (3.4) [°]
Angle for polar coordinates, rotation, tilting the working plane	–360.0000 to 360.0000 (3.4) [°]
Polar coordinate angle for helical interpolation (CP)	–5400.0000 to 5400.0000 (4.4) [°]
Datum numbers in Cycle 7	0 to 2999 (4.0)
Scaling factor in Cycles 11 and 26	0.000001 to 99.999999 (2.6)
Miscellaneous Functions M	0 to 999 (3.0)
Q parameter numbers	0 to 1999 (4.0)
Q parameter values	–99 999.9999 to +99 999.9999 (5.4)
Labels (LBL) for program jumps	0 to 999 (3.0)
Labels (LBL) for program jumps	Any text string in quotes (“ ”)
Number of program section repeats REP	1 to 65 534 (5.0)
Error number with Q parameter function FN14	0 to 1099 (4.0)
Spline parameter K	–9.99999999 to +9.99999999 (1.8)
Exponent for spline parameter	–255 to 255 (3.0)
Surface-normal vectors N and T with 3-D compensation	–9.99999999 to +9.99999999 (1.8)

14.3 Exchanging the Buffer Battery

A buffer battery supplies the TNC with current to prevent the data in RAM memory from being lost when the TNC is switched off.

If the TNC displays the error message **`Exchange buffer battery,`** then you must replace the batteries:



Backup your data before changing the buffer battery

To exchange the buffer battery, first switch off the TNC.

The buffer battery must be exchanged only by trained service personnel.

Battery type:1 Lithium battery, type CR 2450N (Renata)
ID Nr. 315 878-01

1. The buffer battery is on the main board of the MC 320 (see **1**, figure at upper right)
2. Remove the five screws of the MC 320 housing cover
3. Remove the cover
4. The buffer battery is on the right edge of the PCB. Exchange the battery. The socket accepts a new battery only in the correct orientation.
5. Exchange the battery. The socket accepts a new battery only in the correct orientation.

SYMBOLE

U

V

W

Table of Cycles

Cycle number	Cycle designation	DEF-active	CALL-active	Page
1	Pecking		■	
2	Tapping		■	
3	Slot milling		■	
4	Pocket milling		■	Page 226
5	Circular pocket		■	Page 232
7	Datum shift	■		Page 282
8	Mirror image	■		Page 286
9	Dwell time	■		Page 293
10	Rotation	■		Page 288
11	Scaling factor	■		Page 289
12	Program call	■		Page 294
13	Oriented spindle stop	■		Page 295
14	Contour definition	■		Page 256
17	Tapping (controlled spindle)		■	
18	Thread cutting		■	
20	Contour data SL II	■		Page 260
21	Pilot drilling SL II		■	Page 261
22	Rough out SL II		■	Page 262
23	Floor finishing SL II		■	Page 263
24	Side finishing SL II		■	Page 264
26	Axis-specific scaling	■		Page 290
200	Drilling		■	Page 182
201	Reaming		■	Page 184
202	Boring		■	Page 186
203	Universal drilling		■	Page 188
204	Back boring		■	Page 190
205	Universal pecking		■	Page 192

Cycle number	Cycle designation	DEF-active	CALL-active	Page
206	Tapping with a floating tap holder, new		■	Page 197
207	Rigid tapping, new		■	Page 199
208	Bore milling		■	Page 195
209	Tapping with chip breaking		■	Page 201
210	Slot with reciprocating plunge		■	Page 238
211	Circular slot		■	Page 241
212	Rectangular pocket finishing		■	Page 228
213	Rectangular stud finishing		■	Page 230
214	Circular pocket finishing		■	Page 234
215	Circular stud finishing		■	Page 236
220	Point pattern on circle	■		Page 248
221	Hole patterns on lines	■		Page 250
230	Multipass milling		■	Page 268
231	Ruled surface		■	Page 270
232	Face milling		■	Page 273
262	Thread milling		■	Page 205
263	Thread milling/countersinking		■	Page 207
264	Thread drilling/milling		■	Page 211
265	Helical thread drilling/milling		■	Page 215
267	Outside thread milling		■	Page 219

Table of Miscellaneous Functions

M	Effect	Effective at block: Start	End	Page
M00	Stop program/Spindle STOP/Coolant OFF		■	Page 162
M01	Optional program STOP		■	Page 394
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	Page 162
M03	Spindle ON clockwise	■		Page 162
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	
M06	Tool change/Stop program run (machine-dependent function)/Spindle STOP		■	Page 162
M08	Coolant ON	■		Page 162
M09	Coolant OFF		■	
M13	Spindle ON clockwise/Coolant ON	■		Page 162
M14	Spindle ON counterclockwise/Coolant ON	■		
M30	Same function as M02		■	Page 162
M89	Vacant miscellaneous function **or**	■		Page 179
	Cycle call, modally effective (machine-dependent function)		■	
M91	Within the positioning block: Coordinates are referenced to machine datum	■		Page 163
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		Page 163
M94	Reduce display of rotary axis to value under 360°	■		Page 174
M97	Machine small contour steps		■	Page 165
M98	Machine open contours completely		■	Page 167
M99	Blockwise cycle call		■	Page 179
M101	Automatic tool change with replacement tool if maximum tool life has expired	■		Page 108
M102	Cancel M101		■	
M107	Suppress error message for replacement tools	■		Page 107
M108	Cancel M107		■	
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)	■		Page 167
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)	■		
M111	Cancel M109/M110		■	
M116	Feed rate for rotary tables in mm/minn	■		Page 172
M117	Cancel M116		■	
M118	Superimpose handwheel positioning during program run	■		Page 169

M	Effect	Effective at block	Start	End	Page
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)		■		Page 168
M126	Shortest-path traverse of rotary axes		■		Page 173
M127	Cancel M126			■	
M140	Retraction from the contour in the tool-axis direction		■		Page 169
M141	Suppress touch probe monitoring		■		Page 170
M143	Delete basic rotation		■		Page 171
M148	Automatically retract tool from the contour at an NC stop		■		Page 171
M149	Cancel M148			■	

The machine tool builder may add some M functions that are not described in this User's Manual. Also, the machine tool builder can change the meaning and effect of the M functions described here. Refer to your machine manual.

Comparison: Functions of the TNC 320, TNC 310 and iTNC 530

Comparison: User functions

Function	TNC 320	TNC 310	iTNC 530
Program entry with HEIDENHAIN conversational programming	X	X	X
Program entry according to ISO	–	–	X
Program entry with smarT.NC	–	–	X
Position data: Nominal positions for lines and arcs in Cartesian coordinates	X	X	X
Position data: Incremental or absolute dimensions	X	X	X
Position data: Display and input in mm or inches	X	X	X
Position data: Display of handwheel traverse when machining with handwheel superimposition	–	–	X
Tool compensation: In the working plane and tool length	X	X	X
Tool compensation: Radius-compensated contour look ahead for up to 99 blocks	X	–	X
Tool compensation: Three-dimensional tool-radius compensation	–	–	X
Tool table: Save tool data centrally	X	X	X
Tool table: Multiple tool tables with any number of tools	X	–	X
Cutting-data tables: Calculation of spindle speed and feed rate	–	–	X
Constant contouring speed: Relative to the path of the tool center or relative to the tool's cutting edge	X	–	X
Parallel operation Creating programs while another program is being run	X	X	X
Tilt working plane	–	–	X
Rotary-table machining: Programming of cylindrical contours as if in two axes	–	–	X
Rotary-table machining: Feed rate in mm/min	X	–	X
Approaching and departing the contour: Via a straight line or arc	X	X	X
FK (free contour programming): Programming of workpieces not correctly dimensioned for NC programming	X	–	X
Program jumps: Subprograms and program section repeats	X	X	X
Program jumps: Calling any program as subprogram	X	X	X
Test graphics: Plan view, projection in 3 planes, 3-D view	X	X	X
Programming graphics: 2-D line graphics	X	X	X

Function	TNC 320	TNC 310	iTNC 530
Machining graphics: Plan view, projection in 3 planes, 3-D view	X	–	X
Datum tables, for storing workpiece-related datums	X	X	X
Preset table, for saving reference points (presets)	–	–	X
Returning to the contour with mid-program startup	X	X	X
Returning to the contour after program interruption	X	X	X
Autostart	X	–	X
Actual position capture: Actual positions can be transferred to the NC program	X	X	X
Expanded file management: Create multiple directories and subdirectories	X	–	X
Context-sensitive help: Help function for error messages	X	–	X
Pocket calculator	X	–	X
Entry of text and special characters: On the TNC 320 via on-screen keyboard, on the iTNC 530 via regular keyboard	X	–	X
Comment blocks in NC program	X	–	X
Structure blocks in NC program	–	–	X

Comparison: Cycles

Cycle	TNC 320	TNC 310	iTNC 530
1, Pecking	X	X	X
2, Tapping	X	X	X
3, Slot milling	X	X	X
4, Pocket milling	X	X	X
5, Circular pocket	X	X	X
6, Rough out (SL I)	–	X	X
7, Datum shift	X	X	X
8, Mirror image	X	X	X
9, Dwell time	X	X	X
10, Rotation	X	X	X
11, Scaling	X	X	X
12, Program call	X	X	X
13, Oriented spindle stop	X	X	X
14, Contour definition	X	X	X
15, Pilot drilling (SL I)	–	X	X
16, Contour milling (SL I)	–	X	X
17, Tapping (controlled spindle)	X	X	X
18, Thread cutting	X	–	X
19, Working plane	–	–	X
20, Contour data	X	–	X
21, Pilot drilling	X	–	X
22, Rough-out	X	–	X
23, Floor finishing	X	–	X
24, Side finishing	X	–	X
25, Contour train	–	–	X
26, Axis-specific scaling factor	X	–	X
27, Contour train	–	–	X
28, Cylinder surface	–	–	X

Cycle	TNC 320	TNC 310	iTNC 530
29, Cylinder surface ridge	–	–	X
30, 3-D data	–	–	X
32, Tolerance	–	–	X
39, Cylinder surface external contour	–	–	X
200, Drilling	X	X	X
201, Reaming	X	X	X
202, Boring	X	X	X
203, Universal drilling	X	X	X
204, Back boring	X	X	X
205, Universal pecking	X	–	X
206, Tapping with floating tap holder	X	–	X
207, Rigid tapping	X	–	X
208, Bore milling	X	–	X
209, Tapping with chip breaking	X	–	X
210, Slot with reciprocating plunge	X	X	X
211, Circular slot	X	X	X
212, Rectangular pocket finishing	X	X	X
213, Rectangular stud finishing	X	X	X
214, Circular pocket finishing	X	X	X
215, Circular stud finishing	X	X	X
220, Point pattern on circle	X	X	X
221, Point pattern on lines	X	X	X
230, Multipass milling	X	X	X
231, Ruled surface	X	X	X
232, Face milling	X	–	X
240, Centering	–	–	X
247, Datum setting	–	–	X
251, Rectangular pocket (complete)	–	–	X
252, Circular pocket (complete)	–	–	X

Cycle	TNC 320	TNC 310	iTNC 530
253, Slot (complete)	–	–	X
254, Circular slot (complete)	–	–	X
262, Thread milling	X	–	X
263, Thread milling/counter sinking	X	–	X
264, Thread drilling/milling	X	–	X
265, Helical thread drilling/milling	X	–	X
267, Outside thread milling	X	–	X

Comparison: Miscellaneous functions

M	Effect	TNC 320	TNC 310	iTNC 530
M00	Stop program/Spindle STOP/Coolant OFF	X	X	X
M01	Optional program STOP	X	X	X
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1	X	X	X
M03 M04 M05	Spindle ON clockwise Spindle ON counterclockwise Spindle STOP	X	X	X
M06	Tool change/Stop program run (machine-dependent function)/Spindle STOP	X	X	X
M08 M09	Coolant ON Coolant OFF	X	X	X
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON	X	X	X
M30	Same function as M02	X	X	X
M89	Vacant miscellaneous function **or** Cycle call, modally effective (machine-dependent function)	X	X	X
M90	Constant contouring speed at corners	–	X	X
M91	Within the positioning block: Coordinates are referenced to machine datum	X	X	X
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	X	X	X
M94	Reduce display of rotary axis to value under 360°	X	X	X
M97	Machine small contour steps	X	X	X
M98	Machine open contours completely	X	X	X
M99	Blockwise cycle call	X	X	X
M101 M102	Automatic tool change with replacement tool if maximum tool life has expired Cancel M101	X	–	X
M107 M108	Suppress error message for replacement tools Cancel M107	X	–	X
M109 M110 M111	Constant contouring speed at tool cutting edge (increase and decrease feed rate) Constant contouring speed at tool cutting edge (feed rate decrease only) Cancel M109/M110	X	–	X

M	Effect	TNC 320	TNC 310	iTNC 530
M112 M113	Enter contour transition between two contour elements Cancel M112	–	–	X
M114 M115	Automatic compensation of machine geometry when working with tilted axes Cancel M114	–	–	X
M116 M117	Feed rate for rotary tables in mm/minn Cancel M116	X	–	–
M118	Superimpose handwheel positioning during program run	X	–	X
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	X	–	X
M124	Contour filter	–	–	X
M126 M127	Shortest-path traverse of rotary axes Cancel M126	X	–	X
M128 M129	Maintain the position of the tool tip when positioning the tilted axes (TCPM) Cancel M126	–	–	X
M134 M135	Exact stop at nontangential contour transitions when positioning with rotary axes Cancel M134	–	–	X
M138	Select tilting axes	–	–	X
M140	Retraction from the contour in the tool-axis direction	X	–	X
M141	Suppress touch probe monitoring	X	–	X
M142	Delete modal program information	–	–	X
M143	Delete basic rotation	X	–	X
M144 M145	Compensating the machine's kinematic configuration for ACTUAL/NOMINAL positions at end of block Cancel M144	–	–	X
M148 M149	Automatically retract tool from the contour at an NC stop Cancel M148	X	–	X
M150	Suppress limit switch message	–	–	X
M200- M204	Laser cutting functions	–	–	X

Comparison: Touch probe cycles in the Manual and Electronic Handwheel modes

Cycle	TNC 320	TNC 310	iTNC 530
Calibrate the effective length	X	X	X
Calibrate the effective radius	X	X	X
Measure a basic rotation using a line	X	X	X
Set the datum in any axis	X	X	X
Set a corner as datum	X	X	X
Set the center line as datum	–	–	X
Set the circle center as datum	X	X	X
Measure a basic rotation using two holes/cylindrical studs	–	–	X
Set the datum using four holes/cylindrical studs	–	–	X
Set circle center using three holes/cylindrical studs	–	–	X

Comparison: Touch probe cycles for automatic workpiece inspection

Cycle	TNC 320	TNC 310	iTNC 530
0, Reference plane	X	–	X
1, Polar datum	X	–	X
2, Calibrate TS	–	–	X
3, Measuring	X	–	X
9, Calibrate TS length	X	–	X
30, Calibrate TT	–	–	X
31, Measure tool length	–	–	X
32, Measure tool radius	–	–	X
33, Measure tool length and radius	–	–	X
400, Basic rotation	–	–	X
401, Basic rotation from two holes	–	–	X
402, Basic rotation from two studs	–	–	X
403, Compensate a basic rotation via a rotary axis	–	–	X
404, Set basic rotation	–	–	X
405, Compensating workpiece misalignment by rotating the C axis	–	–	X
410, Datum from inside of rectangle	–	–	X
411, Datum from outside of rectangle	–	–	X
412, Datum from inside of circle	–	–	X
413, Datum from outside of circle	–	–	X
414, Datum in outside corner	–	–	X
415, Datum at inside corner	–	–	X
416, Datum circle center	–	–	X
417, Datum in touch probe axis	–	–	X
418, Datum at center of 4 holes	–	–	X
419, Datum in one axis	–	–	X
420, Measure angle	–	–	X
421, Measure hole	–	–	X
422, Measure circle outside	–	–	X

Cycle	TNC 320	TNC 310	iTNC 530
423, Measure rectangle inside	–	–	X
424, Measure rectangle outside	–	–	X
425, Measure inside width	–	–	X
426, Measure ridge outside	–	–	X
427, Boring	–	–	X
430, Measure bolt hole circle	–	–	X
431, Measure plane	–	–	X

HEIDENHAIN

DR. JOHANNES HEIDENHAIN GmbH
Dr.-Johannes-Heidenhain-Straße 5
83301 Traunreut, Germany
☏ +49 (8669) 31-0
FAX +49 (8669) 5061
E-Mail: info@heidenhain.de

Technical support FAX +49 (8669) 31-1000
E-Mail: service@heidenhain.de
Measuring systems ☏ +49 (8669) 31-3104
E-Mail: service.ms-support@heidenhain.de
TNC support ☏ +49 (8669) 31-3101
E-Mail: service.nc-support@heidenhain.de
NC programming ☏ +49 (8669) 31-3103
E-Mail: service.nc-pgm@heidenhain.de
PLC programming ☏ +49 (8669) 31-3102
E-Mail: service.plc@heidenhain.de
Lathe controls ☏ +49 (711) 952803-0
E-Mail: service.hsf@heidenhain.de

www.heidenhain.de

3-D Touch Probe Systems from HEIDENHAIN

help you to reduce non-cutting time:

For example in

- workpiece alignment
- datum setting
- workpiece measurement
- digitizing 3-D surfaces

with the workpiece touch probes
TS 220 with cable
TS 640 with infrared transmission

- tool measurement
- wear monitoring
- tool breakage monitoring

with the tool touch probe
TT 130

Ve 00
550 671-20 · SW01 · 5 · 3/2006 · F&W · Printed in Germany · Subject to change without notice