HEIDENHAIN

TNC 310

NC-Software
286 040-xx

User's Manual
Conversational
Programming

7/2000

Controls on the TNC

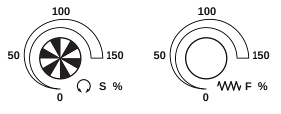
Controls on the visual display unit

- Split screen layout
- Soft keys
- Shift soft-key rows

Machine control buttons

- Axis direction buttons
- Rapid traverse button
- Spindle rotation direction
- Coolant
- Tool release
- Spindle ON/OFF
- NC start/NC stop

Override control knobs for feed rate/spindle speed



Machine operating modes

- MANUAL OPERATION
- POSITIONING WITH MDI
- PROGRAM RUN, SINGLE BLOCK
- PROGRAMMING AND EDITING

Coordinate numbers, editing

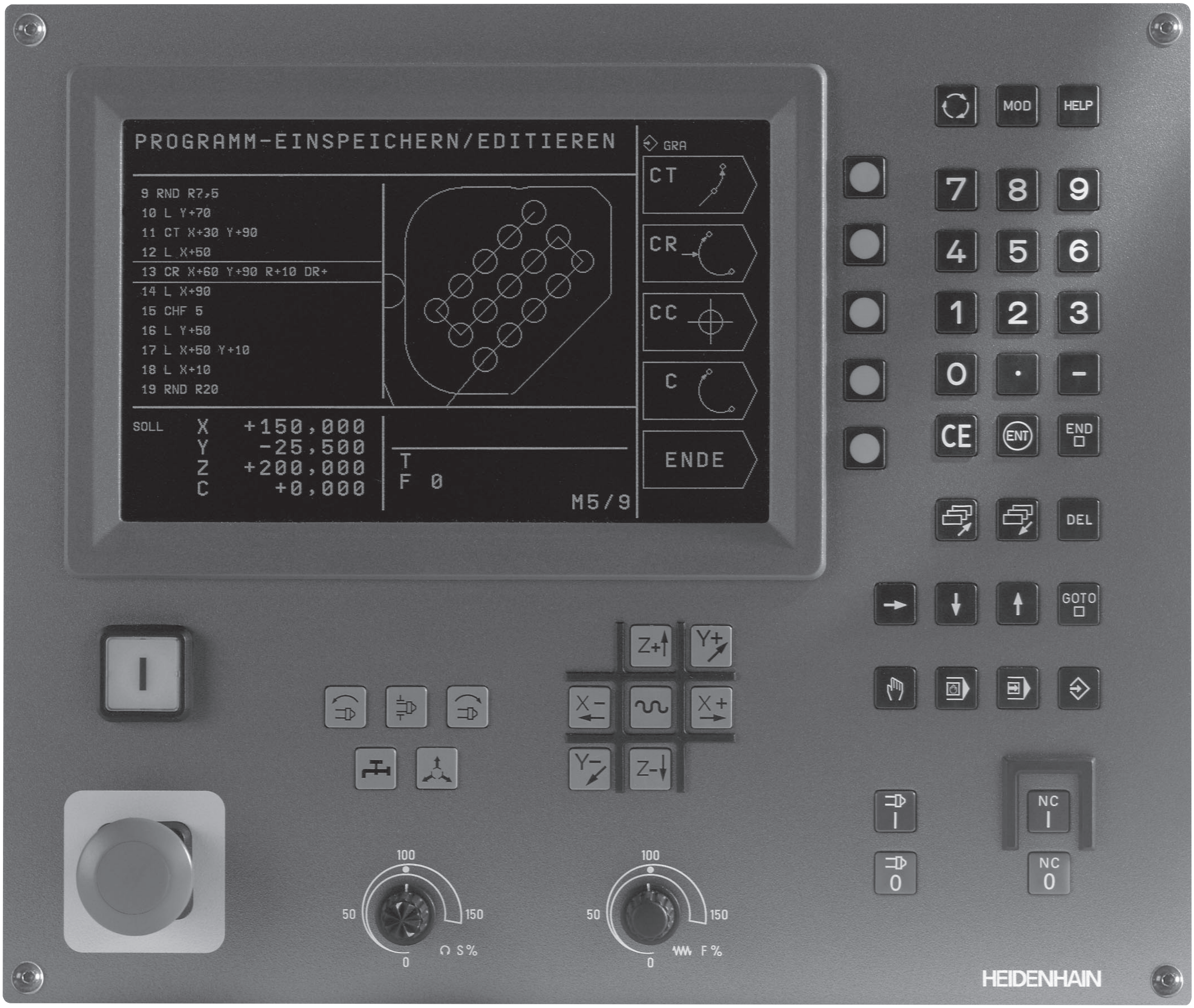
- 0 ... 9 Numbers
- Decimal point
- Change arithmetic sign
- ENT Confirm entry and resume dialog
- END Close block
- CE Clear numerical entry or TNC error message
- DEL Abort dialog, delete program section

Programming aids

- MOD MOD functions
- HELP HELP functions

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

- Move highlight
- Move highlight, skip dialog question
- GOTO Select blocks and cycles directly


PROGRAMM-EINSPEICHERN/EDITIEREN
GRA
9 RND R7,5
10 L Y+70
11 CT X+30 Y+90
12 L X+50
13 CR X+60 Y+90 R+10 DR+
14 L X+90
15 CHF 5
16 L Y+50
17 L X+50 Y+10
18 L X+10
19 RND R20
CT
CR
CC
C
ENDE
SOLL X +150,000
Y -25,500
Z +200,000
C +0,000
T
F 0
M5/9
MOD
HELP
7 8 9
4 5 6
1 2 3
0 · –
CE
ENT
END
DEL
GOTO
Z+
Y+
X–
X+
Y–
Z–
NC
I
NC
0
0
100
50
150
0
S %
100
50
150
0
F %
HEIDENHAIN

TNC Models, Software and Features

This manual describes functions and features provided by the TNCs with the following NC software number.

TNC Model	NC Software No.
TNC 310	286 040 xx

The machine tool builder adapts the useable features of the TNC to his machine by setting machine parameters. Therefore, 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
- Digitizing option
- Tool measurement with the TT 120
- Rigid tapping

Please contact your machine tool builder to become familiar with the individual implementation of the control on 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

Contents

Contents

1

Introduction

1.1 The TNC 310

HEIDENHAIN TNC controls are shop-floor programmable contouring controls for milling, drilling and boring machines.

You can program conventional milling, drilling and boring operations right at the machine with the easily understandable interactive conversational guidance. The TNC 310 can control up to 4 axes. Instead of the fourth axis, you can also change the angular position of the spindle under program control.

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

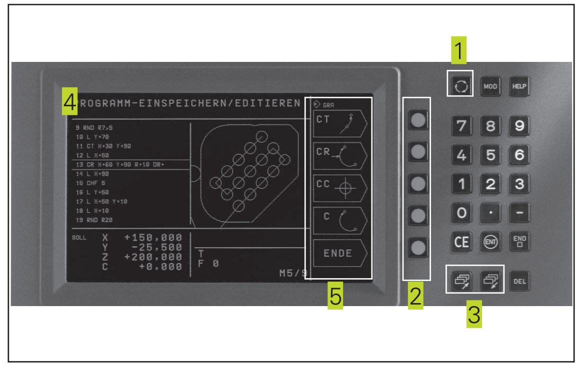
Programming: HEIDENHAIN conversational format
HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. Workpiece machining can be graphically simulated during test run.

You can also enter one program while the TNC is running another.

Compatibility
The TNC can execute all part programs that were written on HEIDENHAIN controls TNC 150 B and later.

1.2 Visual Display Unit and Keyboard

Visual display unit



The figure at right shows the keys and controls on the VDU:

1 Setting the screen layout

2 Soft key selector keys

3 Switching the soft-key rows

4 Header
When the TNC is on, the selected operating mode is shown in the screen header. Dialog prompts and TNC messages also appear here (unless the TNC is showing only graphics).

5 Soft keys
In the right margin the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately beside them 2. Directly beneath the soft-key row are rectangular boxes indicating the number of soft-key rows. These rows can be called with the 3 shift key. The box representing the active soft-key row is filled in.

Screen layout
You select the screen layout yourself: In the PROGRAMMING AND EDITING 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 help graphics for cycle definition in the right window instead, 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.
PGM + GRAPHICS	Select the desired screen layout.

Keyboard

The figure at right shows the keys of the keyboard grouped according to their functions:

1 MOD function,
HELP function

2 Numerical input

3 Dialog buttons

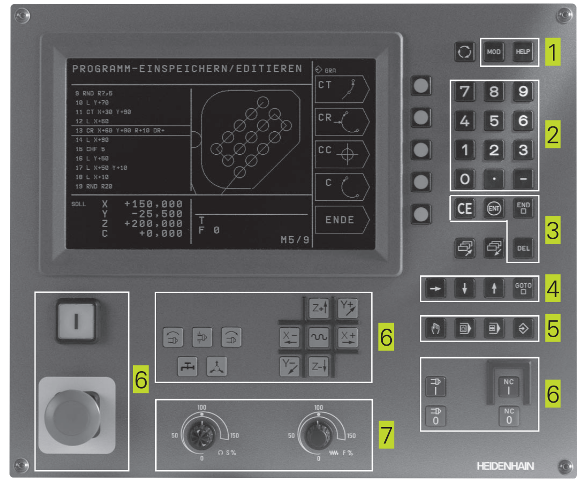
4 Arrow keys and GOTO jump command

5 Modes of Operation

6 Machine control buttons

7 Override control knobs for feed rate/spindle speed

The functions of the individual keys are described in the foldout of the front cover. The exact functioning of the machine control buttons, e.g. NC START, is described in more detail in your Machine Manual.

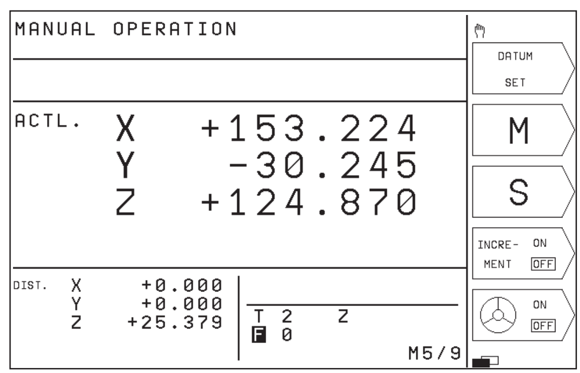


1.3 Modes of Operation

The TNC offers the following modes of operation for the various functions and working steps that you need to machine a workpiece:

MANUAL OPERATION and ELECTRONIC HANDWHEEL Operating Modes

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. Datums can be set by the usual scratching method or by using the TS 220 triggering touch probe. The TNC also supports the manual traverse of the machine axes using a HR electronic handwheel.



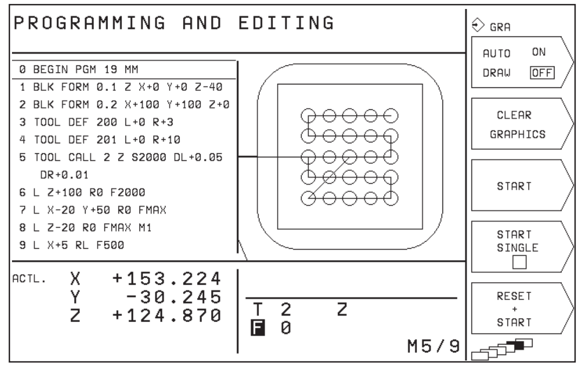
Soft keys for selecting the screen layout
There are no select options available. The TNC always shows the position display.

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
There are no select options available. The TNC always shows the position display.

PROGRAMMING AND EDITING



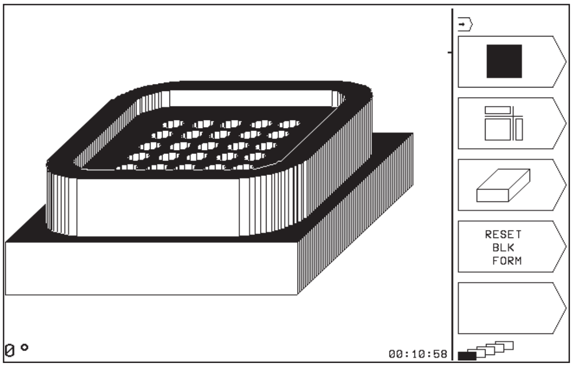
In this mode of operation you can write your part programs. The various cycles help you with programming and add necessary information. If desired, you can have the programming graphics show the individual steps.

Soft keys for selecting the screen layout

Screen windows	Soft key
Program blocks	PGM
Left: program blocks, right: help graphics for cycle programming	PGM + FIGURE
Left: program blocks, right: programming graphics	PGM + GRAPHICS
Interactive Programming Graphics	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. Use a soft key to activate the test run in the PROGRAM RUN operating mode.



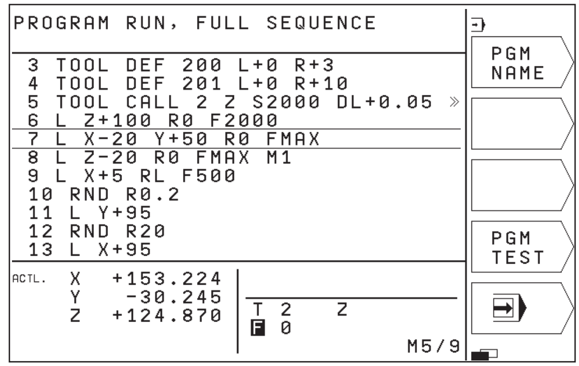
Soft keys for selecting the screen layout

Screen windows	Soft key
Program blocks	PGM
n Test run graphics	GRAPHICS
Left: program blocks, right: general program information	PGM + PGM STATUS
Left: program blocks, right: positions and coordinates	PGM + POS. STATUS
Left: program blocks, right: tool information	PGM + TOOL STATUS
Left: program blocks, right: coordinate transformations	PGM + COORD.TRANS. STATUS

PROGRAM RUN/SINGLE BLOCK and PROGRAM RUN/FULL SEQUENCE

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 NC START button.



Soft keys for selecting the screen layout

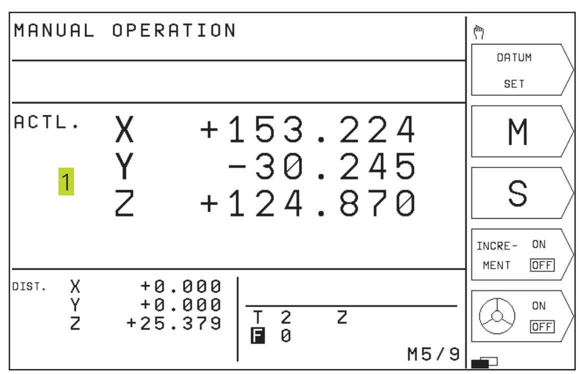
Screen windows	Soft key
Program blocks	PGM
Left: program blocks, right: general program information	PGM + PGM STATUS
Left: program blocks, right: positions and coordinates	PGM + POS. STATUS
Left: program blocks, right: tool information	PGM + TOOL STATUS
Left: program blocks, right: coordinate transformations	PGM + COORD.TRANS. STATUS

1.4 Status Displays

"General" status display

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

In the operating modes MANUAL OPERATION and ELECTRONIC HANDWHEEL and POSITIONING WITH MDI the status display appears in the large window 1.



Information in the status display

Symbol	Meaning
ACTL.	Actual or nominal coordinates of the current position
X Y Z	Machine axes
S F M	Spindle speed S, feed rate F and active M functions
*	Program run started
→|←	Axis locked
ROT	Axes are moving under a basic rotation.

Additional status displays

The additional status displays contain detailed information on the program run. They can be called in all operating modes, except in the MANUAL OPERATION mode.

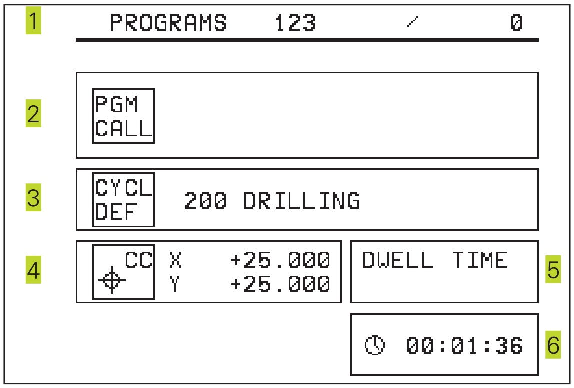
To switch on the additional status display:

	Call the soft-key row for screen layout.
PGM + POS. STATUS	Select the layout option for the additional status display, e.g. positions and coordinates.

You can also choose between the following additional status displays:

PGM + PGM STATUS

General program information



1 Name of main program / Active block number

2 Program called via Cycle 12

3 Active machining cycle

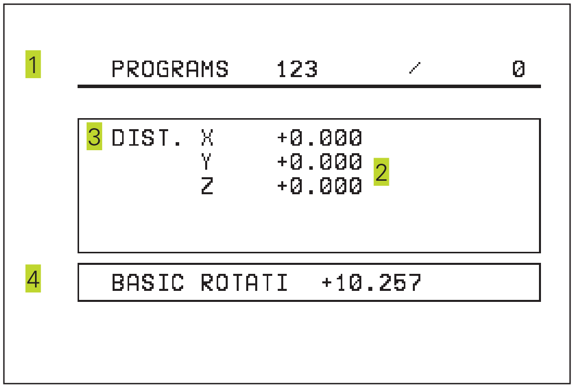
4 Circle center CC (pole)

5 Dwell time counter

6 Operating time

PGM + POS. STATUS

Positions and coordinates

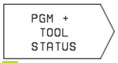


1 Name of main program / Active block number

2 Position display

3 Type of position display, e.g. distance-to-go

4 Angle of a basic rotation



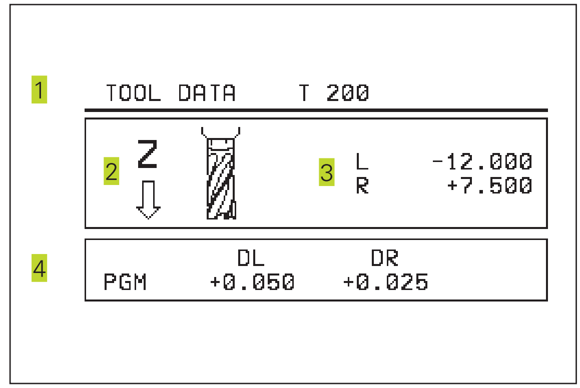
Information on tools

1 T: Tool number

2 Tool axis

3 Tool length and radius

4 Oversizes (delta values) from TOOL CALL block



Coordinate transformations

1 Name of main program / Active block number

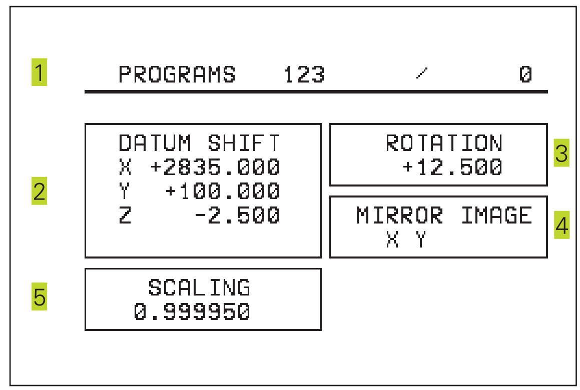
2 Active datum shift (Cycle 7)

3 Active rotation angle (Cycle 10)

4 Mirrored axes (Cycle 8)

5 Active scaling factor (Cycle 11)

See also section 8.7 “Coordinate Transformation Cycles.”



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

3-DTouch Probes

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

- Automatically align workpieces
- Quickly and precisely set datums

TS 220 touch trigger probe

This touch probe is particularly effective for automatic workpiece alignment, datum setting and workpiece measurement. The TS 220 transmits the triggering signals to the TNC via cable.

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-on and traversing the reference points can vary depending on the individual machine tool. Refer to your machine manual for more information.

- Switch on the power supply for control and machine.

The TNC automatically initiates the following dialog

MEMORY TEST

The TNC memory is automatically checked.

POWER INTERRUPTED

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

TRANSLATE PLC PROGRAM

The PLC program of the TNC is automatically translated.

RELAY EXT. DC VOLTAGE MISSING

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

TRAVERSE REFERENCE POINTS

Z+ X+ — 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, or

Z NC I — Cross the reference points with several axes at the same time: Use softkeys to select the axes (axes are then shown highlighted on the screen), and then press the NC START button.

The TNC is now ready for operation in the MANUAL OPERATION mode.

2.2 Moving the Machine Axes

Traversing the machine axes with the axis direction keys is a machine-dependent function. Refer to your machine tool manual for more information on operating times.

Traverse the axis with the axis direction keys

Select the MANUAL OPERATION mode.

X+ Press the axis direction button and hold it as long as you wish the axis to move.

...or move the axis continuously:

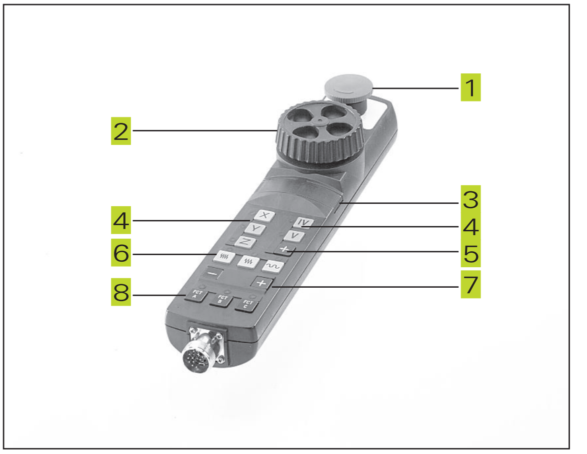
X+ and NC I Press and hold the axis direction button, then press the NC START button: The axis continues to move after you release the keys.

NC 0 Press the NC STOP key to stop the axis.

You can move several axes at a time with these two methods.

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 an permissive button is depressed (machine-dependent function).



The HR 410 handwheel features the following operating elements:

1 EMERGENCY STOP

2 Handwheel Modes

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)

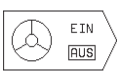
7 Direction in which the TNC moves the selected axis

8 Machine function (set by the machine tool builder)

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

To move an axis:

Select the MANUAL OPERATION mode.



Activate handwheel, set soft key to ON

Press the permissive button.

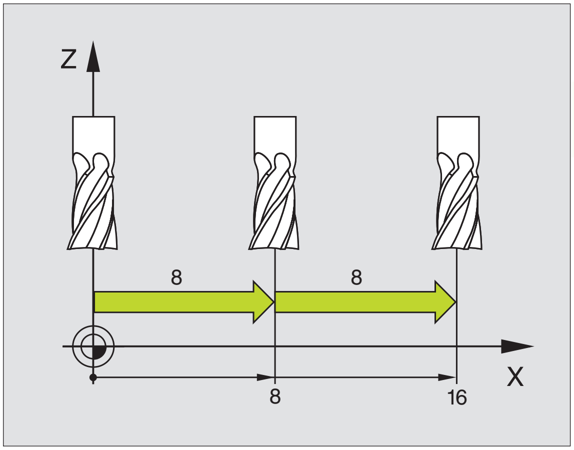
Select the axis on the handwheel

Select the feed rate.

 or  Move the active axis in the positive or negative direction.

Incremental jog positioning

With incremental jog positioning you can move a machine axis by a preset distance each time you press the corresponding axis direction button.



	Select the MANUAL OPERATION mode.
SCHRITT EIN MASS AUS	Select incremental jog positioning, set the softkey to ON

JOG INCREMENT?

8 ENT	Enter the jog increment in millimeters (here, 8 mm), or
1	Select the jog increment via soft key (select 2nd or 3rd soft-key row)
Y+	Press the axis direction button to position as often as desired

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

In the operating mode MANUAL OPERATION, you can enter the spindle speed S and the miscellaneous functions M with soft keys. The miscellaneous functions are described in Chapter 7 "Programming: Miscellaneous Functions." The feed rate is defined in a machine parameter and can be changed only with the override knobs (see next page).

Entering values

Example: Entering the spindle speed S

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

SPINDLE SPEED S=

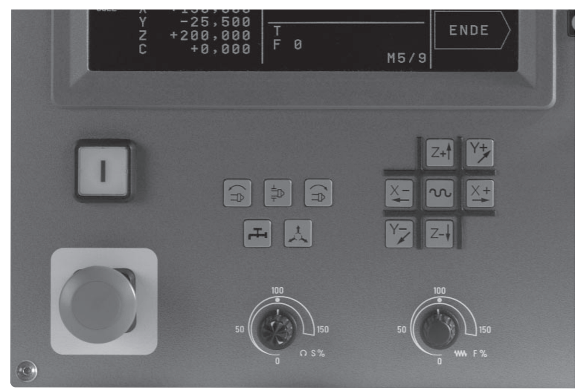
1000	Enter the desired spindle speed,
NC I	and confirm with the NC START button

The spindle speed S with the entered rpm is started with a miscellaneous function.

Proceed in the same way to enter the miscellaneous functions M.

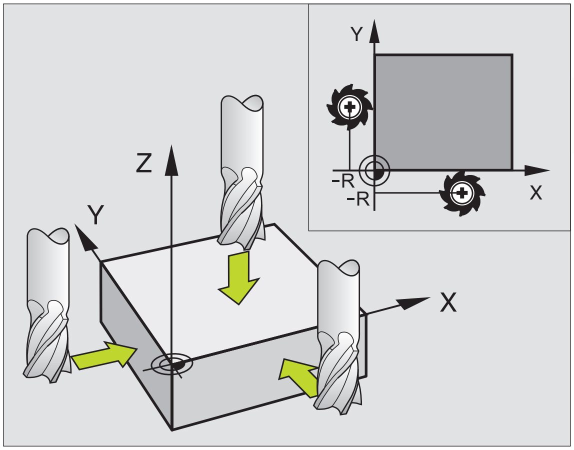
Changing the spindle speed and feed rate

With the override knobs you can vary the spindle speed S and feed rate F from 0% to 150% of the set value.



> The knob for spindle speed override is effective only on machines with a stepless spindle drive.
>
> The machine tool builder determines which miscellaneous functions M are available on your TNC and what effects they have.

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



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.

Setting the datum

Fragile workpiece? If the workpiece surface must not be scratched, you can lay a metal shim of know 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.
	Move the tool slowly until it touches the workpiece surface.
DATUM SET	Select the function for setting the datum
Z	Select the axis.
DATUM SET Z=	
0 ENT	Zero tool: Set the display to a known workpiece position (here, 0) or enter the thickness *d* of the shim.

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.

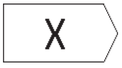
3

Positioning with Manual Data Input (MDI)

3.1 Programming and Executing Simple Positioning Blocks

The operating mode POSITIONING WITH MANUAL DATA INPUT is particularly convenient for simple positioning blocks and for programming a tool call. You can write the individual blocks in HEIDENHAIN conversational programming and execute them immediately. The entered blocks are not stored by the TNC.

Select the POSITIONING WITH MDI mode of operation.



Enter a simple positioning block without radius compensation and feed rate, e.g. X+25 R0 F50

Conclude entry.

Press the NC START button: The TNC executes the entered block.

4

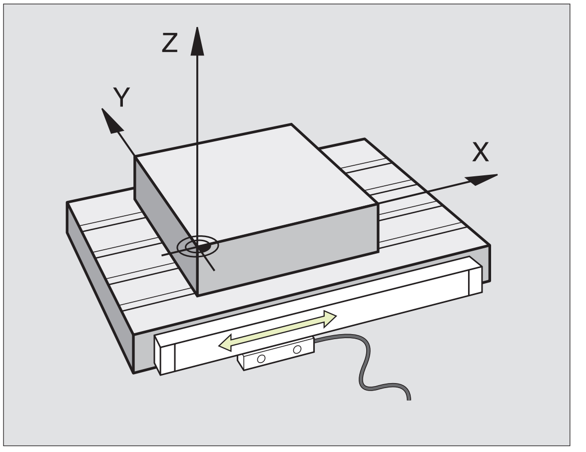
Programming:

Fundamentals of NC, File Management, Programming Aids

4.1 Fundamentals of NC

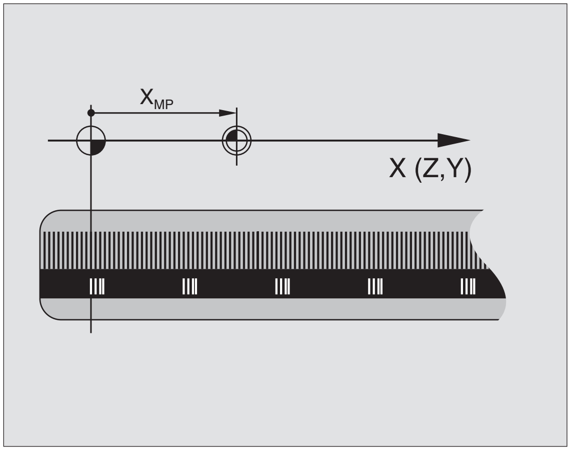
Position encoders and reference marks

The machine axes are equipped with position encoders that register the positions of the machine table or tool. 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 an interruption of power, the calculated position will no longer correspond to the actual position of the machine slide. The TNC can re-establish this relationship with the aid of reference marks when power is returned. 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 identifies that position as the machine-axis reference point and can re-establish the assignment of displayed positions to machine axis positions.

Linear encoders are generally used for linear axes. Rotary tables and tilt axes have angle encoders. If the position encoders feature distance-coded reference marks, you only need to move each axis a maximum of 20 mm (0.8 in.) for linear encoders, and 20° for angle encoders, to re-establish the assignment of the displayed positions to machine axis positions.

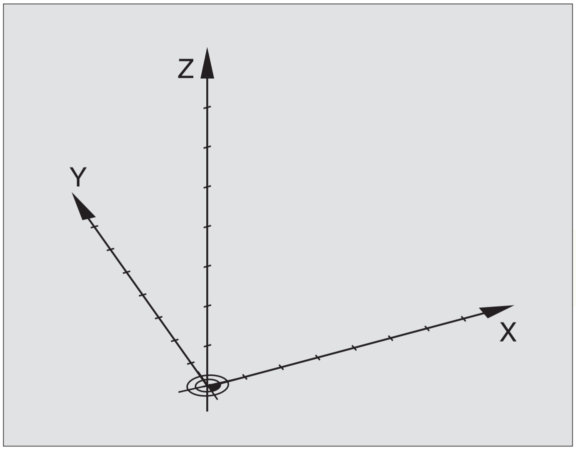


Reference system

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 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.

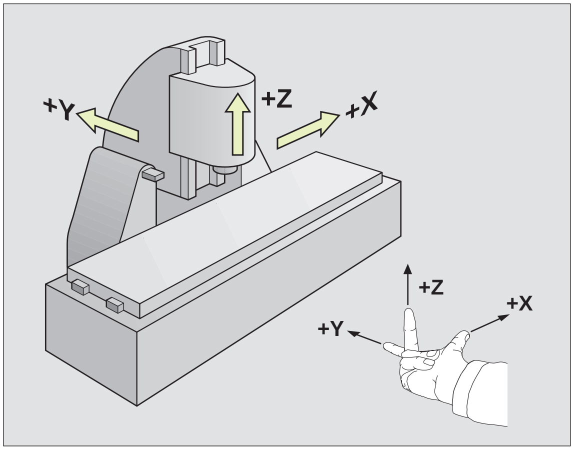
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

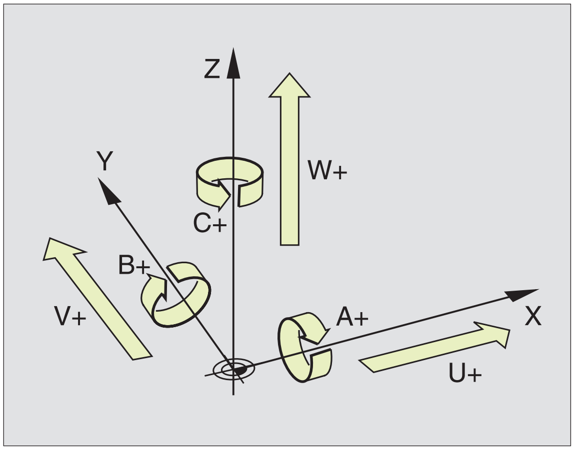


Reference systems on milling machines

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 figure at right illustrates the “right-hand rule” for remembering the three axis directions: the middle finger is pointing in the positive direction of the tool axis from the workpiece toward the tool (the Z axis), the thumb is pointing in the positive X direction, and the index finger in the positive Y direction.

The TNC 310 can control up to 4 axes. The axes U, V and W 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 shows the assignment of secondary axes and rotary axes to the main axes.





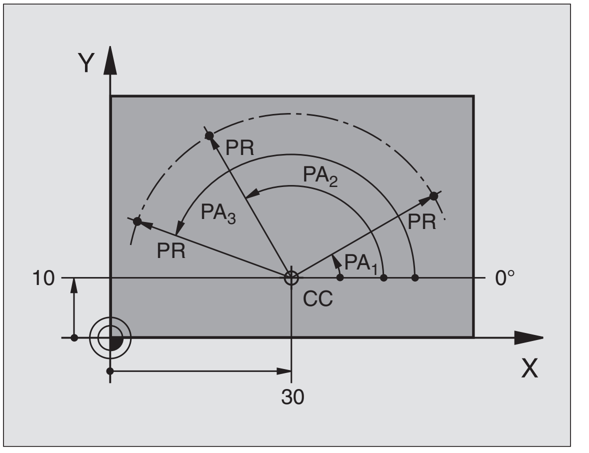
Polar coordinates

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

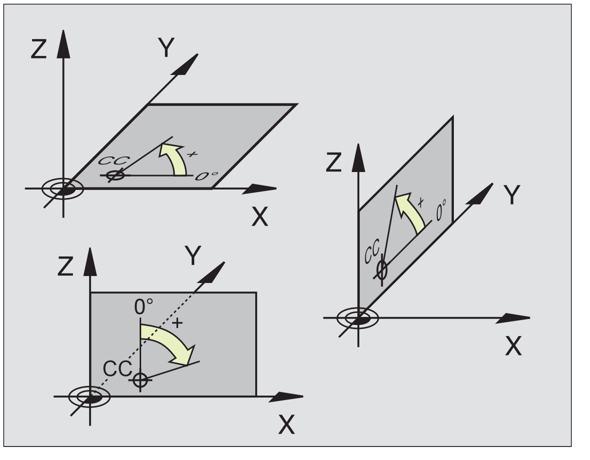
- Polar Radius, the distance from the circle center CC to the position, and the
- Polar Angle, the size of the angle between the reference axis and the line that connects the circle center CC with the position.

See figure to the lower right.



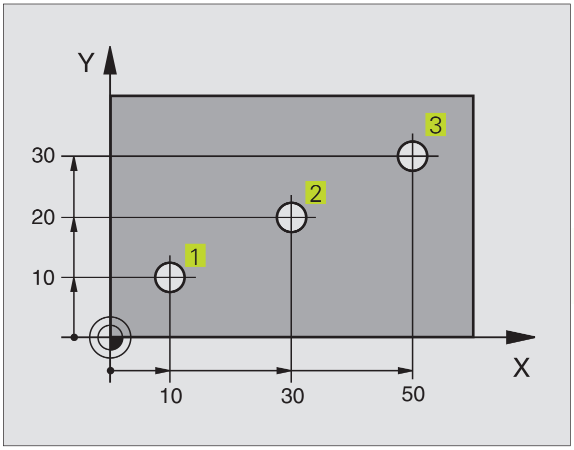
Definition of pole and angle reference axis
The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle PA.

Coordinates of the pole (plane)	Reference axis of the angle
XY	+X
YZ	+Y
ZX	+Z



Absolute and relative 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.

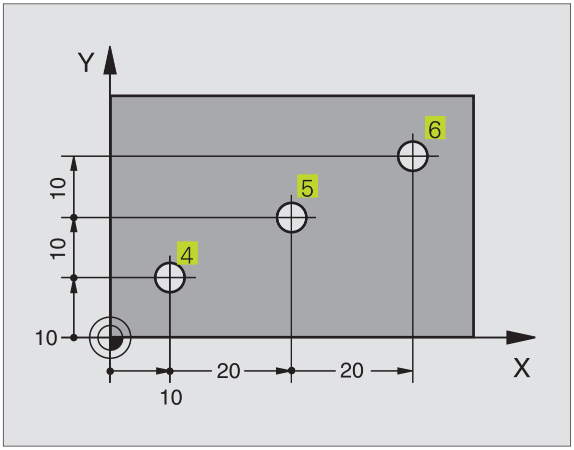


Example 1: Holes dimensioned in absolute coordinates

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

Relative workpiece positions
Relative 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"(soft key) before the axis.



Example 2: Holes dimensioned with relative coordinates

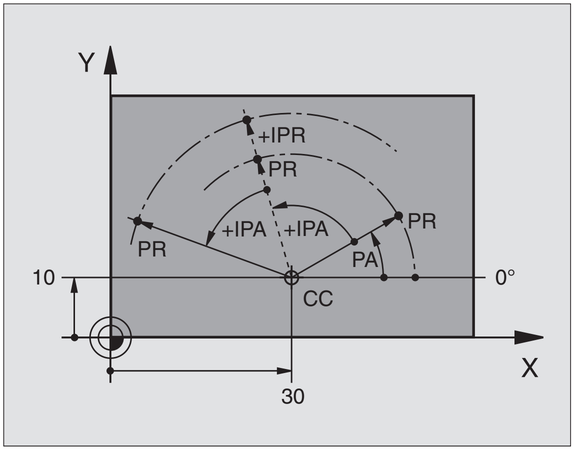
Absolute coordinates of hole 4:

X= 10 mm
Y= 10 mm

Hole 5 referenced to hole 4	Hole 6 referenced to hole 5
IX= 20 mm	IX= 20 mm
IY= 10 mm	IY= 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.



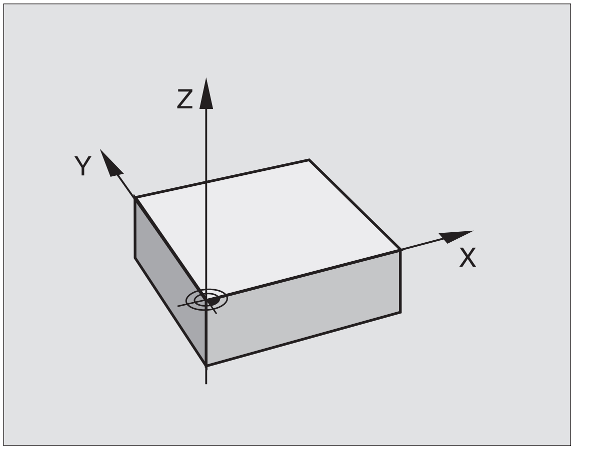
Selecting the datum

A production drawing identifies a certain point on the workpiecee — usually a cornerr — 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. For further information, refer to section 8.6 "Coordinate Transformation Cycles."

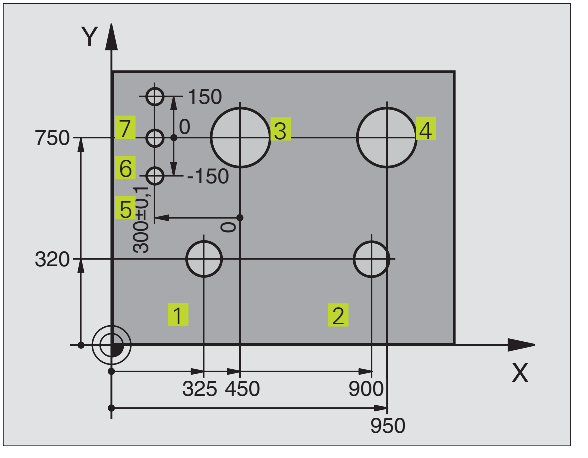
If the production drawing is not dimensioned for NC, set the datum at a position or corner on the workpiece, which is the most suitable for deducing the dimensions of the remaining workpiece positions.

The fastest, easiest and most accurate way of setting the datum is by using a 3-D touch probe from HEIDENHAIN. For further information, refer to section 11.2 "Setting the Datum with a 3-D Touch Probe."



Example

The workpiece drawing at right illustrates the holes 1 to 4, which are dimensioned to an absolute datum with the coordinates X=0 Y=0. The holes 5 to 7 are referenced to a relative datum with the absolute coordinates X=450 Y=750. By using the DATUM SHIFT cycle you can shift the datum temporarily to the position X=450, Y=750 and program the holes 5 to 7 without any further calculations.



4.2 File Management

Files and file management

When you write a part program on the TNC, you must first enter a file name. The TNC then stores the program as a file with the same name. You can also store tables as files.

File names

The name of a file can have up to 8 characters. When you store programs and tables as files, the TNC adds an extension to the file name, separated by a point. This extension identifies the file type (see table at right).

35720	.H
File name	File type

The TNC can manage up to 64 files. Their total size, however, must not exceed 128 MB.

Files in theTNC	Type
Programs in HEIDENHAIN conversational format	.H
Table for Tools	.T

Working with the file manager

This section informs you about the meaning of the individual screen information, and describes how to select files. If you are not yet familiar with the TNC file manager, we recommend that you read this section completely and test the individual functions on your TNC.

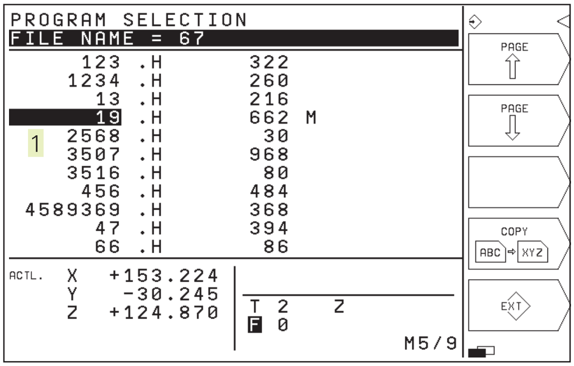
Call the file manager.

PGM MGT

Press the PGM NAME soft key: the TNC displays the file management window

The window shows all of the files 1 that are stored in the TNC. Each file is shown with additional information that is illustrated in the table on the next page.

Display	Meaning
FILE NAME the name:	Name with up to 8 characters and file type Number following File size in bytes
Status M	Properties of the file: Program is selected in a Program Run operating mode
P	Protect a file against editing and erasure (Protected)



Select a file

PGM MGT	Call the file manager.

Use the arrow keys to move the highlight to the desired file:

↑ ↓	Move the highlight up or down

Enter the first or more numbers of the file you wish to select and then press the GOTO key: The highlight moves to the first file that matches these numbers.

ENT	The selected file is opened in the operating mode from which you have the called file manager: Press ENT.

Copy a file

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

COPY ABC → XYZ

▶ Press the COPY soft key to select the copying function.

▶ Enter the name of the destination file and confirm your entry with the ENT key: The TNC copies the file. The original file is retained.

Rename a file

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

RENAME ABC = XYZ

▶ Select the renaming function.

▶ Enter the new file name; the file type cannot be changed.

▶ To execute renaming, press the ENT key.

Delete a file

▶ Move the highlight to the file you want to delete.

DELETE

▶ To select the erasing function, press the DELETE soft key. The TNC inquires whether you really intend to erase the file.

▶ To confirm erasure: Press the YES soft key. Abort with the NO soft key if you do not wish to erase the directory

Protecting a file/Canceling file protection

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

PROTECT/ UNPROTECT

▶ To enable file protection, press the PROTECT/UNPROTECT soft key. The file now has status P.

To cancel file protection, proceed in the same way using the PROTECT/UNPROTECT soft key. You also need to enter the code number 86357.

Read in/read out files

EXT

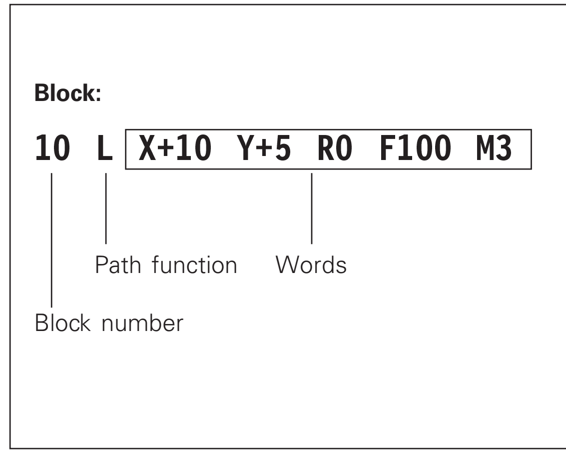
▶ To read in or read out files: Press the ENT soft key. The TNC provides the following functions:

Functions for reading in/reading out files	Soft key
Read in all files	TRANSFER EXT → TNC
Only read in selected files; To accept a file suggested by the TNC, press the YES soft key; Press the NO soft key if you do not want to accept it.	TRANSFER EXT ? → TNC
Read in the selected file: Enter the file name	TRANSFER EXT → TNC
Read out the selected file: Move the highlight to the desired file and confirm with ENT	TRANSFER TNC → EXT
Read out all of the files in the TNC memory	TRANSFER TNC → EXT
Display the file directories of the external unit on your TNC screen	SHOW EXT DIRECTORY

4.3 Creating and Writing Programs

Organization of an NC program in HEIDENHAIN conversational format.

A part program consist 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:

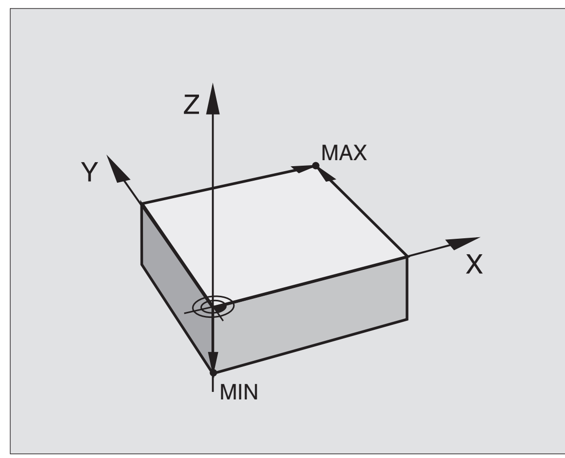
- The blank form
- tool definitions and tool calls,
- Feed rates and spindle speeds as well as
- Path contours, cycles and other functions.

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

Immediately after initiating a new program, you define a cuboid workpiece blank. 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 30 000 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.

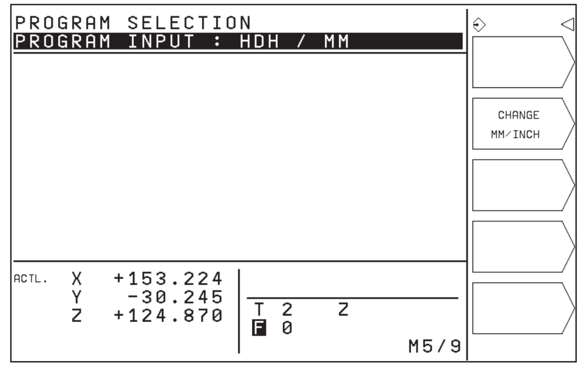


> The TNC can display the graphic only if the short side of the BLK FORM is longer than 1/64 of the long side.

Creating a new part program

You always enter a part program in the PROGRAMMING AND EDITING mode of operation.

Program initiation in an example:



Select the PROGRAMMING AND EDITING mode of operation.

PGM MGT — Call up the file manager: Press the PGM NAME soft key

FILE NAME=

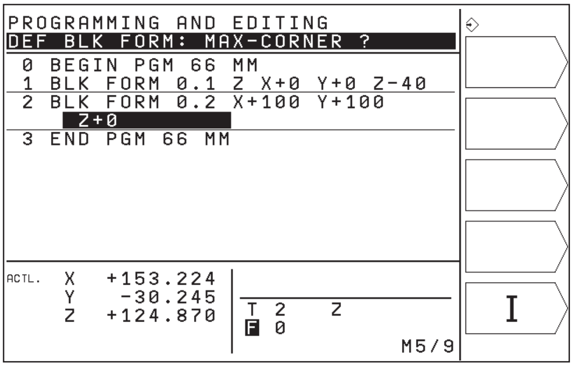
3056 ENT — Enter the new program number and confirm your entry with the ENT key.

Program input : HDH / MM

ENT — Select the default setting for unit of measurement (mm): Press the ENT key, or

MM INCH — Switch to using inches: Press the CHANGE MM/INCH soft key

Define the blank



BLK FORM — Open the dialog for blank definition: Press the BLK FORM soft key

WORKING SPINDLE AXIS X/Y/Z ?

Z — Enter the spindle axis.

DEF BLK FORM: MIN-CORNER?

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

0 ENT

-40 ENT

DEF BLK FORM: MAX-CORNER?

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

100 ENT

0 ENT

The program blocks window shows the following BLK FORM definition

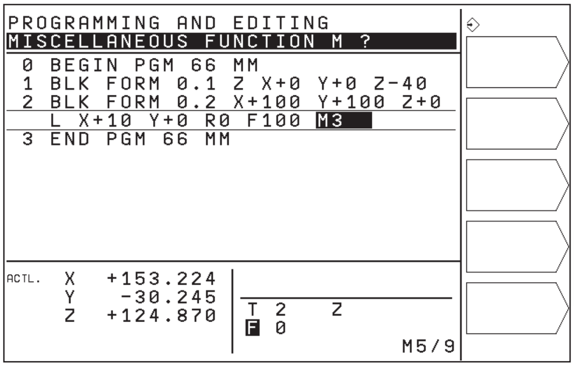
`0 BEGIN PGM 3056 MM`	Program begin, name, unit of measure
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Tool axis, MIN point coordinates
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	MAX point coordinates
`3 END PGM 3056 MM`	Program end, name, unit of measure

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

Programming tool movements in conversational format

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

Example of a dialog



L	Initiate the dialog.

`COORDINATES ?`

X **10**	Enter the target coordinate for the X axis.
Y **5** 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=`

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 will terminate this dialog.

The program blocks window will display the following line:

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

Functions during the dialog	Key
Ignore the dialog question	→
End the dialog immediately	END
Abort the dialog and erase the block	DEL

Editing program lines

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 (see table at right).

Looking for the same words in different blocks

→	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.

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.

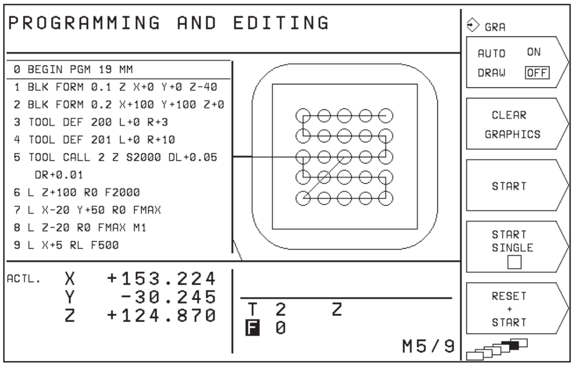
▶ To conclude editing, press the END key.

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

Selecting blocks or words	Keys
Move from one block to the next	↓ ↑
Select individual words in a block	→

Erasing blocks and words	Key
Set the selected word to zero	CE
Erase an incorrect number	CE
Clear a (non-blinking) error message	CE
Delete the selected word	DEL
Delete the selected block (cycle)	DEL
Delete the program sections: First select the last block of the program section to be erased, then erase with the DEL key.	DEL

4.4 Interactive Programming Graphics



While you are writing the part program, you can have the TNC generate a graphic illustration of the programmed contour.

To generate/not generate graphics during programming:

▶ 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 ON DRAW OFF

▶ 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.

AUTO DRAW ON does not simulate program section repeats.

To generate 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 are listed in the table at right.

Functions	Soft key
Generate interactive graphics blockwise	START SINGLE
Generate a complete graphic or complete it after RESET + START	START
Interrupt interactive graphics This soft key only appears while the TNC generates the interactive graphics	STOP

To erase the graphic:

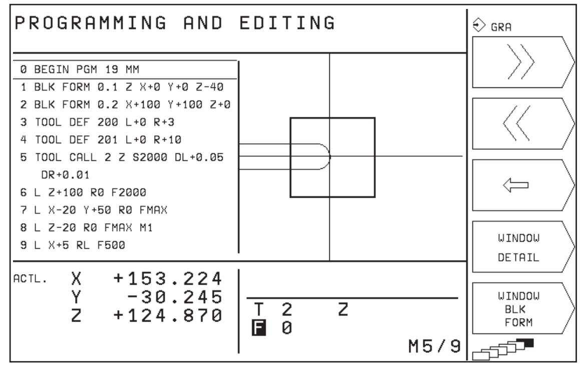
▷

▶ Shift the soft-key row (see figure at right)

CLEAR GRAPHICS

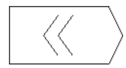
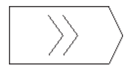
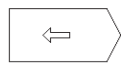
▶ Delete graphic: Press CLEAR GRAPHIC soft key

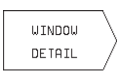
Magnifying or reducing a detail



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 (last row, see figure at right)
The following functions are available:

Function	Soft key
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	
Move the frame overlay to the left: Press and hold the soft key. Move the frame overlay to the right: Press and hold the arrow to the right soft key	



▶ Confirm the selected section with the WINDOW DETAIL soft key

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

4.5 HELP Function

Certain TNC programming functions are explained in more detail in the HELP function. You can select a HELP topic using soft keys.

Select the HELP function

HELP

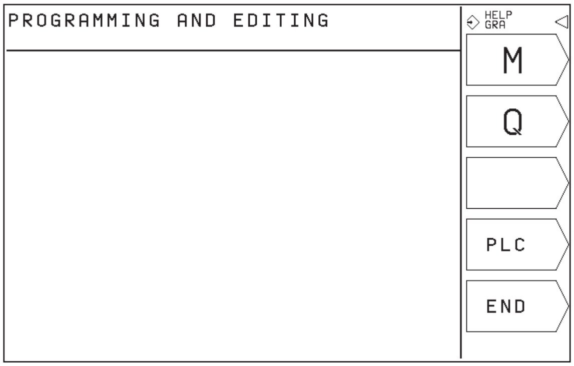
▶ Press the HELP key

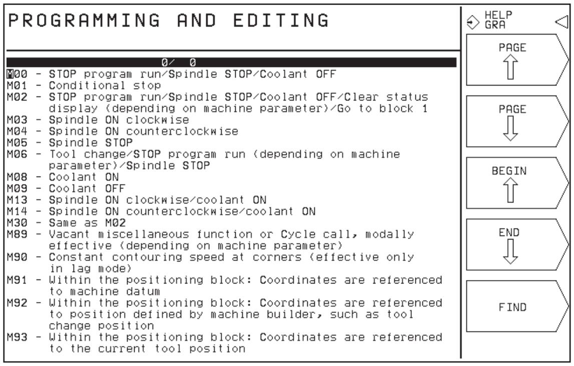
▶ Select a topic: Press one of the available soft keys

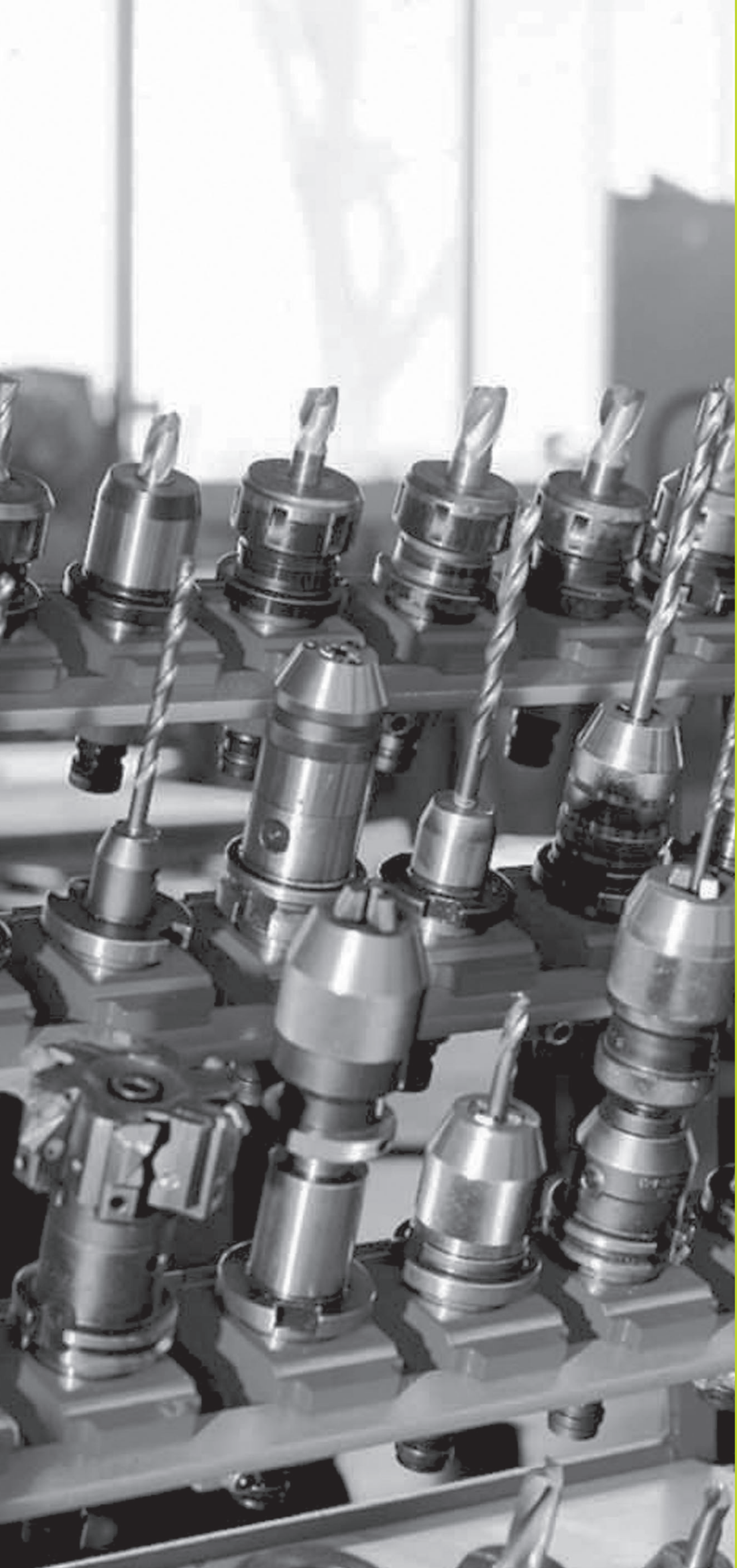
Help topics / Functions	Soft key
M functions	M
Cycle parameters	Q
HELP that is entered by the machine manufacturers (optional)	PLC
Go to previous page	PAGE ⇧
Go to next page	PAGE ⇩
Go to beginning of file	BEGIN ⇧
Go to end of file	END ⇩
Select search functions; Enter a number, Begin search with ENT key	FIND

Leave HELP function

Press the END key or the HELP key.





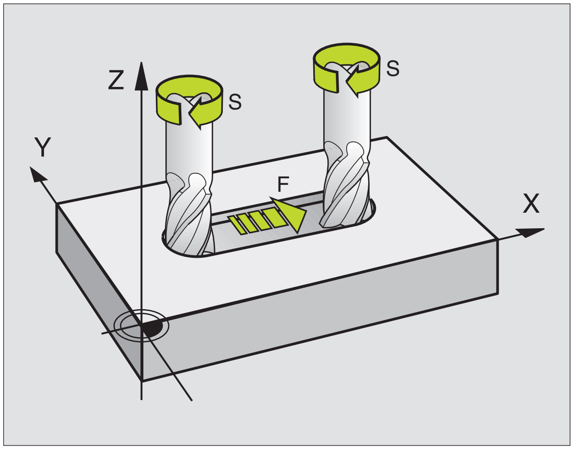


5

Programming:

Tools

5.1 Entering Tool-Related Data



Feed rate F

The feed rate 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 the individual axes and are set in machine parameters.

Input
You can enter the feed rate in every positioning block. For further information refer to section 6.2 "Fundamentals of Path Contours."

Rapid traverse
If you wish to program rapid traverse, enter FMAX. To enter F, press the ENT key or the FMAX soft key as soon as the dialog question "FEED RATE F = ?" appears on the TNC screen.

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

Spindle speed S

The spindle speed S is entered in revolutions per minute (rpm) in a TOOL CALL block.

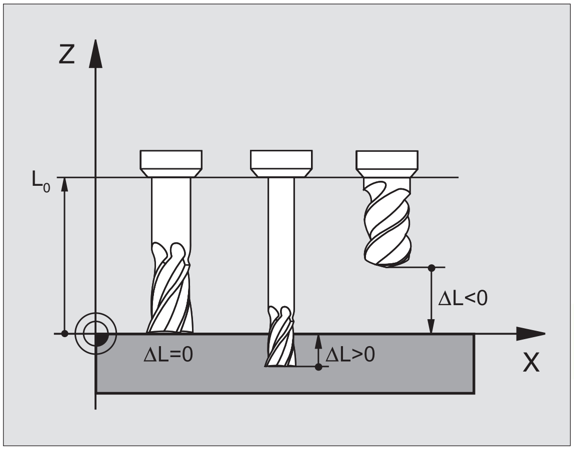
Programmed change
In the part program, you can change the spindle speed in a TOOL CALL block by entering the spindle speed only:

TOOL CALL

- To program a tool call, press the TOOL CALL soft key (3rd soft-key row)
- Ignore the dialog question for „TOOL NUMBER ?" with the right arrow key
- Ignore the dialog question for „WORKING SPINDLE AXIS X/Y/Z ?" with the right arrow key
- Enter the new spindle speed for the dialog question "SPINDLE SPEED S= ?".

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

5.2 Tool Data



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 (and) separately in tool tables. The TNC will consider all of the data entered when executing the part program.

Tool number
Each tool is identified by a number between 0 and 254. If you are working with tool tables, each tool in the table is given a number between 0
and 99.

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 0 should also be defined with L=0 and R=0.

Tool length L
There are two ways to determine the tool length L:

1 The length L is the difference between the length of the tool and that of a zero tool L_0.

For the algebraic sign:

- The tool is longer than the zero tool $L>L_0$
- The tool is shorter than the zero tool: $L<L_0$

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“ key

2 If you determine the length L with a tool presetter, this value can be entered directly in the TOOL DEF block without further calculations.

Tool radius R

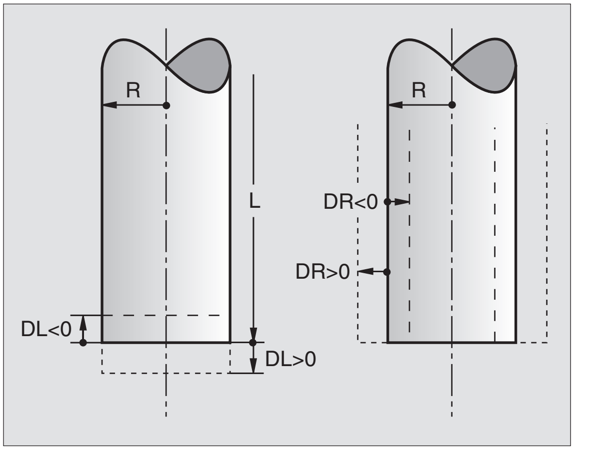
You can enter the tool radius R directly.

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 (DR>0), a negative delta value describes a tool undersize (DR<0). Enter the delta values when you are programming with TOOL CALL.

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



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.

TOOL DEF

- ▶ To select tool definition, press the TOOL DEF key.
- ▶ Enter the TOOL NUMBER: Each tool is uniquely identified by its number. When the tool table is active, enter tool numbers greater than 99 (dependent on MP7260)
- ▶ To enter the TOOL LENGTH, enter the compensation value for the tool length.
- ▶ Enter the TOOL RADIUS.

During the dialog, you can take the values for length and radius directly from the position display with the soft keys „CUR.POS X, CUR.POSY or CUR.POS Z".

Resulting NC block:

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

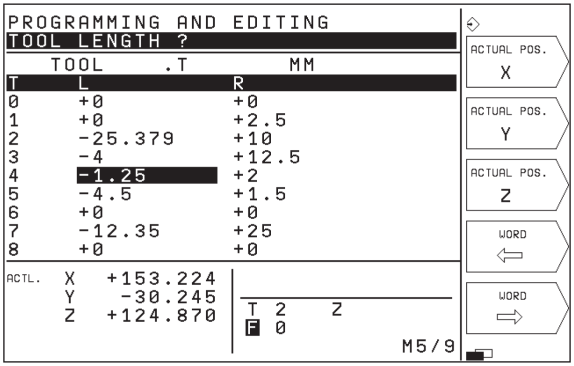
Entering tool data in tables

You can define and store up to 99 tools and their tool data in the tool table TOOL.T. (The maximum number of tools in the table can be set in machine parameter 7260).

Tool table: Available input data

Abbr.	Input	Dialog
T	Number by which the tool is called in the program	–
L	Value for tool length compensation	TOOL LENGTH ?
R	Tool radius R	TOOL RADIUS ?

Editing the tool table



The tool table has the file name TOOL.T. The TOOL.T file can be edited in the PROGRAMMING AND EDITING operating mode. TOOL.T is automatically active in a program run operating mode.

To open the tool table TOOL.T:

▶ Select the PROGRAMMING AND EDITING mode of operation.

▶ call the file manager.

▶ Move the highlight to TOOL.T. Confirm with the ENT 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 (see figure at center right). You can overwrite the stored values, or enter new values at any position. The available editing functions are illustrated in the table on the next page.

To leave the tool table:

▶ Finish editing the tool table: Press the END key.

▶ Call the file manager and select a file of a different type, e.g. a part program.

Editing functions for tool tables	Soft key
Take the value from the position display	ACTUAL POS. Z
Go to the previous page of the table (2nd soft-key row)	PAGE ⇧
Go to the next page of the table (2nd soft-key row)	PAGE ⇩
Move the highlight one column to the left	WORD ⇦
Move the highlight one column to the right	WORD ⇨
Delete incorrect numerical value, re-establish preset value	CE
Re-establish the last value stored	DEL
Move the highlight back to beginning of line	END □

Calling tool data

A TOOL CALL block in the part program is defined with the following data:

TOOL CALL

- Select the tool call function with the TOOL CALL key
- TOOL NUMBER: Enter the number of the tool. The tool must already be defined in a TOOL DEF block or in the tool table.
- WORKING SPINDLE AXIS X/Y/Z: Enter the tool axis.
- SPINDLE SPEED S
- TOOL LENGTH OVERSIZE: Enter the delta value for the tool length.
- TOOL RADIUS OVERSIZE: Enter the delta value for the tool radius.

Example:
Call tool number 5 in the tool axis Z with a spindle speed 2500 rpm. The tool length is to be programmed with an oversize of 0.2 mm, the tool radius with an undersize of 1 mm.

```
20 TOOL CALL 5 Z S2500 DL+0.2 DR-1
```

The character D preceding L and R designates delta values.

Tool change

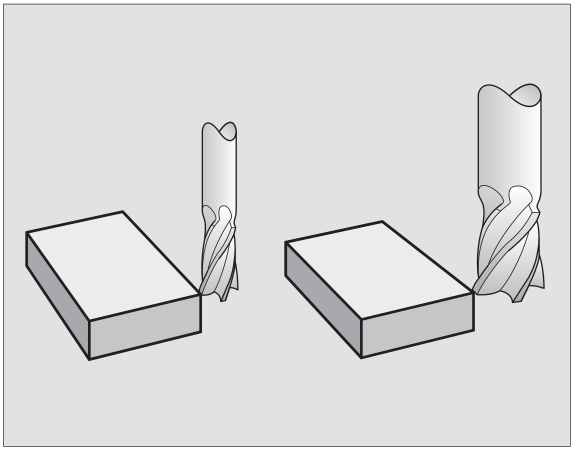
The tool change function can vary depending on the individual machine tool. Your machine manual provides more information on M101.

Tool change position
A 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 section 10.3 "Program Run").
- Change the tool.
- Resume the program run (see section 10.3 "Program Run").

5.3 Tool Compensation



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.

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 the TOOL CALL block into account:

Compensation value = L + $DL_{TOOL\ CALL}$ 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)

Tool radius compensation

The NC block for programming a tool movement contains:

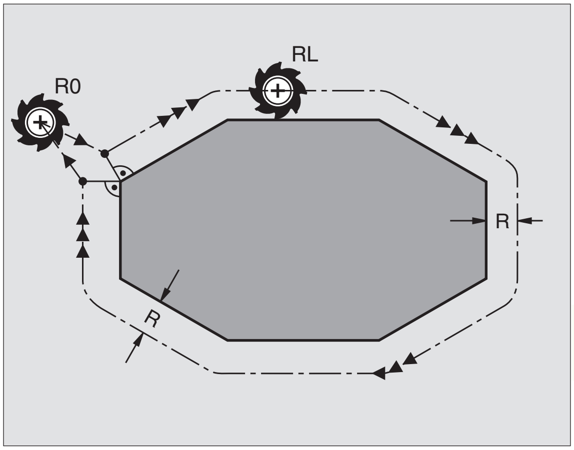
- RL or RR for compensation in the tool radius
- R+ or R– for radius compensation in single-axis movements
- R0 if no radius compensation is required

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with RL or RR. To cancel radius compensation, program a positioning block with R0.

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

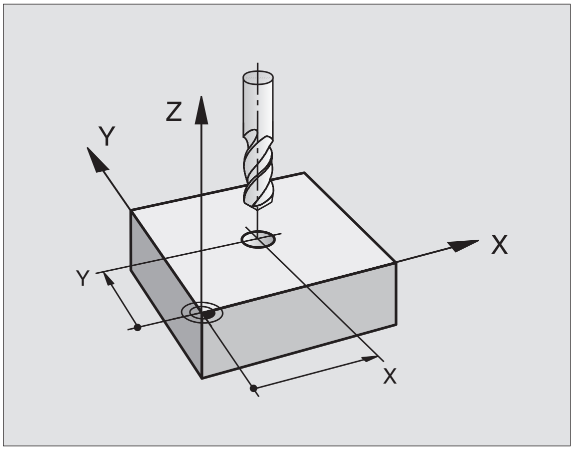
Compensation value = R + $DR_{TOOL\ CALL,}$ where

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)



Tool movements without radius compensation: R0
The tool center moves in the working plane to the programmed path or coordinates.

Applications: Drilling and boring, pre-positioning (see figure at center right)



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 illustrations on the next page).

☞ Between two program blocks with different radius compensations (RR and RL) you must program at least one block without radius compensation (that is, with R0).

Radius compensation does not come into effect until the end of the block in which it is first programmed.

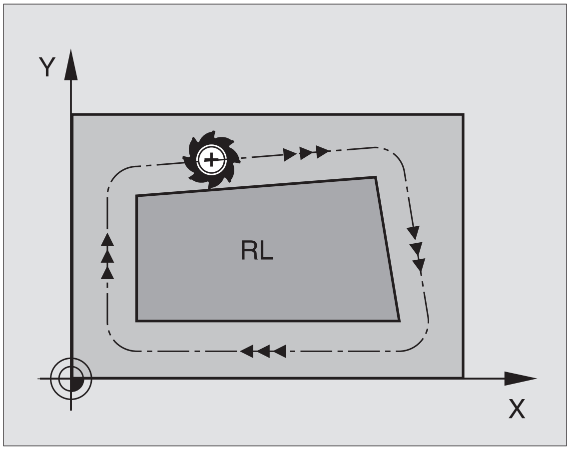
Whenever radius compensation is activated with RR/RL or canceled with R0, the TNC positions the tool perpendicular to the programmed starting or end position. Position the tool at a sufficient distance from the first or last contour point to prevent the possibility of damaging the contour.

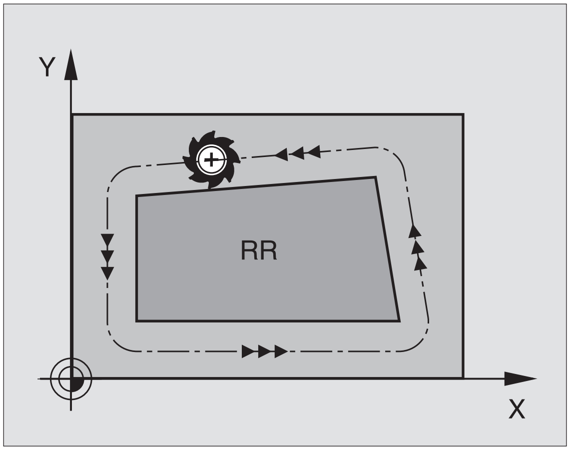
Entering radius compensation

When you program a path contour, the following dialog question is displayed after entry of the coordinates:

RADIUS COMP. RL/RR/NO COMP. ?

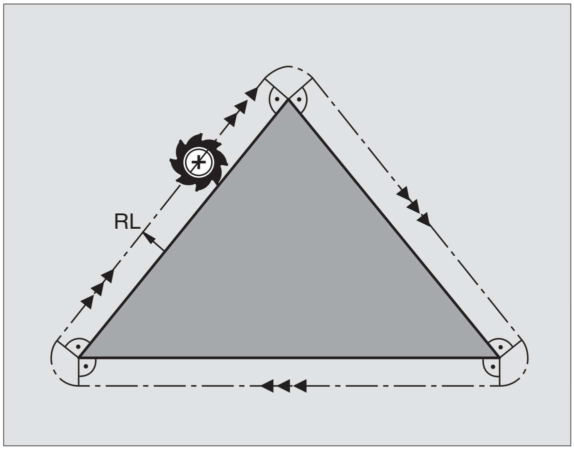
RL	To select tool movement to the left of the contour, press the RL soft key, or
RR	To select tool movement to the right of the contour, press the RR soft key, or
ENT	To select tool movement without radius compensation or to cancel radius compensation, press the ENT key or the R0 soft key.
END	To terminate the dialog, press the END key.





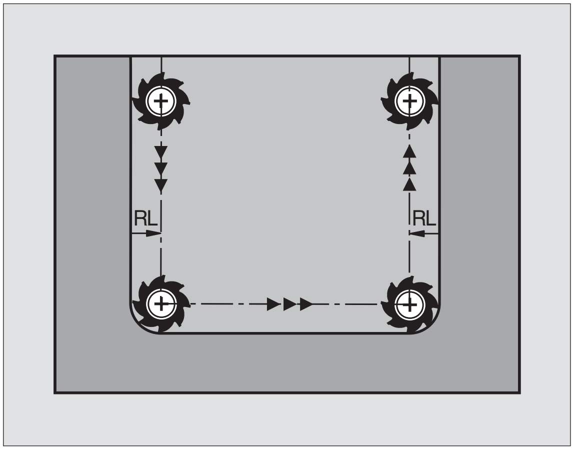
Radius compensation: Machining corners

Outside corners
If you program radius compensation, the TNC moves the tool in a transitional arc around corners. The tool "rolls around" the corner point. 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.



Machining corners without radius compensation
If you program the tool movement without radius compensation, you can change the tool path and feed rate at workpiece corners with the miscellaneous function M90. See "7.4 Miscellaneous Functions for Contouring Behavior."

6

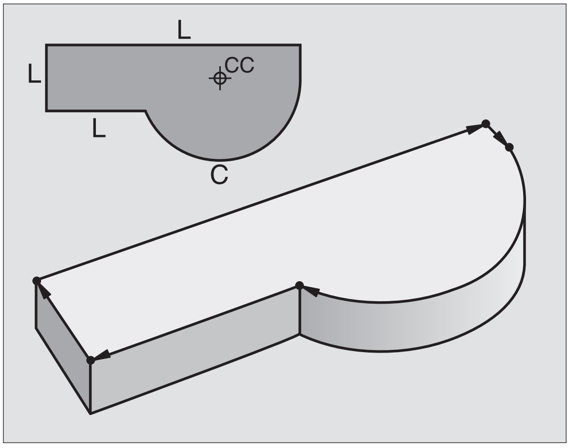
Programming:

Programming Contours

6.1 Overview of 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**.



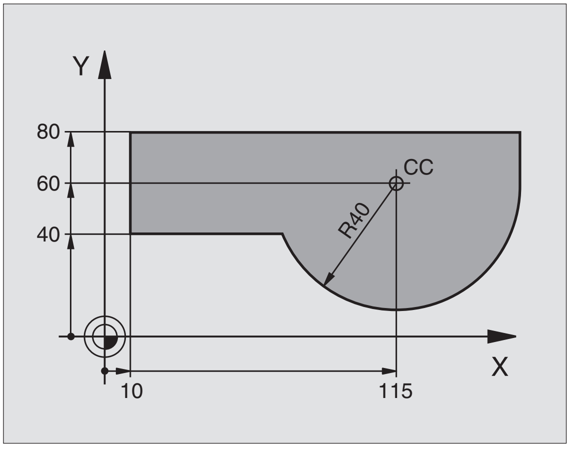
Miscellaneous functions M

With the TNC's miscellaneous functions you can affect

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

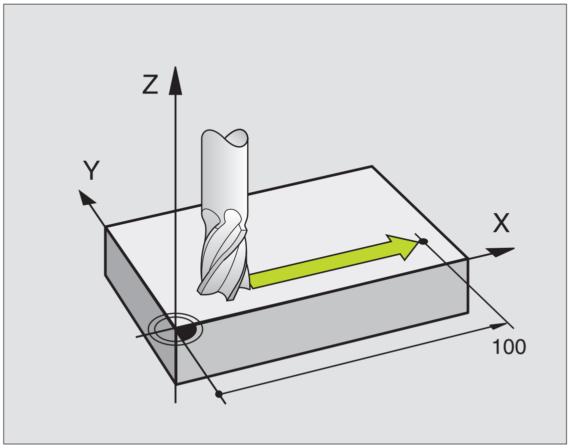
Subprograms and program section repeats

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.



How subprograms and program section repeats are used in programming is described in Chapter 9.

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.

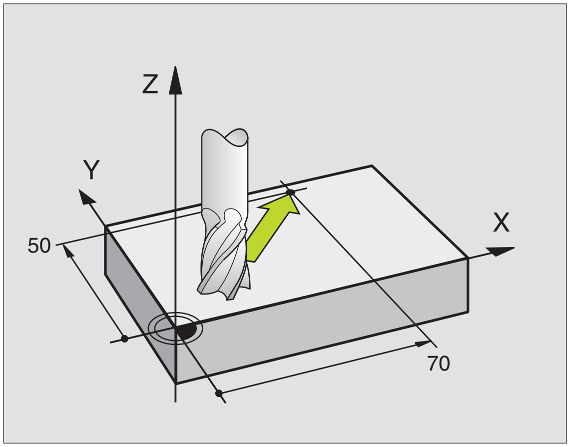
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

```
L X+100
```

L	Path function for "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.



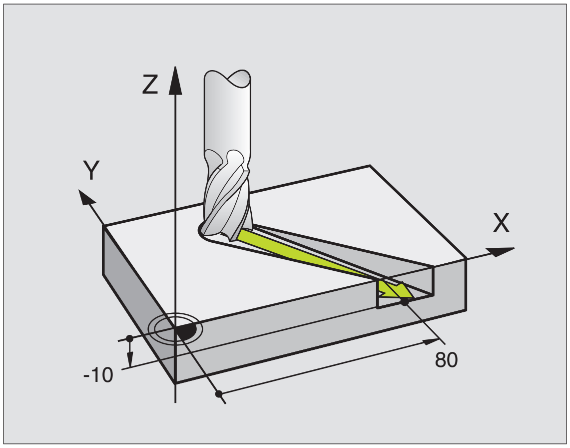
Movement in the main planes

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 position X=70, Y=50. 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
```

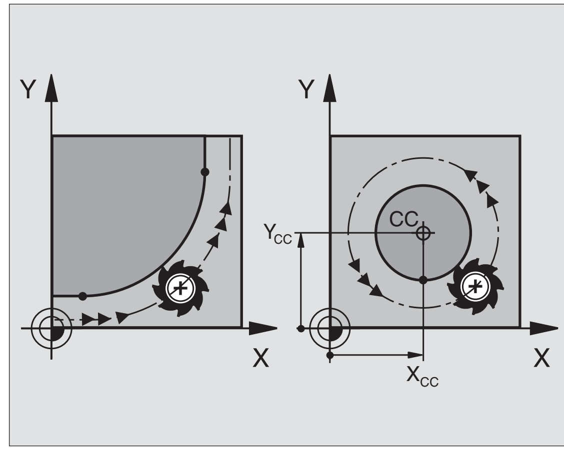
This example is illustrated in the figure at lower right.

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 TNC assigns it to one of the main planes. This plane is defined automatically when you set the spindle axis during a tool call:

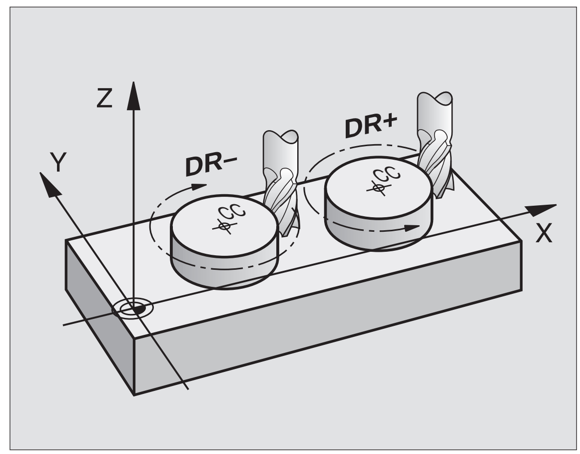
Spindle axis	Main plane
Z	XY
Y	ZX
X	YZ



Direction of rotation DR for circular movements

When 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

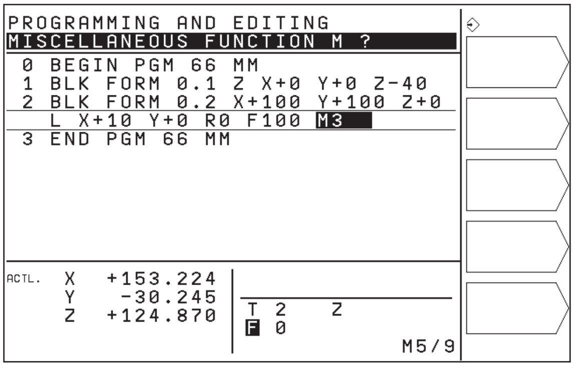
Radius compensation must be programmed before the block containing the coordinates for the first contour element. You cannot begin radius compensation in a circle block. It must be activated beforehand in a straight-line block.

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
Use the path function keys to open a conversational 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:



L — Initiate the programming dialog (here, for a straight line).

COORDINATES ?

X 10 — Enter the coordinates of the straight-line end point.

Y 5

ENT

ACTUAL POSITION — Transfer the coordinates of the selected axis: Press ACTUAL POSITION soft key (second soft-key row)

RADIUS COMP. RL/RR/NO COMP. ?

RL — Select the radius compensation (here, press the RL soft key — the tool moves to the left of the programmed contour).

FEED RATE F=

100 ENT — Enter the feed rate (here, 100 mm/min), and confirm your entry with ENT.

MISCELLANEOUS FUNCTION M ?

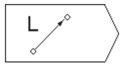
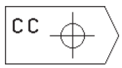
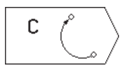
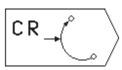
3 ENT — Enter a miscellaneous function (here, M3), and terminate the dialog with ENT.

The part program now contains the following line:

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

6.3 Path Contours — Cartesian Coordinates

Overview of path functions

Function	Contour function soft key	Tool movement	Required input
Line **L**		Straight line	Coordinates of the straight-line end point
CHam**F**er		Chamfer between two straight lines	Chamfer side length
Circle **C**enter		No tool movement	Coordinates of the circle center or pole
Circle		Circular arc around a circle center CC to an arc end point	Coordinates of the arc end point, direction of rotation
Circle by **R**adius		Circular arc with a certain radius	Coordinates of the arc end point, arc radius, direction of rotation
Circle **T**angential		Circular arc with tangential connection to the preceding contour element	Coordinates of the arc end point
Corner **R**ou**ND**ing		Circular arc with tangential connection to the preceding and subsequent contour elements	Rounding-off radius R

Straight line L

The tool moves on a straight line from its current position to the line end point. The starting point for the straight line is the end point that was programmed in the preceding block.

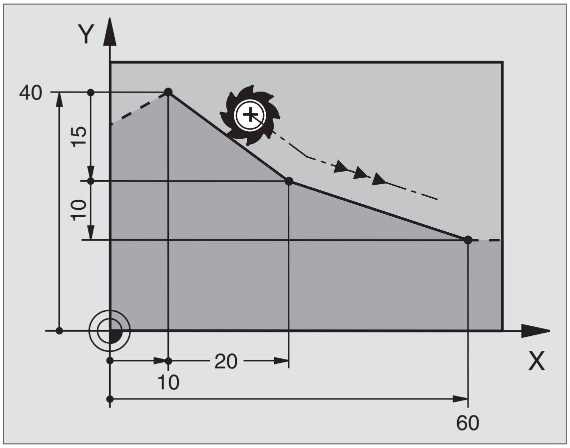
▶ Enter the COORDINATES of the end point.

Further entries, if necessary:

▶ RADIUS COMPENSATION RL/RR/R0

▶ FEED RATE F

▶ MISCELLANEOUS FUNCTION M



Example NC blocks

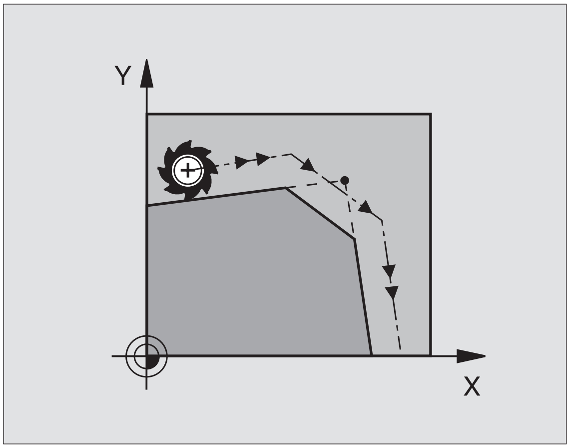
```
7 L X+10 Y+40 RL F200 M3
8 L IX+20 IY-15
9 L X+60 IY-10
```

Inserting a chamfer CHF between two straight lines

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

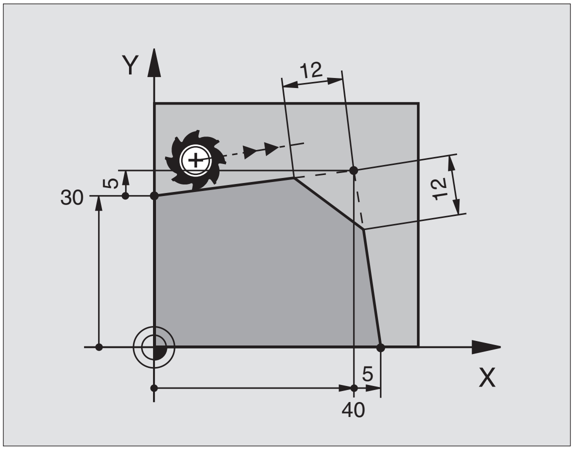
- The blocks before and after the CHF block must be in the same working plane.
- The radius compensation before and after the chamfer block must be the same.
- An inside chamfer must be large enough to accommodate the current tool.

▶ CHAMFER SIDE LENGTH: Enter the length of the chamfer



Example NC blocks

```
7 L X+0 Y+30 RL F300 M3
8 L X+40 IY+5
9 CHF 12
10 L IX+5 Y+0
```



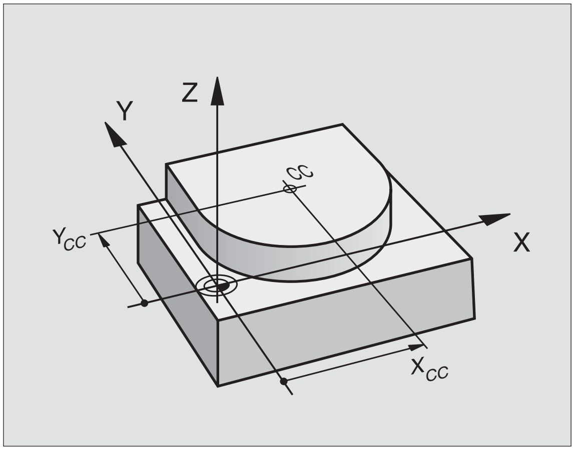
You cannot start a contour with a CHF block

A chamfer is possible only in the working plane.

The feed rate for chamferring is taken from the preceding block.

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

Circle center CC



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

- Entering the Cartesian coordinates of the circle center
- Using the circle center defined in an earlier block
- Capturing the coordinates with the „ACTUAL POSITION" soft key

CIRCLE

▶ Select circle functions: Press the „CIRCLE" soft key (2nd soft-key row)

CC

▶ COORDINATES CC: Enter the circle center coordinates

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 a circle center — the tool does not move to the position.

The circle center also serves as 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.

CIRCLE

▶ Select circle functions: Press the „CIRCLE“ soft key (2nd soft-key row)

CC

▶ Enter the COORDINATES of the circle center.

C

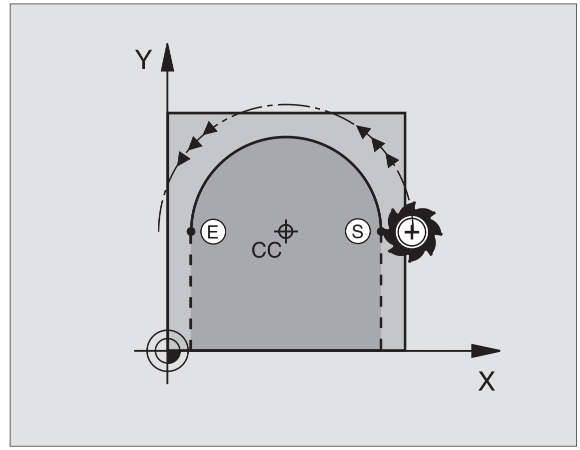
▶ Enter the COORDINATES of the arc end point

▶ DIRECTION OF ROTATION DR

Further entries, if necessary:

▶ FEED RATE F

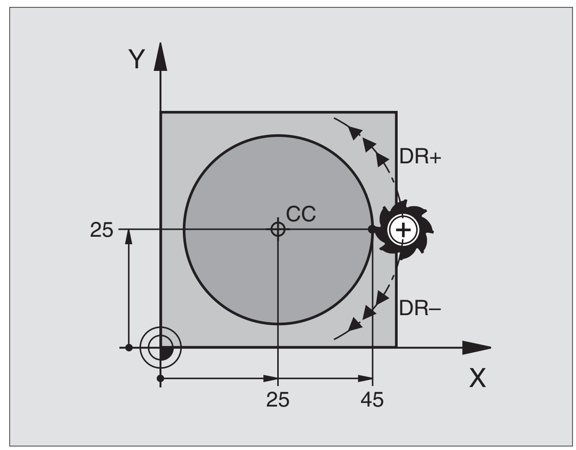
▶ MISCELLANEOUS FUNCTION M



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+
```

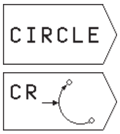
Full circle
Enter the same point you used as the starting point for the end point in a C block.



> The starting and end points of the arc must lie on the circle.
>
> Input tolerance: up to 0.016 mm.

Circular path CR with defined radius

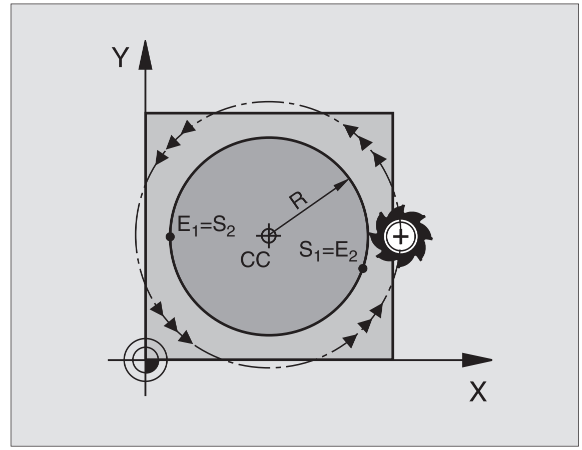
The tool moves on a circular path with the radius R.



- Select circle functions: Press the „CIRCLE" soft key (2nd soft-key row)
- Enter the 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:

- FEED RATE F
- MISCELLANEOUS FUNCTION M



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. See figure at upper right.

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:

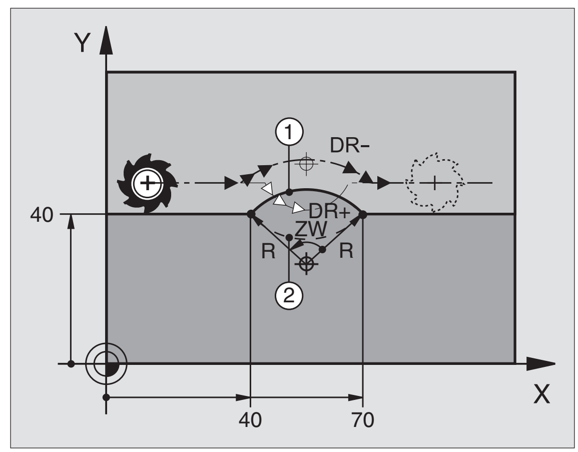
Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

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
See figures at middle and lower right.

```
10 L X+40 Y+40 RL F200 M3
11 CR X+70 Y+40 R+20 DR- (arc 1)
```

or

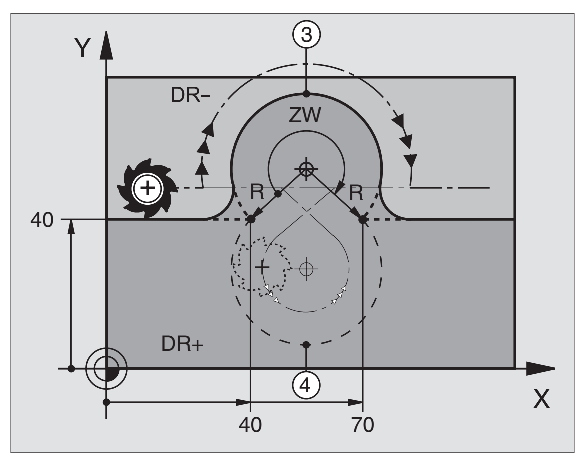
```
11 CR X+70 Y+40 R+20 DR+ (arc 2)
```

or

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

or

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



Please observe the notes on the next page!

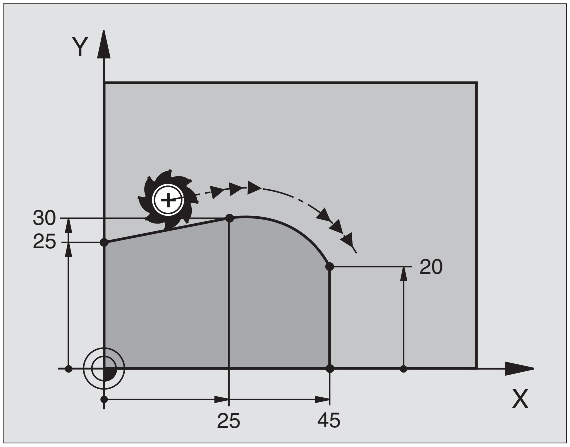
> The distance from the starting and end points of the arc diameter cannot be greater than the diameter of the arc.
>
> The maximum possible radius is 30 m.

Circular path CT with tangential connection

The tool moves on an arc that starts at a tangent with 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.



CIRCLE
- Select circle functions: Press the „CIRCLE“ soft key (2nd soft-key row)

CT
- Enter the 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 of the arc.

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-OFF RADIUS: Enter the radius of the arc.
- FEED RATE for rounding the corner.

Example NC blocks

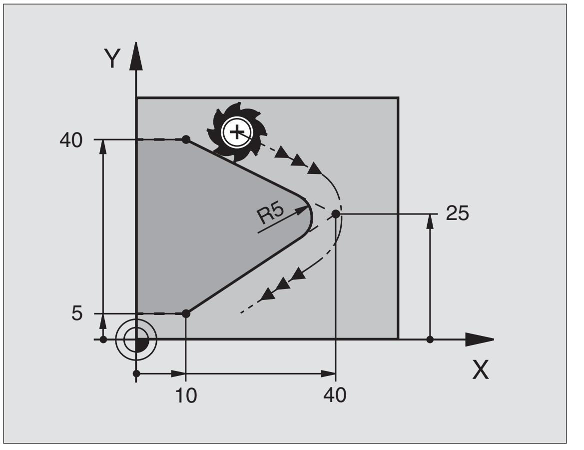
```
5 L X+10 Y+40 RL F300 M3
6 L X+40 Y+25
7 RND R5 F100
8 L X+10 Y+5
```

In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc.

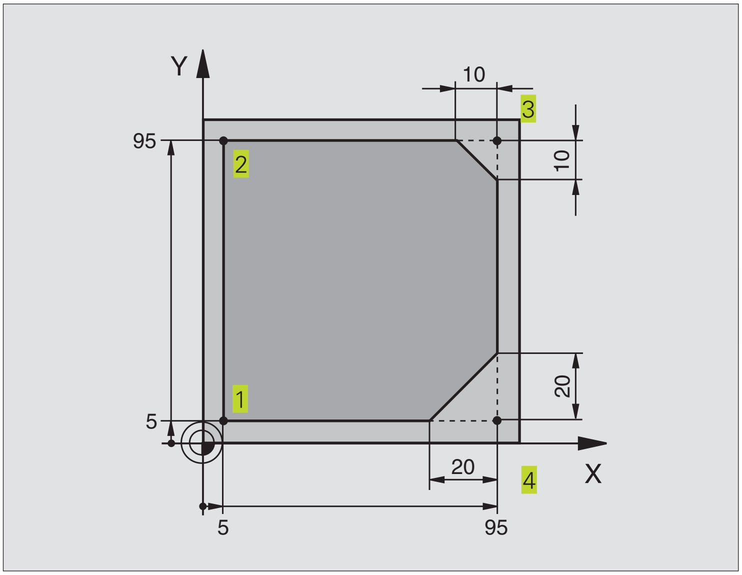
The corner point is cut off by the rounding arc and is not part of the contour.

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.

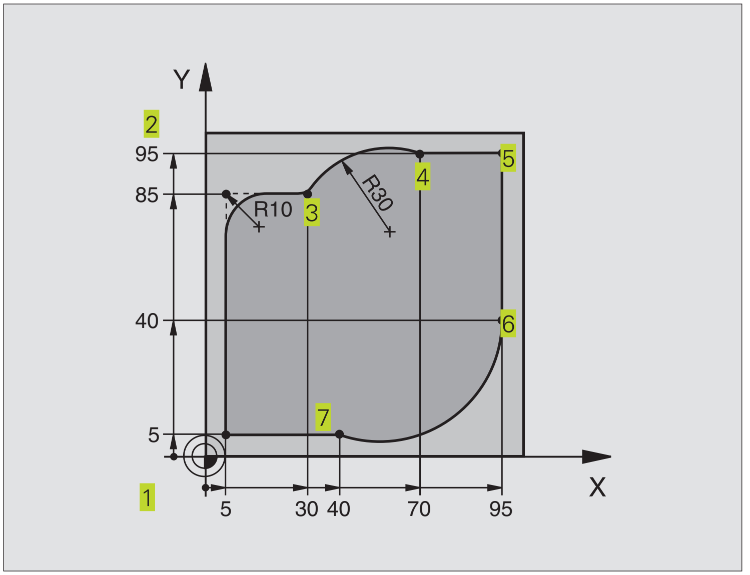


Example: Linear movements and chamfers with Cartesian coordinates



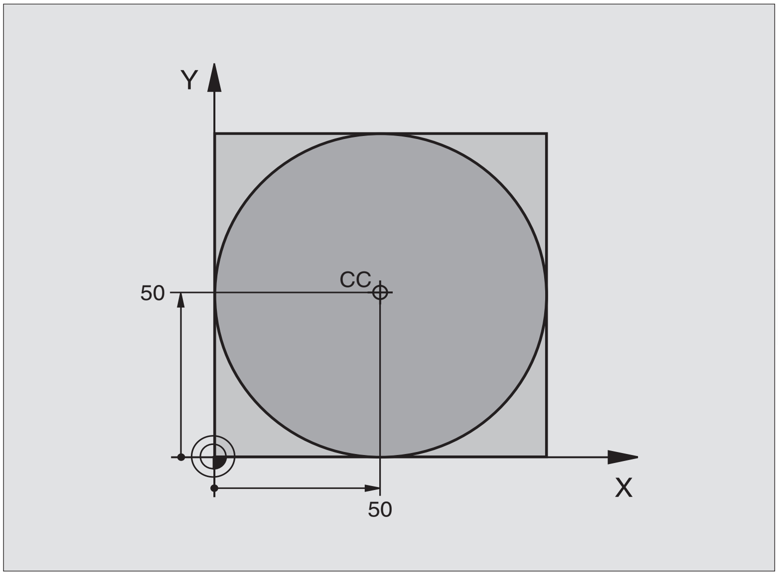
0 BEGIN PGM 10 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define blank form for graphic workpiece simulation
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+10	Define tool in the program
4 TOOL CALL 1 Z S4000	Call tool in the spindle axis and with the spindle speed S
5 L Z+250 R0 F MAX	Retract tool in the spindle axis at rapid traverse FMAX
6 L X-20 Y-10 R0 F MAX	Pre-position the tool
7 L Z-5 R0 F1000 M3	Move to working depth at feed rate F = 1000 mm/min
8 L X+5 Y+5 RL F300	Approach the contour at point 1
9 RND R2	Tangential approach to circle with R=2 mm
10 L Y+95	Move to point 2
11 L X+95	Point 3: first straight line for corner 3
12 CHF 10	Program chamfer with length 10 mm
13 L Y+5	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
14 CHF 20	Program chamfer with length 20 mm
15 L X+5	Move to last contour point 1, second straight line for corner 4
16 RND R2	Tangential departure from circle with R=2 mm
17 L X-20 R0 F1000	Retract tool in the working plane
18 L Z+250 R0 F MAX M2	Retract tool in the spindle axis, end of program
19 END PGM 10 MM	

Example: Circular movements with Cartesian coordinates



`0 BEGIN PGM 20 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 F MAX`	Retract tool in the spindle axis at rapid traverse FMAX
`6 L X-20 Y-20 R0 F MAX`	Pre-position the tool
`7 L Z-5 R0 F1000 M3`	Move to working depth at feed rate F = 1000 mm/min
`8 L X+5 Y+5 RL F300`	Approach the contour at point 1
`9 RND R2`	Tangential approach to circle with R=2 mm
`10 L Y+85`	Point 2: first straight line for corner 2
`11 RND R10 F150`	Insert radius with R = 10 mm, feed rate: 150 mm/min
`12 L X+30`	Move to point 3: Starting point of the arc with CR
`13 CR X+70 Y+95 R+30 DR-`	Move to point 4: End point of the arc with CR, radius 30 mm
`14 L X+95`	Move to point 5
`15 L Y+40`	Move to point 6
`16 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
`17 L X+5`	Move to last contour point 1
`18 RND R2`	Tangential departure from circle with R=2 mm
`19 L X-20 Y-20 R0 F1000`	Retract tool in the working plane
`20 L Z+250 R0 F MAX M2`	Retract tool in the spindle axis, end of program
`21 END PGM 20 MM`	

Example: Full circle with Cartesian coordinates



0 BEGIN PGM 30 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
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	Call the tool
5 CC X+50 Y+50	Define the circle center
6 L Z+250 R0 F MAX	Retract the tool
7 L X-40 Y+50 R0 F MAX	Pre-position the tool
8 L Z-5 R0 F1000 M3	Move to working depth
9 L X+0 Y+50 RL F300	Approach starting point of circle
10 RND R2	Tangential approach to circle with R=2 mm
11 C X+0 DR-	Move to the circle end point (= circle starting point)
12 RND R2	Tangential departure from circle with R=2 mm
13 L X-40 Y+50 R0 F1000	Retract tool in the working plane
14 L Z+250 R0 F MAX M2	Retract tool in the spindle axis, end of program
15 END PGM 30 MM	

6.4 Path Contours— Polar Coordinates

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 section "4.1 Fundamentals of NC."

Polar coordinates are useful with:

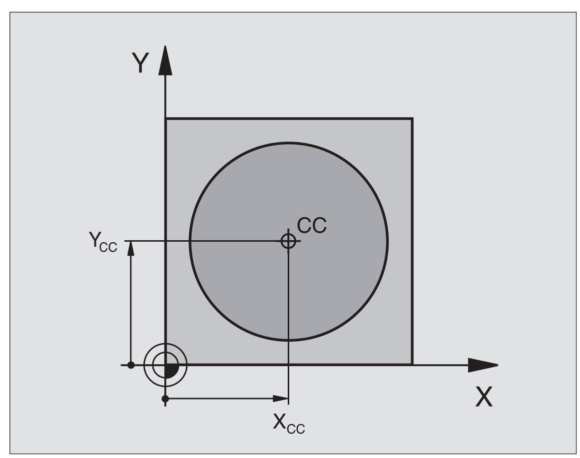
- Positions on circular arcs
- Workpiece drawing dimensions in degrees, e.g. bolt hole circles

Overview of path functions with polar coordinates

Function	Contour function soft keys	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 path with tangential connection to the preceding contour element	Polar radius, polar angle of the arc end point
Helix (Helix)	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.



CIRCLE

▶ Select circle functions: Press the „CIRCLE" soft key

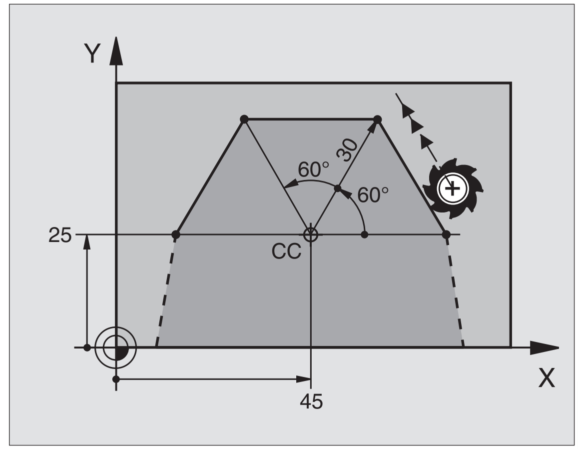
CC

▶ COORDINATES CC: Enter Cartesian coordinates for the pole, or:

If you want to use the last programmed position, do not enter any coordinates.

Straight line LP

The tool moves in a straight line from its current position to the straight-line end point. The starting point for the straight line is the end point that was programmed in the preceding block.



- Select straight line function: Press the L soft key

- Select entry of polar coordinates: Press the P soft key (2nd soft-key row). 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°

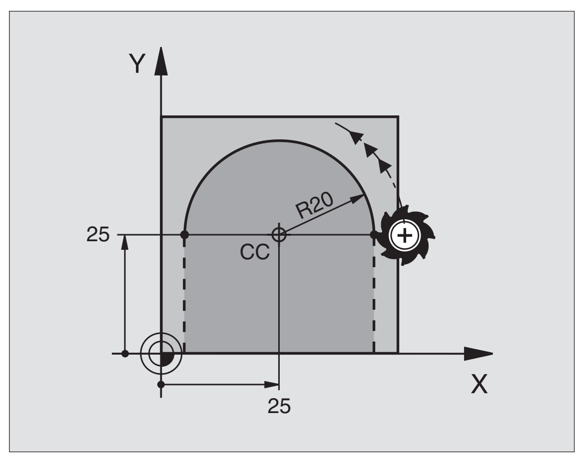
  The sign of PA depends on the angle reference axis:
  Angle from angle reference axis to PR is counterclockwise: PA>0
  Angle from angle reference axis to PR is clockwise: PA<0

Example NC blocks

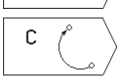
```
12 CC X+45 Y+25
13 LP PR+30 PA+0 RR F300 M3
14 LP PA+60
15 LP IPA+60
16 LP PA+180
```

Circular path CP around pole CC

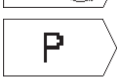
The polar coordinate radius PR is also the radius of the arc. It is defined by the distance from the starting point to the pole CC. The last programmed tool position before the CP block is the starting point of the arc.



- Select circle functions: Press the „CIRCLE" soft key



- Select circular path C: Press the C soft key



- Select entry of polar coordinates: Press the P soft key (2nd soft-key row).
- POLAR COORDINATES-ANGLE PA: Angular position of the arc end point between –5400° and +5400°
- DIRECTION OF ROTATION DR

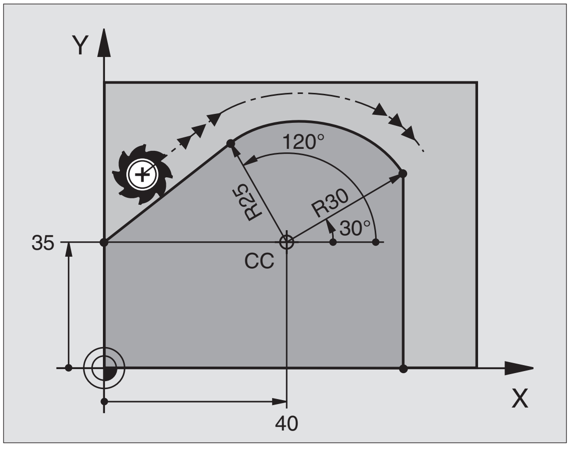
Example NC blocks

```
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.



CIRCLE

▶ Select circle functions: Press the „CIRCLE“ soft key

CT

▶ Select the circular path CT: Press the CT soft key

P

▶ Select entry of polar coordinates: Press the P soft key (2nd soft-key row).

▶ 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

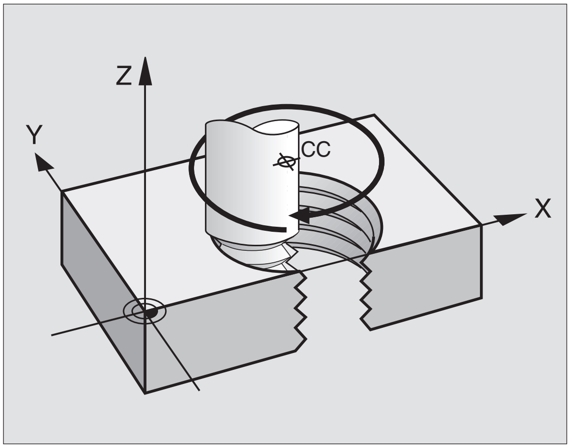
```
12 CC X+40 Y+35
13 L X+0 Y+35 RL F250 M3
14 LP PR+25 PA+120
15 CTP PR+30 PA+30
16 L Y+0
```

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.



Applications

- 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 a upward direction, you need the following data:

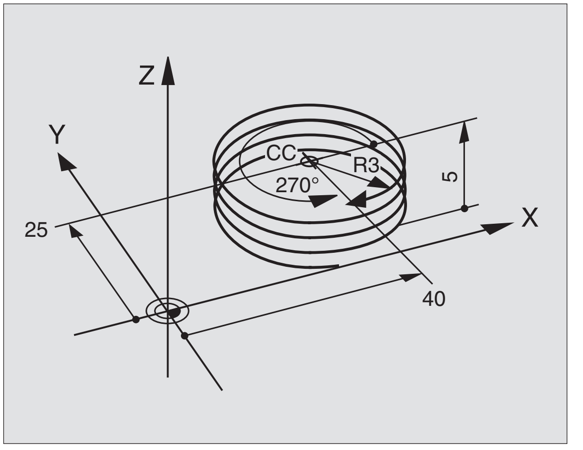
Thread revolutions n	Thread revolutions + thread overrun at the start and end of the thread
Total height h	Thread pitch P x thread revolutions n
Incremental total angle IPA	Thread revolutions x 360° + angle for beginning of thread + angle for thread overrun
Starting coordinate Z	Thread pitch P x (thread 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 direct.	Direction	Radius compens.
Right-handed	Z+	DR+	RL
Left-handed	Z+	DR–	RR
Right-handed	Z–	DR–	RR
Left-handed	Z–	DR+	RL
External thread			
Right-handed	Z+	DR+	RR
Left-handed	Z+	DR–	RL
Right-handed	Z–	DR–	RL
Left-handed	Z–	DR+	RR

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 of more than 15 revolutions, program the helix in a program section repeat (see section 9.2 "Program Section Repeats").

CIRCLE

▶ Select circle functions: Press the „CIRCLE" soft key

C

▶ Select circular path C: Press the C soft key

P

▶ Select entry of polar coordinates: Press the P soft key (2nd soft-key row).

▶ POLAR COORDINATES-ANGLE: Enter the total angle of tool traverse along the helix in incremental dimensions. **After entering the angle, identify the tool axis using a soft key.**

▶ Enter the COORDINATE for the height of the helix in incremental dimensions.

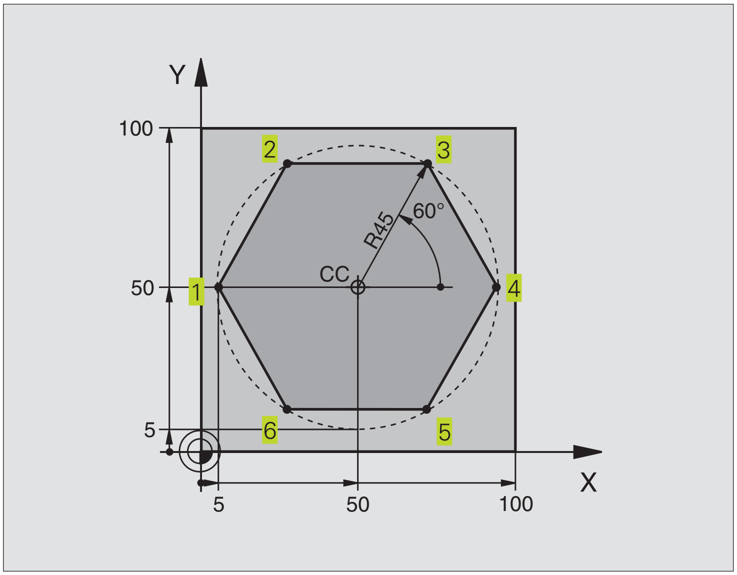
▶ Direction of rotation DR
Clockwise helix: DR–
Counterclockwise helix: DR+

▶ RADIUS COMPENSATION RL/RR/R0
Enter the radius compensation according to the table above.

Example NC blocks

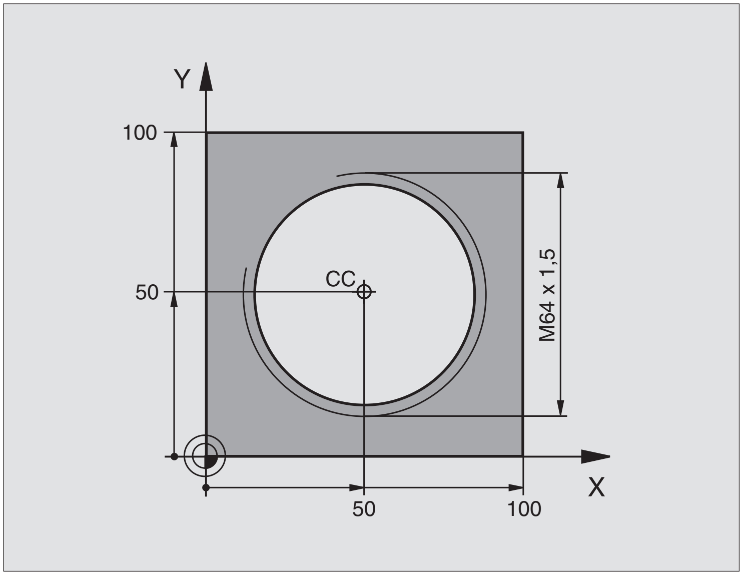
```
12 CC X+40 Y+25
13 Z+0 F100 M3
14 LP PR+3 PA+270 RL F50
15 CP IPA–1800 IZ+5 DR– RL F50
```

Example: Linear movement with polar coordinates



0 BEGIN PGM 40 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	Define the blank form
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	Call the tool
5 CC X+50 Y+50	Define the datum for polar coordinates
6 L Z+250 R0 F MAX	Retract the tool
7 LP PR+60 PA+180 R0 F MAX	Pre-position the tool
8 L Z-5 R0 F1000 M3	Move to working depth
9 LP PR+45 PA+180 RL F250	Approach the contour at point 1
10 RND R1	Tangential approach to circle with R=1 mm
11 LP PA+120	Move to point 2
12 LP PA+60	Move to point 3
13 LP PA+0	Move to point 4
14 LP PA-60	Move to point 5
15 LP PA-120	Move to point 6
16 LP PA+180	Move to point 1
17 RND R1	Tangential departure from circle with R=1 mm
18 LP PR+60 PA+180 R0 F1000	Retract tool in the working plane
19 L Z+250 R0 F MAX M2	Retract tool in the spindle axis, end of program
20 END PGM 40 MM	

Example: Helix



`0 BEGIN PGM 50 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the blank form
`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`	Call the tool
`5 L Z+250 R0 F MAX`	Retract the tool
`6 L X+50 Y+50 R0 F MAX`	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 LP PR+32 PA-180 RL F100`	Approach contour
`10 RND R2`	Tangential approach to circle with R=2 mm
`11 CP IPA+3240 IZ+13,5 DR+ F200`	Helical interpolation
`12 RND R2`	Tangential departure from circle with R=2 mm
`13 L X+50 Y+50 R0 F MAX`	Retract tool in the working plane
`14 L Z+250 R0 F MAX M2`	Retract tool in the spindle axis, end of program
`15 END PGM 50 MM`	

To cut a thread with more than 16 revolutions

`...`	
`8 L Z-12.75 R0 F1000`	
`9 LP PR+32 PA-180 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)

7

Programming:

Miscellaneous Functions

7.1 Entering Miscellaneous Functions M and STOP

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

- Program run, such as 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. Refer to your machine manual for more information.

M functions are always entered at the end of a positioning block. The TNC then displays the following dialog question:

```
MISCELLANEOUS FUNCTION M ?
```

Only enter the number of the M function in the programming dialog.

In the MANUAL OPERATION operating mode, the M functions are entered with the M soft key.

Please note that some F functions become effective at the start of a positioning block, and other at the end.

M functions come into effect in the block in which they are called. Unless the M function is only effective blockwise, it is canceled in a subsequent block or at the end of program. Some M functions are effective only in the block in which they are called.

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 a MISCELLANEOUS FUNCTION M.

Resulting NC block

```
87 STOP M6
```

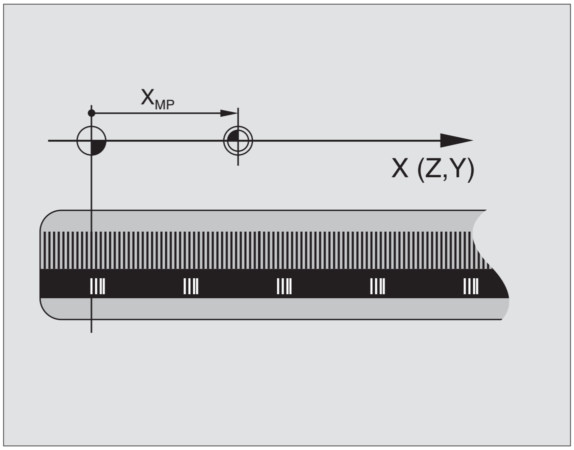
7.2 Miscellaneous Functions for Program Run Control, Spindle and Coolant

M	Effect	Effective at
M00	Stop program run Spindle STOP Coolant OFF	Block end
M01	Stop program run	Block end
M02	Stop program run Spindle STOP Coolant OFF Go to block 1 Clear the status display (dependent on machine parameter 7300)	Block end
M03	Spindle ON clockwise	Block start
M04	Spindle ON counterclockwise	Block start
M05	Spindle STOP	Block end
M06	Tool change Spindle STOP Program run stop (dependent on machine parameter 7440)	Block end
M08	Coolant ON	Block start
M09	Coolant OFF	Block end
M13	Spindle ON clockwise Coolant ON	Block start
M14	Spindle ON counterclockwise Coolant ON	Block start
M30	Same as M02	Block end

7.3 Miscellaneous Functions for Coordinate Data

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").

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.

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 section 1.4 "Status Displays").

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.

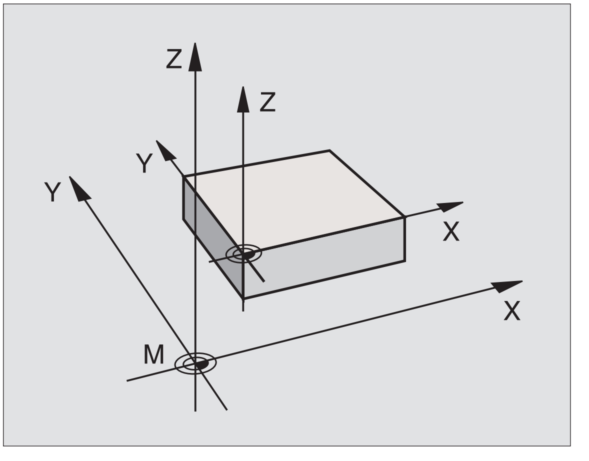
If you want the coordinates in a positioning block to be based on the additional machine datum, end the block with M92.

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 with M91 or M92.

M91 and M92 become effective at the start of block.

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



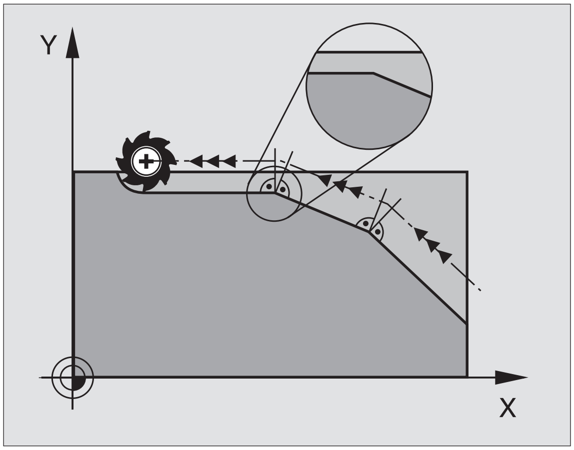
7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

Standard behavior

The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an accurate stop.

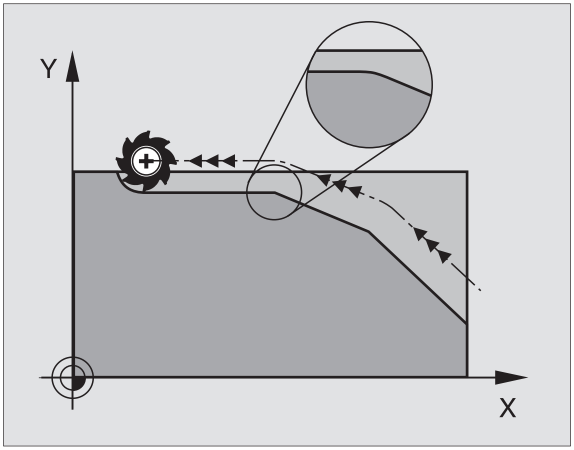
In program blocks with radius compensation (RR/RL), the TNC automatically inserts a transition arc at outside corners.



Behavior with M90

The tool moves at corners with constant speed: This provides a smoother, more continuous surface. Machining time is also reduced. See figure at center right.

Example application: Surface consisting of a series of straight line segments.



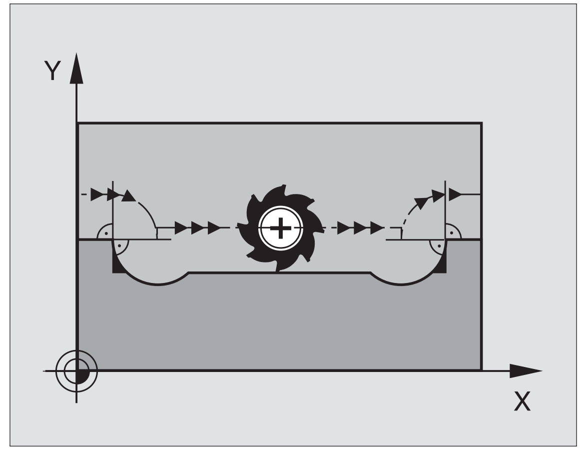
Effect

M90 is effective only in the blocks in which it is programmed with M90.

M90 becomes effective at the start of block. Operation with servo lag must be active.

> ☞ Independently of M90, you can use machine parameter MP7460 to set a limit value up to which the tool moves at constant path speed (effective with servo lag and feedforward control).

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. See figure at upper right.

In such cases the TNC interrupts program run and generates the error message "TOOL RADIUS TOOL LARGE."

Behavior with M97

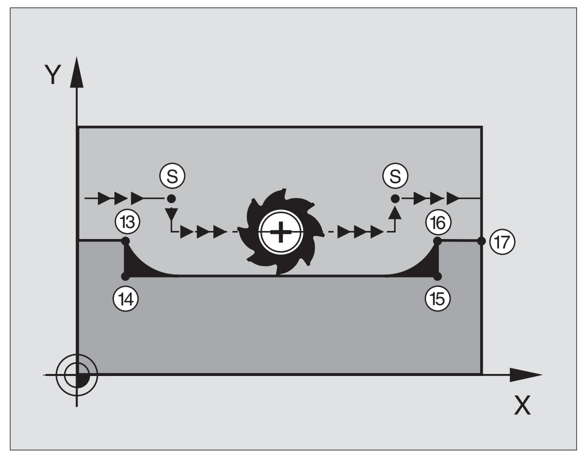
The TNC calculates the intersection of the contour elements — as at inside corners — and moves the tool over this point. See figure at lower right.

Program M97 in the same block as the outside corner.

Effect

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

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



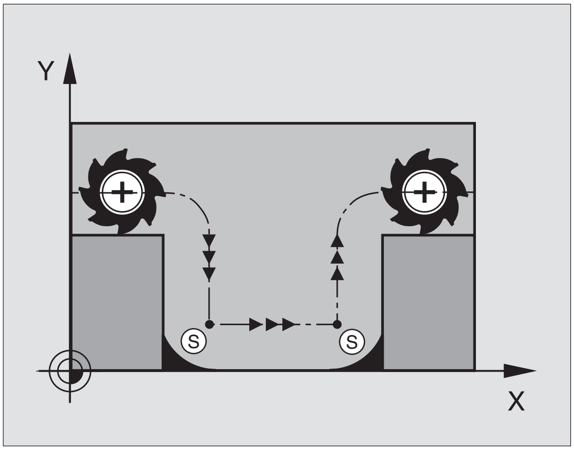
Example NC blocks

`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 (see figure at upper right).



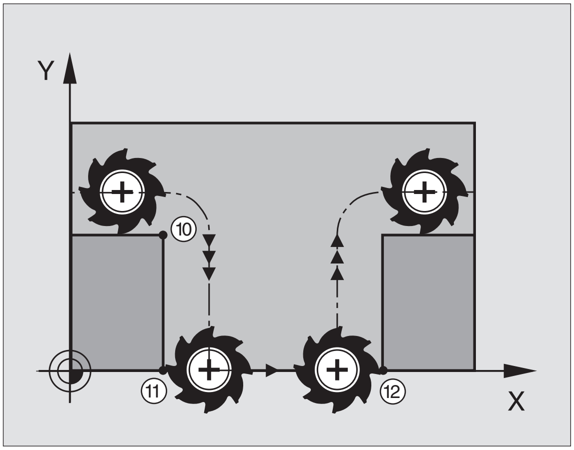
Behavior with M98
With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined (see figure at lower right).

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

M98 becomes effective 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+ ...
```



7.5 Miscellaneous Function for Rotary Axes

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 path 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.

Example NC blocks
To reduce display of all active rotary axes:

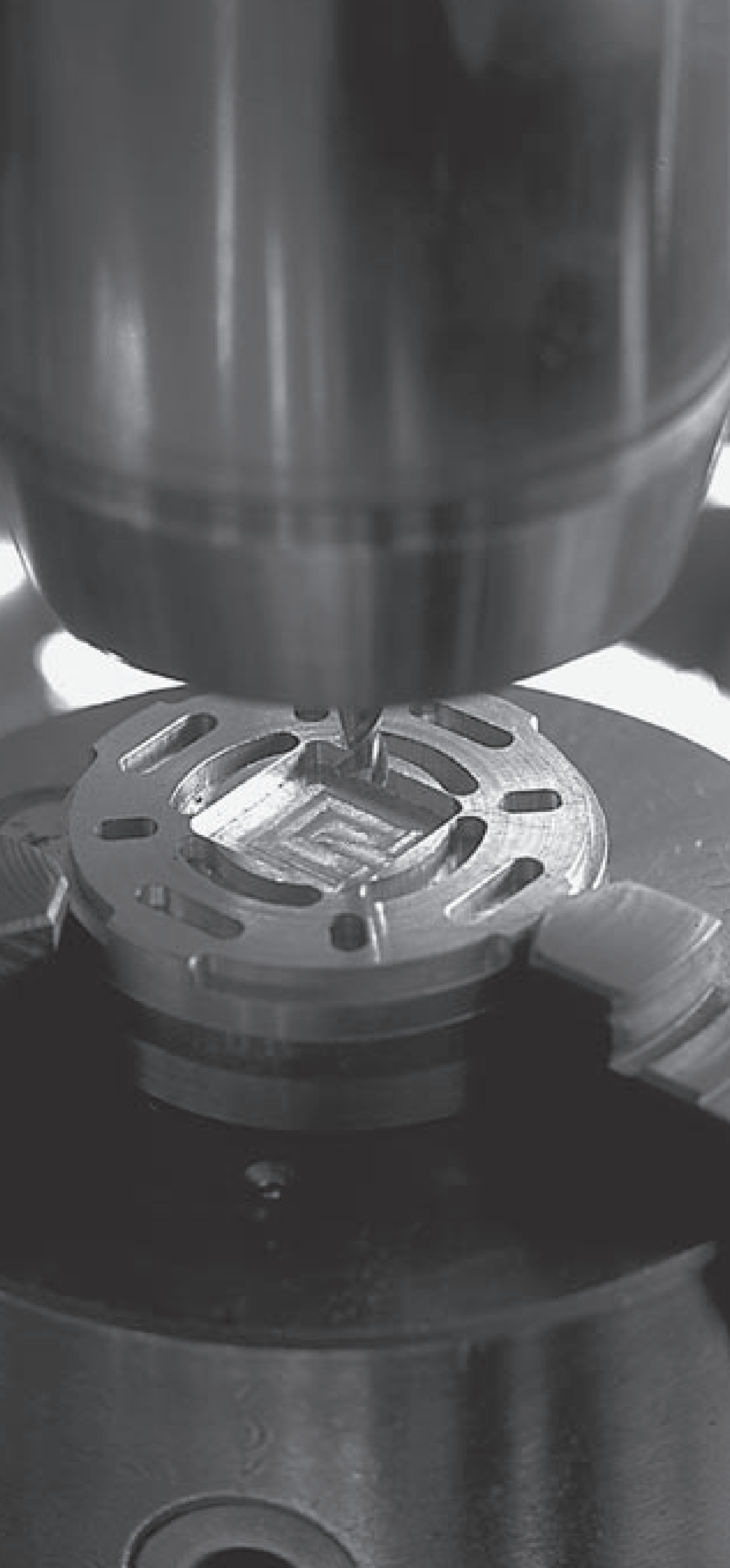
```
L M94
```

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 M94 is programmed.

M94 becomes effective at the start of block.



8

Programming:

Cycles

8.1 General Overview of 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. The table at right shows the various cycle groups.

Fixed cycles with number starting with 200 use Q parameters as transfer parametersParameters with specific functions that are required in several cycles always have the same number: For example, Q200 is always assigned the setup clearance, Q202 the plunging depth, etc.

Group of Cycles	Soft Key
Cycles for pecking, reaming, boring and tapping	DRILLING
Cycles for milling pockets, studs and slots	POCKETS/ ISLANDS
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Cycles for producing hole patterns, such as circular or linear patterns	PATTERN
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
Special cycles such as dwell time, program call and oriented spindle stop	SPECIAL CYCLES

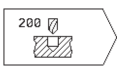
Defining a cycle

CYCL DEF

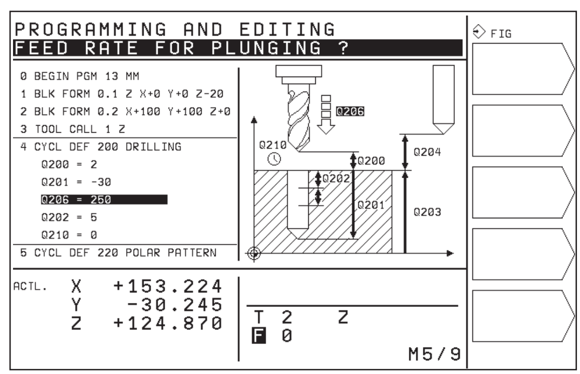
- The soft-key row shows the available groups of cycles

DRILLING

- Press the soft key for the desired group of cycles, for example DRILLING for the drilling cycles



- Select a cycle, e.g. DRILLING The TNC initiates the programming dialog and asks all required input values. At the same time a graphic of the input parameters is displayed in the right screen window. The parameter that is asked for in the dialog prompt is highlighted. Select the screen layout PROGRAM + HELP GRAPHIC.
- Enter all parameters asked by the TNC and conclude each entry with the ENT key
- The TNC terminates the dialog when all required data have been entered



Resulting NC blocks

```
14  CYCL DEF 200 DRILLING
    Q200=2
    Q201=-40
    Q206=250
    Q202=5
    Q210=0
    Q203=-10
    Q204=20
```

Calling the Cycle

Prerequisites

The following data must always be programmed before a cycle call

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

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

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

- Cycles for circular and linear hole patterns
- Coordinate transformation cycles
- DWELL TIME cycle

All other cycles are called as described below.

If the TNC is to execute the cycle once after the last programmed block, program the cycle call with the miscellaneous function M99 or with CYCL CALL:

CYCL CALL

- ▶ Press the CYCL CALL soft key to program a cycle call
- ▶ Enter a miscellaneous function, for example for coolant supply.

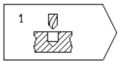
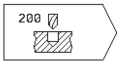
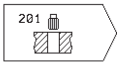
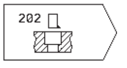
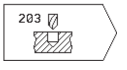
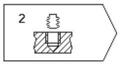
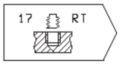
If the TNC is to execute the cycle automatically after every positioning block, program the cycle call with M89 (depending on machine parameter 7440).

To cancel M89, enter

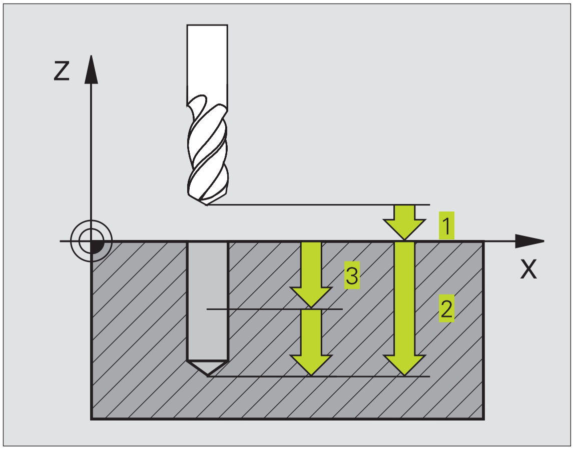
- M99 or
- CYCL CALL or
- CYCL DEF

8.2 Drilling Cycles

The TNC offers 7 cycles for all types of drilling operations:

Cycle	Soft Key
1 PECKING Without automatic pre-positioning	
200 DRILLING With automatic pre-positioning and 2nd set-up clearance	
201 REAMING With automatic pre-positioning and 2nd set-up clearance	
202 BORING With automatic pre-positioning and 2nd set-up clearance	
203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd set-up clearance, chip breaking, and decrement	
2 TAPPING With a floating tap holder	
17 RIGID TAPPING Without a floating tap holder	

PECKING (Cycle 1)



1 The tool drills from the current position to the first PECKING DEPTH at the programmed FEED RATE F.

2 When it reaches the first pecking depth, the tool retracts in rapid traverse FMAX to the starting position and advances again to the first PECKING 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 TOTAL HOLE DEPTH is reached.

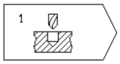
6 After a dwell time at the hole bottom, the tool is returned to the starting position in rapid traverse FMAX for chip breaking.

Before you begin programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

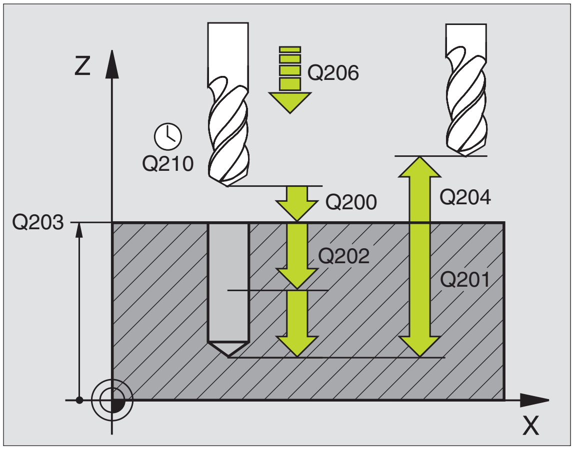
The algebraic sign for the cycle parameter TOTAL HOLE DEPTH determines the working direction.



- ▶ SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- ▶ TOTAL HOLE DEPTH 2 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- ▶ PECKING DEPTH 3 (incremental value): Infeed per cut. The tool will drill to the DEPTH in one movement if:
  - the PECKING DEPTH equals the TOTAL HOLE DEPTH
  - the PECKING DEPTH is greater than the MILLING DEPTH

  The TOTAL HOLE DEPTH does not have to be a multiple of the PECKING DEPTH.
- ▶ DWELL TIME IN SECONDS: Amount of time the tool
  remains at the total hole depth for chip breaking
- ▶ FEED RATE F: Traversing speed of the tool during drilling in mm/min

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 tool is retracted to SET-UP CLEARANCE in FMAX, remains there — if programmed— for the entered dwell time, and advances again in FMAX to 0.2 mm above the first PLUNGING DEPTH.

4 The tool then drills deeper by the PLUNGING DEPTH at the programmed FEED RATE F.

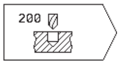
5 The TNC repeats this process (2 to 4) until the programmed TOTAL HOLE DEPTH is reached.

6 At the hole bottom, the tool is retraced to SET-UP CLEARANCE or — if programmed — to the 2ND SET-UP CLEARANCE in rapid traverse FMAX.

Before you begin 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 DEPTH parameter determines the working direction.



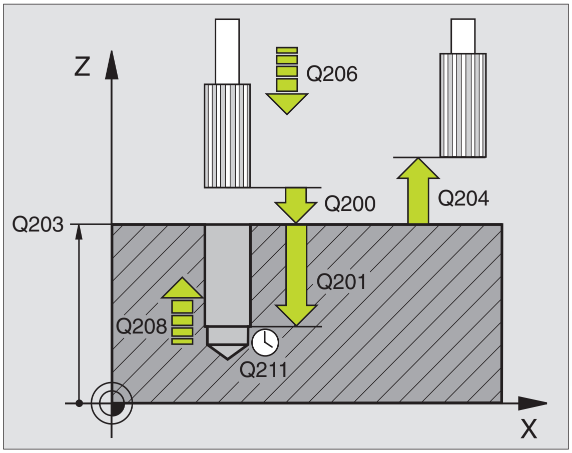
- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.
- DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- FEED RATE FOR PLUNGING Q206: Traversing speed of the tool during drilling in mm/min
- PLUNGING DEPTH Q202 (incremental value): Infeed per cut. The tool will drill to the DEPTH in one movement if:
  - the PLUNGING DEPTH is equal to the DEPTH
  - the PLUNGING DEPTH is greater than the DEPTH

  The DEPTH does not have to be a multiple of the PLUNGING 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): coordinate of 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.

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.

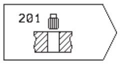
3 If programmed, the tool remains at the hole bottom for the entered dwell time.

4 The tool then retracts to SET-UP CLEARANCE at the FEED RATE F, and from there — if programmed — to the 2ND SET-UP CLEARANCE in FMAX.

Before you begin 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 DEPTH parameter determines the working direction.



▶ 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: Traversingspeed of the tool during reaming in mm/min

▶ DWELL TIME AT DEPTH Q211: Time in seconds that the tool remains at the hole bottom

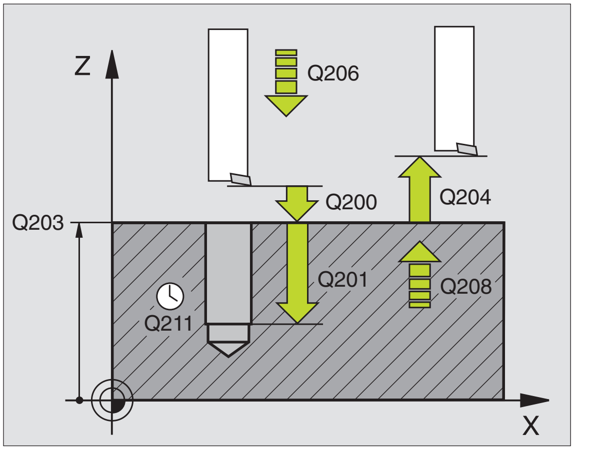
▶ RETRACTION FEED RATE Q208: Traversing speed of the tool in mm/min when retracting from the hole. If you enter Q208 = 0, the tool retracts at the REAMING FEED RATE.

▶ WORKPIECE SURFACE COORDINATE Q203 (absolute): coordinate of 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.

BORING (Cycle 202)

Machine and control must be specially prepared by the machine tool builder to enable Cycle 202.

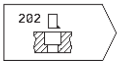


1. The TNC positions the tool in the tool axis at rapid traverse FMAX to SET-UP CLEARANCE above the workpiece surface.
2. The tool drills to the programmed DEPTH at the FEED RATE FOR PLUNGING.
3. If programmed, the tool remains at the hole bottom for the entered dwell time with active spindle rotation for cutting free.
4. The TNC then orients the spindle to the 0° position with an oriented spindle stop.
5. If retraction is selected, the tool retracts in the programmed direction by 0.2 mm (fixed value).
6. The tool then retracts to SET-UP CLEARANCE at the RETRACTION RATE F, and from there — if programmed — to the 2ND SET-UP CLEARANCE in FMAX.

Before you begin 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.



- ▶ 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): coordinate of 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 main axis direction

2: Retract tool in the negative secondary axis direction

3: Retract tool in the positive main axis direction

4: Retract tool in the positive secondary axis direction

Danger of collision

Check the position of the tool tip when you program a spindle orientation to 0° (for example, in the POSITIONING WITH MANUAL DATA INPUT mode of operation). Align the tool tip so that it is parallel to a coordinate axis. Select a DISENGAGING DIRECTION in which the tool moves away from the edge of the hole.

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 0.2 mm. If you are working without chip breaking, the tool retracts at the RETRACTION FEED RATE to SET-UP CLEARANCE, remains there — if programmed — for the entered dwell time, and advances again in FMAX to 0.2 mm 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.

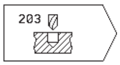
5 The TNC repeats this process (2 to 4) until the programmed TOTAL HOLE DEPTH is reached.

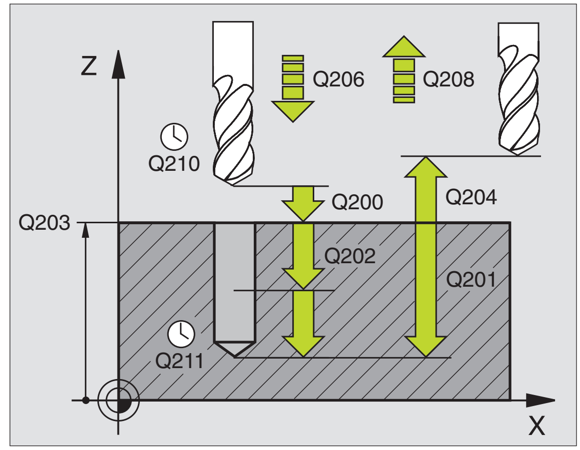
6 The tool remains at the hole bottom — if programmed — for the entered DWELL TIME to cut free, and then retracts to SET-UP CLEARANCE at the RETRACTION FEED RATE. If you have entered a 2ND SET-UP CLEARANCE, the tool subsequently moves to that position in FMAX.

Before you begin 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.





- 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 tool will drill to the DEPTH in one movement if:
  - the PECKING DEPTH equals the DEPTH
  - the PECKING DEPTH is greater than the DEPTH

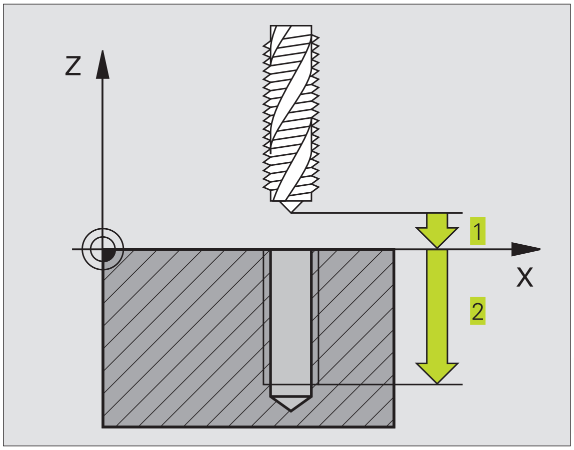
  The DEPTH does not have to be a multiple of the PLUNGING DEPTH.
- DWELL TIME AT TOP Q210: Time in seconds that the tool remains at SET-UP CLEARANCE after having been retracted from the hole for chip breaking.
- WORKPIECE SURFACE COORDINATE Q203 (absolu-te): coordinate of workpiece surface
- 2ND SET-UP 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 after each infeed.
- NR 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 0.2 mm.
- 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 tool retracts in FMAX.

TAPPING with a floating tap holder (Cycle 2)

1 The thread is cut in one pass.

2 Once the tool has reached the TOTAL HOLE DEPTH, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the DWELL TIME.

3 At the starting position, the direction of spindle rotation reverses once again.



Before you begin programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

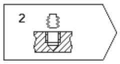
Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the DEPTH parameter determines the working direction.

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).

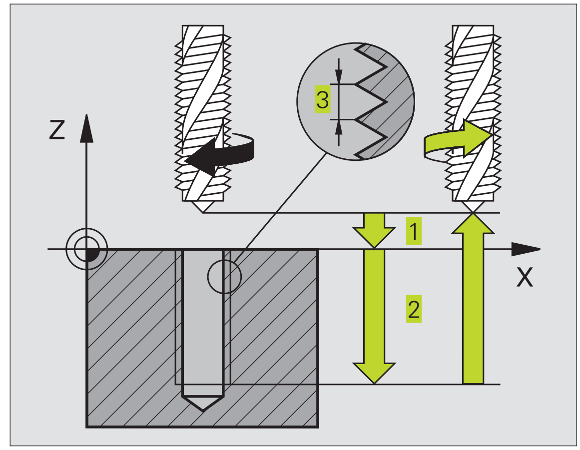
For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.



- SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface. Standard value: approx. 4 times the thread pitch
- TOTAL HOLE DEPTH 2 (thread length, incremental value): Distance between workpiece surface and end of thread
- DWELL TIME IN SECONDS: Enter a value between 0 and 0.5 seconds to avoid wedging of the tool during retraction.
- FEED RATE F: Traversing speed of the tool during tapping

  The feed rate is calculated as follows: F = S x p, where
  F is the feed rate in mm/min),
  S is the spindle speed in rpm,
  and p is the thread pitch in mm

RIGID TAPPING (Cycle 17)



Machine and control must be specially prepared by the machine tool builder to enable rigid tapping.

The TNC cuts the thread without a floating tap holder in one or more passes.

Rigid tapping offers the following advantages over tapping with a floating tap holder

- Higher machining speeds possible
- Repeated tapping of the same thread is possible; repetitions are enabled via spindle orientation to the 0° position during cycle call (depending on machine parameter 7160).
- Increased traverse range of the spindle axis due to absence of a floating tap holder.

Before you begin programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the parameter TOTAL HOLE DEPTH determines the working direction.

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.

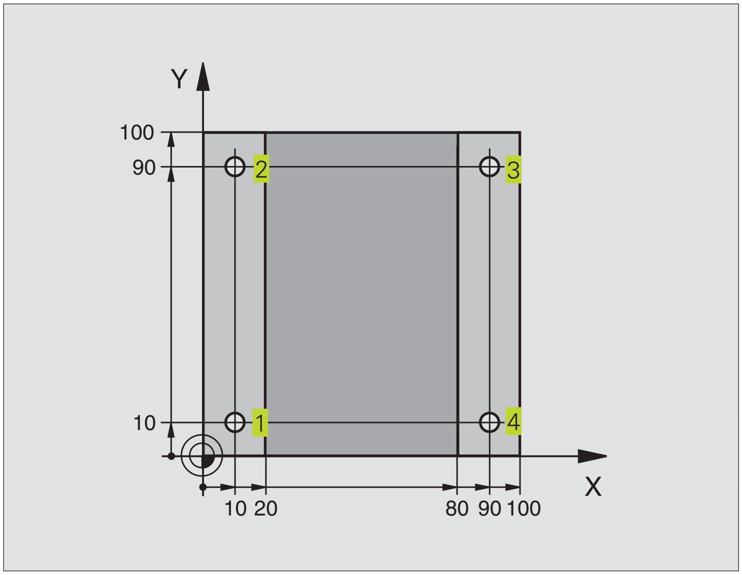
The feed-rate override knob is disabled.

At the end of the cycle the spindle comes to a stop. Before the next operation, restart the spindle with M3 (or M4).

17 RT

- ▶ SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- ▶ TOTAL HOLE DEPTH 2 (incremental value): Distance between workpiece surface (beginning of thread) and end of thread
- ▶ PITCH 3 :
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread
  – = left-hand thread

Example: Drilling cycles

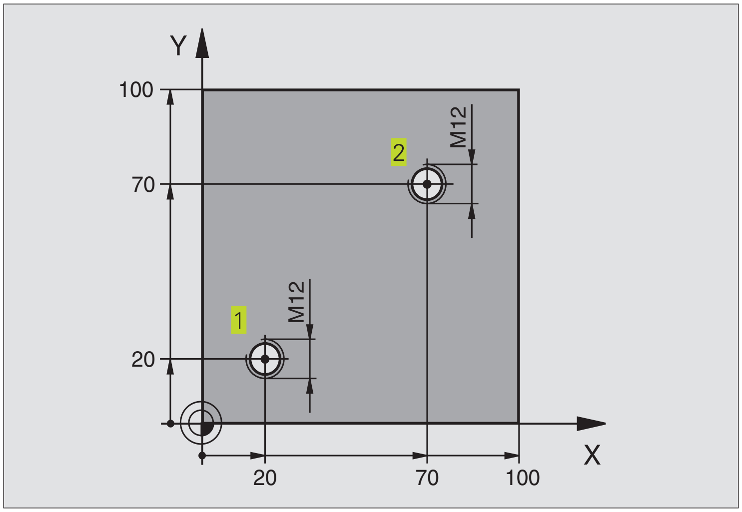


`0 BEGIN PGM 200 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the blank form
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+3`	Tool definition
`4 TOOL CALL 1 Z S4500`	Call the tool
`5 L Z+250 R0 F MAX`	Retract the tool
`6 CYCL DEF 200 DRILLING`	Define cycle
`Q200=2`	Setup clearance
`Q201=-15`	Depth
`Q206=250`	Feed rate for boring
`Q202=5`	Pecking
`Q210=0`	Dwell time above
`Q203=-10`	Coordinate surface
`Q204=20`	2nd set-up clearance
`7 L X+10 Y+10 R0 F MAX M3`	Approach hole 1, spindle ON
`8 CYCL CALL`	Cycle call
`9 L Y+90 R0 F MAX M99`	Approach hole 2, call cycle
`10 L X+90 R0 F MAX M99`	Approach hole 3, call cycle
`11 L Y+10 R0 F MAX M99`	Approach hole 4, call cycle
`12 L Z+250 R0 F MAX M2`	Retract in the tool axis, end of program
`13 END PGM 200 MM`	

Example: Drilling cycles

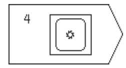
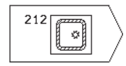
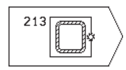
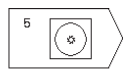
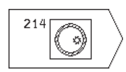
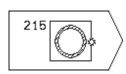
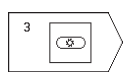
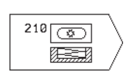
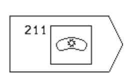
Process

- Plate has already been pilot drilled for M12, depth of the plate: 20 mm
- Program tapping cycle
- For safety reasons, pre-positioning should be done first of all in the main plane and then in the spindle axis

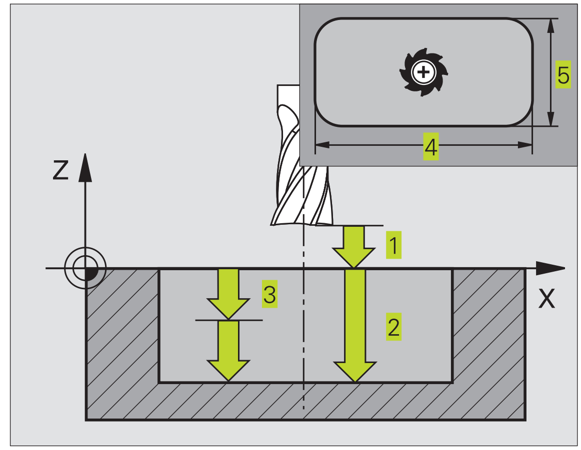


`0 BEGIN PGM 2 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the blank form
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+4.5`	Tool definition
`4 TOOL CALL 1 Z S100`	Call the tool
`5 L Z+250 R0 FMAX`	Retract the tool
`6 CYCL DEF 2 .0 TAPPING`	Cycle definition for tapping
`7 CYCL DEF 2 .1 SET UP 2`	
`8 CYCL DEF 2 .2 DEPTH -25`	
`9 CYCL DEF 2 .3 DWELL 0`	
`10 CYCL DEF 2 .4 F175`	
`11 L X+20 Y+20 R0 FMAX M3`	Approach hole 1 in the machining plane
`12 L Z+2 R0 FMAX M99`	Pre-position in the tool axis
`13 L X+70 Y+70 R0 FMAX M99`	Approach hole 2 in the machining plane
`14 L Z+250 R0 FMAX M2`	Retract in the tool axis, end of program
`15 END PGM 2 MM`	

8.3 Cycles for Milling Pockets, Studs and Slots

Cycle	Soft Key
4 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning	
212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
5 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning	
214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
3 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical downfeed	
210 SLOT WITH RECIPROCATING PLUNGE-CUT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	
211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	

POCKET MILLING (Cycle 4)



1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first PECKING 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 3) is repeated until the DEPTH is reached.

4 At the end of the cycle, the TNC retracts the tool to the starting position.

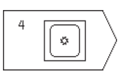
Before you begin programming, note the following:

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

The algebraic sign for the DEPTH parameter determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

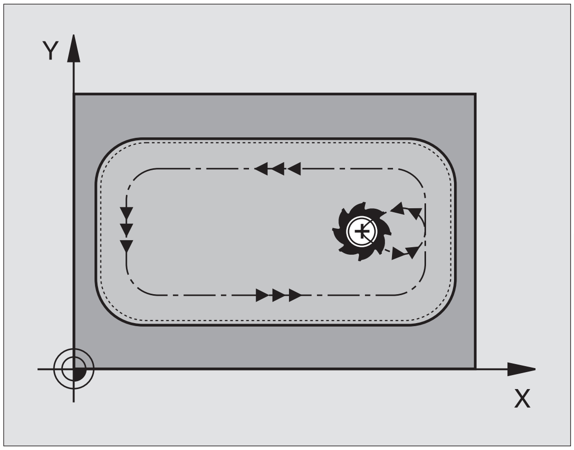


- SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- MILLING DEPTH 2 (incremental value): Distance between workpiece surface and bottom of pocket
- PECKING DEPTH 3 (incremental value): Infeed per cut. The tool will advance to the DEPTH in one movement if:
  - the PECKING DEPTH equals the DEPTH
  - the PECKING DEPTH is greater than the DEPTH
- FEED RATE FOR PECKING: Traversing speed of the tool during penetration
- FIRST SIDE LENGTH 4: Pocket length, parallel to the main axis of the working plane
- SECOND SIDE LENGTH 5: Pocket width
- FEED RATE F: Traversing speed of the tool in the working plane

- DIRECTION OF THE MILLING PATH
  DR **+** : climb milling with M3
  DR **–** : up-cut milling with M3
- Rounding radius: Radius of 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

where
K is the overlap factor, preset in machine parameter 7430, and
R: is the cutter radius



POCKET FINISHING (Cycle 212)

1 The TNC automatically moves the tool in the tool axis to SET-UP CLEARANCE, or, if programmed, to the 2nd SET-UP CLEARANCE and then 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 plunge-cuts into the pocket center.

3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PLUNGING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

5 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.

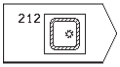
7 At the end of the cycle, the TNC retracts the tool in rapid traverse to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and finally to the center of the pocket (end position = starting position).

Before you begin programming, note the following:

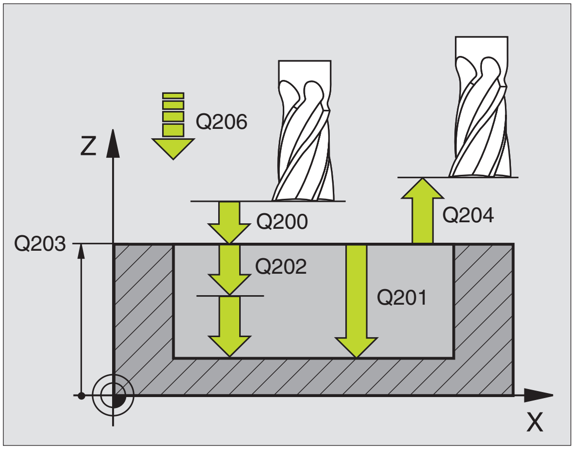
The algebraic sign for the DEPTH parameter determines the working direction.

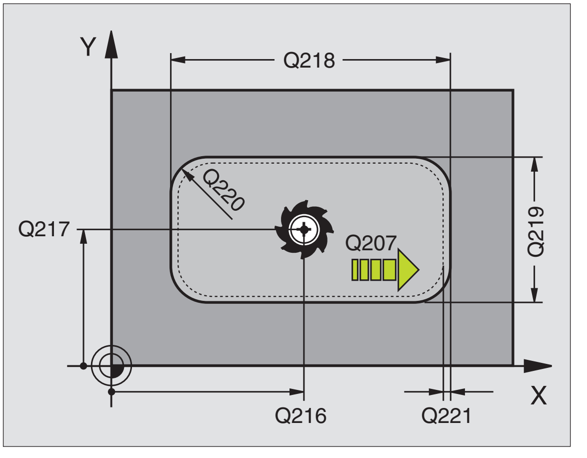
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.

Minimum size of the pocket: 3 times the tool radius.

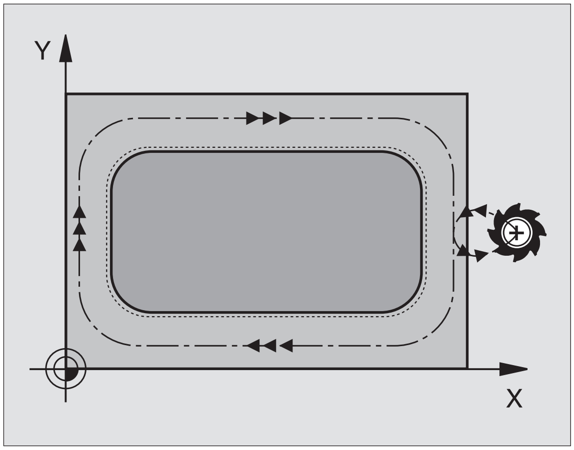


- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.
- DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of pocket
- FEED RATE FOR PLUNGING Q206: Traversing speed of the tool in mm/min when moving to DEPTH. If you are plunge-cutting into the material, enter a low value; if you have already cleared the pocket, 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): coordinate of 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 main axis of the working plane
- CENTER IN 2ND AXIS Q217 (absolute value): Center of the pocket in the secondary 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 secondary 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 in the main axis of the working plane referenced to the length of the pocket. This value is only required by the TNC for calculating the preparatory position.





STUD FINISHING (Cycle 213)



1 The TNC moves the tool in the tool axis to SET-UP CLEARANCE, or — if programmed — to the 2ND SET-UP CLEARANCE, and subsequently to the center of the stud.

2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3.5 times the tool radius.

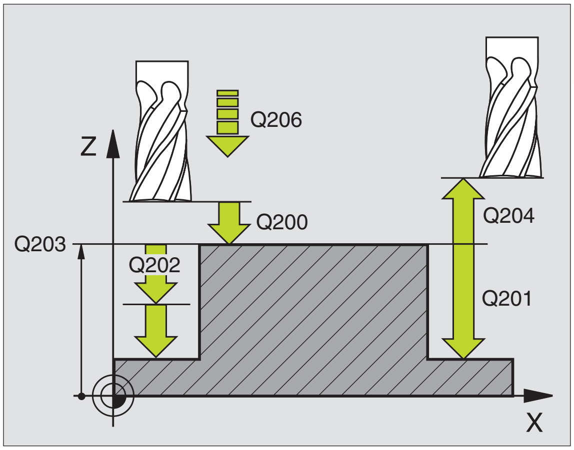
3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PLUNGING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

5 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 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 you begin programming, note the following:

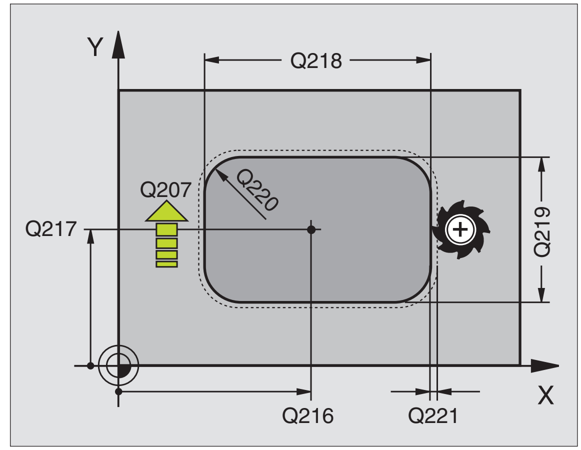
The algebraic sign for the DEPTH parameter determines the working direction.

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.

213

- 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. The tool will drill to the DEPTH in one movement if: 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): coordinate of 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 secondary axis of the working plane
- FIRST SIDE LENGTH Q218 (incremental value): Stud length, parallel to the reference axis of the working plane
- SECOND SIDE LENGTH Q219 (incremental value): Stud length, 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 in the main axis of the working plane referenced to the length of the stud. This value is only required by the TNC for calculating the preparatory position.



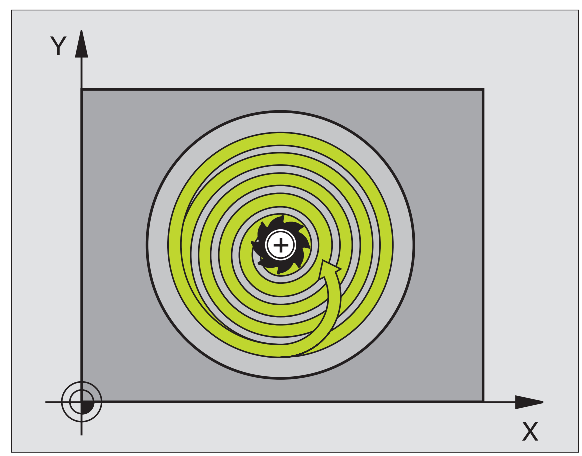
CIRCULAR POCKET MILLING (Cycle 5)

1 The tool penetrates the workpiece at the starting position (pocket center) and advances to the first PECKING DEPTH.

2 The tool subsequently follows a spiral path at the FEED RATE F — see figure at right. For calculating the stepover factor k, see Cycle 4 POCKET MILLING.

3 This process is repeated until the DEPTH is reached.

4 At the end of the cycle, the TNC retracts the tool to the starting position.



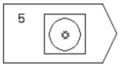
Before you begin programming, note the following:

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION R0.

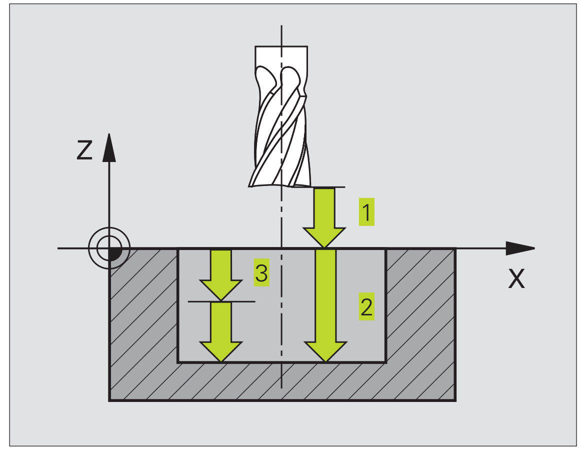
Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

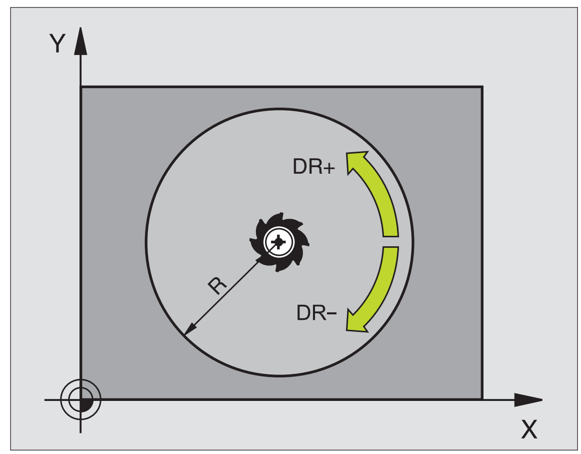
The algebraic sign for the DEPTH parameter determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.



- SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- MILLING DEPTH 2 (incremental value): Distance between workpiece surface and bottom of pocket
- PECKING DEPTH 3 (incremental value): Infeed per cut. The tool will advance to the DEPTH in one movement if:
  n the PECKING DEPTH equals the DEPTH
  n the PECKING DEPTH is greater than the DEPTH
- FEED RATE FOR PECKING: Traversing speed of the tool during penetration
- CIRCLE RADIUS: Radius of the circular pocket
- FEED RATE F: Traversing speed of the tool in the working plane
- DIRECTION OF THE MILLING PATH
  DR **+** : climb milling with M3
  DR **–** : up-cut milling with M3





CIRCULAR POCKET FINISHING (Cycle 214)

1 The TNC automatically moves the tool in the tool axis to SET-UP CLEARANCE, or, if programmed, to the 2nd SET-UP CLEARANCE and then 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.

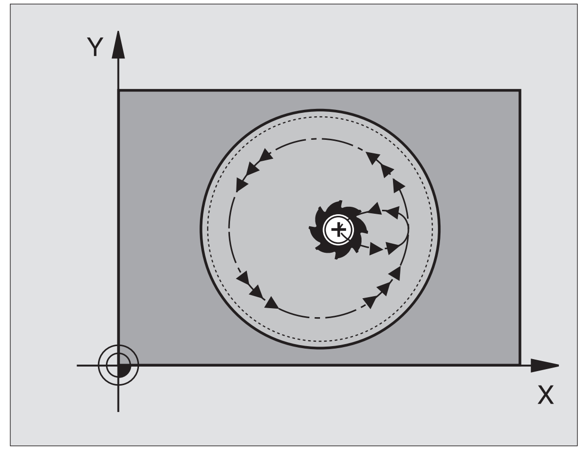
3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PLUNGING DEPTH at the FEED RATE FOR PLUNGING.

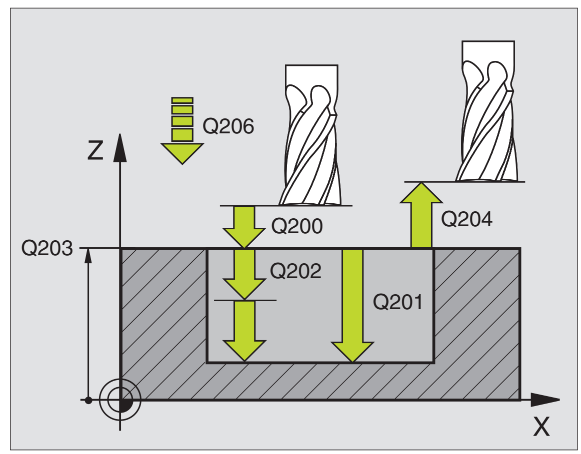
4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

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

6 This process (4 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 pocket (end position = starting position).





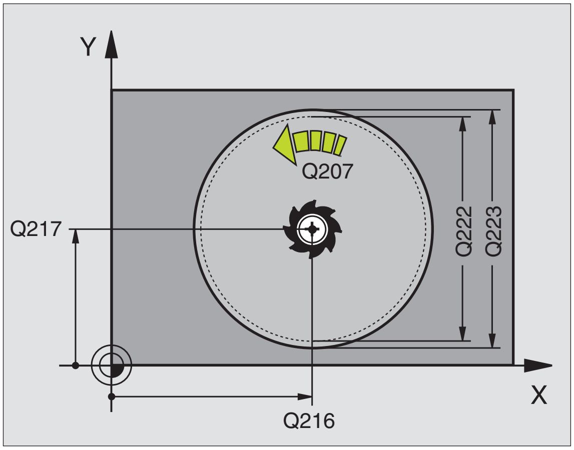
Before you begin programming, note the following:

The algebraic sign for the DEPTH parameter determines the working direction.

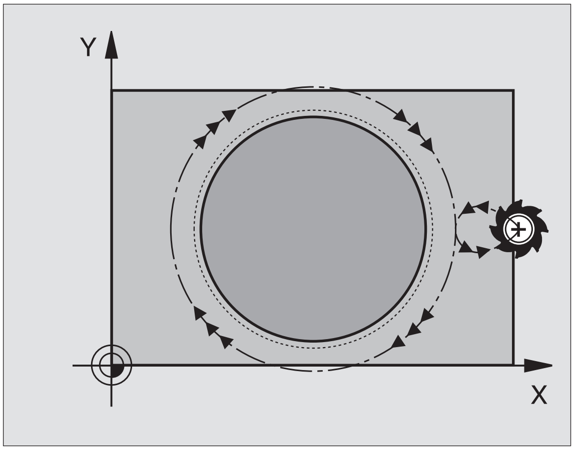
If you want to clear and finish the pocket with the same tool, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.

- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.
- DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of pocket
- FEED RATE Q206 FOR PLUNGING: 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
- FEED RATE FOR MILLING Q207: Traversing speed of the tool in mm/min while milling.

- WORKPIECE SURFACE COORDINATE Q203 (absolute): coordinate of 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 main axis of the working plane
- CENTER IN 2ND AXIS Q217 (absolute value): Center of the pocket in the secondary axis of the working plane
- WORKPIECE BLANK DIA. Q222: Diameter of the premachined pocket. Enter the workpiece blank diameter to be less than the diameter of the finished part. If you enter
  Q222 = 0, then the TNC plunge-cuts into the pocket center.
- FINISHED PART DIA. Q223: Diameter of the finished pocket. Enter the diameter of the finished part to be greater than the workpiece blank diameter.



CIRCULAR STUD FINISHING (Cycle 215)



1 The TNC automatically moves the tool in the tool axis to SET-UP CLEARANCE, or, if programmed, to the
2nd SET-UP CLEARANCE and then to the center of the stud

2 From the stud center, the tool moves in the working plane to the starting point for machining. The starting point lies to the right of the stud by a distance approx. 3.5 times the tool radius.

3 If the tool is at the 2ND SET-UP CLEARANCE, it moves in rapid traverse FMAX to SET-UP CLEARANCE, and from there advances to the first PLUNGING DEPTH at the FEED RATE FOR PLUNGING.

4 The tool then moves tangentially to the contour of the finished part and, using climb milling, machines one revolution.

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

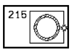
6 This process (4 to 5) is repeated until the programmed DEPTH is reached.

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 then to the center of the pocket (end position = starting position)

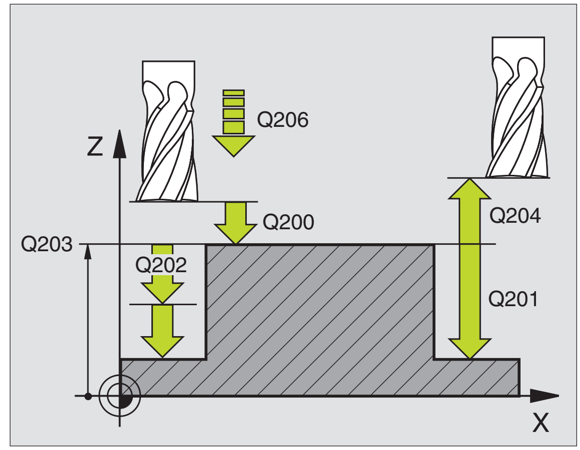
Before you begin programming, note the following:

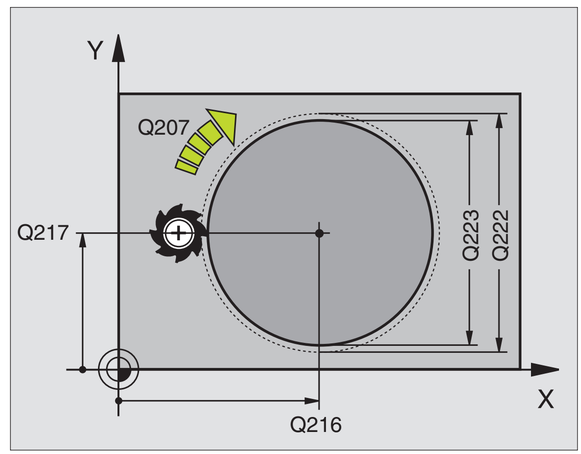
The algebraic sign for the DEPTH parameter determines the working direction.

If you want to clear and finish the stud with the same tool, use a center-cut end mill (ISO 1641) and enter a low FEED RATE FOR PLUNGING.



- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.
- DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of stud
- FEED RATE Q206 FOR PLUNGING: 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): coordinate of 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 secondary axis of the working plane
- WORKPIECE BLANK DIAMETER Q222: Diameter of the premachined stud. 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.





SLOT MILLING (Cycle 3)

Roughing process

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

2 After downfeed at the end of the slot, milling is performed in the opposite direction.

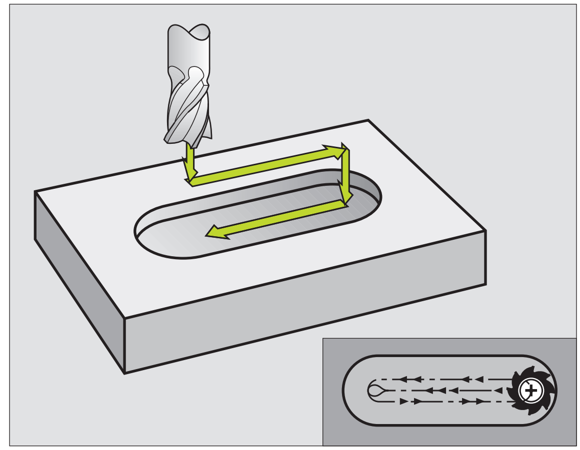
This process is repeated until the programmed MILLING DEPTH is reached.

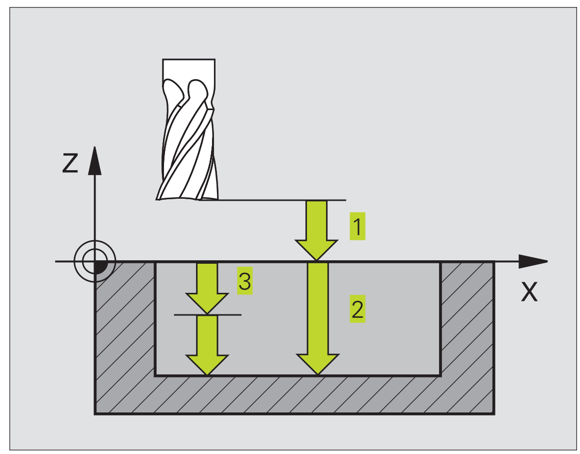
Finishing process

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

4 At the end of the cycle, the tool is retracted in rapid traverse FMAX to SET-UP CLEARANCE.

If the number of infeeds was odd, the tool returns to the starting position at the level of the SET-UP CLEARANCE.





Before you begin programming, note the following:

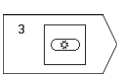
Program a positioning block for the starting point in the working plane — to the center of the slot (SECOND SIDE LENGTH) and, within the slot, offset by the tool radius — with RADIUS COMPENSATION R0.

Program a positioning block for the starting point in the tool axis (SET-UP CLEARANCE above the workpiece surface).

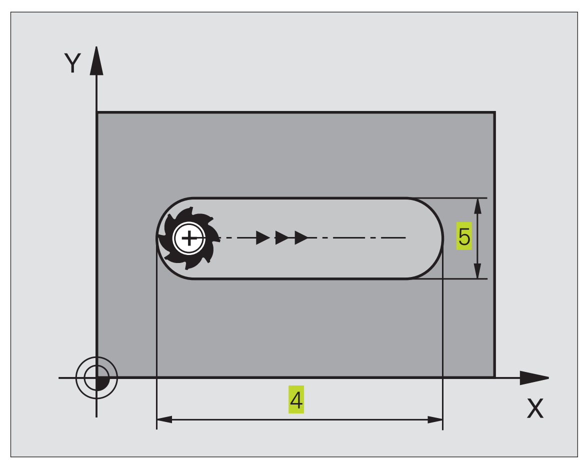
The algebraic sign for the DEPTH parameter determines the working direction.

This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the starting point.

The cutter diameter must be not be larger than the SLOT WIDTH and not smaller than half the SLOT WIDTH.

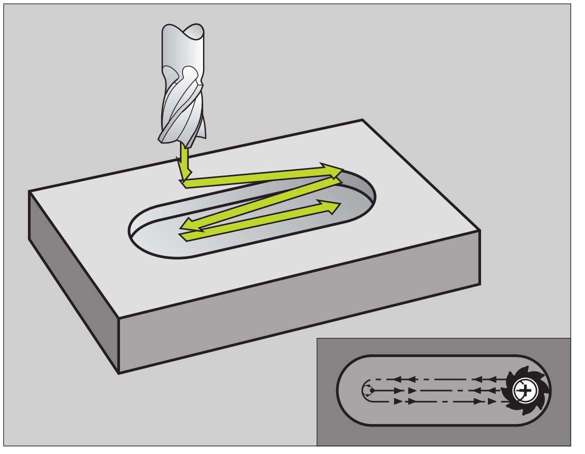


- SET-UP CLEARANCE 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- MILLING DEPTH 2 (incremental value): Distance between workpiece surface and bottom of pocket
- PECKING DEPTH 3 (incremental value): Infeed per cut. The tool will advance to the DEPTH in one operation if:
  - the PECKING DEPTH equals the DEPTH
  - the PECKING DEPTH is greater than the DEPTH



- FEED RATE FOR PECKING: Traversing speed of the tool during penetration
- FIRST SIDE LENGTH 4: Slot length; specify the sign to determine the first milling direction
- SECOND SIDE LENGTH 5: Slot width
- FEED RATE F: Traversing speed of the tool in the working plane

SLOT with reciprocating plunge-cut (Cycle 210)



Before you begin programming, note the following:

The algebraic sign for the DEPTH parameter determines the working direction.

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.

Roughing process

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 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.

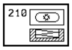
4 At the MILLING DEPTH, the TNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.

Finishing process

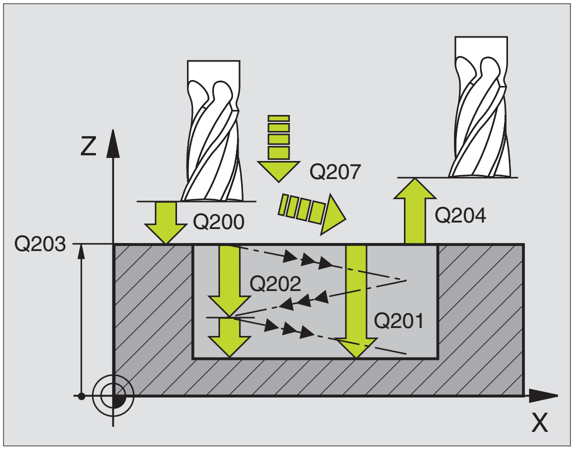
5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3).

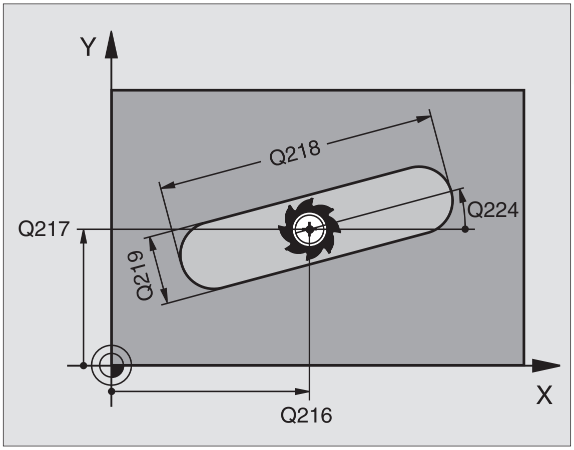
6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.

7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to SET-UP CLEARANCE and, if programmed, to the 2nd SET-UP CLEARANCE

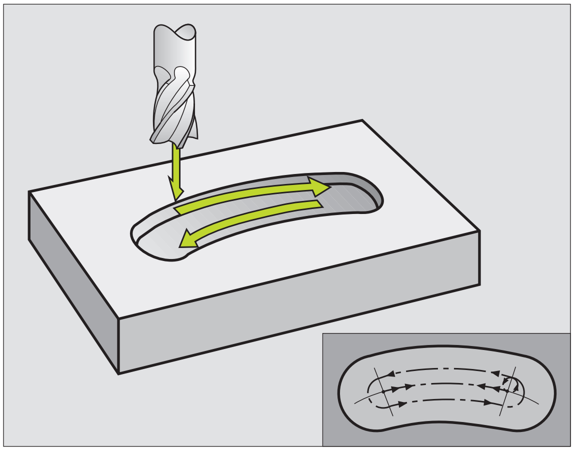


- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.
- DEPTH Q201 (incremental value): Distance between workpiece surface and bottom of slot
- FEED RATE FOR MILLING Q207: Traversing speed of the tool in mm/min while milling.
- PLUNGING DEPTH Q202 (incremental 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 extent of machining:
  0: Roughing and finishing
  1: Roughing only
  2: Finishing only
- 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 secondary axis of the working plane
- FIRST SIDE LENGTH Q218 (value parallel to the main 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.





CIRCULAR SLOT with reciprocating plunge-cut (Cycle 211)



Roughing process

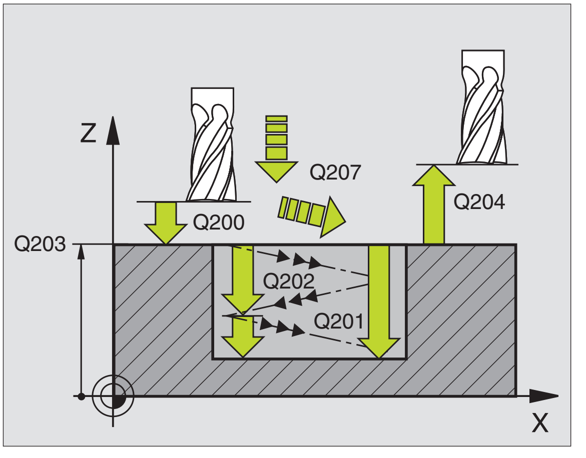
1 At rapid traverse, the TNC positions the tool in the tool axis to the 2ND SET-UP CLEARANCE and subsequently to the center of the right circle.From there, the tool is positioned to the programmed SET-UP CLEARANCE above the workpiece surface.

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 (2 to 3) 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.

Finishing process



5 For finishing the slot, the TNC advances the tool tangentially to the contour of the finished part. The tool subsequently climb-mills the contour (with M3). The starting point for the finishing process is the center of the right circle.

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

7 At the end of the cycle, the tool is retracted in rapid traverse FMAX to SET-UP CLEARANCE and, if programmed, to the 2nd SET-UP CLEARANCE

Before you begin programming, note the following:

The algebraic sign for the DEPTH parameter determines the working direction.

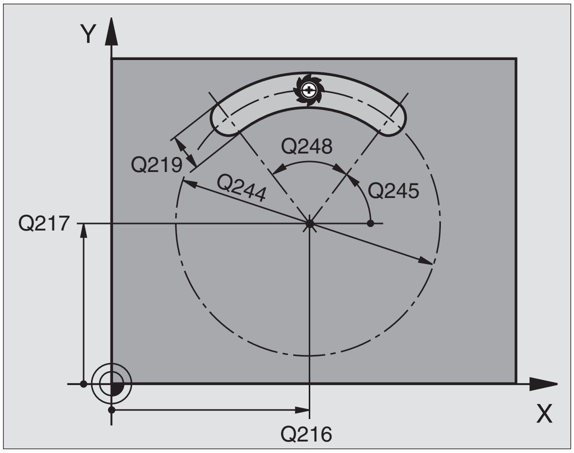
The cutter diameter must not be larger than the slot width and not smaller than a third of the slot width.

The cutter diameter must be smaller than half the slot length. The TNC otherwise cannot execute this cycle.

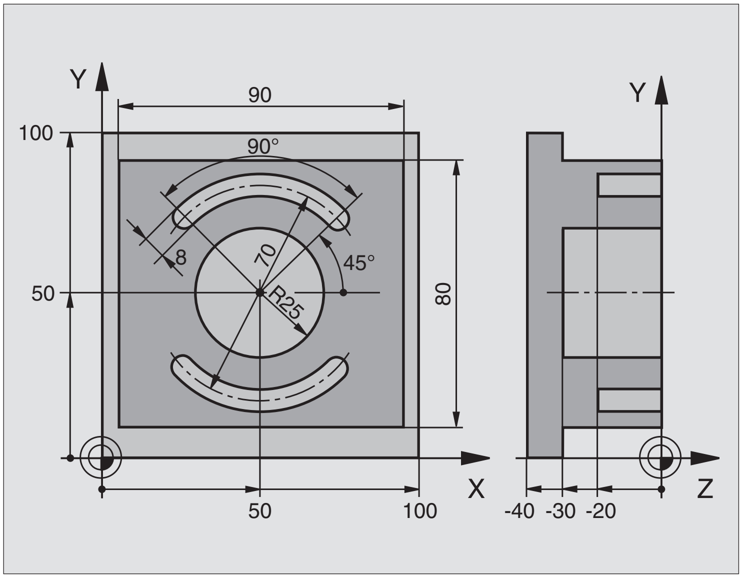
211

- ▶ 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 extent of machining:
  0: Roughing and finishing
  1: Roughing only
  2: Finishing only
- 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 secondary 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



Example: Milling pockets, studs and slots

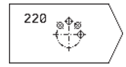
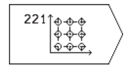


0 BEGIN PGM 210 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-40	Define the blank form
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+6	Define the tool for roughing/finishing
4 TOOL DEF 2 L+0 R+3	Define slotting mill
5 TOOL CALL 1 Z S3500	Call the tool for roughing/finishing
6 L Z+250 R0 F MAX	Retract the tool
7 CYCL DEF 213 STUD FINISHING	Define cycle for machining the contour outside
Q200=2	Setup clearance
Q201=-30	Depth
Q206=250	Feed rate for plunging
Q202=5	Plunging depth
Q207=250	Feed rate for milling
Q203=+0	Coordinate surface
Q204=20	2nd set-up clearance
Q216=+50	Center in X axis
Q217=+50	Center in Y axis
Q218=90	1st side length
Q219=80	2nd side length
Q220=0	Rounding-off radius
Q221=5	Oversize

8 CYCL CALL M3	Cycle call for stud
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	Define cycle for slot 1
Q200=2	Setup clearance
Q201=-20	Depth
Q207=250	Feed rate for plunging
Q202=5	Plunging depth
Q215=0	Machining operation
Q203=+0	Coordinate surface
Q204=100	2nd set-up clearance
Q216=+50	Center in X axis
Q217=+50	Center in Y axis
Q244=70	Pitch circle diameter
Q219=8	2nd side length
Q245=+45	Starting angle
Q248=90	Angular length
19 CYCL CALL M3	Call cycle for slot 1
20 CYCL DEF 211 CIRCULAR SLOT	Cycle definition for slot 2
Q200=2	Setup clearance
Q201=-20	Depth
Q207=250	Feed rate for plunging
Q202=5	Plunging depth
Q215=0	Machining operation
Q203=+0	Coordinate surface
Q204=100	2nd set-up clearance
Q216=+50	Center in X axis
Q217=+50	Center in Y axis
Q244=70	Pitch circle diameter
Q219=8	2nd side length
Q245=+225	**New starting angle**
Q248=90	Angular length
21 CYCL CALL	Call cycle for slot 2
22 L Z+250 R0 F MAX M2	Retract in the tool axis, end of program
23 END PGM 210 MM	

8.4 Cycles for Machining Hole Patterns

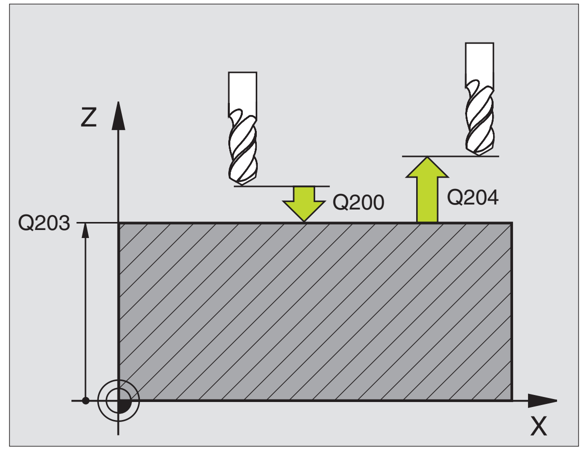
The TNC provides two cycles for machining hole patterns:

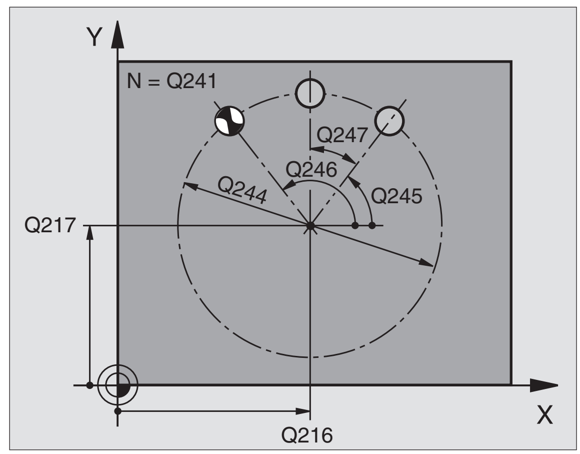
Cycle	Soft Key
220 CIRCULAR PATTERN	
221 LINEAR PATTERN	

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

Cycle 1	PECKING
Cycle 2	TAPPING with a floating tap holder
Cycle 3	SLOT MILLING
Cycle 4	POCKET MILLING
Cycle 5	CIRCULAR POCKET MILLING
Cycle 17	RIGID TAPPING
Cycle 200	DRILLING
Cycle 201	REAMING
Cycle 202	BORING
Cycle 203	UNIVERSAL MILLING CYCLE
Cycle 212	POCKET FINISHING
Cycle 213	STUD FINISHING
Cycle 214	CIRCULAR POCKET FINISHING
Cycle 215	CIRCULAR STUD FINISHING

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.

The tool is positioned in the following sequence:
- Move to 2ND SET-UP CLEARANCE (tool axis)
- Approach starting point in the working plane
- Move to SET-UP CLEARANCE above the workpiece surface (tool axis)

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

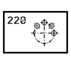
3 The tool then approaches the starting point for the next machining operation on a straight line at SET-UP CLEARANCE (or 2ND SET-UP CLEARANCE).

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

Before you begin programming, note the following:

Cycle 220 is DEF active, which means that Cycle 220 automatically calls the last defined fixed cycle.

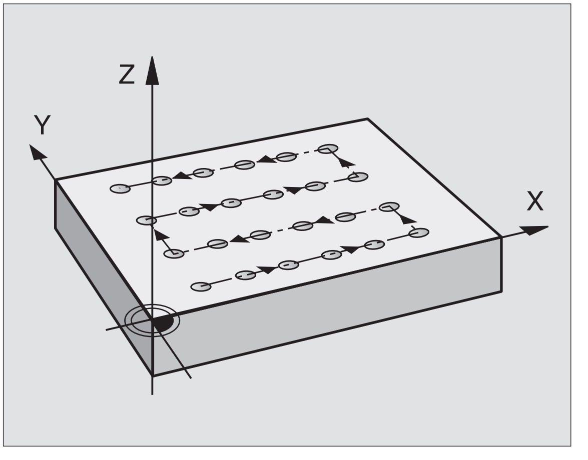
If you combine Cycle 220 with one of the fixed cycles 200 to 204 and 212 to 215, the SET-UP CLEARANCE, workpiece surface and 2ND SET-UP CLEARANCE that you defined in Cycle 220 will be effective for the selected fixed cycle.



- ▶ CENTER IN 1ST AXIS Q216 (absolute value): Center of the pitch circle in the main axis of the working plane
- ▶ CENTER IN 2ND AXIS Q217 (absolute value): Center of the pitch circle in the secondary axis of the working plane
- ▶ PITCH CIRCLE DIAMETER Q244: Diameter of the pitch circle
- ▶ STARTING ANGLE Q245 (absolute value): Angle between the main 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 main axis of the working plane and the starting point for the last machining operation on the pitch circle. 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 a STEPPING ANGLE of 0, the TNC will calculate the STEPPING ANGLE from the STARTING and STOPPING ANGLES. If you enter a value other than 0, the TNC will not take the STOPPING ANGLE into account. The sign for the STEPPING ANGLE 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): coordinate of 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.

LINEAR PATTERN (Cycle 221)



Before you begin programming, note the following:

Cycle 221 is DEF active, which means that Cycle 221 calls the last defined fixed cycle.

If you combine Cycle 221 with one of the fixed cycles 200 to 215, the SET-UP CLEARANCE, workpiece surface and 2ND SET-UP CLEARANCE that you defined in Cycle 221 will be effective for the selected fixed cycle.

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

The tool is positioned in the following sequence:
- Move to 2ND SET-UP CLEARANCE (tool axis)
- Approach starting point in the working plane
- Move to SET-UP CLEARANCE above the workpiece surface (tool 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 main axis direction at SET-UP CLEARANCE (or 2ND SET-UP CLEARANCE).

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

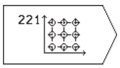
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

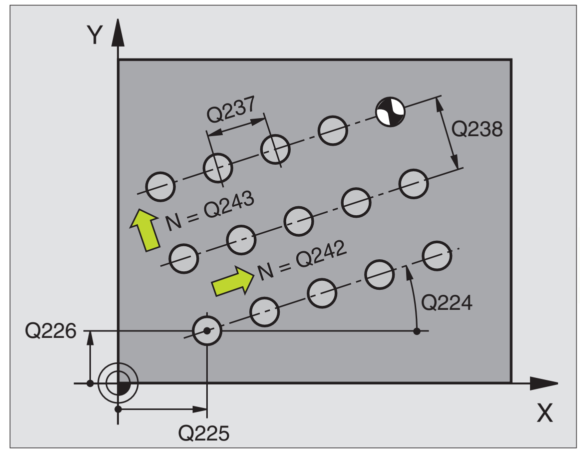
6 From this position, the tool approaches the starting point for the next machining operation in the negative main axis direction.

7 This process (5 to 6) is repeated until all machining operations in the second line have been executed.

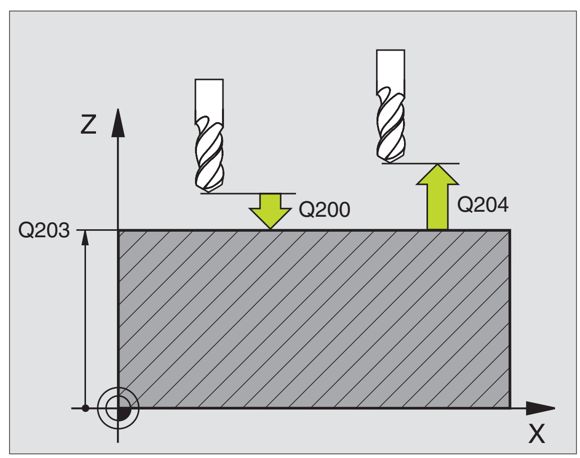
8 The tool then moves to the starting point of the next line.

9 All subsequent lines are processed in a reciprocating movement.

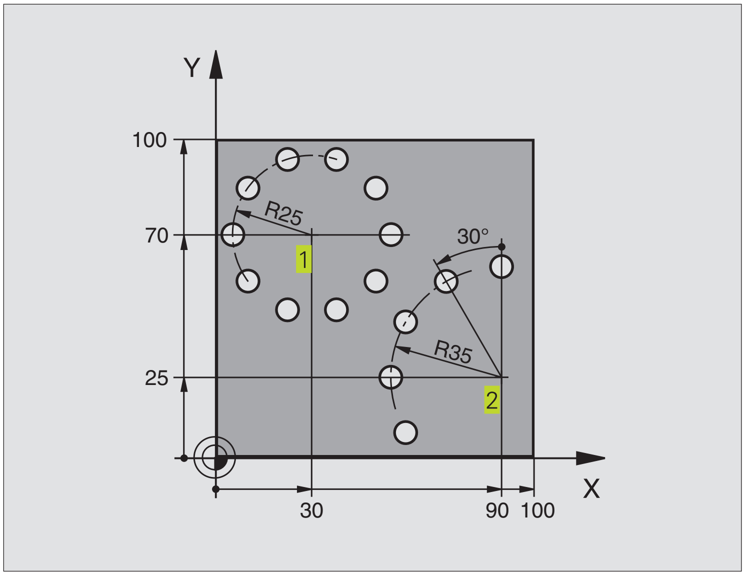




- STARTING POINT 1ST AXIS Q225 (absolute value): Coordinate of the starting point in the main axis of the working plane
- STARTING POINT 2ND AXIS Q226 (absolute value): Coordinate of the starting point in the secondary axis of the working plane
- SPACING IN 1ST AXIS Q237 (incremental value): Spacing between the individual points on a line
- SPACING IN 2ND AXIS Q238 (incremental): Spacing between the individual lines
- NUMBER OF COLUMNS Q242: Number of machining operations on a line
- NUMBER OF LINES Q243: Number of passes
- ANGLE OF ROTATION Q224 (absolute value): Angle by which the entire pattern is rotated. The center of rotation lies in the starting point.
- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and workpiece surface.
- WORKPIECE SURFACE COORDINATE Q203 (absolute): coordinate of 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: Circular hole patterns



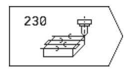
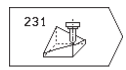
`0 BEGIN PGM 3589M`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	Define the blank form
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+3`	Tool definition
`4 TOOL CALL 1 Z S3500`	Call the tool
`5 L Z+250 R0 F MAX M3`	Retract the tool
`6 CYCL DEF 200 DRILLING`	Cycle definition: drilling
`Q200=2`	Setup clearance
`Q201=-15`	Depth
`Q206=250`	Feed rate for boring
`Q202=4`	Plunging depth
`Q210=0`	Dwell time above
`Q203=+0`	Coordinate surface
`Q204=0`	2nd set-up clearance

7 CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 1, CYCL 200 is called automatically,
	Q200, Q203 and Q204 are effective as defined in Cycle 220
Q216=+30	Center in X axis
Q217=+70	Center in Y axis
Q244=50	Pitch circle diameter
Q245=+0	Starting angle
Q246=+360	Finishing angle
Q247=+0	Angle increment
Q241=10	Machining operations
Q200=2	Setup clearance
Q203=+0	Coordinate surface
Q204=100	2nd set-up clearance
8 CYCL DEF 220 POLAR PATTERN	Define cycle for circular pattern 2, CYCL 200 is called automatically,
	Q200, Q203 and Q204 are effective as defined in Cycle 220
Q216=+90	Center in X axis
Q217=+25	Center in Y axis
Q244=70	Pitch circle diameter
Q245=+90	Starting angle
Q246=+360	Finishing angle
Q247=30	Angle increment
Q241=5	Machining operations
Q200=2	Setup clearance
Q203=+0	Coordinate surface
Q204=100	2nd set-up clearance
9 L Z+250 R0 F MAX M2	Retract in the tool axis, end of program
10 END PGM 3589 MM	

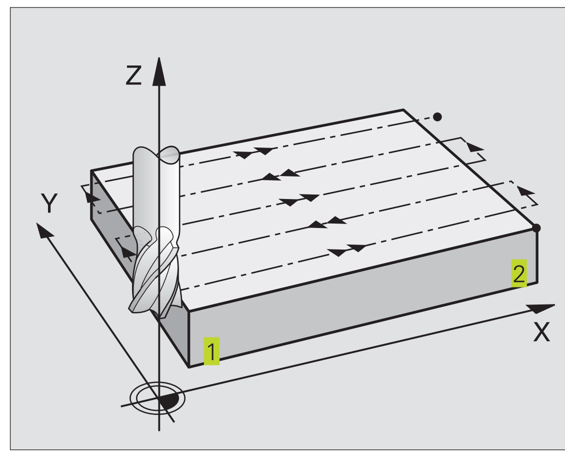
8.5 Cycles for multipass milling

The TNC offers two 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	
231 RULED SURFACE For oblique, inclined or twisted surfaces	

MULTIPASS MILLING (Cycle 230)



1 From the current position, the TNC positions the tool in rapid traverse in the working plane to the starting position. 1 During this movement, the TNC also offsets the tool by its radius to the left and upward.

2 The tool then moves in FMAX in the tool axis to SET-UP CLEARANCE. From there it approaches the programmed starting position in the tool axis at the FEED RATE FOR PLUNGING.

3 The tool subsequently advances to the stopping point 2 at the FEED RATE FOR MILLING. 2 The stopping point is calculated 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 STEPOVER FEED RATE. The offset is calculated from the programmed width and the number of cuts.

5 The tool then returns in the negative X direction

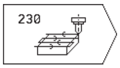
6 Multipass milling is repeated until the programmed surface has been completed.

7 At the end of the cycle, the tool is retracted in FMAX to SET-UP CLEARANCE.

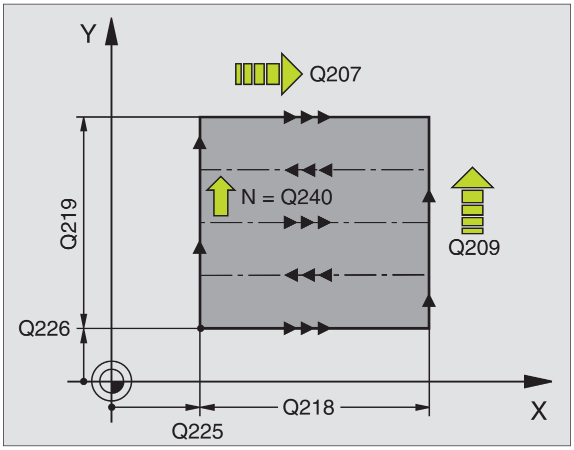
Before you begin programming, note the following:

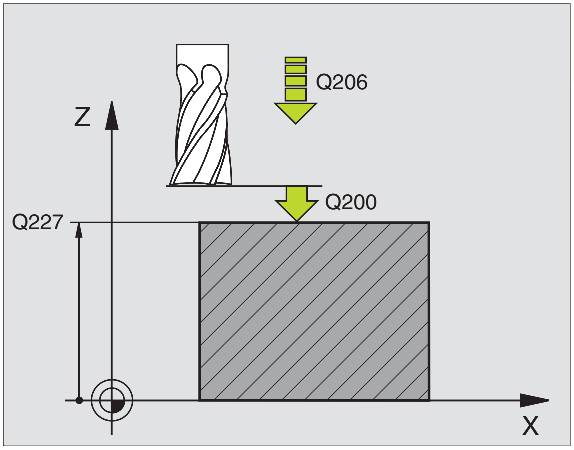
From the current position, the TNC positions the tool at the starting point 1, first in the working plane and then in the tool axis.

Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

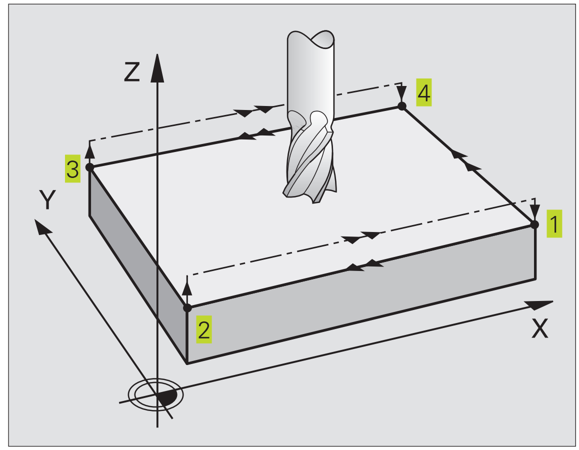


- STARTING POINT IN 1ST AXIS Q225 (absolute value): MIN point coordinate of the surface to be multipass-milled in the main axis of the working plane
- STARTING POINT IN 2ND AXIS Q226 (absolute value): MIN point coordinate of the surface to be multipass-milled in the secondary axis of the working plane
- 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 main axis of the working plane, referenced to the STARTING POINT IN 1ST AXIS
- SECOND SIDE LENGTH Q219 (incremental value): Length of the surface to be multipass-milled in the secondary axis of the working plane, referenced to the STARTING POINT IN 2ND AXIS
- NUMBER OF CUTS Q240: Number of passes to be made over the width
- FEED RATE FOR PLUNGING 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 it transversely in the open, Q209 may be greater than Q207.
- SET-UP CLEARANCE Q200 (incremental value): Distance between tool tip and milling depth for positioning at the start and end of the cycle.

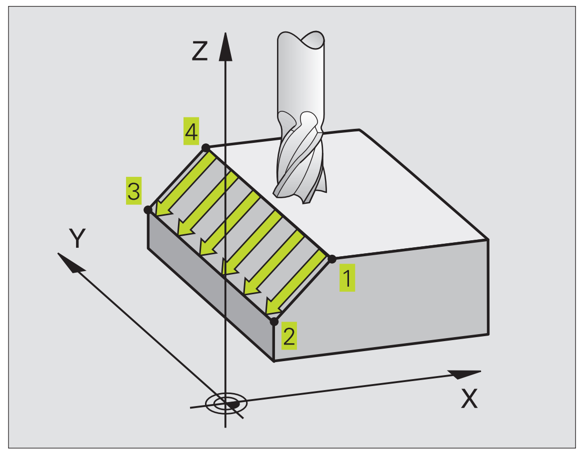




RULED SURFACE (Cycle 231)



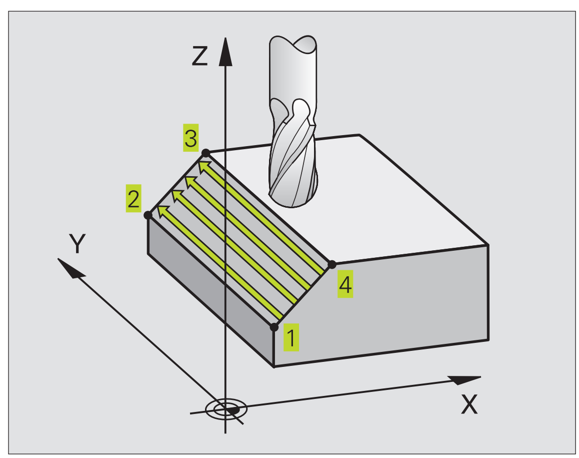
1. From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1. 1
2. The tool then moves to the stopping point 2 at the FEED RATE FOR MILLING. 2
3. From this point, the tool moves in rapid traverse FMAX by the tool diameter in the positive tool axis direction, and then back to starting point 1. 1
4. At the starting position 1 the TNC moves the tool back to the 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 stopping point using point 2 and an offset 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
You can freely choose the starting point and thus the milling direction since the TNC always performs the individual cuts from point 1 to point 2 and the process sequence is executed from points 1 / 2 to point 3 / 4 . You can position point 1 in 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 (tool axis coordinate of point 1 greater than tool axis coordinate of point 2) for slightly inclined surfaces.
- a drawing cut (tool axis coordinate of point 1 less than tool 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. See figure at center right.



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

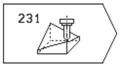
- When milling twisted surfaces, program the main cutting direction (from point 1 to point 2) perpendicular to the direction of the steeper inclination. See figure at lower right.

Before you begin programming, note the following:

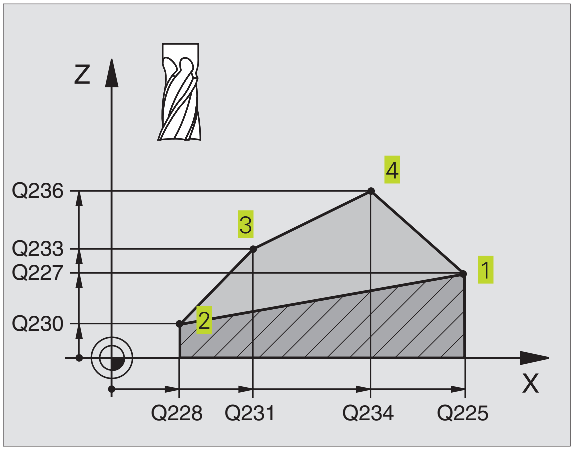
From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1. 1. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

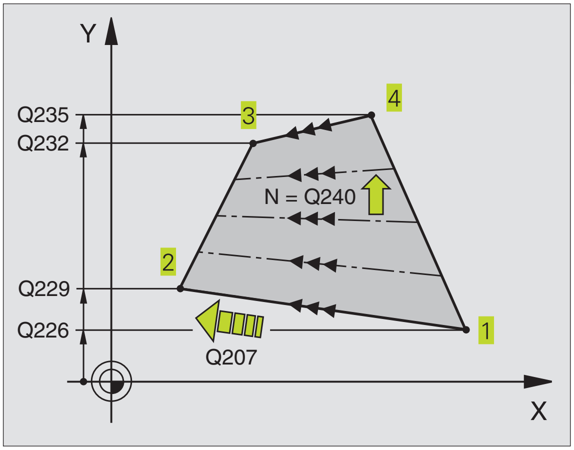
The TNC moves the tool with RADIUS COMPENSATION R0 to the programmed positions.

If required, use a center-cut end mill (ISO 1641).

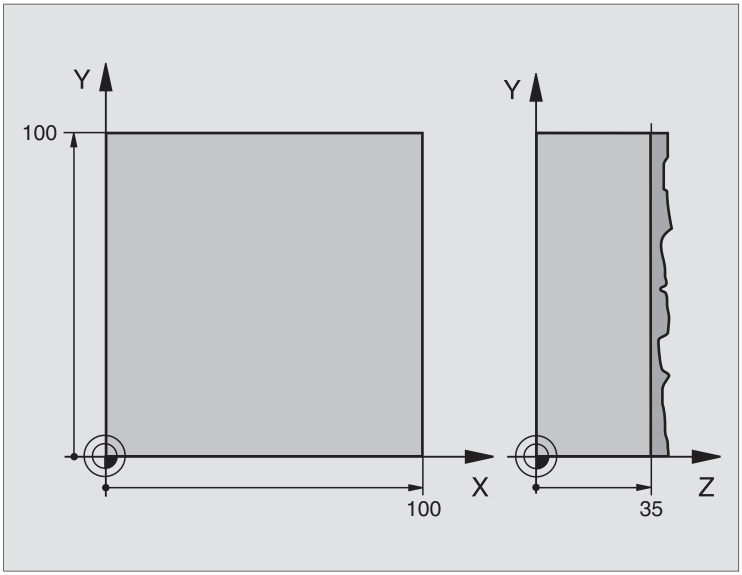


- STARTING POINT IN 1ST AXIS Q225 (absolute value): Starting point coordinate of the surface to be multipass-milled in the main axis of the working plane
- STARTING POINT IN 2ND AXIS Q226 (absolute value): Starting point coordinate of the surface to be multipass-milled in the secondary axis of the working 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 main 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 secondary 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): Coordinate of point3 in the main axis of the working plane
- 3RD POINT IN 2ND AXIS Q232 (absolute): Coordinate of point 3 in the subordinate axis of the working plane
- 3RD POINT IN 3RD AXIS Q233 (absolute): Coordinate of point 3 in the tool axis
- 4TH POINT IN 1ST AXIS Q234 (absolute): Coordinate of point 4 in the main axis of the working plane
- 4TH POINT IN 2ND AXIS Q235 (absolute): Coordinate of point 4 in the subordinate axis of the working plane
- 4TH POINT IN 3RD AXIS Q236 (absolute): Coordinate of point 4 in the tool axis
- NUMBER OF CUTS Q240: Number of passes to be made between points 1 and 4, and between points 2 and 3
- FEED RATE FOR MILLING Q207: Traversing speed of the tool in mm/ min when milling the first pass. The TNC calculates the feed rate for all subsequent passes dependentof the stepover factor of the tool (offset less than tool radius = higher feed rate, high stepover factor = lower feed rate)





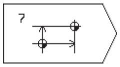
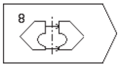
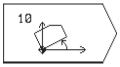
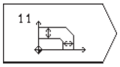
Example: Multipass milling



0 BEGIN PGM 230 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z+0	Define the blank form
2 BLK FORM 0.2 X+100 Y+100 Z+40	
3 TOOL DEF 1 L+0 R+5	Tool definition
4 TOOL CALL 1 Z S3500	Call the tool
5 L Z+250 R0 F MAX	Retract the tool
6 CYCL DEF 230 MULTIPASS MILLNG	Cycle definition: MULTIPASS MILLING
Q225=+0	Starting point for X axis
Q226=+0	Starting point for Y axis
Q227=+35	Starting point for Z axis
Q218=100	1st side length
Q219=100	2nd side length
Q240=25	Number of cuts
Q206=250	Feed rate for plunging
Q207=400	Feed rate for milling
Q209=150	Feed rate for cross pecking
Q200=2	Setup clearance
7 L X-25 Y+0 R0 F MAX M3	Pre-position near the starting point
8 CYCL CALL	Cycle call
9 L Z+250 R0 F MAX M2	Retract in the tool axis, end of program
10 END PGM 230 MM	

8.6 Coordinate transformation cycles

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	
8 MIRROR IMAGE For mirroring contours	
10 ROTATION For rotating contours in the working plane	
11 SCALING FACTOR For increasing or reducing the size of contours	

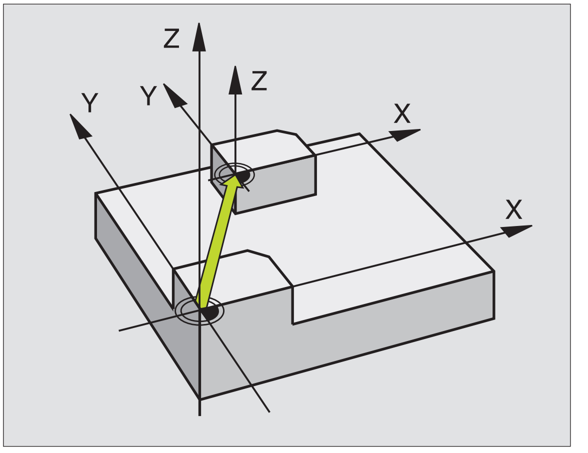
Effect of coordinate transformations
A coordinate transformation becomes effective as soon as it is defined — it is not called. It remains in effect until it is changed or canceled.

To cancel coordinate transformations:

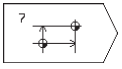
- Define cycles for basic behavior with a new value, such as scaling factor 1.0
- Execute a miscellaneous function M02, M30, or an END PGM block (depending on machine parameter 7300)
- Select a new program

DATUM SHIFT (Cycle 7)

A datum shift allows machining operations to be repeated at various locations on the workpiece.



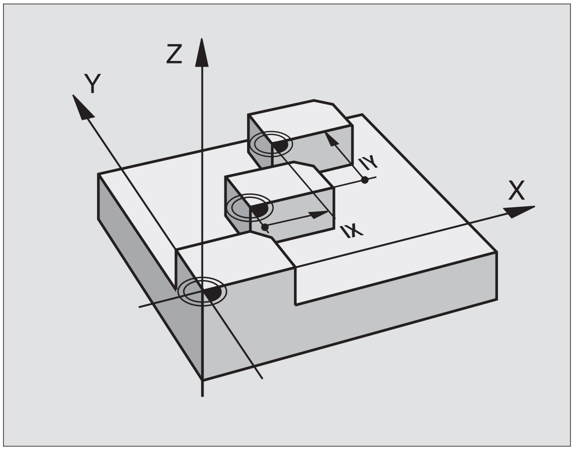
Effect
When the DATUM SHIFT cycle is defined, all coordinate data is based on the new datum. The TNC displays the datum shift in each axis in the additional status display.



▶ 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.

REF

▶ REF: Press the REF soft key (2nd soft-key row) to reference the programmed datum to the machine datum. In this case the TNC indicates the first cycle block with REF



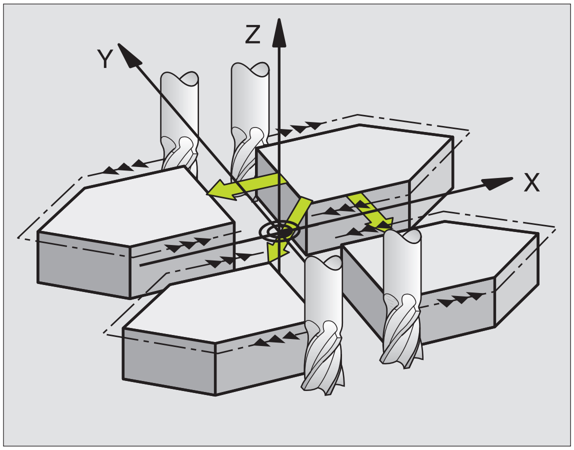
Cancellation
A datum shift is canceled by entering the datum shift coordinates X=0, Y=0 and Z=0.

Status Displays
If datums are referenced to the machine datum, then

- The actual position values are referenced to the active (shifted) datum.
- The datum shown in the additional status display is referenced to the machine datum, whereby the TNC accounts for the manually set datum.

MIRROR IMAGE (Cycle 8)

The TNC can machine the mirror image of a contour in the working plane. See figure at upper right.

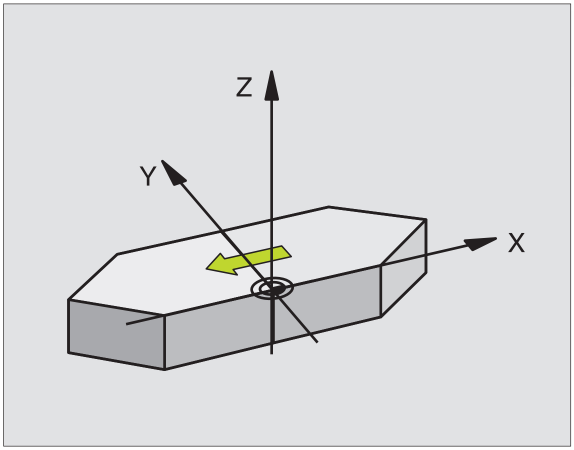


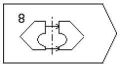
Effect
The MIRROR IMAGE cycle becomes effective as soon as it is defined in the program. Cycle 10 is also effective in the POSITIONING WITH MDI mode of operation. The active mirrored axes are shown in the additional status display.

- 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.

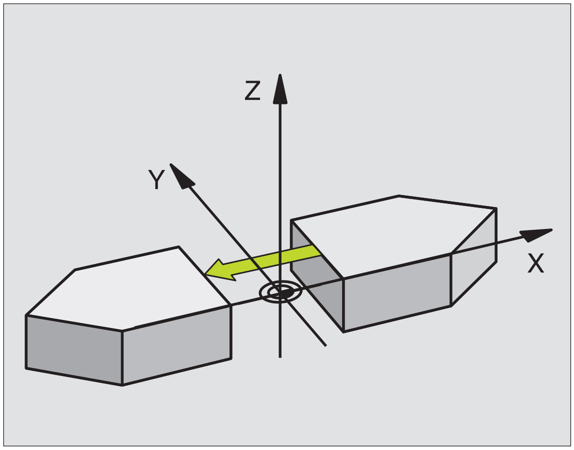
The result of the mirror image depends on the location of the datum

- If the datum lies on the contour to be mirrored, the element simply flips over — see figure at lower right.
- If the datum lies outside the contour to be mirrored, the element also “jumps” to another location — see figure at lower right.





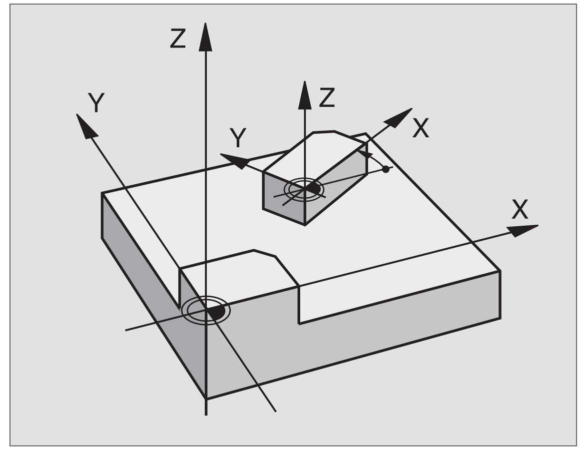
▶ MIRROR IMAGE: Enter the axis you wish to mirror. The tool axis cannot be mirrored.



Cancellation
Program the MIRROR IMAGE cycle once again with NO ENT.

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. Cycle 10 is also effective in the POSITIONING WITH MDI mode of operation. The active rotation angle is shown in the additional status display.

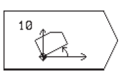
Reference axis for the rotation angle:

- X/Y plane X axis
- Y/Z plane Y axis
- Z/X plane Spindle axis

Before you begin programming, note the following:

An active radius compensation is canceled by defining Cycle 10 and must therefore be reprogrammed, if necessary.

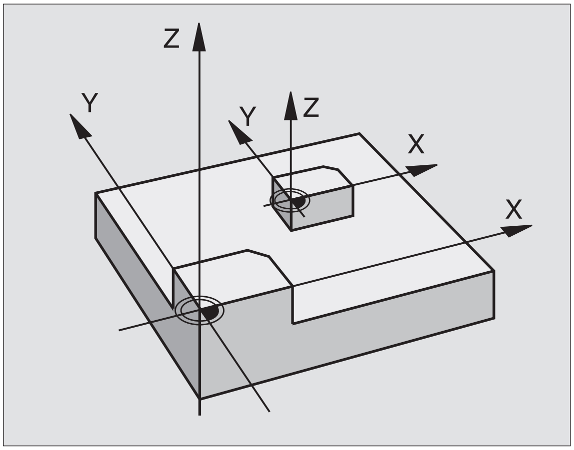
After defining Cycle 10, you must move both axes of the working plane to activate rotation for all axes.



▶ 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°.

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

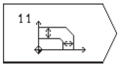
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the POSITIONING WITH MDI mode of operation. The active scaling factor is shown in the additional status display.

The scaling factor can be applied

- in the working plane, or on all three coordinate axes at the same time (depending on machine parameter 7410)
- to the dimensions in cycles
- to the parallel axes U,V,W

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 “Activation” above)

Enlargement: SCL greater than 1 (up to 99.999 999)

Reduction SCL less than 1 (down to 0.000 001)

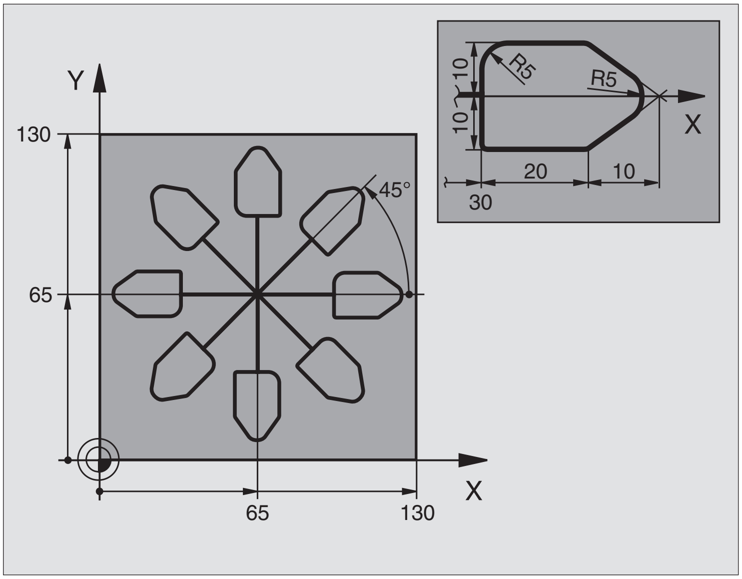
Cancellation

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

Example: Coordinate transformation cycles

Process

- Program the coordinate transformations in the main program
- Program the machining operation in subprogram 1 (see section 9 "Programming: Subprograms and Program Section Repeats")



`0 BEGIN PGM 11 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-20`	Define the blank form
`2 BLK FORM 0.2 X+130 Y+130 Z+0`	
`3 TOOL DEF 1 L+0 R+1`	Tool definition
`4 TOOL CALL 1 Z S4500`	Call the tool
`5 L Z+250 R0 F MAX`	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; execute 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 F MAX M2`	Retract in the tool axis, end of program

21 LBL 1	Subprogram 1:
22 L X+0 Y+0 R0 F MAX	Define milling operation
23 L Z+2 R0 F MAX 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 F500	
35 L Z+20 R0 F MAX	
36 LBL 0	
37 END PGM 11 MM	

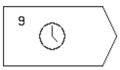
8.7 Special Cycles

DWELL TIME (Cycle 9)

This cycle causes the execution of the next block within a running program to be delayed by the programmed dwell time. A dwell time can be used for such purposes as chip breaking.

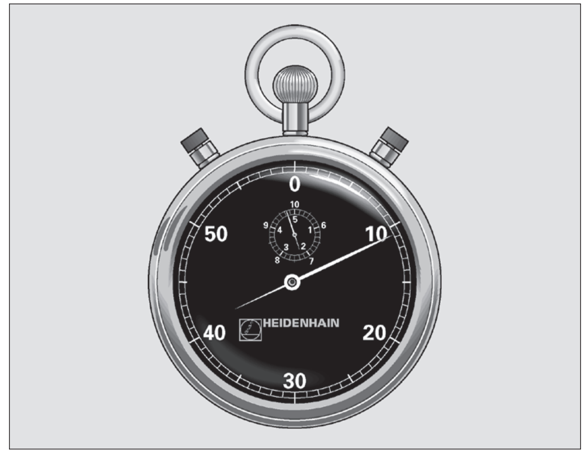
Effect

Cycle 9 becomes effective as soon as it is defined in the program. Modal conditions such as spindle rotation are not affected.



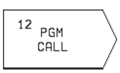
▶ DWELL TIME IN SECONDS: Enter the dwell time in seconds

Input range: 0 to 30 000 seconds (approx. 8.3 hours) in increments of 0.001 seconds



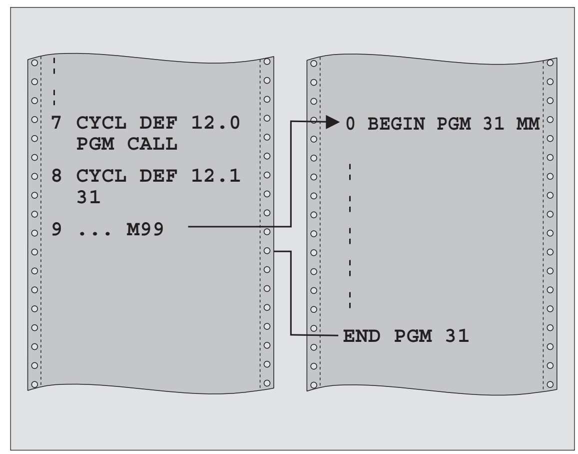
PROGRAM CALL (Cycle 12)

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.



▶ PROGRAM NAME: Number of the program to be called up

The program is called with
- CYCL CALL (separate block) or
- M99 (blockwise) or
- M89 (modally)



Example: Program call

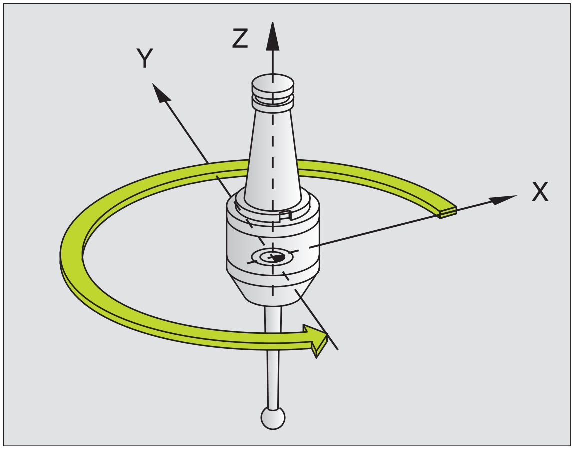
A callable program 50 is to be called into a program via a cycle call.

Resulting NC blocks

55 CYCL DEF 12.0 PGM CALL	Definition:
56 CYCL DEF 12.1 PGM 50	Program 50 is a cycle”
57 L X+20 Y+50 FMAX M99	Call program 50

ORIENTED SPINDLE STOP (Cycle 13)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle 13.



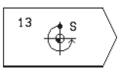
The control can address the machine tool spindle as a 4th axis and rotate it to a given angular position.

Oriented spindle stops are required for

- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission

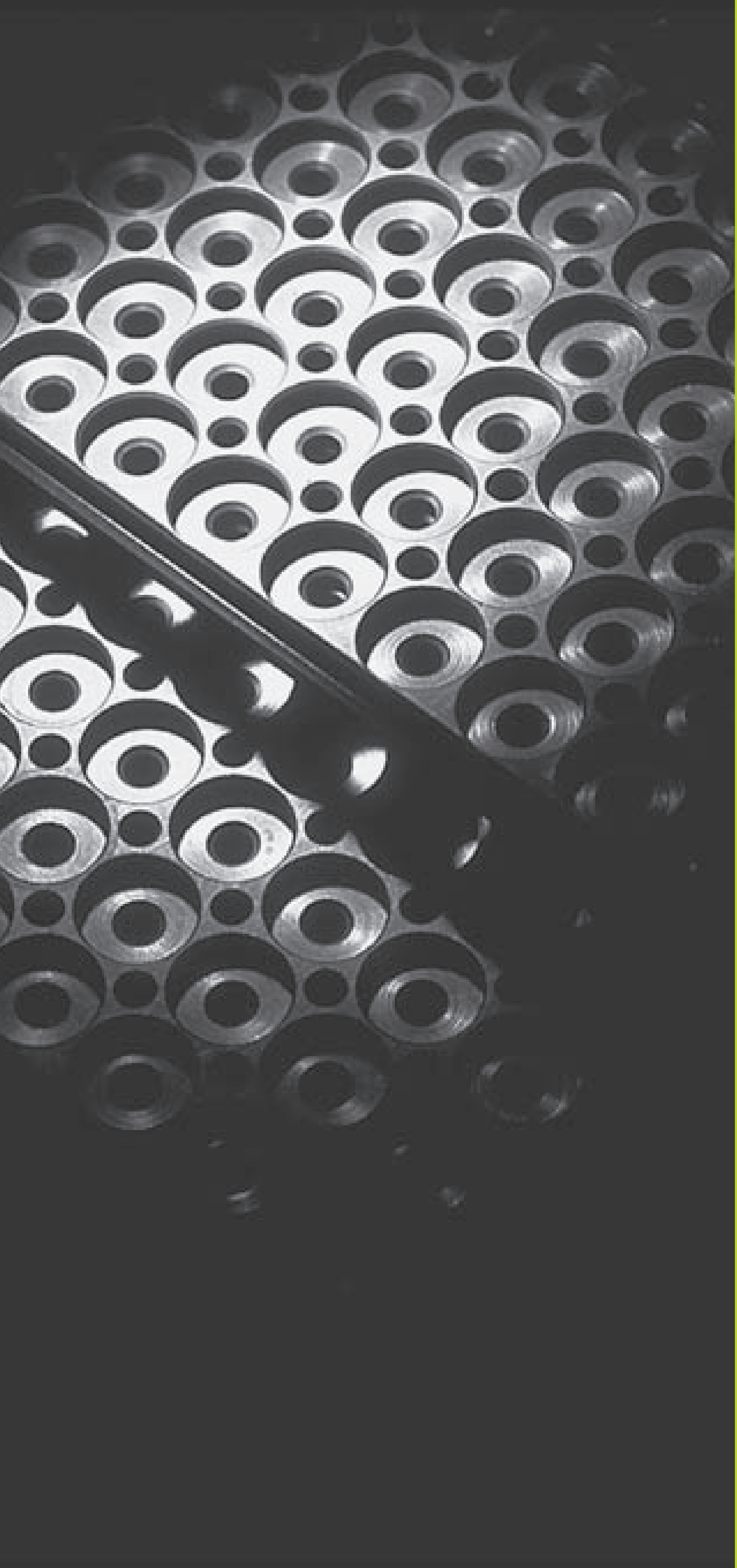
Effect
The angle of orientation defined in the cycle is positioned to by entering M19.

If you program M19 without having defined Cycle 13, the TNC positions the machine tool spindle to an angle that has been set in a machine parameter (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°



9

Programming:

Subprograms and Program Section Repeats

9.1 Marking Subprograms and Program Section Repeats

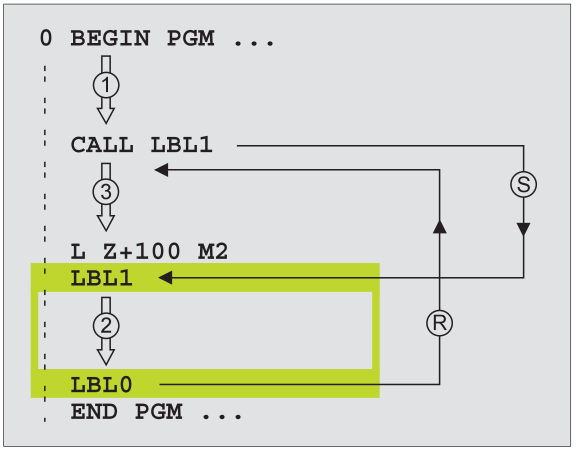
Subprograms and program section repeats enable you to program a machining sequence once and then run it as often as desired.

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 254. Each label can be set only once with LABEL SET in a program.

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 with LBL 0.

3 The TNC then resumes the part program from the block after the subprogram call.

Operating constraints

- 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 M2 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

LBL SET

- To mark the beginning, press the LBL SET key and enter a LABEL NUMBER.
- Enter the subprogram.
- To mark the end, press the LBL SET key and enter the LABEL NUMBER "0".

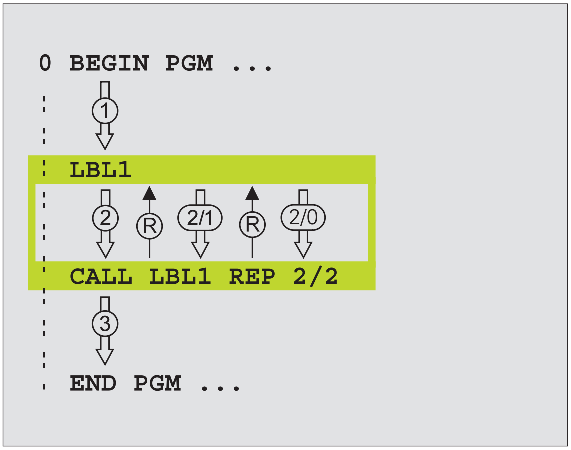
Calling a subprogram

LBL CALL

- To call a subprogram, press the LBL CALL key.
- LABEL NUMBER: Enter the label number of the program you wish to call.
- 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



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 labeled program section, i.e. to the block with 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.

Operating constraints

- You can repeat a program section up to 65 534 times in succession.
- The number behind the slash after REP indicates the number of repetitions remaining to be run.
- The total number of times the program section is executed is always one more than the programmed number of repeats.

Programming a program section repeat

LBL SET

▶ To mark the beginning, press the LBL SET key and enter a LABEL NUMBER for the program section you wish to repeat.

▶ Enter the program section.

Calling a program section repeat

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 Nesting

You can nest subprograms and program section repeats in the following ways:

- 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: 8
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example NC blocks

`0 BEGIN PGM 15 MM`	
`...`	
`17 CALL LBL 1`	Call the subprogram marked with LBL1
`...`	
`35 L Z+100 R0 FMAX M2`	Last program block of the
	main program (with M2)
`36 LBL 1`	Beginning of subprogram 1
`...`	
`39 39 CALL LBL 2`	Call the subprogram marked with LBL2
`...`	
`45 LBL 0`	End of subprogram 1
`46 LBL 2`	Beginning of subprogram 2
`...`	
`62 LBL 0`	End of subprogram 2
`63 END PGM 15 MM`	

Program execution

1st step: Main program 15 is executed up to block 17.

2nd step: Subprogram 1 is called, and executed up to block 39.

3rd step: 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.

4th step: Subprogram 1 is called, and executed from block 40 up to block 45. End of subprogram 1 and return jump to the main program 15.

5th step: Main program 15 is executed from block 18 to block 35. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

0 BEGIN PGM 16 MM	
...	
15 LBL 1	Beginning of program section repeat 1
...	
20 LBL 2	Beginning of program section repeat 2
...	
27 CALL LBL 2 REP 2/2	The program section between this block and LBL 2
...	(block 20) is repeated twice.
35 CALL LBL 1 REP 1/1	The program section between this block and LBL 1
...	(block 15) is repeated once.
50 END PGM 16 MM	

Program execution

1st step: Main program 16 is executed up to block 27.

2nd step: Program section between block 27 and block 20 is repeated twice.

3rd step: Main program 16 is executed from block 28 to block 35.

4th step: Program section between block 35 and block 15 is repeated once (including the program section repeat between 20 and block 27).

5th step: Main program 16 is executed from block 36 to block 50. End of program.

Repeating a subprogram

Example NC blocks

`0 BEGIN PGM 17 MM`	
`...`	
`10  LBL 1`	Beginning of the program section repeat
`11  CALL LBL 2`	Subprogram call
`12  CALL LBL 1 REP 2/2`	The program section between this block and LBL1
`...`	(block 10) is repeated twice
`19  L Z+100 R0 FMAX M2`	Last program block of the main program with M2
`20  LBL 2`	Beginning of subprogram
`...`	
`28  LBL 0`	End of subprogram
`29 END PGM 17 MM`	

Program execution

1st step: Main program 17 is executed up to block 11.

2nd step: Subprogram 2 is called and executed.

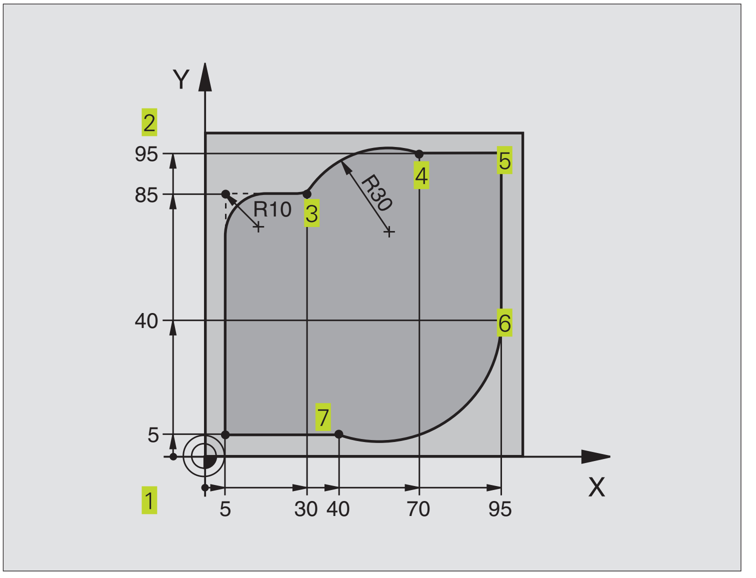
3rd step: Program section between block 12 and block 10 is repeated twice. This means that subprogram 2 is repeated twice.

4th step: Main program 17 is executed from block 13 to block 19. End of program.

Example: Milling a contour in several infeeds

Process

- Pre-position the tool to the workpiece surface
- Enter the infeed depth in incremental values
- Mill the contour
- Repeat downfeed and contour-milling

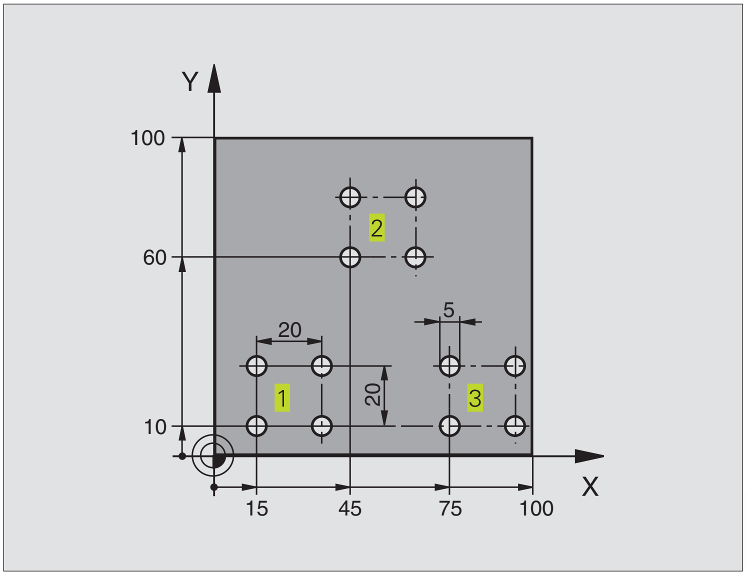


`0 BEGIN PGM 95 MM`	
`1 BLK FORM 0.1 Z X+0 Y+0 Z-40`	
`2 BLK FORM 0.2 X+100 Y+100 Z+0`	
`3 TOOL DEF 1 L+0 R+10`	Define the tool
`4 TOOL CALL 1 Z S4000`	Call the tool
`5 L Z+250 R0 F MAX`	Retract the tool
`6 L X-20 Y-20 R0 F MAX`	Pre-position in the working plane
`7 L Z0 R0 F2000 M3`	Pre-position in the spindle axis
`8 Lbl 1`	Set label for program section repeat
`9 L IZ-4 r0 F2000`	Infeed depth in incremental values (in the open)
`10 L X+5 Y+5 RL F300`	Approach contour
`11 RND R2`	
`12 L Y+85`	Point 2: first straight line for corner 2
`13 RND R10 F150`	Insert radius with R = 10 mm, feed rate: 150 mm/min
`14 L X+30`	Move to point 3
`15 CR X+70 Y+95 R+30 DR-`	Move to point 4
`16 L X+95`	Move to point 5
`17 L Y+40`	Move to point 6
`18 CT X+40 Y+5`	Move to point 7
`19 L X+5`	Move to last contour point 1
`20 RND R2`	
`21 L X-20 Y-20 R0 F1000`	Depart contour
`22 Call LBL 1 REP 4/4`	Return jump to LBL 1; section is repeated a total of 4 times
`23 L Z+250 R0 F MAX M2`	Retract in the tool axis, end of program
`24 END PGM 95 MM`	

Example: Groups of holes

Process

- 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



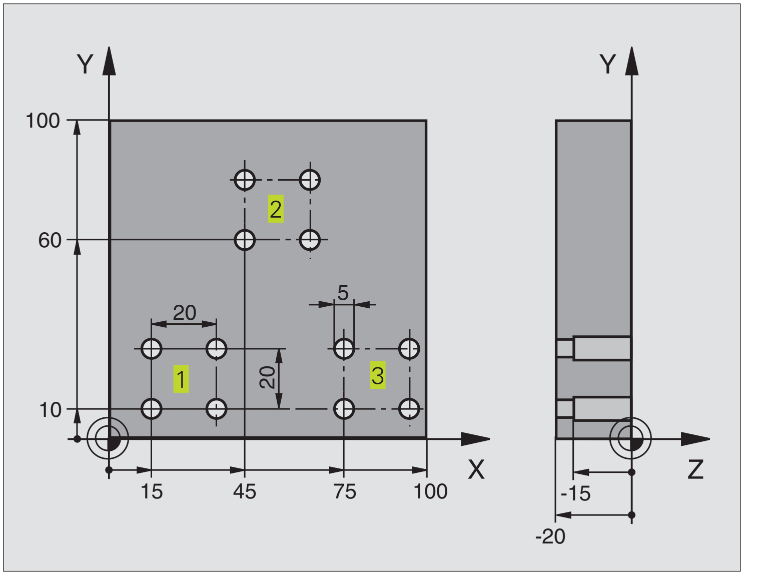
0 BEGIN PGM UP1 MM	
1 BLK FORM 0.1 Z X+0 Y+0 Z-20	
2 BLK FORM 0.2 X+100 Y+100 Z+0	
3 TOOL DEF 1 L+0 R+2.5	Define the tool
4 TOOL CALL 1 Z S5000	Call the tool
5 L Z+250 R0 F MAX	Retract the tool
6 CYCL DEF 200 DRILLING	Cycle definition: drilling
Q200=2	Set-up clearance
Q201=-10	Depth
Q206=250	Feed rate for drilling
Q202=5	Pecking depth
Q210=0	Dwell time at top
Q203=+0	Surface coordinate
Q204=10	2nd set-up clearance
7 L X+15 Y+10 R0 F MAX 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 F MAX	Move to starting point for group 2
10 CALL LBL 1	Call the subprogram for the group
11 L X+75 Y+10 R0 F MAX	Move to starting point for group 3
12 CALL LBL 1	Call the subprogram for the group
13 L Z+250 R0 F MAX M2	End of main program

14 LBL 1	Beginning of subprogram 1: Group of holes
15 CYCL CALL	1st hole
16 L IX+20 R0 F MAX M99	Move to 2nd hole, call cycle
17 L IY+20 R0 F MAX M99	Move to 3rd hole, call cycle
18 L IX-20 R0 F MAX M99	Move to 4th hole, call cycle
19 LBL 0	End of subprogram 1
20 END PGM UP1 MM	

Example: Groups of holes with several tools

Process

- 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 3 L+0 R+3.5	Define tool: reamer
6 TOOL CALL 1 Z S5000	Call tool: center drill
7 L Z+250 R0 F MAX	Retract the tool

8 CYCL DEF 200 DRILLING	Cycle definition: Centering
Q200=2	Set-up clearance
Q201=-3	Depth
Q206=250	Feed rate for drilling
Q202=3	Pecking depth
Q210=0	Dwell time at top
Q203=+0	Surface coordinate
Q204=10	2nd set-up clearance
9 CALL LBL 1	Call subprogram 1 for the entire hole pattern
10 L Z+250 R0 F MAX M6	Tool change
11 TOOL CALL 2 Z S4000	Call the drilling tool
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 F MAX M6	Tool change
16 TOOL CALL 3 Z S500	Tool call: reamer
17 CYCL DEF 201 REAMING	Cycle definition: REAMING
Q200=2	Set-up clearance
Q201=-15	Depth
Q206=250	Feed rate for reaming
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 F MAX M2	End of main program
20 LBL 1	Beginning of subprogram 1: Entire hole pattern
21 L X+15 Y+10 R0 F MAX 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 F MAX	Move to starting point for group 2
24 CALL LBL 2	Call subprogram 2 for the group
25 L X+75 Y+10 R0 F MAX	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 IX+20 R0 F MAX M99	Move to 2nd hole, call cycle
31 L IY+20 R0 F MAX M99	Move to 3rd hole, call cycle
32 L IX-20 R0 F MAX M99	Move to 4th hole, call cycle
33 LBL 0	End of subprogram 2
34 END PGM UP2 MM	

10

Test Run and Program Run

10.1 Graphics

In the TEST RUN mode of operation the TNC graphically simulates the machining of the workpiece. 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.

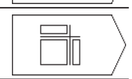
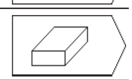
The TNC will not show a graphic if

- the current program has no valid blank form definition
- no program is selected

> A 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

After you have pressed the PGM TEST soft key in the operating mode PROGRAM RUN, the TNC displays the following soft keys:

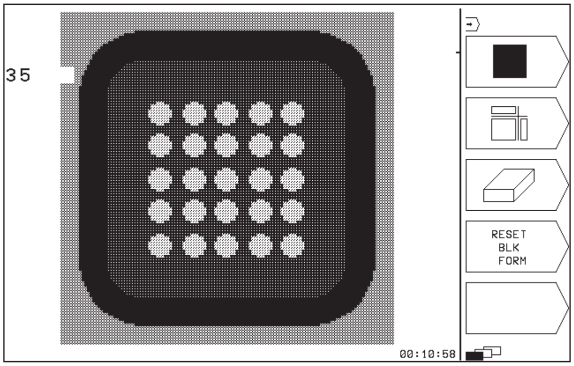
Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Plan view

▶ Press the soft key for plan view.

The deeper the surface, the darker the shade.

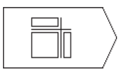
Plan view is the fastest of the three graphic display modes.



Projection in 3 planes

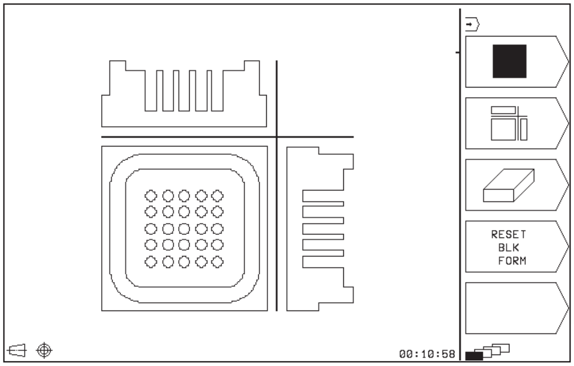
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

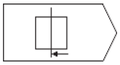
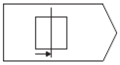
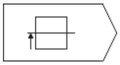
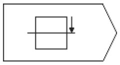
In addition, you can shift the sectional planes with the corresponding soft keys:



▶ Press the soft key for projection in three planes.

▶ Shift the soft-key row until the TNC displays the following soft keys:



Function	Soft keys
Shift the vertical sectional plane to the left or to the right	 
Shift the horizontal sectional plane upwards or downwards	 

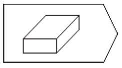
The positions of the sectional planes are visible during shifting.

3-D view

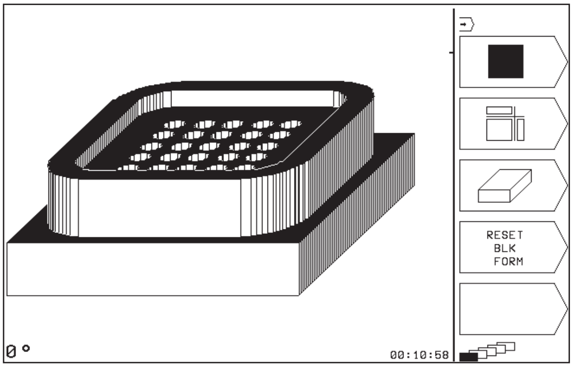
The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.

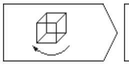
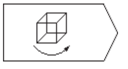
In the TEST RUN mode of operation you can isolate details for magnification (see "Magnifying details").



▶ Press the soft key for 3-D view.



To rotate the 3-D view
Shift the soft-key row until the following soft keys appear:

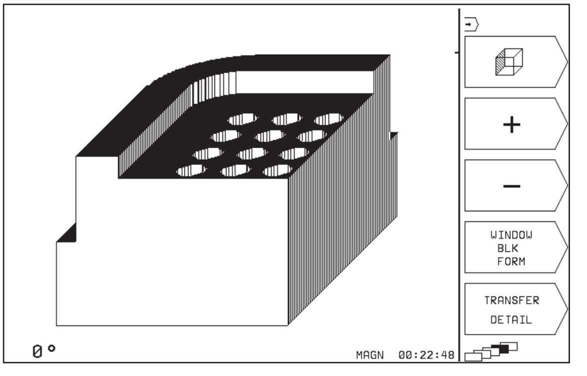
Function	Soft keys
Rotate the workpiece in 90° steps about the vertical axis	 

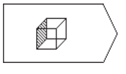
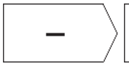
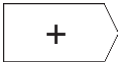
Magnifying details

You can isolate a detail in the TEST RUN operating mode, when the 3-D display mode is selected.

The graphic simulation must first have been stopped. A detail magnification is always effective in all display modes.

Shift the soft-key row in the TEST RUN mode of operation until the following soft keys appear:



Function	Soft keys
Select the workpiece surface to be trimmed: Press the soft key several times	
Shift the sectional plane to reduce or magnify the blank form	 
Select the isolated detail	TRANSFER DETAIL

To change the detail magnification:
The soft keys are listed in the table above.

- ▶ Interrupt the graphic simulation, if necessary.
- ▶ Select the workpiece surface with the corresponding soft key (see table).
- ▶ To reduce or magnify the blank form, press the minus or plus soft key, respectively.
- ▶ To select the isolated detail, press the TRANSFER DETAIL soft key
- ▶ Restart the test run or program run.

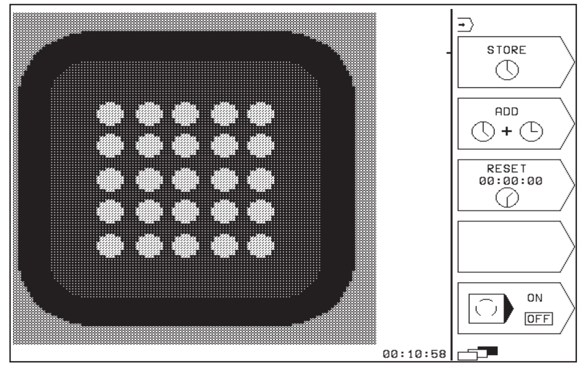
Repeating graphic simulation

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

☞ The WINDOW BLK FORM soft key will return the blank form to its original shape or size, even if a detail has been isolated and not yet magnified with TRANSFER DETAIL.

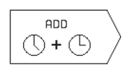
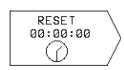
Measuring the machining time



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 approximate time which the TNC calculates from the duration of tool movements. The time calculated by the TNC cannot be used for calculating the production time because the TNC does not account for the duration of machine-dependent interruptions, such as tool change.

To activate 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	
Display the sum of stored time and displayed time	
Clear displayed time	

10.2 Test Run

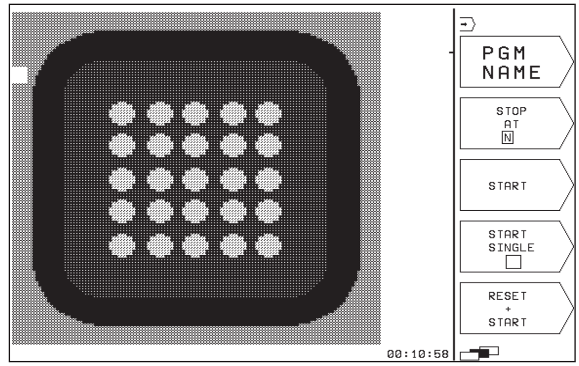
In the TEST RUN mode of operation you can simulate programs and program sections to prevent errors from occurring during program run. The TNC checks the programs for the following errors:

- Geometrical incompatibilities
- Missing data
- Impossible jumps
- Violation of the machine's working space

The following functions are also available:

- Blockwise test run
- Interrupt test at any block
- Functions for graphic simulation
- Additional status display

Running a program test



- ▶ Select the PROGRAM RUN operating mode

PGM TEST

- ▶ Select the TEST RUN mode of operation.
- ▶ Call the file manager with the PGM NAME key and select the file you wish to test, or
- ▶ Go to the program beginning: Select line "0" with the GOTO key and confirm you entry with the ENT key.

The TNC displays the following soft keys (1st or 2nd soft-key row):

Function	Soft key
Test the entire program	START
Test each program block individually	START SINGLE
Show the blank form and test the entire program	RESET + START
Interrupt the test run	STOP

Running a program test up to a certain block

With the STOP AT N function the TNC does a test run up to the block with block number N.

- ▶ Go to the program beginning in the TEST RUN mode of operation.
- ▶ To do a test run up to a certain block, press the STOP AT N soft key.

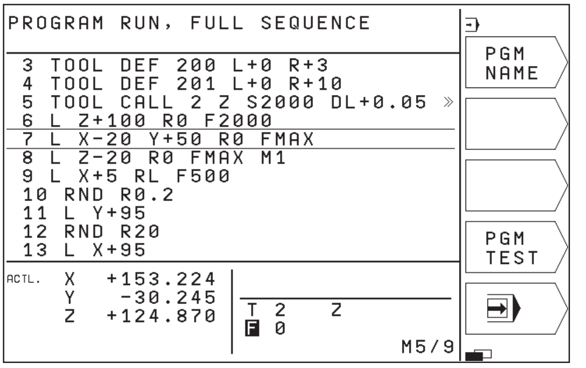
STOP AT N

- ▶ TO BLOCK NUMBER =: Enter the block number at which you wish the test to stop.
- ▶ To test a program section, press the ENT soft key. The TNC will test the program up to the entered block.

10.3 Program Run

In the PROGRAM RUN operating mode the TNC executes a part program either in single block or continuously.

Function	Soft key
PROGRAM RUN, SINGLE BLOCK (default setting)	
PROGRAM RUN, FULL SEQUENCE	



In PROGRAM RUN, SINGLE BLOCK you must start each block separately by pressing the NC START button.

In PROGRAM RUN, FULL SEQUENCE the TNC executes a part program continuously to its end or up to a program stop.

The following TNC functions can be used in the program run modes of operation:

- Interrupt program run
- Start program run from a certain block
- Additional status display

To run a part program:

To prepare the TNC:

1 Clamp the workpiece to the machine table.

2 Set the datum

3 Select the part program (status M).

> You can adjust the feed rate and spindle speed with the override knobs.

PROGRAM RUN, FULL SEQUENCE

▶ Start machining program with the NC start button

PROGRAM RUN, SINGLE BLOCK

▶ Start each block of the part program individually with the NC START button.

Interrupt machining.

There are several ways to interrupt a program run:

- Programmed interruptions
- Machine STOP button
- Switching to PROGRAM RUN, SINGLE BLOCK

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 functions M0, M1 (see „10.4 Optional Program Run Interruption"), M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

Interruption with NC STOP button

- ▶ Press the NC STOP button: The block which the TNC is currently executing is not completed. The asterisk in the status display blinks.
- ▶ If you do not wish to continue the machining process you can reset the TNC with the STOP soft key. The asterisk in the status display goes out. In this case, the program must be restarted from the program beginning.

Interruption of machining by switching to the PROGRAM RUN, SINGLE BLOCK mode of operation.

You can interrupt a program that is being run in the PROGRAM RUN, FULL SEQUENCE mode of operation by switching to PROGRAM RUN, SINGLE BLOCK. The TNC interrupts the machining process at the end of the current block.

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.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations
- The coordinates of the circle center that was last defined
- The current count of program section repeats
- The number of the block where a subprogram or a program section repeat was last called

Resuming program run with the NC START button
You can resume program run by pressing the NC START button if the program was interrupted in one of the following ways:

- NC STOP button was pressed
- A programmed interruption
- The EMERGENCY STOP button was pressed (machine-dependent function)

> If you interrupted program run with the STOP soft key, you can select another block with the GOTO key and resume machining there.
>
> If you select block 0, the TNC resets all stored data (tool data, etc.)
>
> If you interrupted program run during a program section repeat, you can only use GOTO to select other blocks within the program section repeat.

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 at the place at which it was interrupted.

- If the error message *is blinking*

▶ Switch off the TNC and the machine.

▶ 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.

10.4 Optional Program Run Interruption

The TNC optionally interrupts the program or test run at blocks containing M01:

- ▶ 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

10.5 Blockwise Transfer: Running Longer Programs

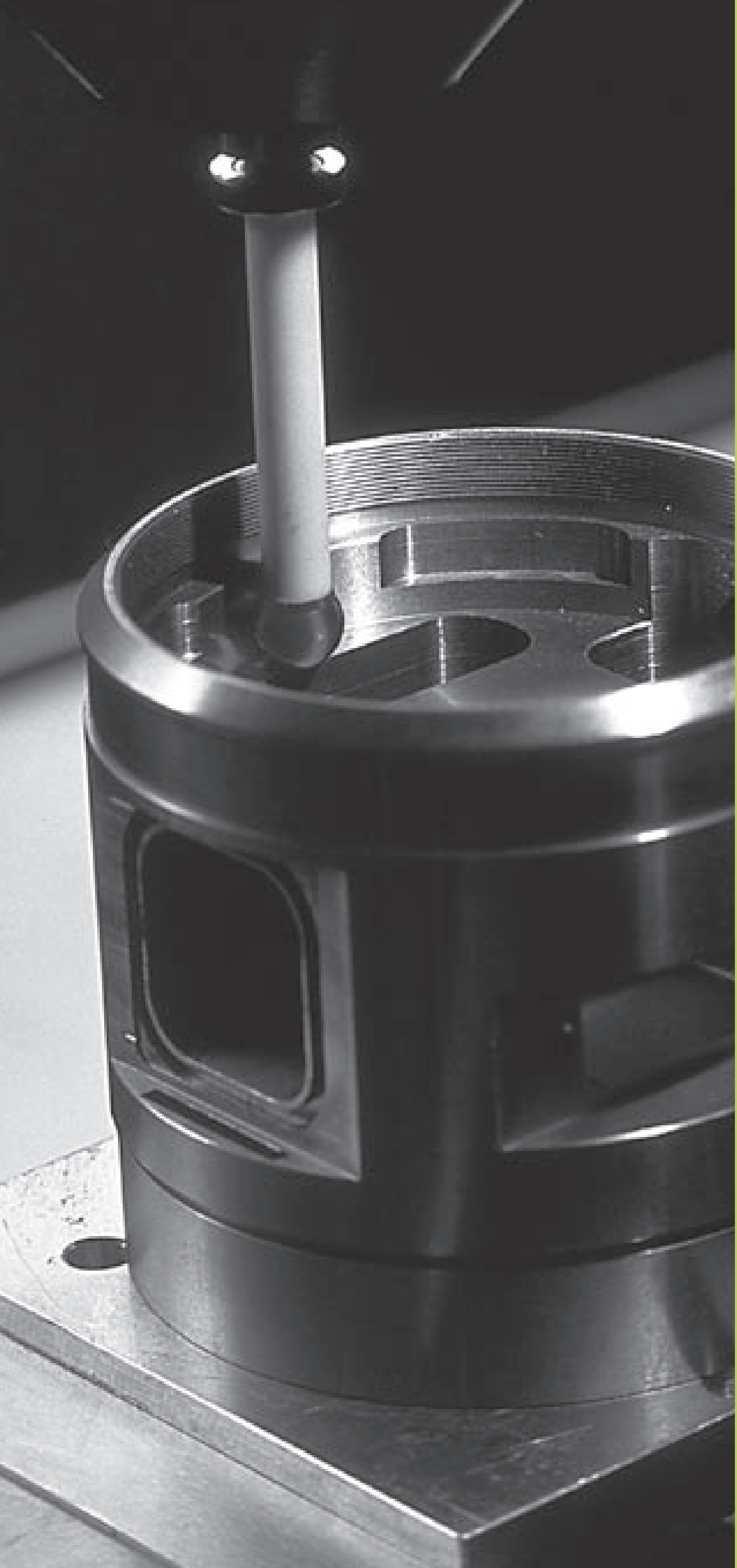
Machine programs that require more memory space than is available in the TNC can be transferred „blockwise" from an external memory.

The program blocks are read in via data interface and are then deleted immediately after being executed. In this way programs of unlimited length can be executed.

> The program may have a maximum of 20 TOOL DEF blocks. If you require more tools then use a tool table.
>
> If the program contains a PGM CALL block, the called program must be stored in the TNC memory.
>
> The program may not include:
> - Subprograms
> - Program section repeats
> - Function FN15:PRINT

Blockwise program transfer
Configure the data interface with the "Set block buffer" MOD function (see section 12.4 "Setting the Data Interface").

- ▶ Select the Program Run, Full Sequence mode or the Program Run, Single Block mode.
- ▶ Begin blockwise transfer: Press the BLOCKWISE TRANSFER soft key.
- ▶ Enter the program name and confirm your entry with the ENT key. The TNC reads in the selected program via data interface
- ▶ Start the part program with the machine start button. If you have defined a block buffer greater than 0, the TNC waits with the program start until the defined number of NC blocks has been read in.

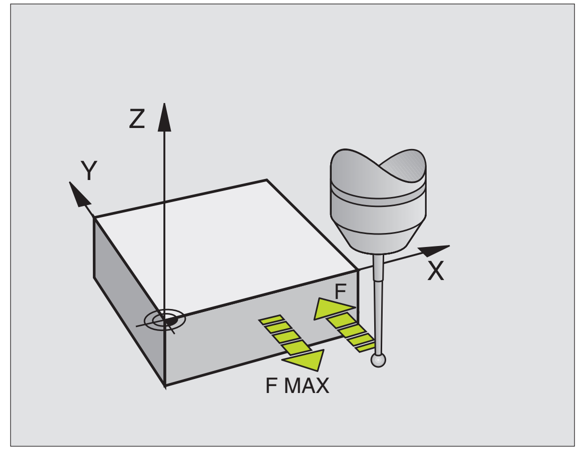


11

3-D Touch Probes

11.1 Touch probe cycles in the operating mode MANUAL OPERATION

The TNC must be specially prepared by the machine tool builder for the use of a 3-D touch probe.



After you press the NC START button, the touch probe approaches the workpiece in a paraxial movement in the selected probe function. The machine tool builder sets the probe feed rate (see figure at right). When the probe contacts the workpiece, it

- transmits a signal to the TNC: the coordinates of the probed position are stored,
- The touch probe stops moving, and
- returns to its starting position in rapid traverse.

If the stylus is not deflected within a defined distance, the TNC displays an error message (MP 6130).

Select the touch probe function

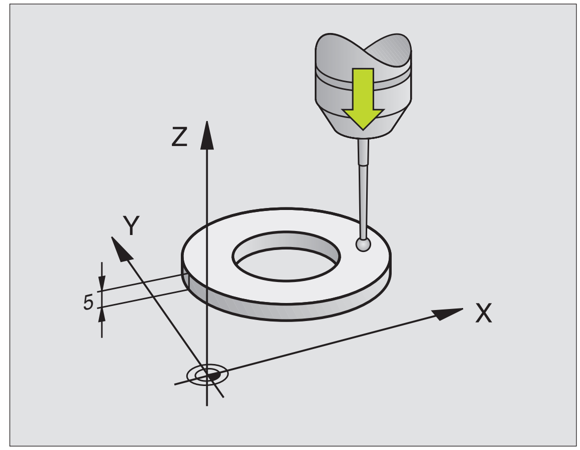
▶ Select the MANUAL OPERATION mode.

TOUCH PROBE

▶ To choose the touch probe functions, press the TOUCH PROBE soft key (2nd soft-key row). The TNC displays additional soft keys — see table at right.

Function	Soft key
Calibrate the effective length (2nd soft-key row)	CAL L
Calibrate the effective radius (2nd soft-key row)	CAL R
Basic rotation	PROBING ROT
Set the datum	PROBING POS
Set the datum at a corner	PROBING P
Set the datum at a circle center	PROBING CC

Calibrating a touch trigger probe



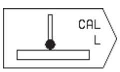
The touch probe must be calibrated

- commissioning
- when the stylus breaks
- when the stylus is changed
- when the probe feed rate is changed
- in case of irregularities such as those resulting from thermal changes in the machine

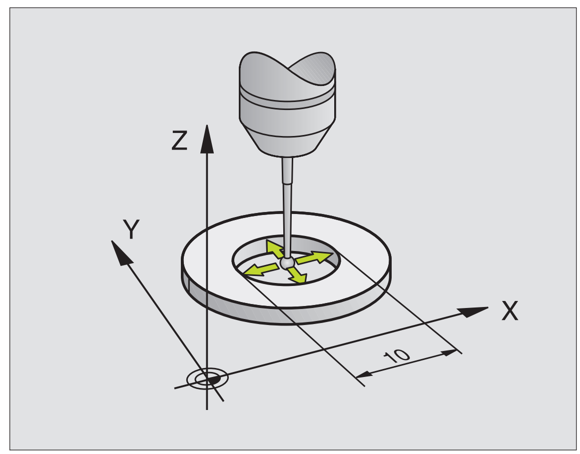
During calibration, the TNC finds the "effective" length of the stylus and the "effective" radius of the ball tip. To calibrate the touch probe, clamp a ring gauge of known height and known internal radius to the machine table.

To calibrate the effective length:

- ▶ Set the datum in the tool 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 fields.
- ▶ Use a soft key to select the TOOL AXIS
- ▶ 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 displayed traverse direction (if necessary), press an arrow key.
- ▶ To probe the upper surface, press the NC START button



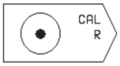
Calibrating the effective radius and compensating center misalignment

After the touch probe is inserted it normally needs to be exactly aligned with the spindle axis. The misalignment is measured with this calibration function and compensated electronically.

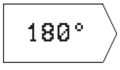
For this operation the TNC rotates the 3-D touch probe by 180°. The rotation is initiated by a miscellaneous function that is set by the machine tool builder in the machine parameter 6160.

The center misalignment is measured after the effective ball tip radius is calibrated.

- In the MANUAL OPERATION mode, position the ball tip in the bore of the ring gauge.



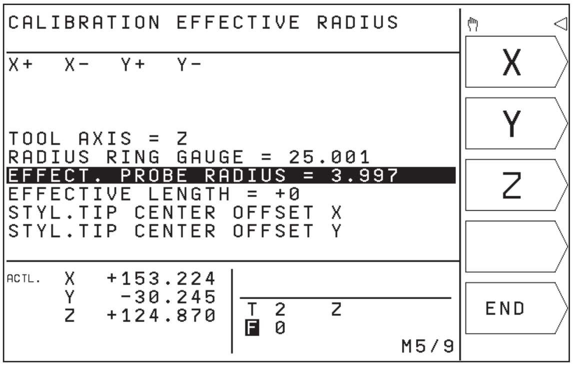
- To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL R soft key.
- Enter the TOOL AXIS. The remaining menu items do not require input.
- To start probing: 4 x NC START button. The touch probe contacts a position on the bore in each axis direction and calculates the effective ball-tip radius.
- If you want to terminate the calibration function at this point, press the END soft key.



- If you want to determine the ball-tip center misalignment, press the 180° soft key. The TNC rotates the touch probe by 180°.
- To start probing: 4 x NC START button. 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, and the center misalignment for use when the touch probe is needed again. You can display the values on the screen with the soft keys CAL L and CAL R.

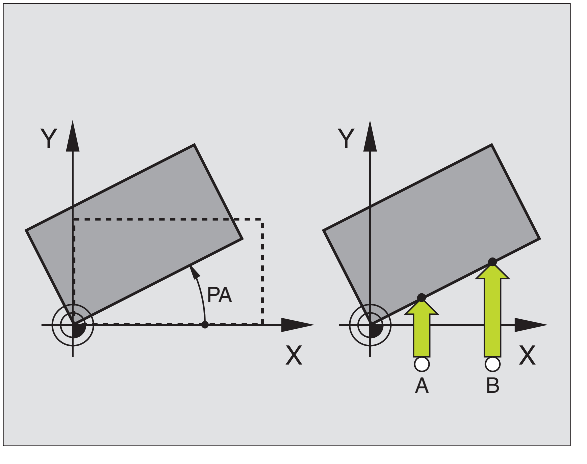


Compensating workpiece misalignment

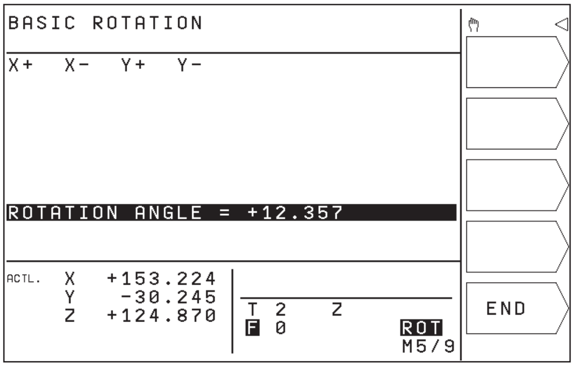
The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

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 lower 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.



- To select the touch probe function: Press 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 with an arrow key.
- To probe the workpiece, press the NC START button
- Position the ball tip at a starting position near the second touch point.
- To probe the workpiece, press the NC START button



The TNC saves the basic rotation in non-volatile memory. The basic rotation is effective for all subsequent program runs and test runs.

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:

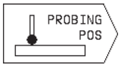
- To select the touch probe function: Press the PROBING ROT soft key
- Enter a ROTATION ANGLE of zero and confirm with the ENT key.
- To terminate the probe function, press the END key.

11.2 Setting the Datum with a 3-D Touch Probe

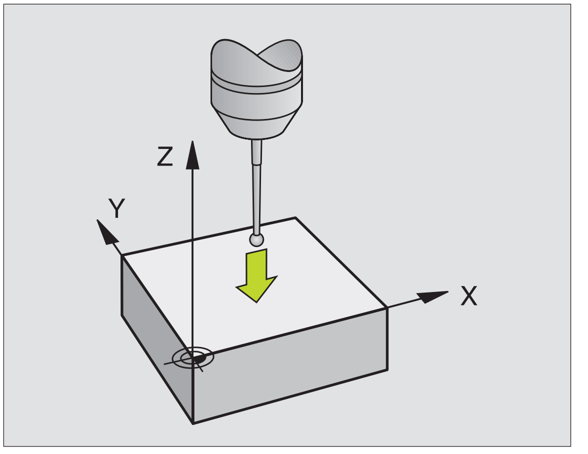
The following functions are available for setting the datum on an aligned workpiece:

- 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

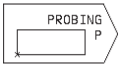
To set the datum in any axis (see figure at upper right)



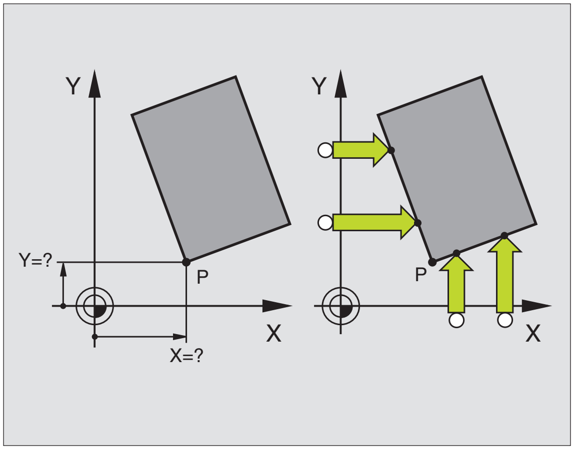
- To select the touch probe function: Press the PROBING POS soft key
- Move the touch probe to a starting position near the touch point.
- Select the probe axis and direction in which you wish to set the datum, such as Z in direction Z–. Selection is made with the arrow keys.
- To probe the workpiece, press the NC START button
- DATUM: Enter the nominal coordinate and confirm you entry with ENT.



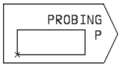
Corner as datum — using points that were already probed for a basic rotation (see figure at center right)



- Select the touch probe function: Press the PROBING P soft key
- TOUCH POINTS OF BASIC ROTATION?: Press YES to transfer the touch point coordinates to memory.
- Position the touch probe at a starting position near the first touch point of the side that was not probed for basic rotation.
- Select the probe direction: Select the axis with an arrow key.
- To probe the workpiece, press the NC START button
- Position the touch probe near the second touch point on the same side.
- To probe the workpiece, press the NC START button
- DATUM: Enter both datum coordinates into the menu window, and confirm your entry with the ENT key.
- To terminate the probe function, press the END key.



Corner as datum — without using points that were already probed for a basic rotation



- Select the touch probe function: Press the PROBING P soft key
- TOUCH POINTS OF BASIC ROTATION?: Press NO to ignore the previous touch points. (The dialog question only appears if a basic rotation was made previously.)
- Probe both workpiece sides twice.
- Enter the datum coordinates and confirm your entry with ENT.
- To terminate the probe function, press the END key.

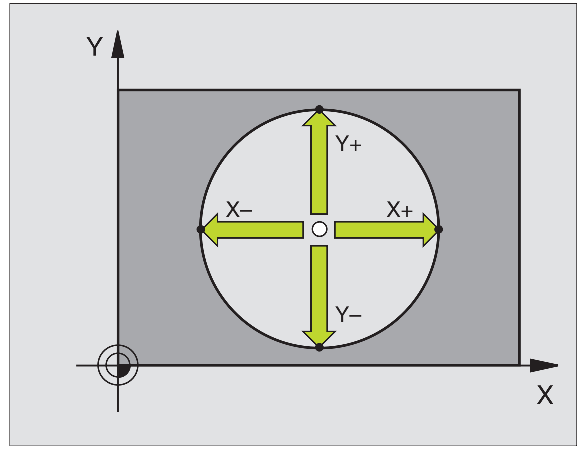
Circle center as datum

With this function, the centers of bore holes, circular pockets, cylinders, studs, circular islands, etc. can be defined as datums.

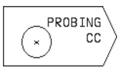
Inside circle

The TNC automatically probes the inside wall in all four coordinate axis directions.

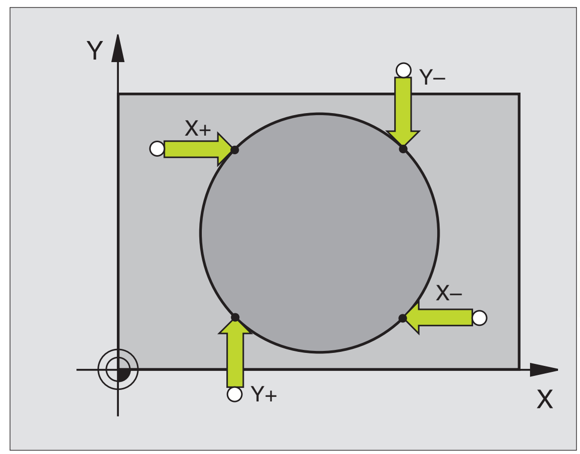
For incomplete circles (circular arcs) you can choose the appropriate probing direction.



- Position the touch probe approximately in the center of the circle.



- Select the touch probe function: Press the PROBING CC soft key
- To probe the workpiece, press the NC 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 available on machines with spindle orientation, depending on MP6160), 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: Enter both circle center coordinates into the menu window, and confirm your entry with ENT.
- 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 with a soft key.
- To probe the workpiece, press the NC START button
- Repeat the probing process for the remaining three points.See figure at center right.
- Enter the datum coordinates and confirm your entry with ENT.

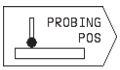
After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

11.3 Measuring Workpieces with a 3-D Touch Probe

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:



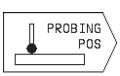
- To select the touch probe function: Press the PROBING POS soft key
- Move the touch probe to a starting position near the touch point.
- Select the probe direction and axis of the coordinate. Use the arrow keys for selection.
- To start probing, press the NC START button

The TNC shows the coordinates of the touch point as DATUM.

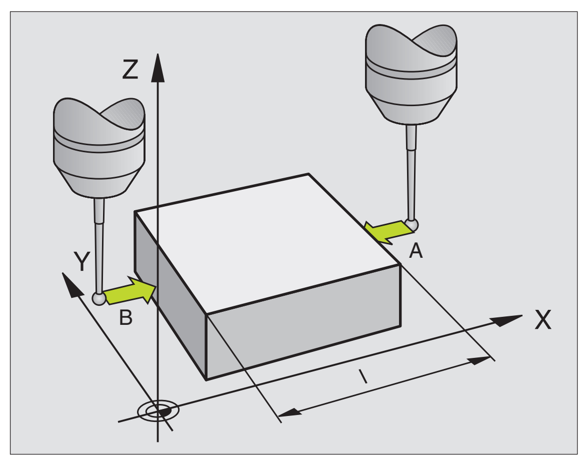
Finding the coordinates of a corner in the working plane

Find the coordinates of the corner point as described under "Corner as datum." The TNC displays the coordinates of the probed corner as datum.

Measuring workpiece dimensions



- To select the touch probe function: Press the PROBING POS soft key
- Position the touch probe at a starting position near the first touch point A.
- Select the probe direction with the arrow key.
- To probe the workpiece, press the NC 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 touch probe function again: Press the PROBING POS soft key



- Position the touch probe at a starting position near the second touch point B.
- Select the probe direction with an arrow key: Same axis but opposite direction as for A.
- To probe the workpiece, press the NC 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:

- To select the touch probe function: Press 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.

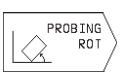
Measuring angles

You can also 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°.

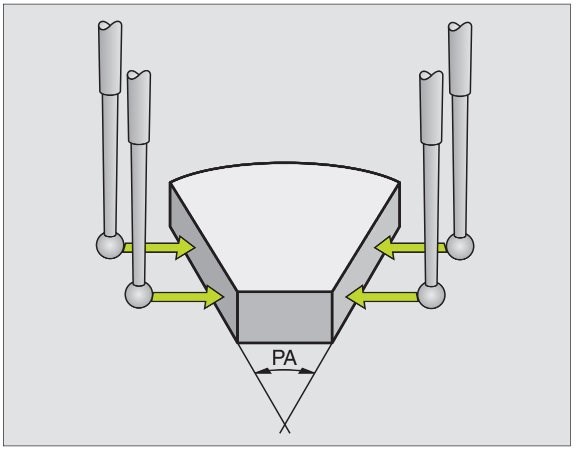
To find the angle between the angle reference axis and a side of the workpiece:



- Select the touch probe function: Press 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”).
- 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:

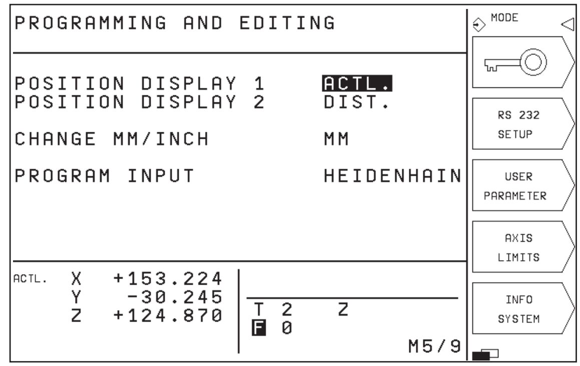
- To select the touch probe function: Press the PROBING ROT soft key
- ROTATION ANGLE: If you will need the current basic rotation later, write down the value that appears under ROTATION ANGLE.
- Make a basic rotation for the first side (see "Compensating workpiece misalignment").
- Probe the second 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.



12

MOD Functions

12.1 Selecting, Changing and Exiting the MOD Functions



The MOD functions provide additional displays and input possibilities.

To select the MOD functions
Call the mode of operation in which you wish to change the MOD function.

MOD

▶ To select the MOD functions, press the MOD key. The figure at upper right shows the „MOD screen".

You can make the following changes:

- Select position display
- Unit of measurement (mm/inches)
- Enter code number
- Set data interface
- Machine-specific user parameters
- Axis traverse limits
- Display NC software number
- Display PLC software number

To change the MOD functions:

▶ Select the desired MOD function in the displayed menu with the arrow keys.

▶ Press the ENT key repeatedly until the desired function is highlighted, or enter the appropriate numbers and confirm your entry with ENT.

To exit the MOD functions:

▶ To exit the MOD function, press the END key.

12.2 System Information

You can use the soft key INFO SYSTEM to display the following information:

- Free program memory
- NC software number
- PLC software number

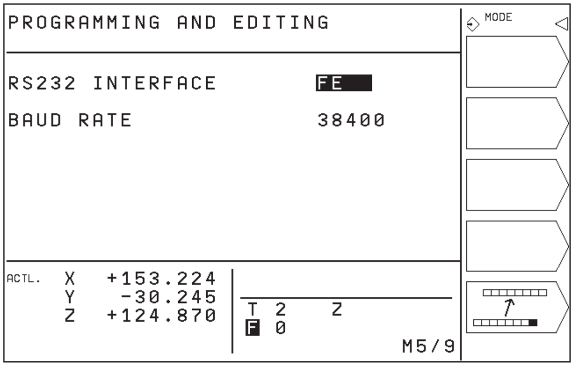
are displayed on the TNC screen after the functions have been selected.

12.3 Enter Code Number

To enter the code number, press the soft key with the key symbol. The TNC requires a code number for the following functions:

Function	Code number
Selecting user parameters	123
Cancel file protection	86357
Operating hours counter for: CONTROL ON PROGRAM RUN SPINDLE ON	857282

12.4 Setting the Data Interface



Press the soft key marked RS 232 SETUP to call a menu with the following options for setting the data interfaces:

Setting the OPERATING MODE of the external device

External device	RS232 INTERFACE
HEIDENHAIN floppy disk unit FE 401 and FE 401B	FE
Non-HEIDENHAIN devices such as tape punchers, PC without TNC.EXE	EXT1, EXT2
PC with HEIDENHAIN software TNC.EXE	FE
No data transfer; e.g. working without a connected external device	none

Setting the BAUD RATE

You can set the BAUD-RATE (data transfer speed) from 110 to 115,200 baud. The TNC stores an individual BAUD RATE for each operating mode (FE, EXT1 etc.). When you select BAUD RATE using an arrow key, the TNC recalls the value that was last stored for this operating mode.

12.5 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 USER PARAMETERS. Refer to your machine manual.

12.6 Position Display Types

In the MANUAL OPERATION mode and in the program run modes of operation, you can select the type of coordinates to be displayed.

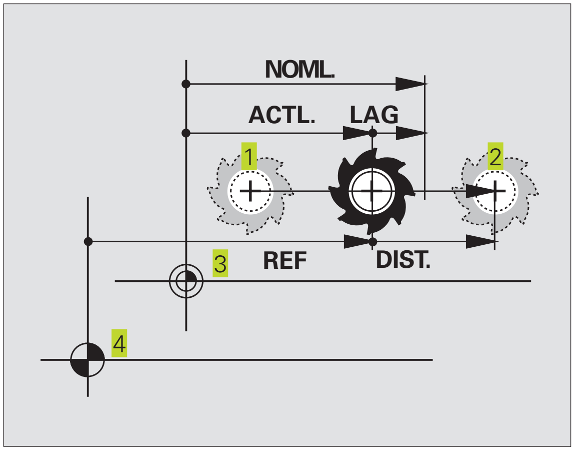
The figure at right shows the different tool positions:

1 Starting position

2 Target position of the tool

3 Workpiece datum

4 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 as referenced to the machine datum	REF
Distance remaining to the programmed position; difference between actual and target positions	DIST.
Servo lag: difference between nominal and actual positions	LAG

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.

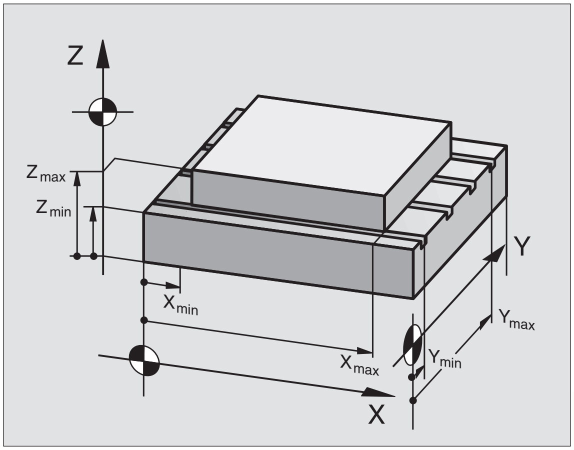
12.7 Unit of Measurement

The MOD function CHANGE MM/INCHES 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 with 3 digits after the decimal point.
- To select the inch system (e.g. X = 0.6216 inch) set the CHANGE MM/INCHES function to INCHES. The value is displayed to 4 decimal places.

This MOD function also determines the unit of measurement when you open a new program.

12.8 Enter Axis Traverse Limits



The AXIS LIMIT mod function allows you to set limits to axis traverse within the machine's actual working envelope.

Possible application:
to protect an indexing fixture against tool collision.

The maximum range of traverse of the machine tool is defined by software limit switches. This range can be additionally limited through the AXIS LIMIT mod function. With this function, you can enter the maximum and minimum traverse positions for each axis, referenced to the machine datum.

Working without additional traverse limits
To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 30 000 mm) as the AXIS LIMIT.

To find and enter the maximum traverse:

- ▶ Select POSITION DISPLAY REF
- ▶ Move the spindle to the positive and negative end positions of the X, Y and Z axes.
- ▶ Write down the values, including the algebraic sign.
- ▶ To select the MOD functions, press the MOD key.

AXIS LIMITS

- ▶ To enter the traverse limits, press the AXIS LIMIT soft key and enter the values that you wrote down as LIMITS in the corresponding axes.
- ▶ To exit the MOD function, press the END key.

> The tool radius is not automatically compensated in the axis traverse limit value.
>
> The traverse range limits and software limit switches become active as soon as the reference points are traversed.

13

Tables and Overviews

13.1 General User Parameters

General user parameters are machine parameters affecting TNC settings that the user may want to change in accordance with his requirements.

Some examples of user parameters are:

- Dialog language
- Interface behavior
- Traversing speeds
- Sequence of machining
- Effect of overrides

Input possibilities for machine parameters

Enter the machine parameters as decimal numbers

Some machine parameters have more than one function. The input value for these machine parameters is the sum of the individual values. For these machine parameters the individual values are preceded by a plus sign.

Selecting general user parameters

General user parameters are selected with code number 123 in the MOD functions.

> The MOD functions also include machine-specific user parameters (USER PARAMETERS).

External data transfer

Determining the control character for blockwise transfer

Integrating TNC interfaces EXT1 (5020.0) and EXT2 (5020.1) to an external device

	MP5020.x
	7 data bits (ASCII code, 8th bit = parity): **+0** 8 data bits (ASCII code, 9th bit = parity): **+1**
	Block Check Character (BCC) any: **+0** Block Check Character (BCC) control character not permitted: **+2**
	Transmission stop through RTS active: **+4** Transmission stop through RTS inactive: **+0**
	Transmission stop through DC3 active: **+8** Transmission stop through DC3 inactive: **+0**
	Character parity even: **+0** Character parity odd: **+16**
	Character parity not desired: **+0** Character parity desired: **+32**
	$1^1/_2$ stop bits: **+0** 2 stop bits: **+64**
	1 stop bit: **+128** 1 stop bit: **+192**
	RTS always active: **+0** RTS only active if data transfer has been started: **+256**
	Send EOT after ETX: **+0** Do not send EOT after ETX: **+512**

Example:
Use the following setting to adjust the TNC interface EXT2 (MP 5020.1) to an external non-HEIDENHAIN device:

8 data bits, any BCC, transmission stop through DC3, even character parity, character parity desired, 2 stop bits
Input for **MP 5020.1**: 1+0+8+0+32+64 = **105**

3-D Touch Probes

Probing feed rate for triggering touch probes	**MP6120** **80** to **3000** [mm/min]
Maximum traverse to first probe point	**MP6130** **0.001** to **30 000** [mm]
Safety clearance to probing point during automatic measurement	**MP6140** **0.001** to **30 000** [mm]
Rapid traverse for triggering touch probes	**MP6150** **1** to **30 000** [mm/min]
Measure center misalignment of the stylus when calibrating a triggering touch probe	**MP6160** No 180° rotation of the 3-D touch probe during calibration: **0** M function for 180° rotation of the touch probe during calibration: **1** to **88**

TNC displays, TNC editor

Programming station	**MP7210** TNC with machine: **0** TNC as programming station with active PLC: **1** TNC as programming station with inactive PLC: **2**
Acknowledgment of POWER INTERRUPTED after switch-on	**MP7212** Acknowledge with key: **0** Acknowledge automatically: **1**
Dialog language	**MP7230** German: **0** English: **1**
Configure tool tables	**MP7260** Inactive: **0** Number of tools in the tool table: **1** to **99**

MANUAL OPERATION MODE: Display of feed rate	**MP7270** Display feed rate F only if an axis direction button is pressed: **+0** Display feed rate F even if no axis direction button is pressed (feed rate of the „slowest“ axis): **+1** Spindle speed S and miscellaneous function M effective after STOP: **+0** Spindle speed S and miscellaneous function M no longer effective after STOP: **+2**
Decimal character	**MP7280** The decimal character is a comma: **0** The decimal character is a point: **1**
Position display in the tool axis	**MP7285** Display is referenced to the tool datum: **0** Display in the tool axis is referenced to the tool face: **1**
Display step for the X axis	**MP7290.0** 0.1 mm or 0.1°: **0** 0.05 mm or 0.05°: **1** 0.01 mm or 0.01°: **2** 0.005 mm or 0.005°: **3** 0.001 mm or 0.001°: **4**
Display step for the Y axis	**MP7290.1** see MP 7290.0
Display step for the Z axis	**MP7290.2** see MP 7290.0
Display step for the IVth axis	**MP7290.3** see MP 7290.0
Reset status display, Q parameters and tool data	**MP7300** Do not cancel Q parameters and status display: **+0** Q parameters and status display with M02, M30, END PGM: **+1** After a power interruption do not activate tool data that was last active: **+0** After a power interruption re-activate tool data that was last active: **+4**

Graphic display mode

MP7310
Projection in three planes according to ISO 6433, projection method 1: **+0**
Projection in three planes according to ISO 6433, projection method 2: **+1**
Do not rotate coordinate system for graphic display: **+0**
Rotate coordinate system for graphic display by 90°: **+2**

Machining and program run

Cycle 17: Oriented spindle stop at beginning of cycle

MP7160
Spindle orientation: **0**
No spindle orientation: **1**

Effect of Cycle 11 SCALING FACTOR

MP7410
SCALING FACTOR effective in 3 axes: **0**
SCALING FACTOR effective in the working plane only: **1**

Cycle 4 POCKET MILLING and Cycle 5 CIRCULAR POCKET MILLING: Overlap factor

MP7430
0.1 to **1.414**

Angle of tool-path directional change up to which the feed rate will remain constant (applies for radius-compensated inside corners and corners with R0)
This feature works both during operation with servo lag as well as with velocity feedforward.

MP7460
0.000 to **179.999** [°]

Maximum contouring speed at a feed rate override setting of 100% in the program run modes

MP7470
0 to **99 999** [mm/min]

Electronic handwheels

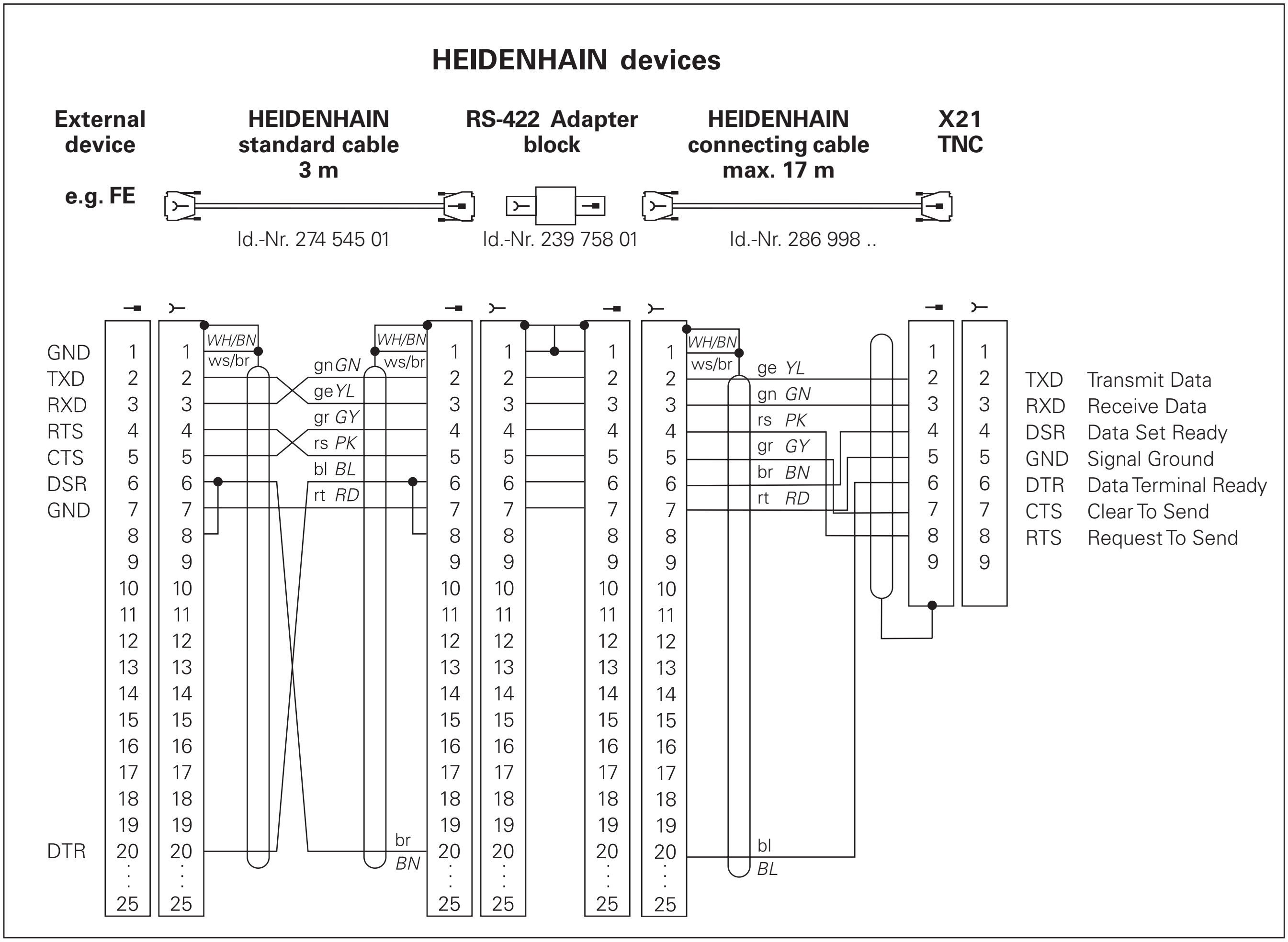
Handwheel type

MP7640
Machine without handwheel: **0**
HR 330 with additional keys _ the handwheel keys for traverse direction and rapid traverse are evaluated by the NC: **1**
HR 130 without additional keys: **2**
HR 330 with additional keys _ the handwheel keys for traverse direction and rapid traverse are evaluated by the PLC: **3**
HR 332 with twelve additional keys: **4**
Multi-axis handwheel with additional keys: **5**
HR 410 with miscellaneous functions: **6**

13.2 Pin Layout and Connecting Cable for the Data Interface

RS-232-C/V.24 Interface

HEIDENHAIN devices



☞ The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

Non-HEIDENHAIN devices

The connector pin layout of a non-HEIDENHAIN device may differ considerably from that of a HEIDENHAIN device.

This often depends on the unit and type of data transfer. The figure above shows the connector pin layout on the adapter block.

13.3 Technical Information

TNC features

Description	Contouring control for machines with: 4 controlled axes and non-controlled spindle 3 controlled axes and controlled spindle
Components	Compact control with integrated flat-panel display (192 mm x 120 mm, 640 x 400 pixels) and integrated operating keys
Data interface	■ RS-232-C / V.24
Simultaneous axis control for contour elements	■ Straight lines: up to 3 axes ■ Circles: up to 2 axes ■ Helixes: 3 axes
Background programming	One part program can be edited while the TNC runs another program
Graphics	■ Interactive programming graphics ■ Test run graphics
File types	■ HEIDENHAIN conversational programming ■ Tool table
Program memory	■ Buffer battery back-up for approx. 6000 NC blocks (dependent on length of block), 128 KB ■ Up to 64 files possible
Tool definitions	Up to 254 tools in the program or up to 99 in the table
Programming support	■ Functions for approaching and departing the contour ■ HELP

Programmable functions

Contour elements	■ Straight line ■ Chamfer ■ Circular arc ■ Circle center ■ Circle radius ■ Tangentially connecting circle ■ Corner rounding ■ Straight lines and circular arcs for contour approach and departure
Program jumps	■ Subprograms ■ Program section repeats
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring, tapping with a floating tap holder, rigid tapping ■ Milling and finishing rectangular and circular pockets ■ Cycles for milling linear and circular slots ■ Linear and circular hole patterns ■ Cycles for multipass milling of flat and twisted surfaces
Coordinate transformations	■ Datum shift ■ Mirroring ■ Rotation ■ Scaling
3-D touch probe applications	■ Touch probe functions for setting the datum

TNC Specifications

Block processing time	40 ms/block
Control loop cycle time	Contouring interpolation: 6 ms
Data transfer rate	Maximum 115 200 baud
Ambient temperature	■ In operation: 0°C to +45°C ■ Storage: –30°C to +70°C
Traverse range	Maximum 30 m (1181 inches)
Traversing speed	Maximum 30 m/min (1181 inches/min)
Spindle speed	Maximum 30 000 rpm
Input range	■ Minimum 1 µm (0.0001 inches) or 0.001° ■ Maximum 30 000 mm (1181 inches) or 30 000°

13.4 TNC Error Messages

The TNC automatically 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

Some of the more frequent TNC error messages are explained in the following list.

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. To clear the TNC error message, first correct the error and then press the CE key.

TNC error messages during programming

FURTHER PROGRAM ENTRY IMPOSSIBLE	Erase some old files to make room for new ones.
ENTRY VALUE INCORRECT	■ Enter a correct label number. ■ Note the input limits.
EXT. IN-/OUTPUT NOT READY	■ Connect the data transfer cable ■ Transfer cable is defective or not soldered properly ■ Switch on the connected device (PC, printer). ■ The data transfer speeds (baud rates) are not identical
PROTECTED PGM !	Cancel edit protection if you wish to edit the program.
LABEL NUMBER ALREADY ASSIGNED	A given label number can only be entered once in a program.
JUMP TO LABEL 0 NOT PERMITTED	Do not program CALL LBL 0.

TNC error messages during test run and program run

AXIS DOUBLE PROGRAMMED	Each axis can have only one value for position coordinates.
SELECTED BLOCK NOT ADDRESSED	Before a test run or program run, you must enter GOTO 0.
TOUCH POINT INACCESSIBLE	■ Pre-position the 3-D touch probe to a position nearer the model
ARITHMETICAL ERROR	You have calculated with non-permissible values ■ Define values within the range limits. ■ Choose probe positions for the 3-D touch probe that are farther apart.
PATH OFFSET WRONGLY ENDED	Do not cancel tool radius compensation in a block with a circular path.
PATH OFFSET WRONGLY STARTED	■ Use the same radius compensation before and after an RND and CHF block. ■ Do not begin tool radius compensation in a block with a circular path

CYCL INCOMPLETE	■ Define the cycles with all data in the proper sequence ■ Do not call the coordinate transformation cycles ■ Define a cycle before calling it ■ Enter a pecking depth other than 0
BLK FORM DEFINITION INCORRECT	■ Program the MIN and MAX points according to the instructions ■ Choose a ratio of sides that is less than 200:1
PLANEWRONGLY DEFINED	■ Do not change the tool axis while a basic rotation is active ■ Correctly define the main axes for circular arcs ■ Define both main axes for CC
WRONGAXIS PROGRAMMED	■ Do not attempt to program locked axes. ■ Program a rectangular pocket or slot in the working plane. ■ Do not mirror rotary axes. ■ Enter a positive chamfer length.
WRONG RPM	Program a spindle speed within the programmed range.
CHAMFER NOT PERMITTED	A chamfer block must be located between two straight-line blocks with identical radius compensation.
FAULTY PROGRAM DATA	The program that was read in through the data interface contains incorrect block formats.
GROSS POSITIONING ERROR	The TNC monitors positions and movements. If the actual position deviates excessively from the nominal position, this blinking error message is displayed. You must press the END key for a few seconds to acknowledge the error message (system reset).
NO EDITING OF RUNNING PROGRAM	A program cannot be edited while it is being executed.
CIRCLE END POS. INCORRECT	■ Enter complete information for the connecting arc. ■ Enter end points that lie on the circular path.
CIRCLE CENTER UNDEFINED	■ Define a circle center with the CC key ■ Define a pole with the CC key
LABEL NUMBER NOT FOUND	Only call label numbers that have been set.
SCALING FACTOR NOT PERMITTED	Enter identical scaling factors for coordinate axes in the plane of the circle.
PGM-SECTION CANNOT BE SHOWN	■ Enter a smaller tool radius. ■ Enter a tool axis for simulation that is the same as the axis in the BLK FORM.
RADIUS COMP. UNDEFINED	Radius compensation RR or RL is only possible when tool radius is not equal to 0
ROUNDING-OFF NOT PERMITTED	Enter tangentially connecting arcs and rounding arcs correctly.
ROUNDING RADIUSTOOL LARGE	Rounding arcs must fit between contour elements.

KEY NON-FUNCTIONAL	This message always appears when you press a key that is not needed for the current dialog.
STYLUS ALREADY IN CONTACT	Before probing, pre-position the stylus where it is not touching the workpiece surface.
PROBE SYSTEM NOT READY	■ Check whether the touch probe is ready for operation
PROGRAM START UNDEFINED	■ Begin the program only with a TOOL DEF block ■ Do not resume an interrupted program at a block with a tangential arc or if a previously defined pole is needed.
FEED RATE IS MISSING	■ Enter the feed rate for the positioning block. ■ Enter FMAX in each block.
TOOL RADIUSTOO LARGE	Enter a tool radius that ■ lies within the given limits ■ permits the contour elements to be calculated and machined
ANGLE REFERENCE MISSING	■ Complete your definition of the arc and its end points. ■ If you enter polar coordinates, define the polar angle correctly.
EXCESSIVE SUBPROGRAMMING	■ Conclude all subprograms with LBL0. ■ Program CALL LBL for subprograms without REP. ■ Program CALL LBL for program section repeats to include the repetitions (REP). ■ Subprograms cannot call themselves. ■ Subprograms cannot be nested in more than eight levels.

13.5 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. The batteries are located in the control housing (refer also to your Machine Manual). The TNC also has an power storage device that provide the control with current while you are exchanging the batteries (for a maximum of 24 hours).

> ☞ To exchange the buffer battery, first switch off the TNC!
>
> The buffer battery must be exchanged only by trained service personnel!

Battery type: Three AA-size cells, leak-proof, IEC designation „LR6“

M	Effect of M Function	Effective at block start	end	Page
M00	Stop program run/Spindle STOP/Coolant OFF		■	**77**
M01	Optional program run interruption		■	**158**
M02	Stop program run/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	**77**
M03	Spindle ON clockwise	■		
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	**77**
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	**77**
M08	Coolant ON	■		
M09	Coolant OFF		■	**77**
M13	Spindle ON clockwise/Coolant ON	■		
M14	Spindle ON counterclockwise/Coolant ON	■		**77**
M30	Same function as M02		■	**77**
M89	Vacant miscellaneous function **or**	■		
	Cycle call, modally effective (depending on machine parameter)		■	**85**
M90	Only in lag mode: Constant contouring speed at corners	■		**79**
M91	Within the positioning block: Coordinates are referenced to machine datum		■	**77**
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		**77**
M94	Reduce display of rotary axis to value less than 360°	■		**82**
M97	Machine small contour steps		■	**80**
M98	Completely machine open contours		■	**87**
M99	Blockwise cycle call		■	**85**

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

Ve 00
313 509-22 · 7/2000 · pdf · Subject to change without notice