HEIDENHAIN

TNC 410
TNC 426
TNC 430

NC Software
286 060-xx
286 080-xx
280 472-xx
280 473-xx
280 474-xx
280 475-xx

User's Manual
ISO Programming

4/99

Controls on the TNC

Controls on the visual display unit

- Split screen layout
- Switch between machining or programming modes
- Soft keys for selecting functions in screen
- Switching the soft-key rows
- Changing the screen settings (only BC 120)

Typewriter keyboard for entering letters and symbols

Q W E R T Y	File name Comments
G F S T M	ISO programs

Machine operating modes

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

Programming modes

- Programming and Editing
- Test run

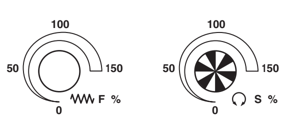
Program/file management, TNC functions

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

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

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

Override control knobs for feed rate/spindle speed



Programming path movements (only conversational)

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

Tool data (only conversational)

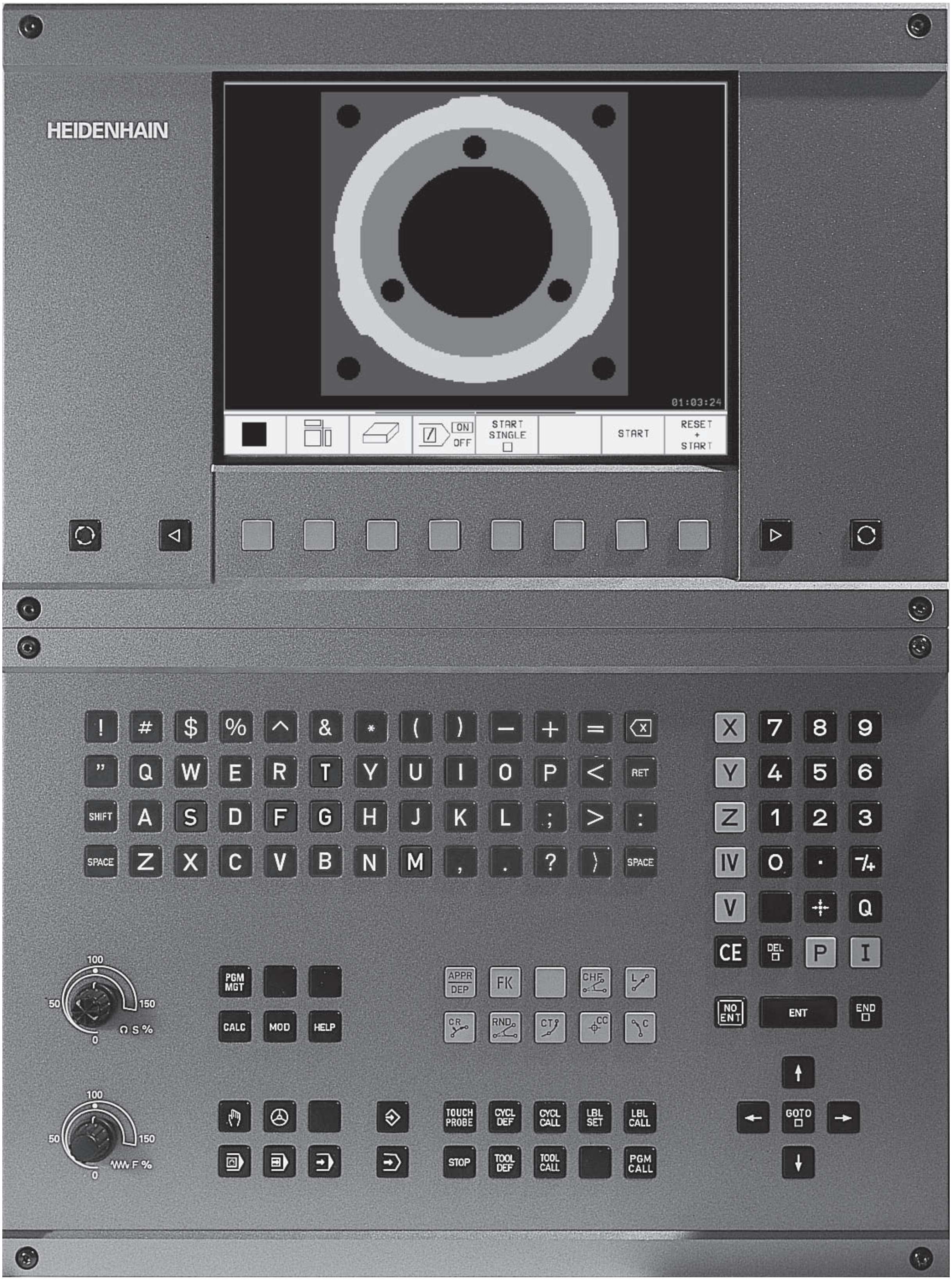
TOOL DEF, TOOL CALL	Entering and calling tool length and radius

Cycles, subprograms, and program section repeats (only conversational)

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

Coordinate axes and numbers, editing

X ... V	Select coordinate axes or enter them in a program
0 ... 9	Numbers
.	Decimal point
-/+	Change arithmetic sign
P	Polar coordinates
I	Incremental dimensions
Q	Q parameters
	Capture actual position
NO ENT	Skip dialog questions, delete words
ENT	Confirm entry and resume dialog
END	End block
CE	Clear numerical entry or TNC error message
DEL	Abort dialog, delete program section


HEIDENHAIN
01:03:24
ON
OFF
START
SINGLE
START
RESET
+
START
! # $ % ^ & * ( ) — + =
" Q W E R T Y U I O P < RET
SHIFT A S D F G H J K L ; > :
SPACE Z X C V B N M , . ?) SPACE
X 7 8 9
Y 4 5 6
Z 1 2 3
IV 0 . -/+
V Q
CE DEL P I
NO ENT
ENT
END
GOTO
PGM MGT
CALC
MOD
HELP
APPR DEP
FK
CHF
RND
CT
CC
TOUCH PROBE
CYCL DEF
CYCL CALL
LBL SET
LBL CALL
STOP
TOOL DEF
TOOL CALL
PGM CALL
100
50
150
0
S %
F %

TNC Models, Software and Features

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

TNC Model	NC Software No.
TNC 410	286 060-xx
TNC 410	286 080-xx
TNC 426 CB, TNC 426 PB	280 472-xx
TNC 426 CF, TNC 426 PF	280 473-xx
TNC 430 CA, TNC 430 PA	280 472-xx
TNC 430 CE, TNC 430 PE	280 473-xx
TNC 426 CB, TNC 426 PB	280 474-xx
TNC 426 CF, TNC 426 PF	280 475-xx
TNC 426 M	280 474-xx
TNC 426 ME	280 475-xx
TNC 430 CA, TNC 430 PA	280 474-xx
TNC 430 CE, TNC 430 PE	280 475-xx
TNC 430 M	280 474-xx
TNC 430 ME	280 475-xx

The suffixes E and F indicate the export versions of the TNC which have the following limitations:

- Linear movement is possible in no more than 4 axes simultaneously

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

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

- Probing function for the 3-D touch probe
- Digitizing option (conversational programming only)
- Tool measurement with the TT 120 (conversational programming only)
- Rigid tapping
- Returning to the contour after an interruption

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.

Touch Probe Cycles User's Manual:

In addition to this manual, another manual is available describing all the touch probe functions of the TNC 426 / TNC 430. Please contact HEIDENHAIN if you require a copy of this User's Manual. ID number: 329 203-xx.

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.

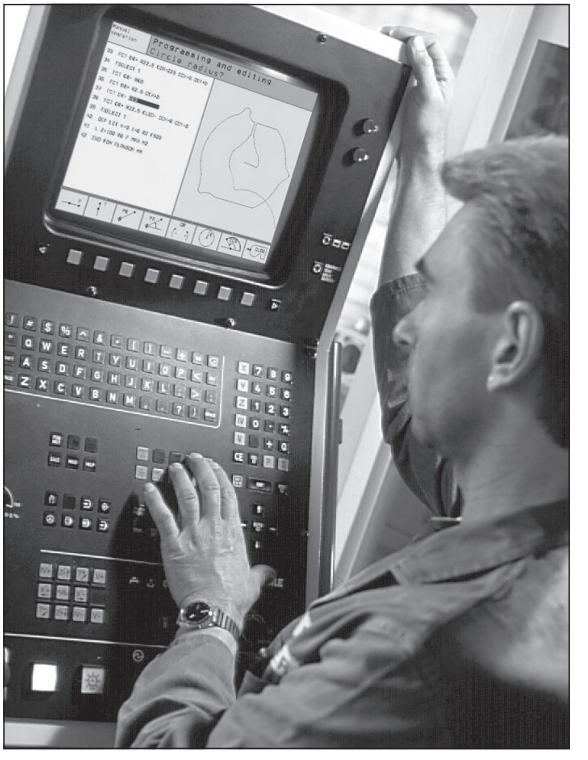
Contents

Contents

1

Introduction

1.1 The TNC 410, The TNC 426, and The TNC 430



HEIDENHAIN TNC controls are workshop-oriented contouring controls that enable you to program conventional machining operations right at the machine in an easy-to-use conversational programming language. They are designed for milling, drilling and boring machines, as well as for machining centers. The TNC 410 can control up to 4 axes, the TNC 426 up to 5 axes, and the TNC 430 up to 9 axes. 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 and ISO formats
HEIDENHAIN conversational programming is an especially easy method of writing programs. Interactive graphics illustrate the individual machining steps for programming the contour. If a production drawing is not dimensioned for NC, the HEIDENHAIN FK free contour programming carries out the necessary calculations automatically. Workpiece machining can be graphically simulated either during or before actual machining. It is also possible to program in ISO format or DNC mode.

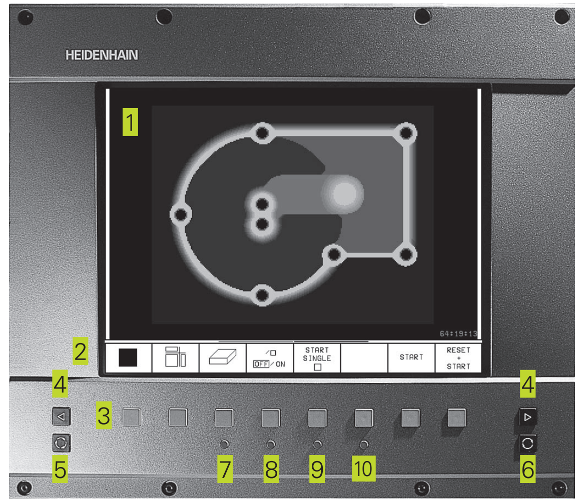
You can enter a program while the control is running another. With the TNC 426, TNC 430 it is also possible to test one program while another is being run.

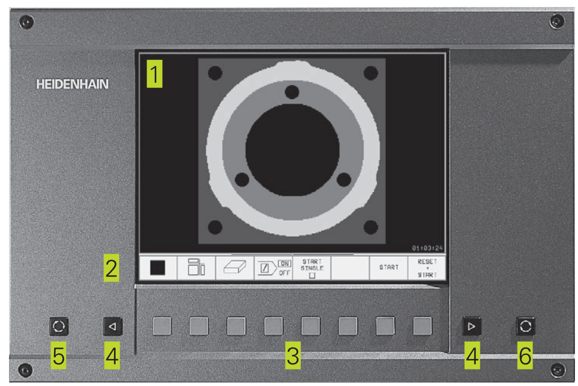
Compatibility
The TNC can execute all part programs that were written on HEIDENHAIN controls TNC 150 B and later.

1.2 Visual Display Unit and Keyboard

Visual display unit

The TNC is available with either a color CRT screen (BC 120) or a TFT flat panel display (BF 120. The figures at right show the keys and controls on the BC 120 (upper right) and the BF 120 (middle right).





1 Header
When the TNC is on, the selected operating modes are shown in the screen header. With the TNC 426, TNC 430, the machine operating modes are on the left and the programming modes are on the right. The currently active mode is displayed in the larger box, where the dialog prompts and TNC messages also appear (unless the TNC is showing only graphics).

2 Soft keys
In the footer the TNC indicates additional functions in a soft-key row. You can select these functions by pressing the keys immediately below them 3. The lines immediately above the soft-key row indicate the number of soft-key rows that can be called with the black arrow keys to the 4 outside right and left. The line representing the active soft-key row is highlighted.

3 Soft key selector keys

4 Switching the soft-key rows

5 Setting the screen layout

6 Shift key for switchover between machining and programming modes

Keys on BC 120 only

7 Screen demagnetization;
Exit main menu for screen settings

8 Select main menu for screen settings;

In the main menu:	Move highlight downward
In the submenu:	Reduce value
	Move picture to the left or downward

9

In the main menu:	Move highlight upward
In the submenu:	Increase value
	Move picture to the right or upward

10

In the main menu:	Select submenu
In the submenu:	Exit submenu

See next page for the screen settings.

Main menu dialog	Function
BRIGHTNESS	Adjust brightness
CONTRAST	Adjust contrast
H-POSITION	Adjust horizontal position
H-SIZE	Adjust picture width
V-POSITION	Adjust vertical position
V-SIZE	Adjust picture height
SIDE-PIN	Correct barrel-shaped distortion
TRAPEZOID	Correct trapezoidal distortion
ROTATION	Correct tilting
COLORTEMP	Adjust color temperature
R-GAIN	Adjust strength of red color
B-GAIN	Adjust strength of blue color
RECALL	No function

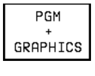
The BC 120 is sensitive to magnetic and electromagnetic noise, which can distort the position and geometry of the picture. Alternating fields can cause the picture to shift periodically or to become distorted.

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 (only TNC 410). The available screen windows depend on the selected operating mode.

To change the screen layout:

Press the switch-over key: The soft-key row shows the available layout options (see section 1.3 "Modes of Operation").



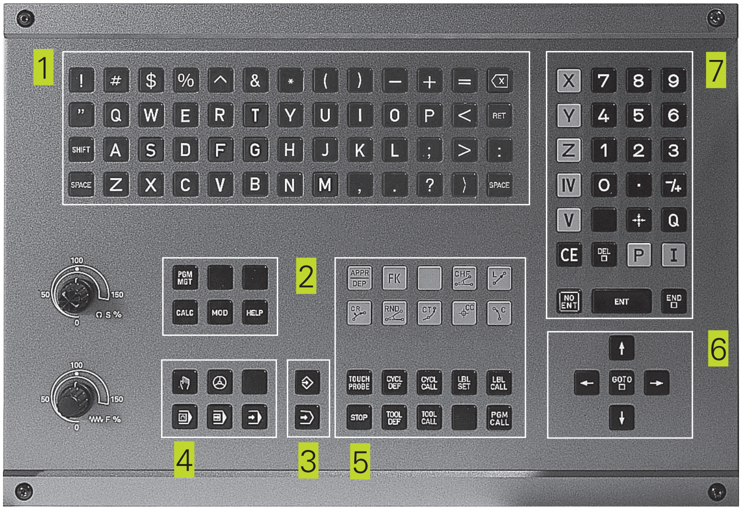
Select the desired screen layout.

Keyboard

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

- 1 Alphanumeric keyboard
  for entering texts and file names, as well as for programming in ISO format
- 2 File management,
  pocket calculator (not TNC 410),
  MOD function,
  HELP function
- 3 Programming modes
- 4 Machine operating modes
- 5 Initiation of programming dialog
- 6 Arrow keys and GOTO jump command
- 7 Numerical input and axis selection

The functions of the individual keys are described on the inside front cover. Machine panel buttons, e.g. NC START, are described in the manual for your machine tool.



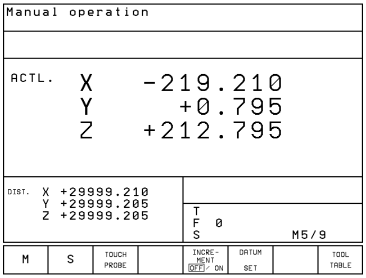
1.3 Modes of Operation

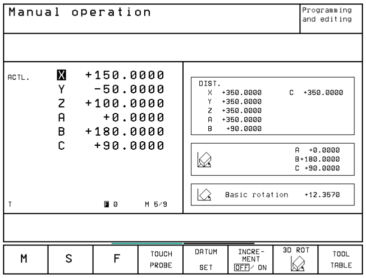
The TNC offers the following modes of operation for the various functions and working steps that you need to machine a workpiece:

Manual Operation and Electronic Handwheel

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, set the datums, and tilt the working plane.

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



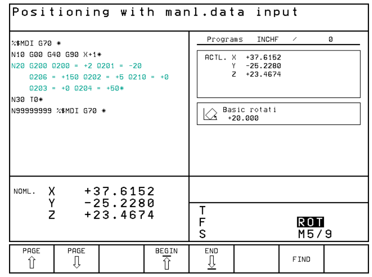


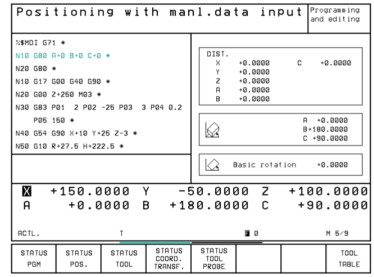
Soft keys for selecting the screen layout
(select as describe above, TNC 410: see screen layout with program run, full sequence)

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

Positioning with Manual Data Input (MDI)

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



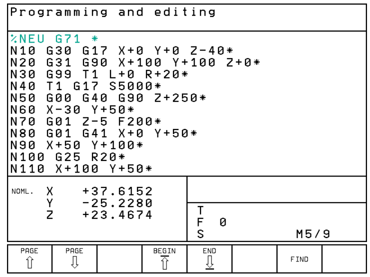


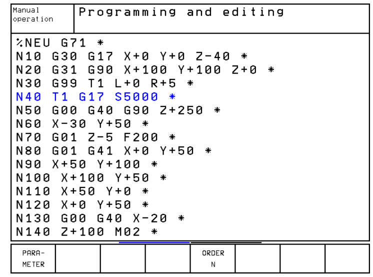
Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PROGRAMM
Left: positions. Right: status display. (only TNC 426, TNC 430)	POSITION + STATUS
Left: program. Right: general program information (only TNC 410)	PGM + PGM STATUS
Left: program. Right: positions and coordinates (only TNC 410)	PGM + POS. STATUS
Left: program. Right: information on tools (only TNC 410)	PGM + TOOL STATUS
Left: program. Right: coordinate transformations (only TNC 410)	PGM + C.TRANS. STATUS

Programming and Editing

In this mode of operation you can write your part programs. The various cycles and Q-parameter functions help you with programming and add necessary information.



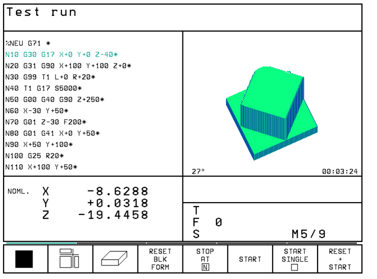


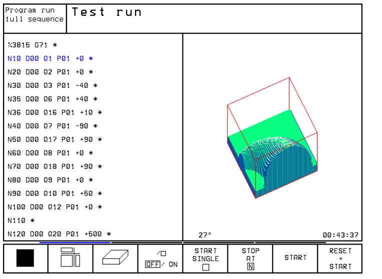
Soft keys for screen layout (not for TNC 426, TNC 430)

Screen windows	Soft key
Program	PROGRAMM
Left: program. Right: help graphics for cycle programming	PGM + FIGURE
Left: program. 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.





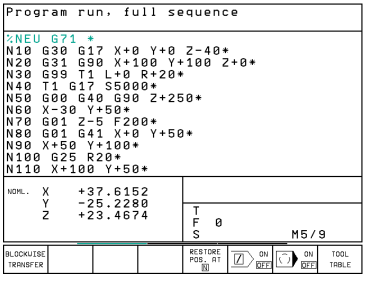
Soft keys for selecting the screen layout

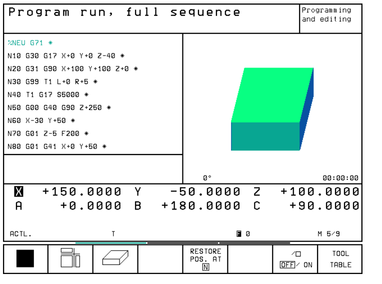
See Program Run, Full Sequence.

Program Run, Full Sequence and Program Run, Single Block

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

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





Soft keys for selecting the screen layout

Screen windows	Soft key
Program	PROGRAMM
Left: program. Right: STATUS (only TNC 426, TNC 430)	PGM + STATUS
Left: program blocks, right: graphics (only TNC 426, TNC 430)	PGM + GRAPHICS
Graphics (only TNC 426, TNC 430)	GRAPHICS

Screen windows	Soft key
Left: program. Right: general Program information (only TNC 410)	PGM + PGM STATUS
Left: program. Right: positions and coordinates (only TNC 410)	PGM + POS. STATUS
Left: program. Right: information on tools (only TNC 410)	PGM + TOOL STATUS
Left: program. Right: coordinate transformations (only TNC 410)	PGM + C.TRANS. STATUS
Left: program Right: tool measurement (only TNC 410)	PGM + T.PROBE STATUS

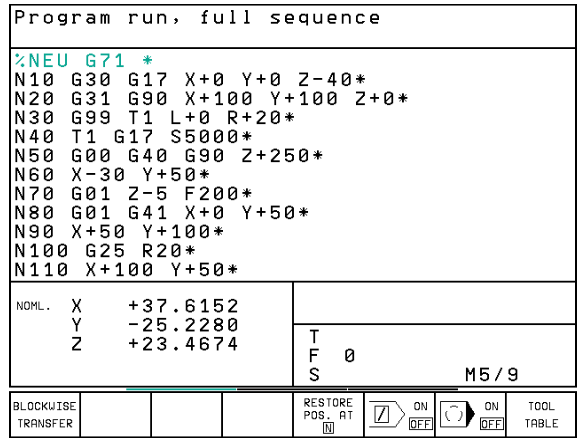
1.4 Status Displays

"General" status display

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

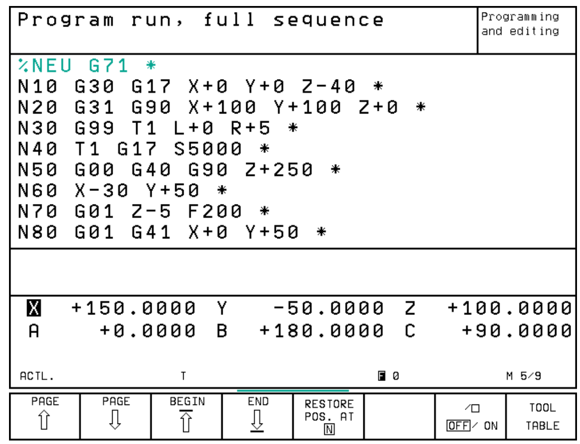
- Program Run, Single Block and Program Run, Full Sequence, except if the screen layout is set to display graphics only, and
- Positioning with Manual Data Input (MDI).

In the operating modes Manual and Electronic Handwheel, the status display is shown in the large window.



Information in the status display

The	Meaning
ACTL.	Actual or nominal coordinates of the current position
X Y Z	Machine axes; the TNC displays auxiliary axes in lower-case letters. The sequence and quantity of displayed axes is determined by the machine tool builder. Refer to your machine manual for more information
F S M	The displayed feed rate in inches corresponds to one tenth of the effective value. Spindle speed S, feed rate F and active M functions
*	Program run started
→|← ■	Axis locked
	Axis can be moved with the handwheel
	Axes are moving in a tilted working plane (not TNC 410)
	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 Programming and Editing mode of operation.

To switch on the additional status display:

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

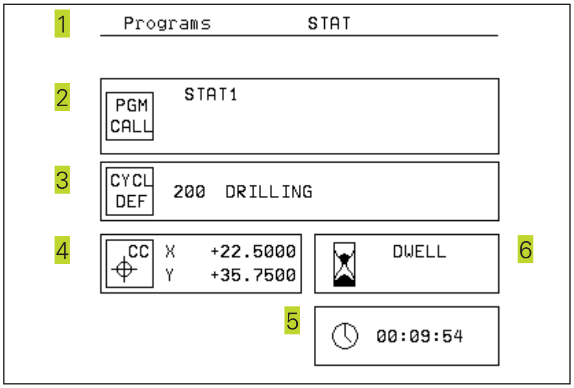
You can choose between several additional status displays with the following soft keys:

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

STATUS PGM

General program information

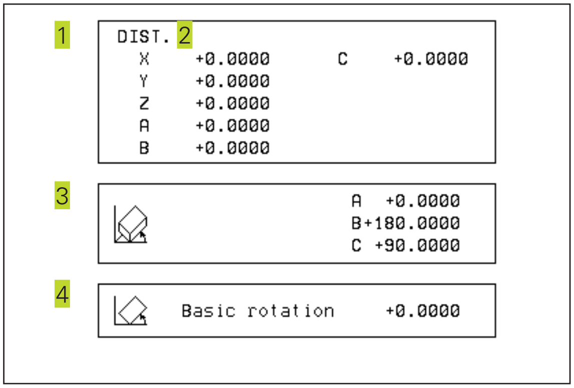
1 Name of main program
2 Active programs
3 Active machining cycle
4 Circle center CC (pole)
5 Operating time
6 Dwell time counter



STATUS POS.

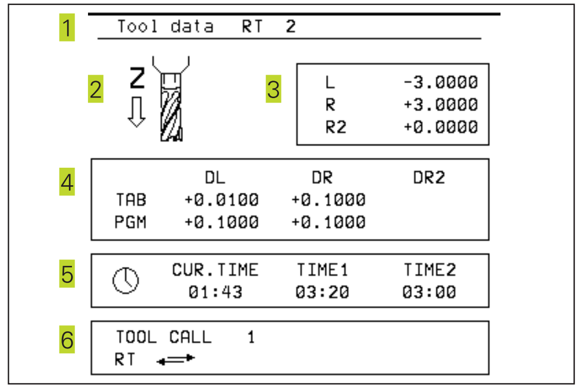
Positions and coordinates

1 Position display
2 Type of position display, e.g. actual positions
3 Tilting angle for the working plane (not TNC 410)
4 Angle of a basic rotation



STATUS TOOL

Information on tools



1 T: Tool number and name
RT: Number and name of a replacement tool

2 Tool axis

3 Tool length and radii

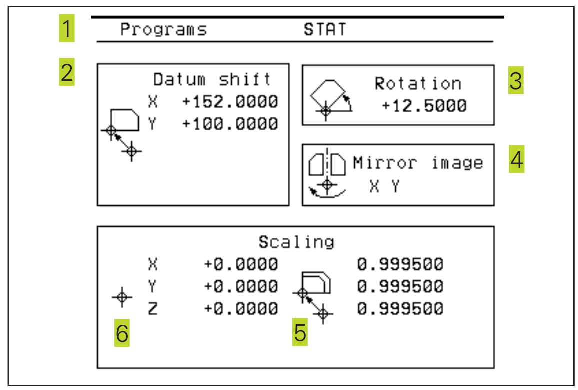
4 Oversizes (delta values) from TOOL CALL (PGM) and the tool table (TAB)

5 Tool life, maximum tool life (TIME 1) and maximum tool life for TOOL CALL (TIME 2)

6 Display of the active tool and the (next) replacement tool

STATUS COORD. TRANSF.

Coordinate transformations



1 Name of main program

2 Active datum shift (Cycle 7)

3 Active rotation angle (Cycle 10)

4 Mirrored axes (Cycle 8)

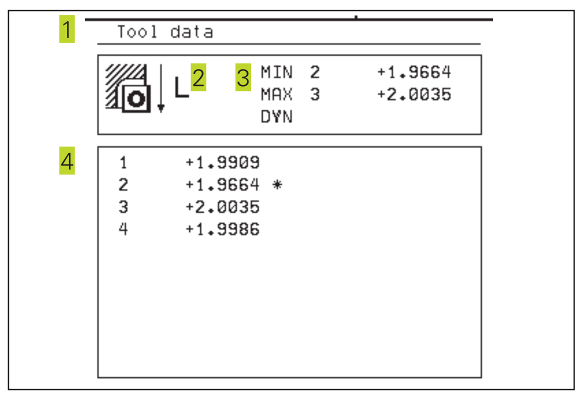
5 Active scaling factor(s) (Cycles 11 / 26)

6 Scaling datum

For further information, refer to section 8.8 “Coordinate Transformation Cycles.”

STATUS TOOL PROBE

Tool measurement



1 Number of the tool to be measured

2 Display whether the tool radius or the tool length is being measured

3 MIN and MAX values of the individual cutting edges and the result of measuring the rotating tool (DYN = dynamic measurement)

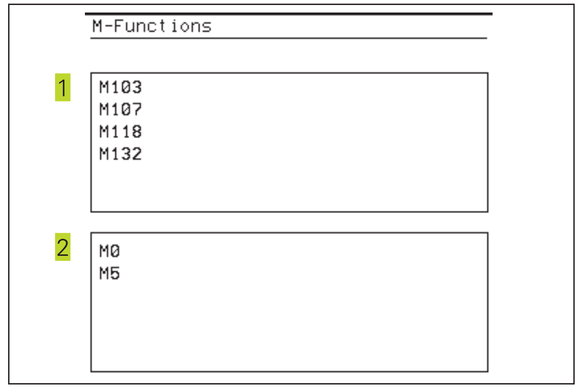
4 Cutting edge number with the corresponding measured value. If the measured value is followed by an asterisk, the allowable tolerance in the tool table was exceeded.

STATUS OF M FUNCT.

Active miscellaneous functions M (only TNC 426, TNC 430 with NC software 280 474-xx)

1 List of the active M functions with fixed meaning.

2 List of the active M functions with function assigned by machine manufacturer.



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

3-D Touch Probes

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

- Automatically align workpieces
- Quickly and precisely set datums
- Measure the workpiece during program run
- Digitize 3-D surfaces (option), and
- Measure and inspect tools

TS 220 and TS 630 touch trigger probes
These touch probes are particularly effective for automatic workpiece alignment, datum setting, workpiece measurement and for digitizing. The TS 220 transmits the triggering signals to the TNC via cable and is a cost-effective alternative for applications where digitizing is not frequently required.

The TS 630 features infrared transmission of the triggering signal to the TNC. This makes it highly convenient for use on machines with automatic tool changers.

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

During digitizing the TNC generates a program containing straight line blocks in HEIDENHAIN format from a series of measured position data. You can then output the program to a PC for further processing with the SUSA evaluation software. This evaluation software enables you to calculate male/female transformations or correct the program to account for special tool shapes and radii that differ from the shape of the stylus tip. If the tool has the same radius as the stylus tip you can run these programs immediately.

TT 120 tool touch probe for tool measurement
The TT 120 is a triggering 3-D touch probe for tool measurement and inspection. Your TNC provides three cycles for this touch probe with which you can measure the tool length and radius automatically—either with the spindle rotating or stopped (only for conversational programming).

The TT 120 features a particularly rugged design and a high degree of protection, which make it insensitive to coolants and swarf. The triggering signal is generated by a wear-resistant and highly reliable optical switch.

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

2

Manual Operation and Setup

2.1 Switch-on, Switch-off

Switch-On

Switch-on and traversing the reference points can vary depending on the individual machine tool. Your machine manual provides more detailed 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 compiled.

Relay Ext. DC Voltage Missing

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

Manual Operation
Traverse Reference Points

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

X Y — Cross the reference points in any sequence: Press and hold the external direction button for each axis until the reference point has been traversed, or additionally for the TNC 410

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

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

For the TNC 426, TNC 430:

The reference points need only be traversed if the machine axes are to be moved. If you intend only to write, edit or test programs, you can select the Programming and Editing or Test Run modes of operation immediately after switching on the control voltage.

You can then traverse the reference points later by pressing the PASS OVER REFERENCE soft key in the Manual Operation mode.

Traversing the reference point in a tilted working plane

The reference point of a tilted coordinate system can be traversed by pressing the machine axis direction buttons. The "tilting the working plane" function (see section 2.5 "Tilting the Working Plane") must be active in the Manual Operation mode. The TNC then interpolates the corresponding axes.

The NC START button is not effective. Pressing this button may result in an error message.

Make sure that the angle values entered in the menu for tilting the working plane match the actual angle of the tilted axis.

Switch-off

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

▶ Select the Manual mode

OFF

▶ Select the function for run-down, confirm again with the YES soft key.

▶ When the TNC displays the message „Now you can switch off the TNC" in a superimposed window, you may cut off the power supply to the TNC.

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

2.2 Moving the Machine Axes

Traversing with the machine axis direction buttons is a machine-dependent function. Refer to your machine tool manual.

To traverse with the machine axis direction buttons:

Select the Manual Operation mode.

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

...or move the axis continuously:

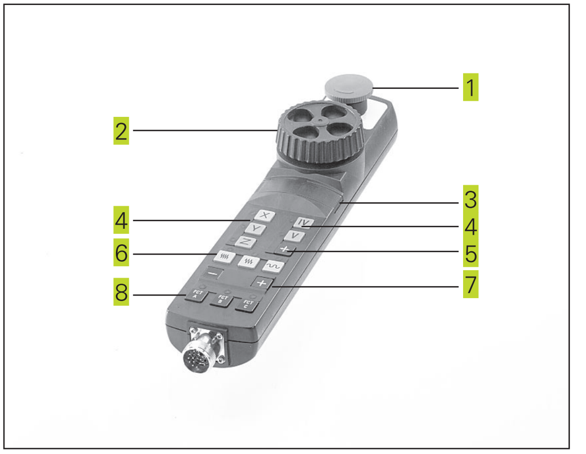
X and I — Press and hold the machine axis direction button, then press the machine START button: The axis continues to move after you release the keys.

0 — To stop the axis, press the machine STOP button.

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

You can change the feed rate at which the axes are traversed with the F soft key (see section 2.3 "Spindle Speed S, Feed Rate F and Miscellaneous Functions M"). This function is not available on TNC 410.

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

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.

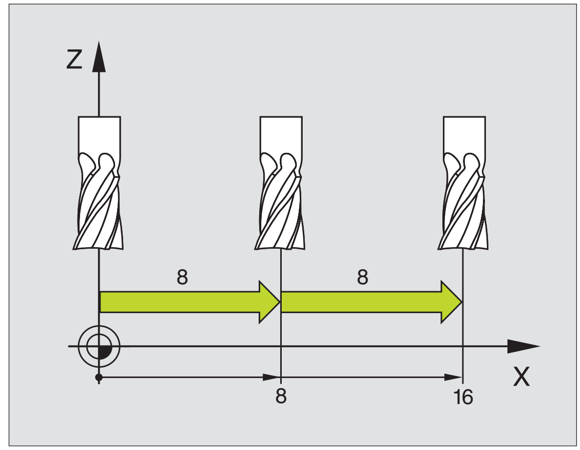
It is also possible to move the machine axes with the handwheel during a program run.

To move an axis:

	Select the Electronic Handwheel mode of operation
	Press and hold the permissive button.
X	Select the axis.
	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.



	Select Manual or Electronic Handwheel mode of operation
INCRE-MENT OFF / ON	Select incremental jog positioning: Switch the INCREMENT soft key to ON

Jog increment:

8 ENT	Enter the jog increment in millimeters (here, 8 mm).
2.5	Enter the jog increment via soft key (preset soft-key values). Not for TNC 426, TNC 430.
X	Press the machine axis direction button as often as desired.

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

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

Entering values
Example: Entering the spindle speed S

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

`Spindle speed S =`

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

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

Proceed in the same way to enter the feed rate F and the miscellaneous functions M.

For the feed rate F (not true for TNC 410):

- If you enter F=0, then the lowest feed rate from MP1020 is effective
- F is not lost during a power interruption

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

> The knob for spindle speed override is effective only on machines with an infinitely variable 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)

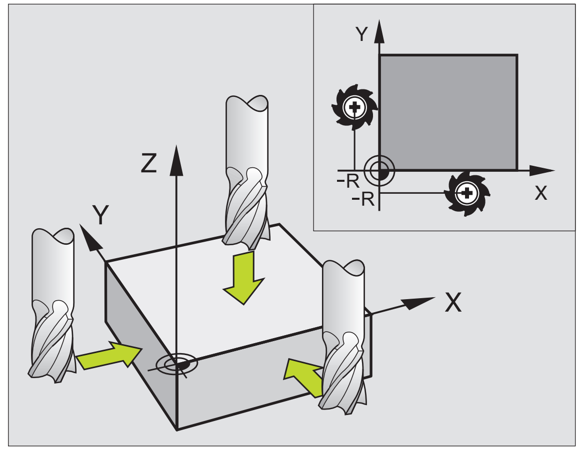
You fix a datum by setting the TNC position display to the coordinates of a known position on the workpiece.

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.

Datum setting

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.

X Y Z

Move the tool slowly until it touches the workpiece surface.

Z

Select an axis (all axes can also be selected via the ASCII keyboard)

```
Datum Set Z=
```

0 ENT

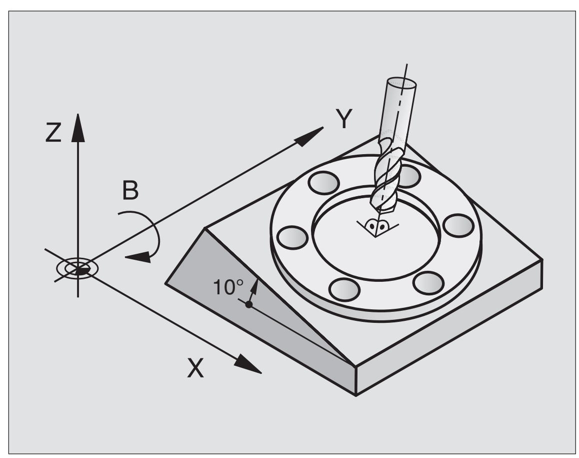
Zero tool in spindle axis: Set the display to a known workpiece position (here, 0) or enter the thickness *d* of the shim. In the tool axis, offset the tool radius.

Repeat the process for the remaining axes.

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

2.5 Tilt the working plane (not TNC 410)

The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With specific swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the tilt axes or as solid angles. Your machine manual provides more detailed information.



The TNC supports the tilting functions on machine tools with swivel heads and/or tilting tables. Typical applications are, for example, oblique holes or contours in an oblique plane. The working plane is always tilted around the active datum. The program is written as usual in a main plane, such as the X/Y plane, but is executed in a plane that is tilted relative to the main plane.

There are two functions available for tilting the working plane

- 3-D ROT soft key in the Manual mode and Electronic Handwheel mode (described below)
- Tilting under program control: Cycle G80 WORKING PLANE in the part program: see section "8.9 Coordinate Transformation Cycles."

The TNC functions for "tilting the working plane" are coordinate transformations in which the working plane is always perpendicular to the direction of the tool axis.

When tilting the working plane, the TNC differentiates between two machine types

Machines with tilting tables:

- You must tilt the **workpiece** into the desired position for machining by positioning the tilting table, for example with an L block.
- The position of the transformed tool axis **does not change** in relation to the machine-based coordinate system. Thus if you rotate the table — and therefore the workpiece — by 90° for example, the coordinate system **does not rotate**. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in Z+ direction.
- In calculating the transformed coordinate system, the TNC considers only the mechanically influenced offsets of the particular tilting table (the so-called "translational" components).

Machines with swivel heads

- You must bring the **tool** into the desired position for machining by positioning the swivel head, for example with an L block.
- The position of the transformed tool axis **changes** in relation to the machine-based coordinate system. Thus if you rotate the swivel head — and therefore the tool — in the B axis by 90° for example, **the coordinate system rotates also**. If you press the Z+ axis direction button in the Manual Operation mode, the tool moves in X+ direction of the machine-based coordinate system.
- In calculating the transformed coordinate system, the TNC considers both the mechanically influenced offsets of the particular swivel head (the so-called "translational" components) **and** offsets caused by tilting of the tool (3-D tool length compensation).

Traversing the reference points in tilted axes
With tilted axes, you use the machine axis direction buttons to cross over the reference points. The TNC interpolates the corresponding axes. Be sure that the function for tilting the working plane is active in the Manual Operation mode and the actual angle of the tilted axis was entered in the menu field.

After you have positioned the rotary axes, set the datum in the same way as for a non-tilted system. The TNC then converts the datum for the tilted coordinate system. If your machine tool features axis control, the angular values for this calculation are taken from the actual position of the rotary axis.

You must not set the datum in the tilted working plane if in machine parameter 7500 bit 3 is set. If you do, the TNC will calculate the wrong offset.

If your machine tool is not equipped with axis control, you must enter the actual position of the rotary axis in the menu for manual tilting: The actual positions of one or several rotary axes must match the entry. Otherwise the TNC will calculate an incorrect datum.

Datum setting on machines with rotary tables

The behavior of the TNC during datum setting depends on the machine.Your machine manual provides more detailed information.

The TNC automatically shifts the datum if you rotate the table and the tilted working plane function is active.

MP 7500, bit 3=0
To calculate the datum, the TNC uses the difference between the REF coordinate during datum setting and the REF coordinate of the tilting axis after tilting. The method of calculation is to be used when you have clamped your workpiece in proper alignment when the rotary table is in the 0° position (REF value).

MP 7500, bit 3=1
If you rotate the table to align a workpiece that has been clamped in an unaligned position, the TNC must no longer calculate the offset of the datum from the difference of the REF coordinates. Instead of the difference from the 0° position, the TNC uses the REF value of the tilting table after tilting. In other words, it assumes that you have properly aligned the workpiece before tilting.

Position display in a tilted system
The positions displayed in the status window (ACTL. and NOML.) are referenced to the tilted coordinate system.

Limitations on working with the tilting function
- The touch probe function Basic Rotation cannot be used.
- PLC positioning (determined by the machine tool builder) is not possible.
- Positioning blocks with M91/M92 are not permitted.

To activate manual tilting:

3D ROT

To select manual tilting, press the 3-D ROT soft key.
You can now select the desired menu option with the arrow keys.

Enter the tilt angle.

To set the desired operating mode in menu option "Tilt working plane" to Active, select the menu option and shift with the ENT key.

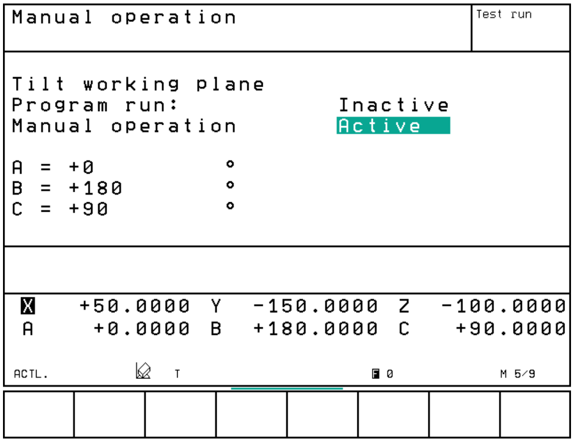
END

To conclude entry, press the END soft key.

To reset the tilting function, set the desired operating modes in menu "Tilt working plane" to Inactive.

If the Working Plane function is active and the TNC moves the machine axes in accordance with the tilted axes, the status display shows the symbol .

If you set the function "Tilt working plane" for the operating mode Program Run to Active, the tilt angle entered in the menu becomes active in the first block of the part program. If you are using Cycle G80 WORKING PLANE in the part program, the angular values defined in the cycle (starting at the cycle definition) are effective. Angular values entered in the menu will be overwritten.



3

Positioning with Manual Data Input (MDI)

3.1 Program and Run Simple Machining Operations

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

Select the operating mode Positioning with MDI.
Program the $MDI file as desired.

I

To start the selected block: Press the machine START button.

Limitations for TNC 410:

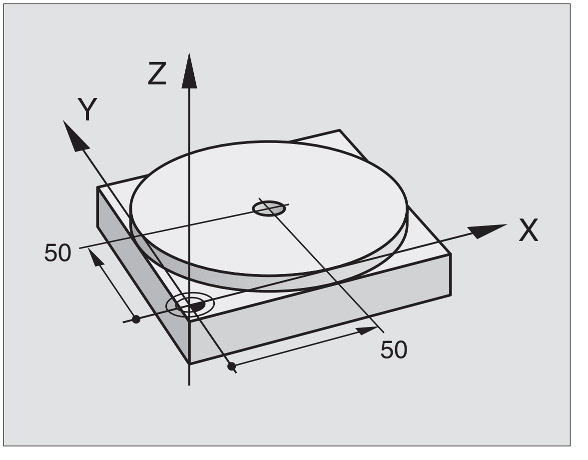
The following functions are not available:

- Tool radius compensation
- Programming and program run graphics
- Programmable probe functions
- Subprograms, program section repeats
- Contouring functions G06, G02 and G03 with R, G24 and G25
- Program call with %

Limitations of the TNC 426, TNC 430:

The following functions are not available:

- Program call with %
- Program run graphics



Example 1
A hole with a depth of 20 mm is to be drilled into a single workpiece. After clamping and aligning the workpiece and setting the datum, you can program and execute the drilling operation in a few lines.

First you pre-position the tool with G00 and G01 blocks (straight-line blocks) to the hole center coordinates at a setup clearance of 5 mm above the workpiece surface. Then drill the hole with Cycle G83 PECKING.

`%$MDI G71 *`	
`N10 G99 T1 L+0 R+5 *`	Define tool: zero tool, radius 5
`N20 T1 G17 S2000 *`	Call tool: spindle axis Z,
	Spindle speed 2000 rpm
`N30 G00 G40 G90 Z+200 *`	Retract tool (rapid traverse)
`N40 X+50 Y+50 M3 *`	Move the tool at rapid traverse to a position above the
	hole. Spindle on.
`N50 G01 Z+2 F2000 *`	Position tool to 5 mm above hole
`N60 G83`	Define Cycle G83 PECKING:
`P01 +2`	Setup clearance of the tool above the hole
`P02 -20`	Total hole depth (Algebraic sign=working direction)
`P03 +10`	Depth of each infeed before retraction
`P04 0.5`	Dwell time in seconds at the hole bottom
`P05 250 *`	Feed rate for pecking
`N70 G79 *`	Call Cycle G83
`N80 G00 G40 Z+200 M2 *`	Retract tool
`N99999 %$MDI G71 *`	End of program

The straight-line function is described in section 6.4 "Path Contours — Cartesian Coordinates," the G83 PECKING cycle in section 8.3 "Drilling Cycles."

Example 2

Correcting workpiece misalignment on machines with rotary tables

Use the 3-D touch probe to rotate the coordinate system. See section "12.1 Touch Probe Cycles in the Manual and Electronic Handwheel Modes," section "Compensating Workpiece Misalignment."

Write down the Rotation Angle and cancel the Basic Rotation.

Select operating mode: Positioning with MDI.

G 0

Select the axis of the rotary table, enter the rotation angle you wrote down previously and set the feed rate.
For example: G00 G40 G90 C+2.561 F50

END

Conclude entry.

I

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

Save or delete programs from %$MDI

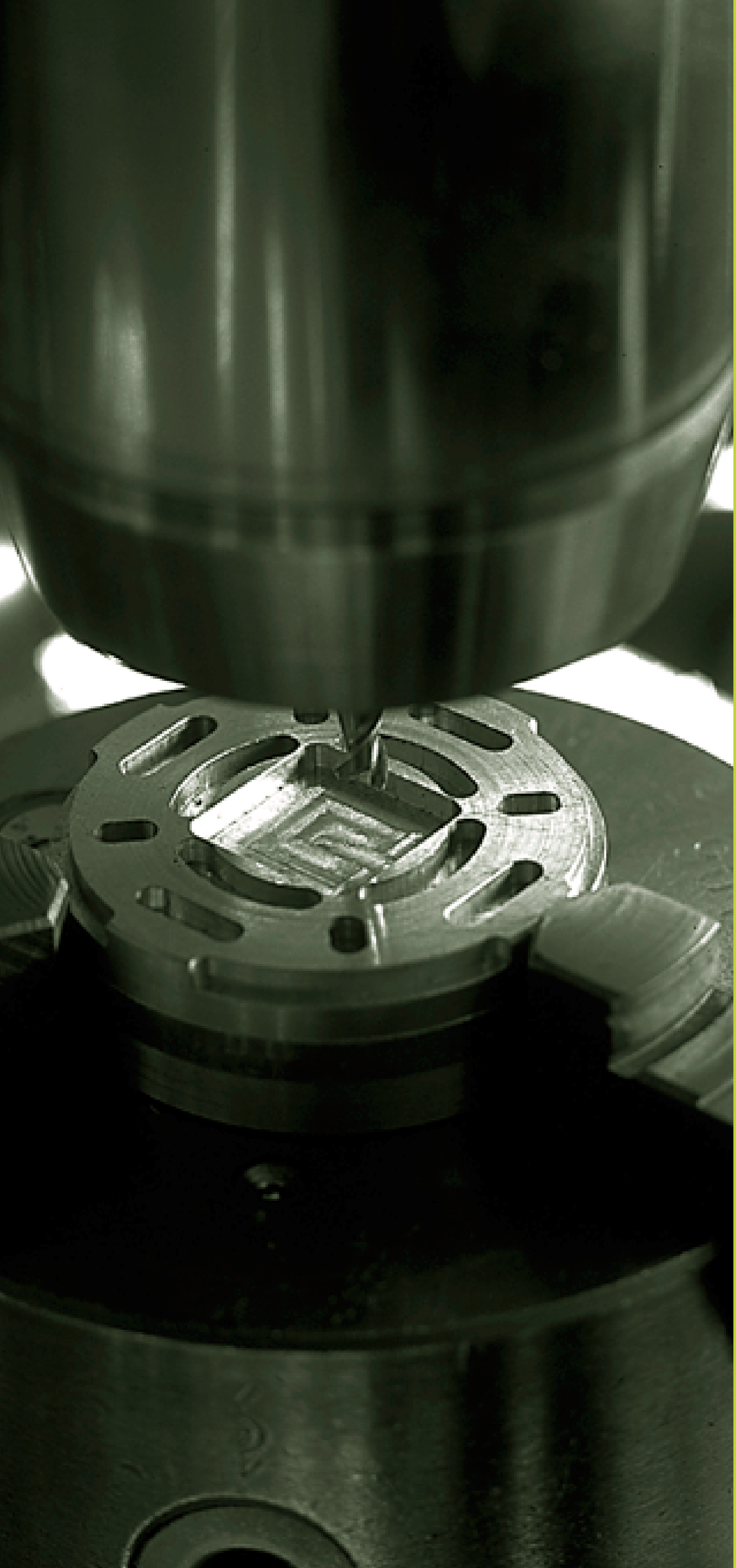
The %$MDI file is generally intended for short programs that are only needed temporarily. Nevertheless, you can store a program, if necessary, by proceeding as described below:

	Select the Programming and Editing mode of operation
PGM MGT	To call the file manager, press the PGM MGT key (program management).
	Tag the %$MDI file
COPY ABC ⇒ XYZ	Select „Copy file“: Press the COPY soft key
Target file =	
Hole	Enter the name under which you want to save the current contents of the $MDI file.
ENT	TNC 410: Start copying by pressing the ENT key
EXECUTE	TNC 426, TNC 430: Press the EXECUTE soft key to start copying
END	To close the file manager, press the END soft key.

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

TNC 426, TNC 430:

The %$MDI file may not be selected in the Programming and Editing mode during the erasure procedure.



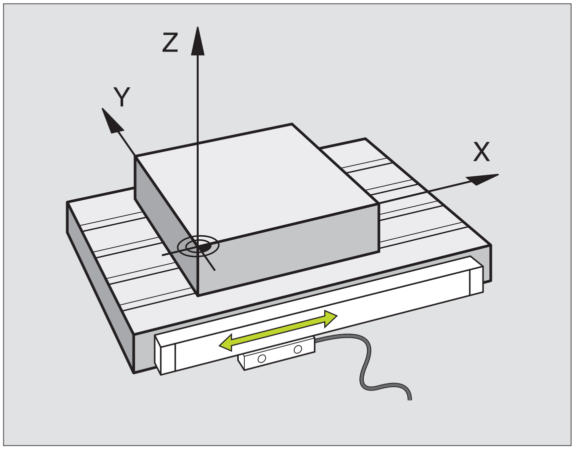
4

Programming:

Fundamentals of NC, File Management, Programming Aids, Pallet Management

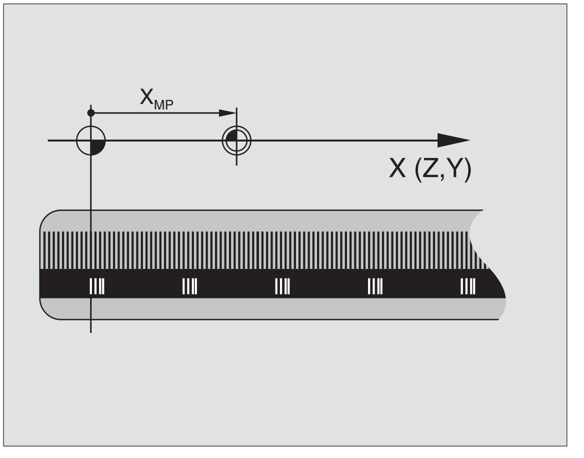
4.1 Fundamentals of NC

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.

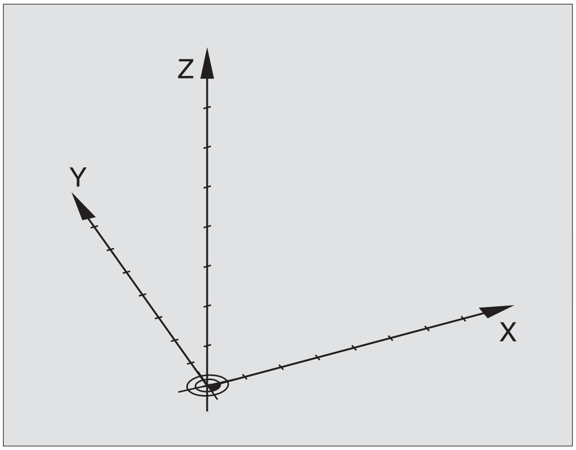
If there is an interruption of power, the calculated position will no longer correspond to the actual position of the machine slide. The CNC can re-establish this relationship with the aid of reference marks when power is returned. The scales of the position encoders contain one or more reference marks 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.

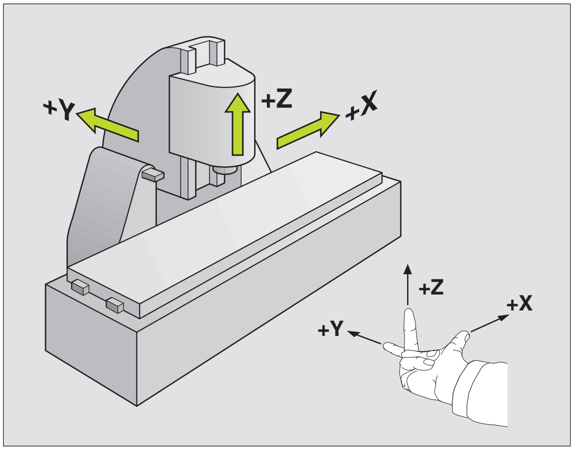


The Cartesian coordinate system (a rectangular coordinate system) is based on three coordinate axes X, Y and Z. The axes are mutually perpendicular and intersect at one point called the datum. A coordinate identifies the distance from the datum in one of these directions. A position in a plane is thus described through two coordinates, and a position in space through three coordinates.

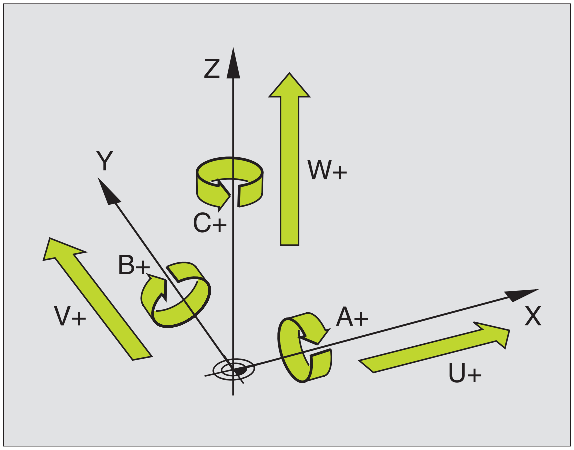
Coordinates that are referenced to the datum are referred to as absolute coordinates. Relative coordinates are referenced to any other known position (datum) you define within the coordinate system. Relative coordinate values are also referred to as incremental coordinate values.

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 410 can control a maximum of 4 axes, the TNC 426 a maximum of 5 axes and the TNC 430 a maximum of 9 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 at lower right shows the assignment of secondary axes and rotary axes to the main axes.



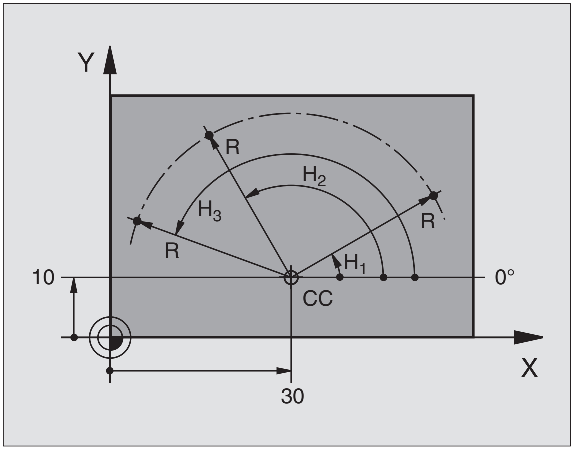
Polar coordinates

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

While the Cartesian coordinates X, Y and Z are three-dimensional and can describe points in space, polar coordinates are two-dimensional and describe points in a plane. Polar coordinates have their datum at the so-called pole. A position in a plane can be clearly defined by the

- Polar radius R: the distance from the pole to the position, and the
- Polar angle H, the size of the angle between the reference axis and the line that connects the pole with the position.

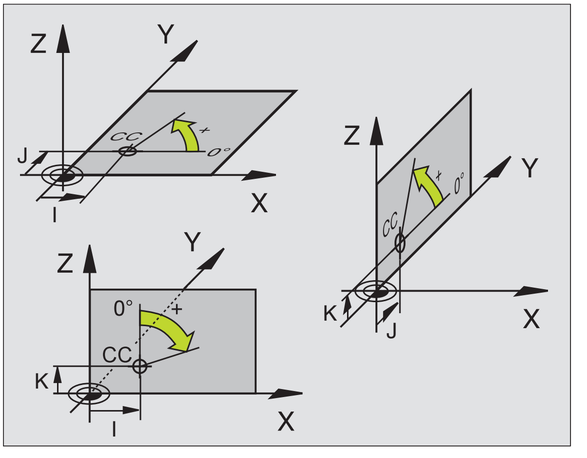
See figure at lower right.



Definition of pole and angle reference axis

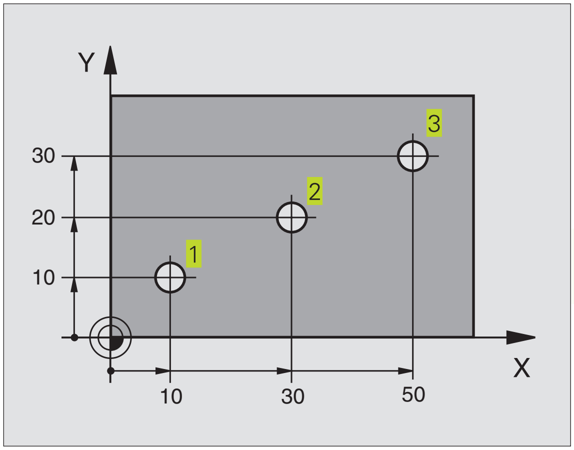
The pole is set by entering two Cartesian coordinates in one of the three planes. These coordinates also set the reference axis for the polar angle H.

Coordinates of the pole (plane)	Reference axis of the angle
I and J	+X
J and K	+Y
K and I	+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.

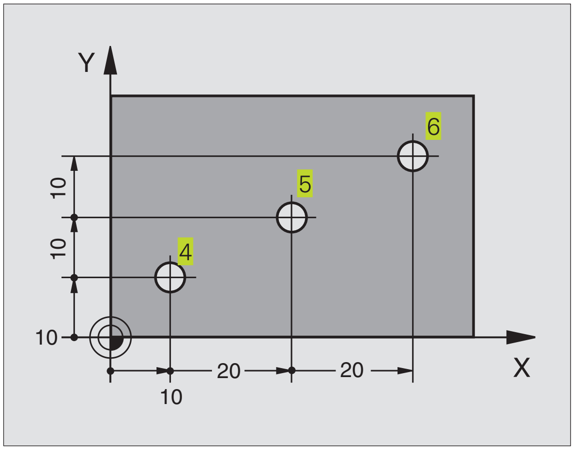


Example 1: Holes dimensioned in absolute coordinates

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 function G91 before the axis.



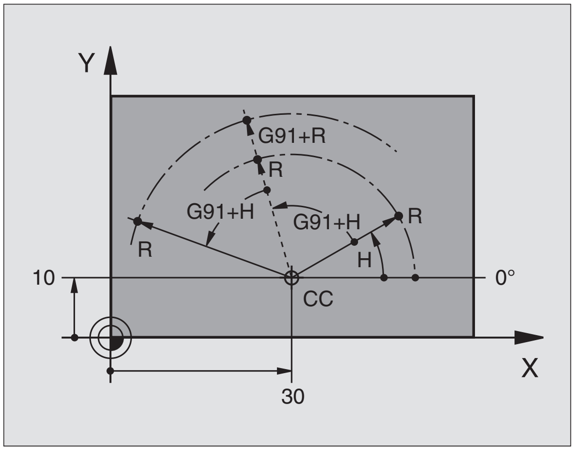
Example 2: Holes dimensioned with relative coordinates
Absolute coordinates of hole 4:

X= 10 mm
Y= 10 mm

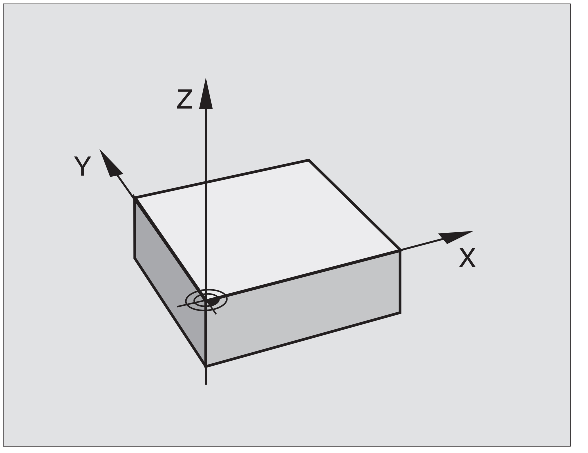
Hole 5 referenced to hole 4	Hole 6 referenced to hole 5
G91 X= 20 mm	G91 X= 20 mm
G91 Y= 10 mm	G91 Y= 10 mm

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

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



Selecting the datum



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

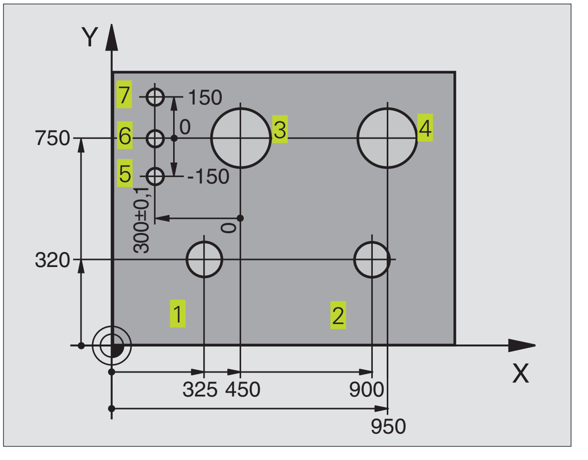
If the production drawing is dimensioned in relative coordinates, simply use the coordinate transformation cycles. For further information, refer to section 8.9 "Coordinate Transformation Cycles."

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 12.2 "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: Fundamentals

Files

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 text and tables as files.

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

In the TNC 410 you can manage a max. 64 files with a total of up to 128 KB.

The TNC 426, TNC 430 can manage any number of files. However, their total size must not exceed **1.5 gigabytes**.

Files in theTNC	Type
Programs	
in HEIDENHAIN conversational format	.H
in ISO format	.I
Tables for	
Tools	.T
Tool changer (TNC 410: 1 table)	.TCH
Datums	.D
Points	.PNT
Pallets (not TNC 410)	.P
Text as	
ASCII files (not TNC 410)	.A

File names

The name of a file can have up to 16 characters (TNC 410: 8 characters). When you store programs, tables and texts 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).

PROG20	.H
File name	File type

Data backup TNC 426, TNC 430

We recommend saving newly written programs and files on a PC at regular intervals. You can do this with the cost-free backup program TNCBACK.EXE from HEIDENHAIN. Your machine tool builder can provide you with a copy of TNCBACK.EXE.

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

> Saving the contents of the entire hard disk (up to 1.5 GB) can take up to several hours. In this case, it is a good idea to save the data outside of working hours, (e.g. overnight), or to use the PARALLEL EXECUTE function to copy in the background while you work.

4.3 Standard file management TNC 426, TNC 430

Use the standard file manager if you want to store all of the files in one directory, or if you are used to working with the file manager on old TNC controls.

Set the MOD function PGM MGT to Standard (see section 13.9) .

Calling the file manager

PGM MGT	Press the PGM MGT: The TNC displays the file management window (see Fig. at top right)

The window shows you all of the files that are stored in the TNC. Each file is shown with additional information, see table at center right.

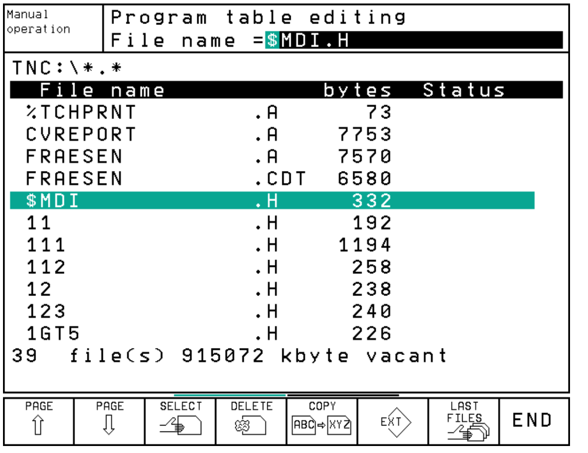
Selecting a file

PGM MGT	Calling the file manager

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

↑ ↓	Move the highlight up or down.

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



Display	Meaning
FILE NAME	Name with max. 16 characters and file type
BYTE	File size in bytes
STATUS	Property of the file:
E	Program is in the Programming and Editing mode of operation
S	Program is in the Program is selected in the Test RUN mode of operation
M	Program is in the Program Run mode of operation.
P	File is protected against editing and erasure (Protected)

Display of long file directories	Soft key
Move pagewise up through the file directory.	PAGE ⇧
Move pagewise down through the file directory	PAGE ⇩

Deleting a file

PGM MGT	Calling the file manager

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

↑ ↓	Move the highlight up or down.
DELETE	Delete a file: Press the DELETE soft key

Delete .... file ?

YES	Press the YES soft key to confirm, or
NO	the NO soft key to abort.

Copying a file

PGM MGT	Calling the file manager

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

↑ ↓	Move the highlight up or down.
COPY ABC⇨XYZ	Copy a file: Press the COPY soft key

Target file =

Enter the name of the new file and confirm your entry with the ENT key or EXECUTE soft key. A status window appears on the TNC, informing about the copying progress. As long as the TNC is copying, you can no longer work, or

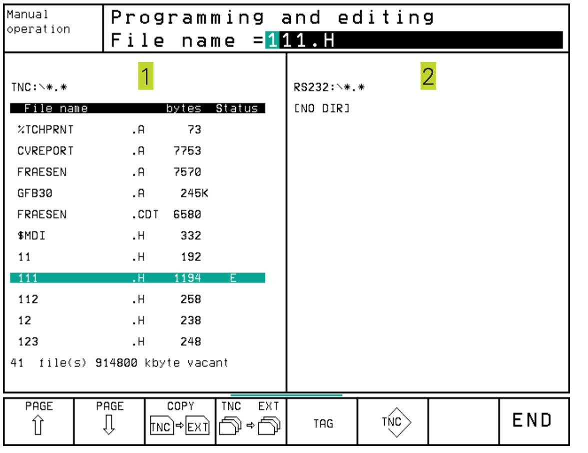
If you wish to copy very long programs, enter the new file name and confirm with the PARALLEL EXECUTE soft key. The file will now be copied in the background, so you can continue to work while the TNC is copying.

Data transfer to or from an external data medium

Before you can transfer data to an external data medium, you must set the interface (see section 13.6 "Setting the Data Interfaces for TNC 426, TNC 430").

PGM MGT — Calling the file manager

EXT — Activate data transfer: press the EXT soft key. In the left half of the screen, the TNC shows all of the 1 files that are stored on the TNC, and in the right half of the screen, 2 all of the files that are stored on the external data medium.



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

↑ ↓ — Move the highlight up and down within a window

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

If you are transferring from the TNC to the external medium, move the highlight in the left window onto the file that is to be transferred.

If you are transferring from the external medium to the TNC, move the highlight in the right window onto the file that is to be transferred.

COPY ABC ⇨ XYZ — Transfer a single file: Press the COPY soft key, or

TAG — Transfer several files: Press TAG (marking functions, see table on right), or

TNC ⇨ EXT — transfer all files by pressing the TNC EXT soft key

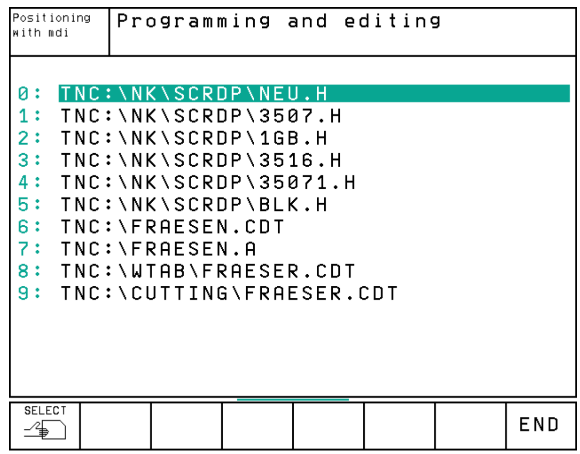
Tagging functions	Soft key
Tag a single file	TAG FILE
Tag all files	TAG ALL FILES
Untag a single file	UNTAG FILE
Untag all files	UNTAG ALL FILES
Copy all tagged files	COPY TAG

Confirm with the EXECUTE or with the ENT key. A status window appears on the TNC, informing about the copying progress, or

If you wish to transfer more than one file or longer files, press the PARALLEL EXECUTE soft key. The TNC then copies the file in the background.

TNC — To stop transfer, press the TNC soft key. The standard file manager window is displayed again.

Selecting one of the last 10 files selected



PGM MGT — Calling the file manager

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

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

↑ ↓ — Move the highlight up or down.

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

Renaming a file

PGM MGT — Calling the file manager

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

↑ ↓ — Move the highlight up or down.

RENAME ABC = XYZ — To rename the file, press the RENAME key.

Target file =

Enter the name of the new file and confirm your entry with the ENT key or EXECUTE soft key.

Protect file / Cancel file protection

PGM MGT — Calling the file manager

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

↑ ↓ — Move the highlight up or down.

PROTECT — Press the PROTECT soft key to enable file protection The file now has status P, or

UNPROTECT — To cancel file protection, press the UNPROTECT soft key. The P status is canceled.

4.4 Expanded File Management TNC 426, TNC 430

Select the file manager with additional functions if you wish to store files in various different directories.

Set the MOD function PGM MGT (see section 13.9) to **Enhanced**!

See also section 4.2 "File Management: Fundamentals"!

Directories

To ensure that you can easily find your files, we recommend that you organize your hard disk into directories. You can divide a directory up into further directories, which are called subdirectories.

The TNC can manage up to 6 directory levels!

If you save more than 512 files in one directory, the TNC no longer sorts them alphabetically!

Directory names
The name of a directory can contain up to 8 characters and does not have an extension. If you enter more than 8 characters for the directory name, the TNC will shorten the name to 8 characters.

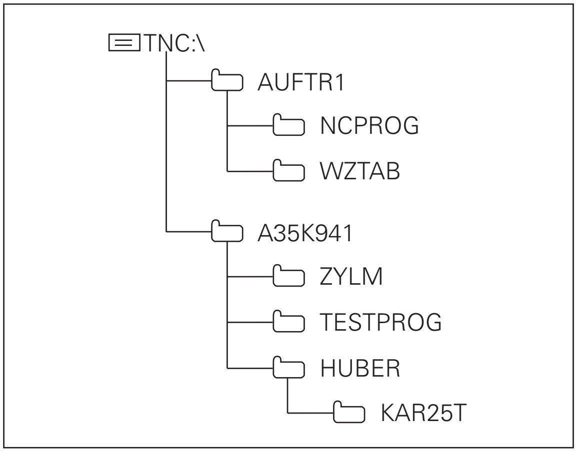
Paths

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

Example: On drive TNC:\, the directory AUFTR1 was created. Under this directory, the subdirectory NCPROG was created, and the part program PROG1.I copied into this subdirectory. The part program now has the following path:

TNC:\AUFTR1\NCPROG\PROG1.I

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



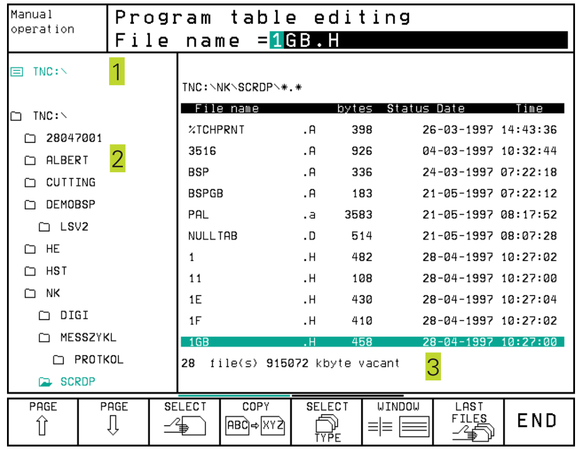
Overview: Functions of the expanded file manager

Function	Soft key
Copy (and convert) individual files	COPY ABC ⇨ XYZ
Display a specific file type	SELECT TYPE
Display the last 10 files that were selected	LAST FILES
Erase a file or directory	DELETE
Tag a file	TAG
Renaming a file	RENAME ABC = XYZ
Protect a file against editing and erasure	PROTECT
Cancel file protection	UNPROTECT
Network drive management (Ethernet option only)	NET
Copying a directory	COPY DIR
Display all the directories of a particular drive	SHOW TREE
Delete directory with all its subdirectories	DELETE ALL

Calling the file manager

PGM MGT

Press the PGM MGT:
The TNC displays the file management window (see Fig. at top right for default setting. If the TNC displays a different screen layout, press the WINDOW soft key)



The narrow window at left shows three drives 1 . If the TNC is connected to a network, it also displayed the connected network drives. Drives designate devices with which data are stored or transferred. One drive is the hard disk of the TNC. Other drives are the interfaces (RS232, RS422, Ethernet), which can be used, for example, to connect a personal computer. The selected (active) drive is shown in a different color.

In the lower part of the narrow window the TNC shows all directories 2 of the selected drive. A drive is always identified by a file symbol to the left and the directory name to the right. The TNC displays a subdirectory to the right of and below its parent directory. The selected (active) directory is depicted in a different color.

The wide window at on the right side shows all the files 3 that are stored in the selected directory. Each file is shown with additional information that is illustrated in the table on the next page.

Display	Meaning
FILE NAME	Name with max. 16 characters and file type
BYTE	File size in bytes
STATUS E	Property of the file: Program is in the Programming and Editing mode of operation
S	Program is in the Program is selected in the Test RUN mode of operation
M	Program is in the Program Run mode of operation.
P	File is protected against editing and erasure (Protected)
DATE	Date the file was last changed
TIME	Time the file was last changed

To select drives, directories and files:

PGM MGT	Calling the file manager

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

← →	Move the highlight from the left to the right window, and vice versa.
↑ ↓	Move the highlight up and down within a window
PAGE ⇧ PAGE ⇩	Move the highlight one page up or down within a window

1st step: select drive:

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

SELECT or ENT	Select drive: Press the SELECT soft key or ENT

2nd step: select directory:

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

3rd step: select a file:

SELECT TYPE	Press the SELECT TYPE soft key
SHOW .A	Press the soft key for the desired file type, or
SHOW ALL	Press the SHOW ALL soft key to display all files

Move the highlight to the desired file in the right window

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

To create a new directory (only possible on the TNC's hard disk drive):

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

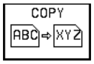
NEW ENT	Enter the new file name, and confirm with ENT.

Create \NEW directory ?

YES	Press the YES soft key to confirm, or
NO	the NO soft key to abort.

Copying a file

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



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

▶ Enter the name of the destination file and confirm your entry with the ENT key or EXECUTE soft key: The TNC copies the file into the active directory. The original file is retained.
Press the PARALLEL EXECUTE soft key to copy the file in the background. Copying in the background permits you to continue working while the TNC is copying. This can be useful if you are copying very large files that take a long time. While the TNC is copying in the background you can press the INFO PARALLEL EXECUTE soft key (under MORE FUNCTIONS, second soft-key row) to check the progress of copying.

Copying a table
If you are copying tables, you can overwrite individual lines or columns in the target table with the REPLACE FIELDS soft key. Prerequisites:

- The target table must exist.
- The file to be copied must only contain the columns or lines you want to replace.

Example:
With a tool presetter you have measured the length and radius of 10 new tools. The tool presetter then generates the tool table TOOL.T with 10 lines (for the 10 tools) and the columns

- Tool number
- Tool length
- Tool radius

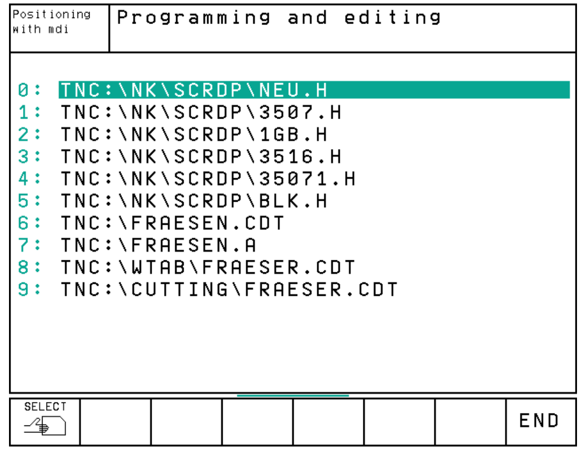
If you wish to copy this file to the TNC, the TNC asks if you wish to overwrite the existing TOOL.T tool table:

- If you press the YES soft key, the TNC will completely overwrite the current TOOL.T tool table. After this copying process the new TOOL.T table consists of 10 lines. The only remaining columns in the table are tool number, tool length and tool radius.
- If you press the REPLACE FIELDS soft key, the TNC merely overwrites the first 10 lines of the columns number, length and radius in the TOOL.T file. The data of the other lines and columns is not changed.

Copying a directory

Move the highlight in the left window onto the directory you want to copy. Press the COPY DIR soft key instead of the COPY soft key. Subdirectories are also copied at the same time.

Selecting one of the last 10 files selected



PGM MGT — Calling the file manager

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

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

↑ ↓ — Move the highlight up or down.

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

Deleting a file

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

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

▶ To confirm, press the YES soft key;
To abort erasure, press the NO soft key.

Erase a directory

▶ Erase all files and subdirectories stored in the directory that you wish to erase.

▶ Move the highlight to the directory 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 directory.

▶ To confirm, press the YES soft key;
To abort erasure, press the NO soft key.

Tagging files

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

Move the highlight to the first file.

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

TAG FILE — Tag a file by pressing the TAG FILE soft key.

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

TAG FILE — You can tag several files in this way, as desired.

COPY TAG — To copy the tagged files, press the COPY TAG soft key, or

END, DELETE — Delete the tagged files by pressing END to end the marking function, and then DELETE to delete the tagged files.

Tagging functions	Soft key
Tagging single files	TAG FILE
Tag all files in the directory	TAG ALL FILES
Untag a single file	UNTAG FILE
Untag all files	UNTAG ALL FILES
Copy all tagged files	COPY TAG

Renaming 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.

Additional Functions

Protecting a file/Canceling file protection

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

MORE FUNCTIONS

▶ To select the additional functions, press the MORE FUNCTIONS key.

PROTECT

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

To cancel file protection, proceed in the same way using the UNPROTECT soft key.

Erase a directory together with all its subdirectories and files.

▶ Move the highlight in the left window onto the directory you want to erase.

MORE FUNCTIONS

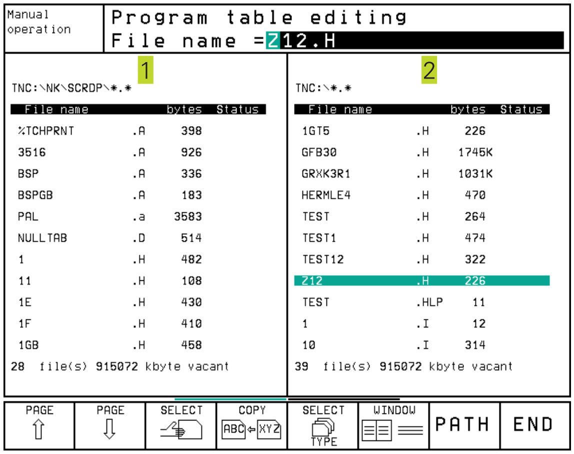
▶ To select the additional functions, press the MORE FUNCTIONS key.

DELETE ALL

▶ Press DELETE ALL to erase the directory together with its subdirectories.

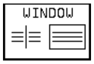
▶ To confirm, press the YES soft key;
To abort erasure, press the NO soft key.

Data transfer to or from an external data medium



Before you can transfer data to an external data medium, you must set the interface (see section 13.6 "Setting the Data Interfaces for TNC 426, TNC 430").

Calling the file manager



Select the screen layout for data transfer: press the WINDOW soft key. In the left half of the screen, the TNC shows all of the 1 files that are stored on the TNC, and in the right half of the screen, 2 all of the files that are stored on the external data medium.

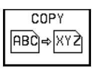
Use the arrow keys to highlight the file(s) that you want to transfer:

Move the highlight up and down within a window

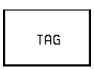
Move the highlight from the left to the right window, and vice versa.

If you are transferring from the TNC to the external medium, move the highlight in the left window onto the file that is to be transferred.

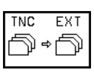
If you are transferring from the external medium to the TNC, move the highlight in the right window onto the file that is to be transferred.



Transfer a single file: Press the COPY soft key, or



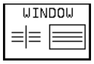
To transfer several files, use the TAG soft key (in the second soft-key row, see also Tagging functions earlier on in this chapter), or



transfer all files by pressing the TNC EXT soft key

Confirm with the EXECUTE or with the ENT key. A status window appears on the TNC, informing about the copying progress, or

If you wish to transfer more than one file or longer files, press the PARALLEL EXECUTE soft key. The TNC then copies the file in the background.



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

To select another directory, press the PATH soft key and then select the desired directory using the arrow keys and the ENT key!

Copying files into another directory

▶ Select the screen layout with the two equally sized windows.

▶ To display directories in both windows, press the PATH soft key.

In the right window:

▶ Move the highlight to the directory into which you wish to copy the files, and display the files in this directory with the ENT key

In the left window:

▶ Select the directory with the files that you wish to copy and press ENT to display them.

TAG ▶ Display the file tagging functions.

TAG FILE ▶ Move the highlight to the file you want to copy and tag it. You can tag several files in this way, as desired.

COPY TAG ▶ Copy the tagged files into the target directory.

For additional tagging functions see „Tagging files".

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

Overwriting files
If you copy files into a directory in which other files are stored under the same name, the TNC will ask whether the files in the target directory should be overwritten:

▶ Press the YES soft key to overwrite all files, or

▶ Press the NO soft key if no file is to be overwritten

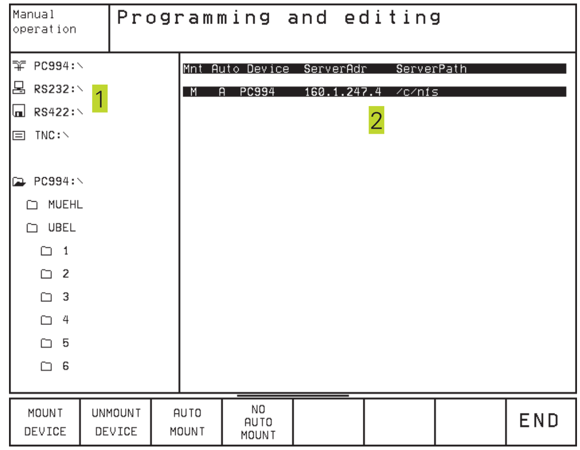
▶ To confirm each file separately before overwriting it, press the CONFIRM key.

If you wish to overwrite a protected file, this must also be confirmed or aborted separately.

The TNC in a network (applies only for Ethernet interface option)

> To connect the Ethernet card to your network, refer to Chapter "13.8 Ethernet Interface"!
>
> The TNC logs error messages during network operation (see section "13.8 Ethernet Interface").

If the TNC is connected to a network, the directory window displays up to 7 drives 1 (see screen at upper right). All the functions described above (selecting a drive, copying files, etc.) also apply to network drives, provided that you have been given the corresponding rights.



Connecting and disconnecting network drives

PGM MGT

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

NET

- To manage the network drives: Press the "Network" soft key. In the right-hand window 2 the TNC shows the network drives available for access. With the following soft keys you can define the connection for each drive.

Function	Soft key
Establish network connection. If the connection is active, the TNC shows an M in the Mnt column. You can connect up to 7 additional drives with the TNC.	MOUNT DEVICE
Delete network connection	UNMOUNT DEVICE
Automatically establish connection whenever the TNC is switched on. The TNC show in the Auto column an A if the connection is established automatically.	AUTO MOUNT
Do not network connection automatically when the TNC is switched on	NO AUTO MOUNT

It may take some time to mount a network device. At the upper right of the screen the TNC displays [READ DIR] to indicate that a connection is being established. The maximum data transmission rate lies between 200 and 1000 kilobaud, depending on the file type being transmitted.

Printing the file with a network printer

If you have defined a network printer (see section "13.8 Ethernet Interface"), you can print the files directly:

- To call the file manager, press the PGM MGT key.
- Move the highlight to the file you wish to print.
- Press the COPY soft key.
- Press the PRINT soft key: If you have define only one printer, the TNC will print the file immediately.

  If you have defined more than one printer, the TNC opens a window listing all defined printers. Use the arrow keys to select the desired printer, then press ENT.

4.5 File Management for the TNC 410

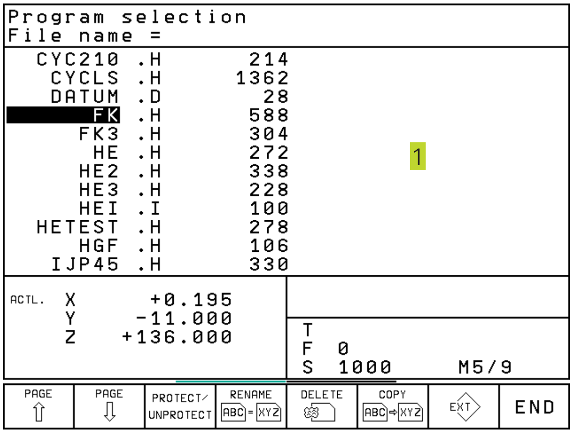
Files in the TNC 410	Type
Programs	
in HEIDENHAIN conversational format	.H
in ISO format	.I
Tables for	
Tools	.T
Tool pockets	.TCH
Datums	.D
Points	.PNT

This section informs you about the meaning of the individual screen information, and describes how to select files and directories. 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.

Calling the file manager

PGM MGT	Press the PGM MGT key: the TNC displays the file management window

The window 1 shows you all of the files 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	Name with up to 8 characters and file type
	Properties of the file:
M	Program is in the Program Run mode of operation.
P	File is protected against editing and erasure (Protected)

Display of long file directories	Soft key
Move pagewise up through the file directory.	PAGE ⇧
Move pagewise down through the file directory	PAGE ⇩

Selecting a file

PGM MGT	Calling 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 letters of the file you wish to select and then press the GOTO key: The highlight moves to the first file that matches these letters.

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

Copying 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.

Renaming 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.

Deleting 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 file.

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

- To read in or read out files: Press the ENT soft key. The TNC provides the functions described below.

> If a file to be read in already exists in the memory of the TNC, the TNC displays the message "File xxx already exists. Read in file? In this case, answer the dialog question with YES (file is the read in) or NO (file is not read in).
>
> Likewise, if a file to be read out already exists on the external device, the TNC asks whether you wish to overwrite the external file.

Read in all files (file types: .H, .I, .T, .TCH, .D, .PNT)

- Read in all of the files that are stored on the external data medium.

Read in offered file

- List all files of a certain file type.
- For example: list all HEIDENHAIN conversational programs. To read-in the listed program, press the YES soft key. If you do not wish the read-in the program, press NO.

Read in a specific file

- Enter the file name. Confirm with the ENT key.
- Select the file type, e.g. HEIDENHAIN dialog program.

If you with to read-in the tool table TOOL.T, press the TOOL TABLE soft key. If you with to read-in the tool-pocket table TOOLP.TCH, press the POCKET TABLE soft key.

Read out a specific file

- Select the function for reading out a single file.
- Move the highlight to the file that you wish to read out. Press ENT or TRANSFER soft key to start the transfer.
- To terminate the function for reading out specific files: press the END key.

Read out all files (file types: .H, .I, .T, .TCH, .D, .PNT)

- Output all files stored in the TNC to an external device.

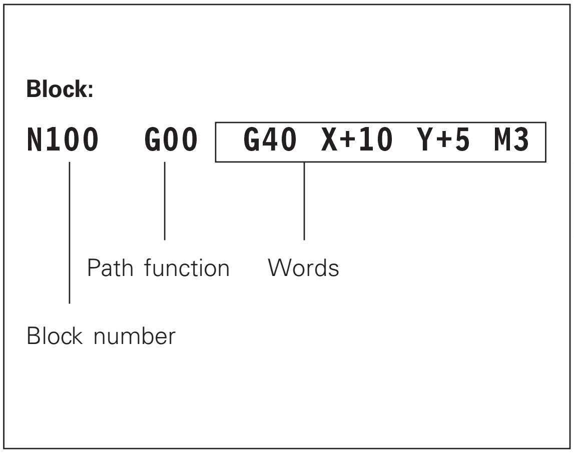
Display a file directory of the external device (File types: .H, .I, .T, .TCH, .D, .PNT)

- Display a list of files stored in the external device. The files are displayed pagewise. To show the next page: press the YES soft key. To return to the main menu: press the NO soft key.

4.6 Creating and Writing Programs

Organization of an NC program in ISO format.

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



The TNC automatically numbers the program blocks in ascending sequence, if you have set a block number increment in MP7220 (see „14.1 General User Parameters")

The first block of a program is identified by "%" the program name and the active unit of measure G70/G71.

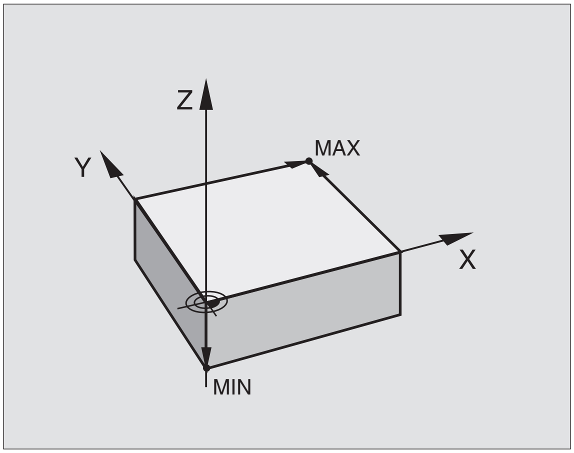
The subsequent blocks contain information on:

- 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 begins with N999 999 and is identified with "%", the program name and the active unit of measure.

Define blank form: G30/G31

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 may be max. 100 000 mm long (TNC 410: 30 000 mm) and lie parallel to the axes X, Y and Z. The ratio of the side lengths must be less than 200:1. The blank form is defined by two of its corner points:



- MIN point G30: the smallest X, Y and Z coordinates of the blank form, entered as absolute values.
- MAX point G31: the largest X, Y and Z coordinates of the blank form, entered as absolute or incremental values.

> You only need to define the blank form if you wish to run a graphic test for the program!

Opening a new part program TNC 426, TNC 430

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

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

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

`File name = OLD.I`

NEW ENT

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

MM

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

Open new program for the TNC 410

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

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

`File name =`

NEW ENT

Entering new program names

.I

Select the file type, e.g. ISO program: Press the .I soft key.

MM INCH

If necessary, switch to inches as unit of measure: Press the MM/INCH soft key.

ENT

Confirm your entry with the ENT key.

Define the workpiece blank

G 30	Define MIN point
G 17	Define spindle axis (here Z)
X 0 Y 0 Z -40	Enter in sequence the X, Y and Z coordinates of the MIN point.
END	To terminate the block, press the END key.
G 31	Define MAX point
G 90	Define absolute/incremental input
X 100 Y 100 Z 0	Enter in sequence the X, Y and Z coordinates of the MAX point.
END	To terminate the block, press the END key.

The program blocks window shows the following BLK FORM definition

`%NEW G71 *`	Program begin, name, unit of measure
`N10 G30 G17 X+0 Y+0 Z-40 *`	Tool axis, MIN point coordinates
`N20 G31 G90 X+100 Y+100 Z+0 *`	MAX point coordinates
`N999999 %NEW G71 *`	Program end, name, unit of measure

The TNC automatically generates the first and last blocks of the program.

Program tool movements

To program a block, select an ISO function key on the alphabetic keyboard. With the TNC 410 you can also use the gray path function keys to get the corresponding G code.

Example of a positioning block

G 1	Start block
G 40	Enter „No radius compensation“
X 10	Enter the target coordinate for the X axis.
Y 5	Enter the target coordinate for the Y axis.
F 100	Enter a feed rate of 100 mm/min for this path contour.
M 3 END	Enter the miscellaneous function M3 “spindle ON”; pressing the END key will terminate the block.

The program blocks window will display the following line:

```
N30 G01 G40 X+10 Y+5 F100 M3 *
```

Editing a program with TNC 426, TNC 430

While you are creating or editing a part program, you can select any desired line in the program or individual words in a block with the arrow keys or the soft keys (see table at right).

Inserting blocks at any desired location

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

Editing and inserting words

▶ Select a word in a block and overwrite it with the new one. The plain-language dialog is available while the word is highlighted.

▶ To accept the change, press the END key.

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

Looking for the same words in different blocks

→	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.

Selecting blocks or words	Soft keys/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	NO ENT
Delete the selected block	DEL
Erase cycles and program sections: First select the last block of the cycle or program section to be erased, then erase with the DEL key.	DEL

Marking, copying, deleting and inserting program sections
The TNC provides certain functions (listed in table at right) for copying program sections within an NC program or into another NC program.

To copy a program section, proceed as follows:

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

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

- ▶ To insert the block: Press the INSERT BLOCK soft key

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

Regenerating the block number increment
If you have deleted, moved or added program sections, you can have the TNC renumber the blocks through the ORDER N function.

- ▶ To regenerate the block numbering: Press the ORDER N soft key. The TNC displays the conversational prompt "Block nr. increment =."
- ▶ Enter the desired block number increment. The value defined in MP7220 is overwritten.
- ▶ To number the blocks: Press the ENT key.
- ▶ To cancel the change: Press the END key or the END soft key.

Editing a program with the TNC 410

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). While you are entering a new block the TNC identifies the block with a * as long as the block has not been saved.

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 save your changes, press the END key.

▶ To reject the change, press the DEL key.

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

Looking for the same words in different blocks

→	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.

Finding any text

▶ To select the search function, press the FIND soft key. The TNC displays the dialog prompt FIND TEXT:

▶ Enter the text that you wish to find.

▶ To find the text, press the EXECUTE soft key.

Inserting the previously edited (deleted) block at any location

▶ Select the block after which you want to insert the block you have just edited (deleted) and press the INSERT NC BLOCK soft key.

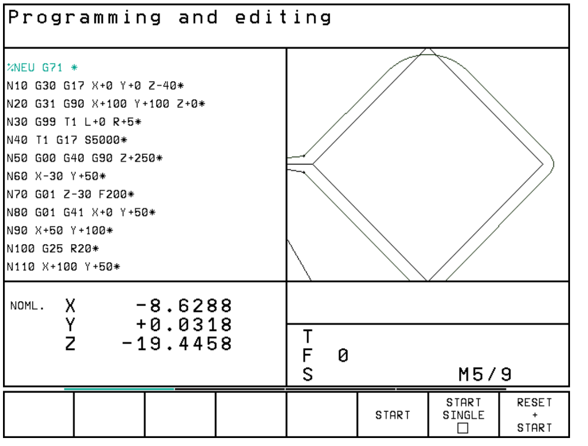
Block display

If a block is so long that the TNC cannot display it in one line (for example in a fixed cycle), this will be indicated with ”>>” at the right edge of the screen.

Function	Soft keys/keys
Go to the previous page	PAGE ⇧
Go to the next page	PAGE ⇩
Go to beginning of program	BEGIN ⇧
Go to end of program	END ⇩
Move from one block to the next	↓ ↑
Select individual words in a block	← →
Search for a sequence of characters	FIND

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	NO ENT
In a block: Restore previously saved version	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

4.7 Programming Graphics (not TNC 426, TNC 430)



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 DRAW OFF / ON

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

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

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

Generating a graphic for an existing program

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

RESET + START

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

Additional functions are listed in the table at right.

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

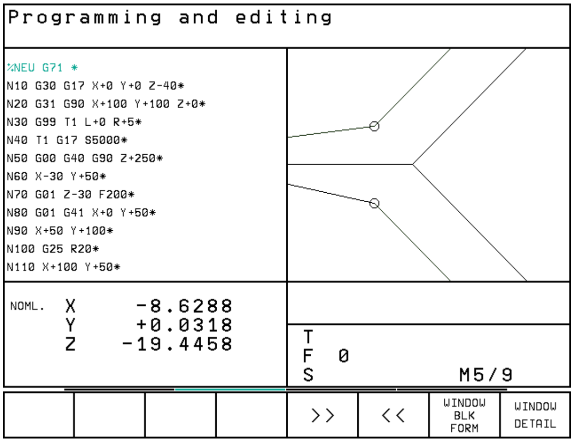
Function	Soft key
Reduce the frame overlay — press and hold the soft key to reduce the detail	<<
Enlarge the frame overlay — press and hold the soft key to magnify the detail	>>

WINDOW DETAIL

- With the WINDOW DETAIL soft key, Confirm the selected area.

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

Functions	Soft key
Generate interactive graphic 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



4.8 Adding Comments

You can add comments to any desired block in the part program to explain program steps or make general notes. There are three possibilities to add comments:

1. Adding comments during program input (not TNC 410)

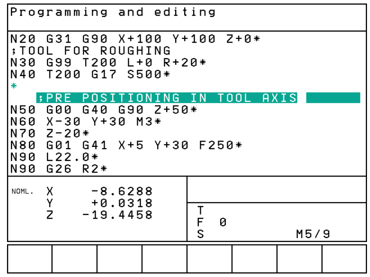
- Enter the data for a program block, then press the semicolon key ";" on the alphabetic keyboard — the TNC displays the dialog prompt COMMENT ?
- Enter your comment and conclude the block by pressing the END key.

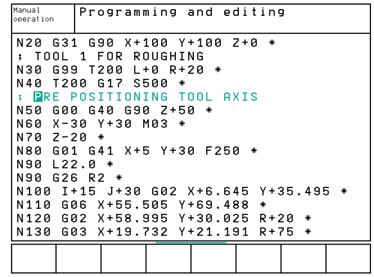
2. Adding comments after program input (not TNC 410)

- Select the block to which a comment is to be added.
- Move to the required block using the arrow keys, then press the semicolon key on the alphabetic keyboard _ the TNC displays the dialog prompt COMMENT ?
- Enter your comment and conclude the block by pressing the END key.

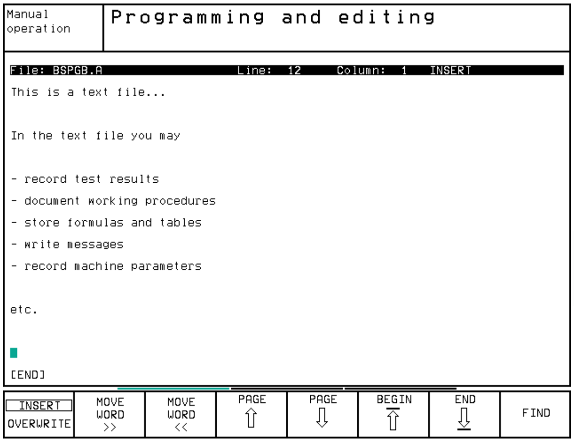
3. To enter a comment in a separate block:

- Select the block after which the comment is to be inserted.
- Initiate the programming dialog with the semicolon key ";" on the alphabetic keyboard.
- Enter your comment and conclude the block by pressing the END key.





4.9 Creating Text Files (not TNC 410)



You can use the TNC's text editor to write and edit texts. Typical applications:

- Recording test results
- Documenting working procedures
- Creating formularies

Text files are type .A files (ASCII files). If you want to edit other types of files, you must first convert them into type .A files.

Opening and exiting text files

- Select the Programming and Editing mode of operation.
- To call the file manager, press the PGM MGT key.
- To display type .A files, press the SELECT TYPE and then the SHOW .A soft keys.
- Select a file and open it with the SELECT soft key or ENT key, **or** create a new file by entering the new file name and confirming your entry with the ENT key.

To leave the text editor, call the file manager and select a file of a different file type, for example a part program.

Editing texts

The first line of the text editor is an information headline which displays the file name, and the location and writing mode of the cursor:

File:	Name of the text file
Line:	Line in which the cursor is presently located
Column:	Column in which the cursor is presently located
Insert:	Insert new text, pushing the existing text to the right
Overwrite:	Write over the existing text, erasing it where it is replaced with the new text.

The text is inserted or overwritten at the location of the cursor. You can move the cursor to any desired position in the text file by pressing the arrow keys.

The line in which the cursor is presently located is depicted in a different color. A line can have up to 77 characters. To start a new line, press the RET key or the ENT key.

Cursor movements	Soft key
Move one word to the right	MOVE WORD >>
Move one word to the left	MOVE WORD <<
Go to the next screen page	PAGE ⇩
Go to the previous screen page	PAGE ⇧
Go to beginning of file	BEGIN ⇧
Go to end of file	END ⇩

Editing functions	Key
Begin a new line	RET
Erase the character to the left of the cursor	X
Insert a blank space	SPACE
Switch between upper and lower case letters	SHIFT + SPACE

Erasing and inserting characters, words and lines

With the text editor, you can erase words and even lines, and insert them at any desired location in the text. See the table at right.

To move a word or line to a different location:

- ▶ Move the cursor to the word or line you wish to erase and insert at a different place in the text.
- ▶ Press the DELETE WORD or DELETE LINE soft key: the text is put in the buffer memory
- ▶ Move the cursor to the location where you wish insert the text, and press the RESTORE LINE/WORD soft key.

Delete functions	Soft key
Delete and temporarily store a line	DELETE LINE
Delete and temporarily store a word	DELETE WORD
Delete and temporarily store a character	DELETE CHAR
Insert a line or word from temporary storage	INSERT LINE / WORD

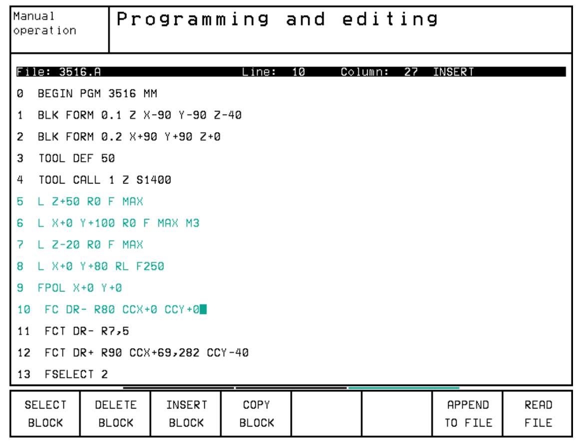
Editing text blocks

You can copy and erase text blocks of any size, and insert them at other locations. Before carrying out any of these editing functions, you must first select the desired text block:

- ▶ To select a text block, move the cursor to the first character of the text you wish to select.

SELECT BLOCK

- ▶ Press the SELECT BLOCK soft key.
- ▶ Move the cursor to the last character of the text you wish to select. You can select whole lines by moving the cursor up or down directly with the arrow keys — the selected text is shown in a different color.



After selecting the desired text block, you can edit the text with the following soft keys:

Function	Soft key
Delete the selected text and store temporarily	DELETE BLOCK
Store marked block temporarily without erasing (copy)	COPY BLOCK

If necessary, you can now insert the temporarily stored block at a different location

- ▶ Move the cursor to the location where you want to insert the temporarily stored text block.

INSERT BLOCK

- ▶ Press the INSERT BLOCK soft key _ the text block in inserted.

You can insert the temporarily stored text block as often as desired.

To transfer the selected text to a different file:

▶ Select the text block as described previously.

APPEND TO FILE

▶ Press the APPEND TO FILE soft key — the TNC displays the dialog prompt Destination file =

▶ Enter the path and name of the target file. The TNC appends the selected text to the end of the specified file. If no target file with the specified name is found, the TNC creates a new file with the selected text.

To insert another file at the cursor position:

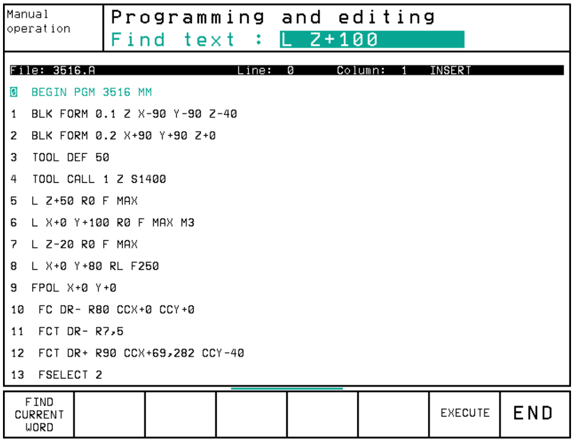
▶ Move the cursor to the location in the text where you wish to insert another file.

READ FILE

▶ Press the READ FILE soft key. The TNC displays the dialog prompt File name =

▶ Enter the path and name of the file you want to insert.

Finding text sections



With the text editor, you can search for words or character strings in a text. Two functions are available:

1. Finding the current text

The search function is to find the next occurrence of the word in which the cursor is presently located:

▶ Move the cursor to the desired word.

▶ To select the search function, press the FIND soft key.

▶ Press the FIND CURRENT WORD soft key

2. Finding any text

▶ To select the search function, press the FIND soft key. The TNC displays the dialog prompt Find text:

▶ Enter the text that you wish to find.

▶ To find the text, press the EXECUTE soft key.

To leave the search function, press the END soft key.

4.10 The Pocket Calculator (not TNC 410)

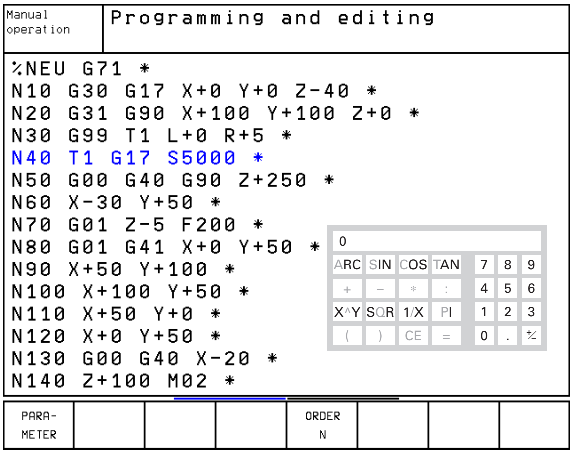
The TNC features an integrated pocket calculator with the basic mathematical functions.

With the CALC key you can open and close an additional window for calculations. You can move the window to any desired location on the TNC screen with the arrow keys.

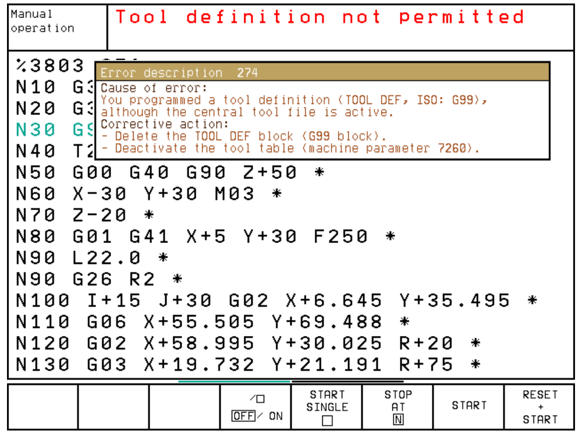
The calculator is operated with short commands through the alphabetic keyboard. The commands are shown in a special color in the calculator window:

Mathematical function	Command
Addition	+
Subtraction	–
Multiplication	*
Division	:
Sine	S
Cosine	C
Tangent	T
Arc sine	AS
Arc cosine	AC
Arc tangent	AT
Powers	^
Square root	Q
Inversion	/
Parenthetic calculations	( )
p (3.14159265359)	P
Display result	=

If you are writing a program and the programming dialog is active, you can use the actual-position-capture key to transfer the result to the highlight position in the current block.



4.11 Direct Help for NC Error Messages (not TNC 410)



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

An error message that contains a program block number was caused by an error in the indicated block or in the preceding block. The TNC error messages can be canceled with the CE key, after the cause of the error has been removed.

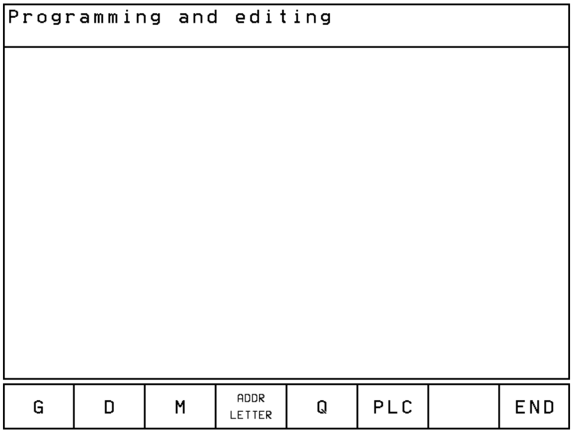
If you require more information on a particular error message, press the HELP key. A window is then superimposed where the cause of the error is explained and suggestions are made for correcting the error.

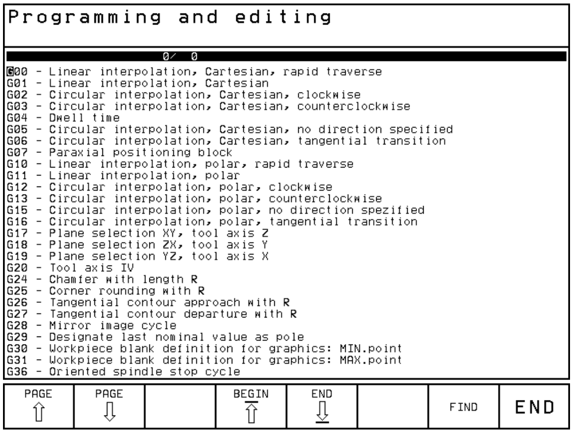
Display HELP
if an error message appears at the top of screen:

- ▶ To display Help, press the HELP key
- ▶ Read the description of the error and the possibilities for correcting it. Close the Help window with the CE, thus canceling the error message
- ▶ Remove the cause of the error as described in the Help window.

The TNC displays the Help text automatically if the error message is flashing. The TNC needs to be restarted after flashing error messages. Press the END key and hold for two seconds.

4.12 Help Function (not TNC 426, TNC 430)





The help function of the TNC includes a description of all of the ISO functions. You can select a HELP topic using the 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
ISO programming: G functions	G
ISO programming: D functions	D
ISO programming: M functions	M
ISO programming: Address letters	ADDR LETTER
Cycle parameters	Q
HELP that is entered by the machine manufacturers (optional, not executable)	PLC
Go to next page	PAGE ⇩
Go to previous page	PAGE ⇧
Go to beginning of file	BEGIN ⇧
Go to end of file	END ⇩
Select search functions; Enter text, Begin search with ENT key	FIND

End the HELP function
Press the END key twice.

4.13 Pallet Management (not TNC 410)

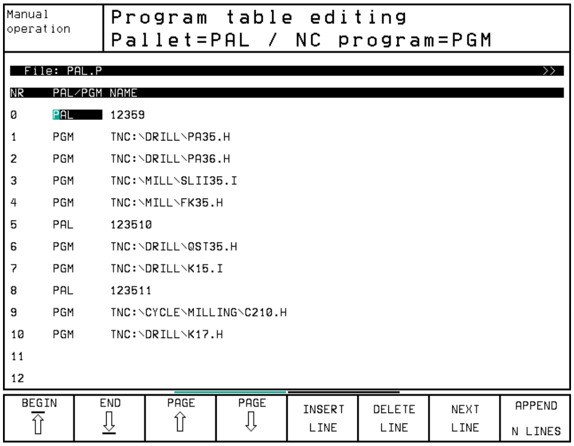
Pallet table management is a machine-dependent function. The standard functional range will be described in the following. Refer to your machine manual for more information.

Pallet tables are used for machining centers with pallet changer: The pallet table calls the part programs that are required for the different pallets, and activates datum shifts or datum tables.

You can also use pallet tables to run in succession several programs that have different datums.

Pallet tables contain the following information:

- PAL/PGM (entry obligatory): Identification for pallet or NC program (select with ENT or NO ENT)
- NAME (entry obligatory): Pallet or program name. The machine tool builder determines the pallet name (see Machine Manual). The program name must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the program.
- DATUM (entry optional): Name of the datum table. The datum table must be stored in the same directory as the pallet table. Otherwise you must enter the full path name for the datum table. Datums from the datum table can be activated in the NC program with Cycle 7 DATUM SHIFT
- X, Y, Z (entry optional; other axes also possible): For pallet names, the programmed coordinates are referenced to the machine datum. For NC programs, the programmed coordinates are referenced to the pallet datum. These entries overwrite the datum that you last set in the Manual mode of operation. With the miscellaneous function M104 you can reactivate the datum that was last set. With the "actual-position-capture" key, the TNC opens a window that enables you to have the TNC enter various points as datums (see next page):



Function	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Insert the last line in the table	INSERT LINE
Delete the last line in the table	DELETE LINE
Go to the beginning of the next line	NEXT LINE
Add the entered number of lines to the end of the table	APPEND N LINES
Copy the highlighted field (2nd soft-key row)	COPY FIELD
Insert the copied field (2nd soft-key row)	PASTE FIELD

Position	Meaning
Actual values	Enter the coordinates of the current tool position relative to the active coordinate system.
Reference values	Enter the coordinates of the current tool position relative to the machine datum.
ACTL measured values	Enter the coordinates relative to the active coordinate system of the datum last probed in the Manual operating mode.
REF measured values	Enter the coordinates relative to the machine datum of the datum last probed in the Manual operating mode.

With the arrow keys and ENT, select the position that you wish to confirm. Then press the ALL VALUES soft key so that the TNC saves the respective coordinates of all active axes in the pallet table. With the PRESENT VALUE soft key, the TNC saves the coordinates of the axis on which the highlight in the pallet table is presently located.

If you have not defined a pallet before an NC program, the programmed coordinates are then referenced to the machine datum.

To select a pallet table:

- Call the file manager in the operating mode Programming and Editing: Press the PGM MGT key.
- Display all .P files: Press the soft keys SELECT TYPE and SHOW .P.
- Select a pallet table with the arrow keys, or enter a new file name to create a new table.
- Confirm your entry with the ENT key.

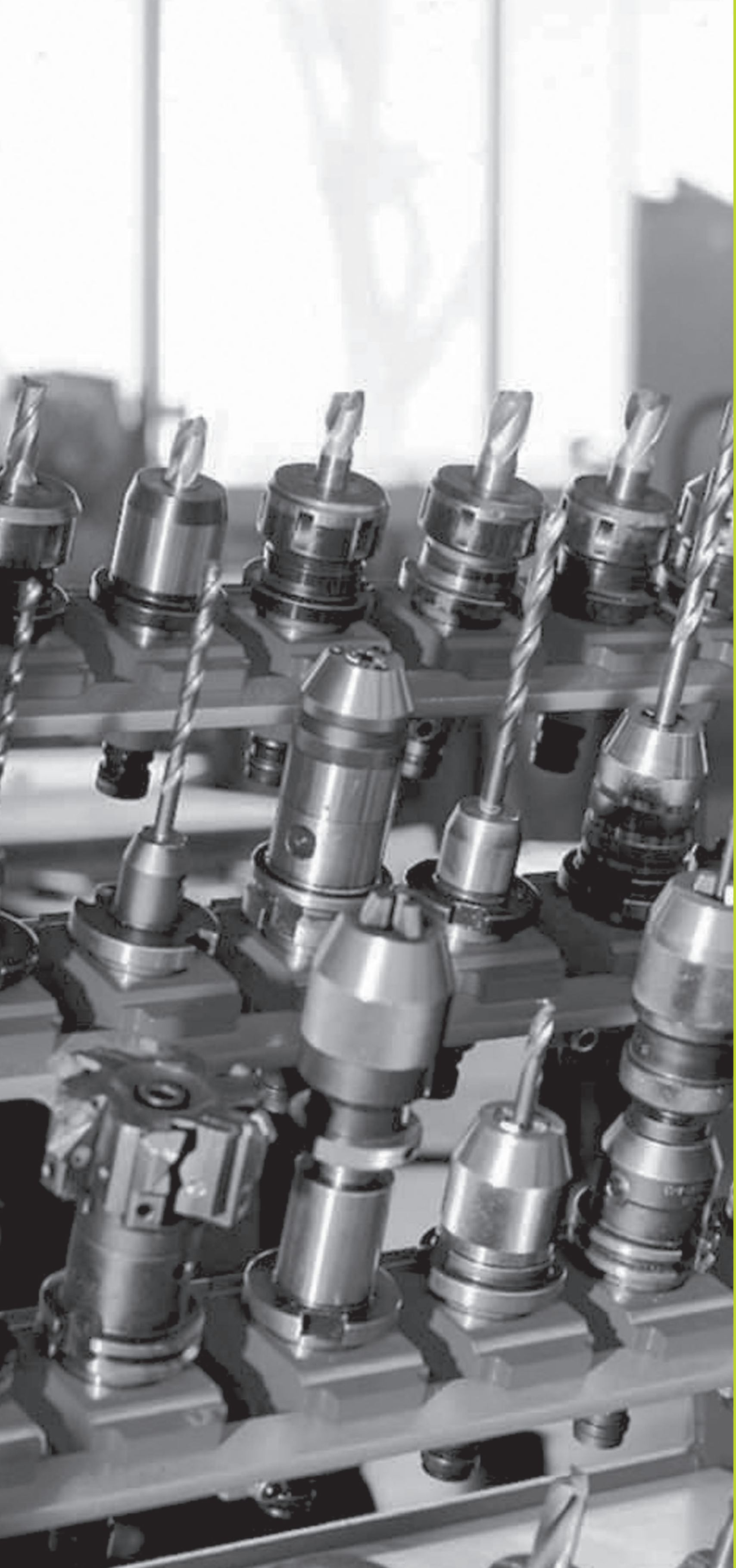
To leave the pallet file:

- To select the file manager, press the Taste PGM MGT key.
- To select a different type of file, press the SELECT TYPE soft key and the soft key for the desired file type, for example SHOW.H.
- Select the desired file.

To execute the pallet file

In machine parameter 7683, set whether the pallet table is to be executed blockwise or continuously (see „14.1 General User Parameters“).

- Select the file manager in the operating mode Program Run, Full Sequence or Program Run, Single Block: Press the PGM MGT key.
- Display all .P files: Press the soft keys SELECT TYPE and SHOW .P.
- Select pallet table with the arrow keys and confirm with ENT.
- Execute pallet table: Press the NC Start button. The TNC executes the pallets as set in Machine Parameter 7683.



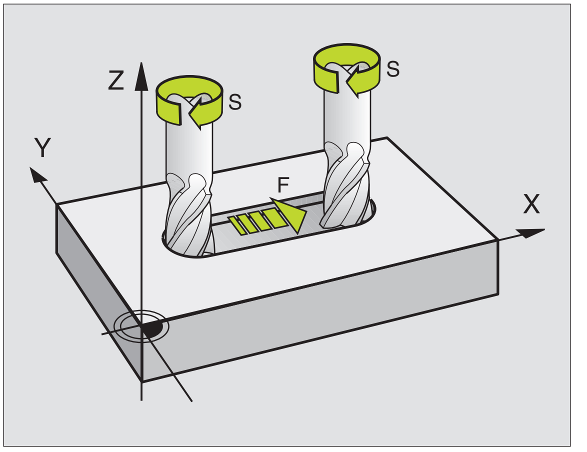
5

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 or in a separate block. Press the F key on the alphabetic keyboard.

Rapid traverse
If you wish to program rapid traverse, enter G00.

Duration of effect
A feed rate entered as a numerical value remains in effect until a block with a different feed rate is reached. If the new feed rate is G00 (rapid traverse), the last programmed feed rate is once again valid after the next block with G01.

Changing 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 any block (e.g. during tool call).

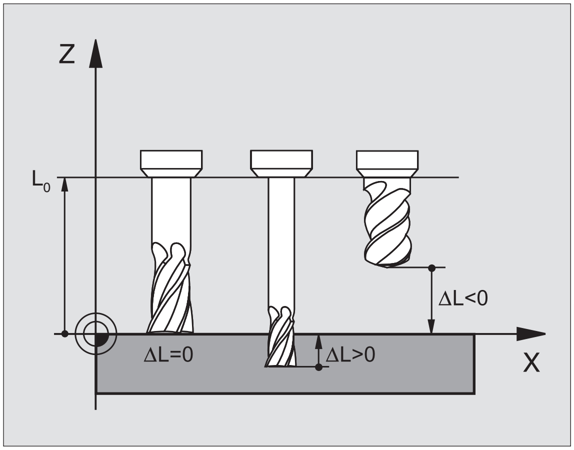
Programmed change
In the part program, you can change the spindle speed with an S block:

S

- ▶ Press the S key on the alphabetic keyboard
- ▶ Enter the new spindle speed

Changing 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 G99 or separately in tool tables. In a tool table, you can also enter additional data on the specific tool. The TNC will consider all the data entered for the tool when executing the part program.

Tool numbers and tool names

Each tool is identified by a number between 0 and 254. If you are working with tool tables, you can use higher numbers (not TNC 410) and you can also enter a tool name for each tool (not TNC 410).

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.
- ▶ Using the key for "actual position capture" (TNC 426 B, TNC 430) or the soft key ACT. POS. Z (TNC 410), transfer the value to the G99 block or the tool table.

2 Determine the tool length L with a tool presetter. This allows you to enter the determined value directly in the G99 tool definition block without further calculations.

Tool radius R

You can enter the tool radius R directly.

Delta values for lengths and radii

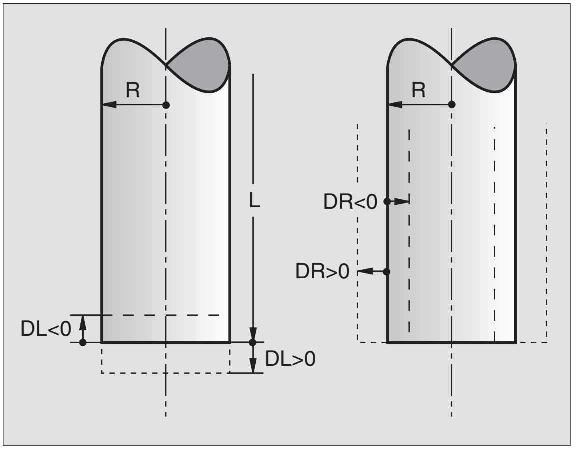
Delta values are offsets in the length and radius of a tool.

A positive delta value describes a tool oversize (DR>0). If you are programming the machining data with an allowance, enter the oversize value with T.

A negative delta value describes a tool undersize (DR<0). An undersize is entered in the tool table for wear.

Delta values are usually entered as numerical values. In a T block, you can also assign the values to Q parameters.

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 G99 block of the part program.

G 99 ▶ Select tool definition. Press ENT to confirm.

▶ Enter the Tool number: Each tool is uniquely identified by its number.

▶ Enter the tool length: Enter the compensation value for the tool length.

▶ Enter the Tool radius.

In the programming dialog, you can transfer the value for tool length directly into the input line.

TNC 426, TNC 430:

Press the actual-position-capture key. You only need to make sure that the highlight in the status display is placed on the tool axis.

TNC 410:

Press the ACT. POS. Z soft key.

Resulting NC block:

```
N40 G99 T5 L+10 R+5 *
```

Entering tool data in tables

You can define and store up to 32767 tools and their tool data in a tool table (TNC 410: 254 tools). In Machine Parameter 7260, you can define how many tool places are to be reserved by the TNC when a new table is set up. See also the Editing Functions at a later stage in this Chapter. For the TNC 426, TNC 430 with the NC software number 280 474-xx, in order to be able to assign various compensation data to a tool (indexing tool number), machine parameter 7262 must not be equal to 0.

You must use tool tables if

- your machine tool has an automatic tool changer,
- you want to measure tools automatically with the TT 120 touch probe (only conversational programming)

Tool table: Available input data

Abbr.	Input	Dialog	Width of column
T	Number by which the tool is called in the program	–	
NAME	Name by which the tool is called in the program	Tool name?	
L	Value for tool length compensation	Tool length?	
R	Compensation value for the tool radius R	Tool radius?	
R2	Tool radius R2 for toroid cutters (only for 3-D radius compensation or graphical representation of a machining operation with spherical or toroid cutters, not TNC 410)	Tool radius 2?	
DL	Delta value for tool length	Tool length oversize?	
DR	Delta value for tool radius R	Tool radius oversize?	
DR2	Delta value for tool radius R2 (not TNC 410)	Tool radius oversize 2 ?	
LCUTS	Tooth length of the tool for Cycle G122	Tool length in the tool axis?	
ANGLE	Maximum plunge angle of the tool for reciprocating plunge-cut in Cycles G122 and G208	Maximum plunge angle?	
TL	Set tool lock (**TL**:Tool Lock	Tool locked? Yes = ENT / No = NO ENT	
RT	Number of replacement tool, if available **(see alsoTIME2)**	Replacement tool?	
TIME1	Maximum tool life in minutes. This function can vary depending on the individual machine tool. Your machine manual provides more information on TIME1.	Maximum tool age ?	
TIME2	Maximum tool life in minutes during TOOL CALL. If the current tool age exceeds this value, the TNC changes the tool during the next TOOL CALL (see also CUR.TIME)	Maximum tool life for TOOL CALL?	
CUR.TIME	Time in minutes the tool has been in use: The TNC automatically counts **the current tool age.** **A starting value can be entered for used tools.**	Current tool life?	

Continued on next page

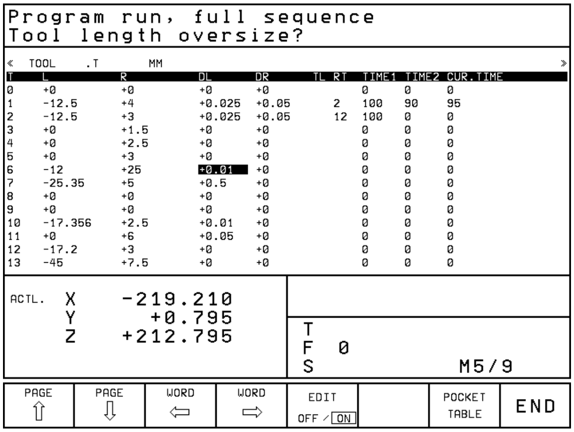
Abbr.	Input	Dialog	Width of column
DOC	Comment on tool (up to 16 characters)	Tool description?	
PLC	Information on this **tool** that is to be sent to the PLC	PLC status?	
Only TNC 426, TNC 430 with NC Software 280 474-xx			
PLC-VAL	Value of this tool that is to be sent to the PLC	PLC value?	

Tool table: Tool data required for automatic tool measurement (only conversational programming)

Abbr.	Input	Dialog
CUT.	Number of teeth (20 teeth maximum)	Number of teeth ?
LTOL	Permissible deviation from tool length L for wear detection. If the entered value is exceeded, the TNC locks the tool(status L). Input range: 0 to 0.9999 mm	Wear tolerance: length ?
RTOL	Permissible deviation from tool radius R for wear detection. If the entered value is exceeded, the TNC locks the tool(status L). Input range: 0 to 0.9999 mm	Wear tolerance: radius ?
DIRECT.	Cutting direction of the tool for measuring the tool during rotation	Cutting direction (M03 = –) ?
TT:R-OFFS	Tool length measurement: tool offset between stylus center and tool center. Preset value: Tool radius R	Tool offset: radius ?
TT:L-OFFS	Tool radius measurement: tool offset in addition to MP6530 (see „14.1 General User Parameters) between upper surface of stylus and lower surface of tool. Preset value: 0	Tool offset: length ?
LBREAK	Permissible deviation from tool length L for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: length ?
RBREAK	Permissible deviation from tool radius R for breakage detection. If the entered value is exceeded, the TNC locks the tool (status L). Input range: 0 to 0.9999 mm	Breakage tolerance: radius ?

Editing tool tables

The tool table that is active during execution of the part program is designated TOOL.T. TOOL.T must be stored in the directory TNC:\ and can be edited in any of the machine operating modes. Other tool tables that are used for archiving or test runs are given different file names with the extension .T .



To open the tool table TOOL.T:

▶ Select any machine operating mode.

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

EDIT OFF/ON ▶ Set the EDIT soft key to ON.

To open any other tool table

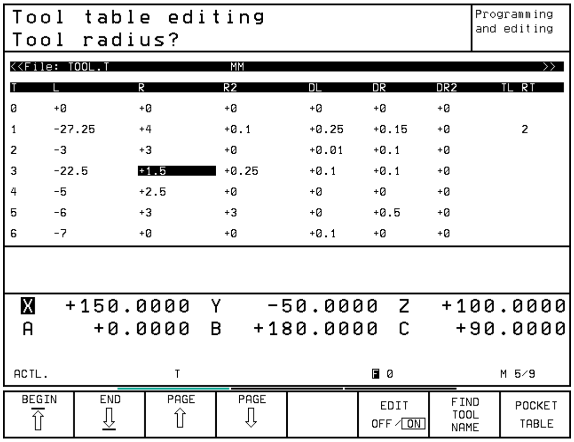
▶ Select the Programming and Editing mode of operation.

PGM MGT ▶ Calling the file manager

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

▶ To show type .T files, press the SHOW soft key.

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



When you have opened the tool table, you can edit the tool data by moving the cursor to the desired position in the table with the arrow keys or the soft keys (see figures at upper and center right). You can overwrite the stored values, or enter new values at any position. Refer to the table (on the next page) for additional editing functions.

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

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 TNC 426, TNC 430	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Look for the tool name in the table	FIND TOOL NAME
Show tool information in columns or show the information on one tool on one screen page	LIST FORMULAR
Move to beginning of line	BEGIN LINE ⇦
Move to end of line	END LINE ⇨
Copy the highlighted field	COPY FIELD
Insert the copied field	PASTE FIELD
Add the entered number of lines (tools) to the end of the table	APPEND N LINES
Only TNC 426 B, TNC 430 with the NC software 280 474-xx: Insert a line for the indexed tool number after the active line. The function is only active if you are permitted to store various compensation data for a tool (machine parameter 7262 not equal to 0). The TNC inserts a copy of the tool data after the last available index and increases the index by 1	ZEILE EINFÜGEN
Delete current line (tool)	DELETE LINE
Display / Do not display pocket numbers	SHOW OMIT POCKET NR
Display all tools / only those tools that are stored in the pocket table	HIDE TOOLS OFF/ON

Editing functions for Tool Table TNC 410	Soft key
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Move highlight to the left	WORD ⇦
Move highlight to the right	WORD ⇨
Lock tool in TL column	YES
Do not lock tool in TL column	NO
Confirm actual positions, e.g. for Z axis	ACT.POS. Z
Confirm entered value Select next column in the table	ENT
Delete incorrect value, restore previous value	CE
Restore last value stored	DEL

Additional notes on tool tables

Machine parameter 7266.x defines which data can be entered in the tool table and in what sequence the data is displayed. Note when configuring the tool table that the total width cannot be more than 250 characters. Wider tables cannot be transferred over the interface. The width of the individual columns is given in the description of MP7266.x.

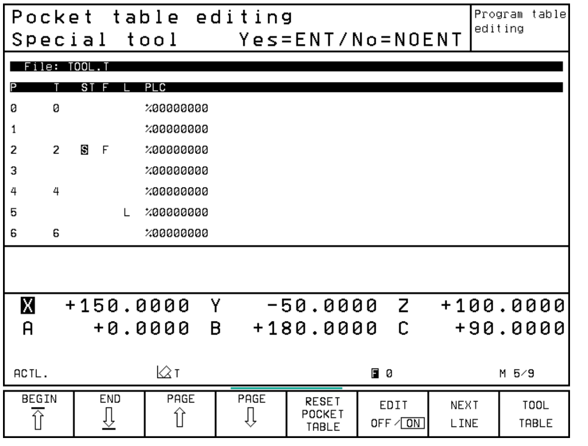
You can overwrite individual columns or lines of a tool table with the contents of another file. Prerequisites:

- The target file must exist.
- The file to be copied must contain only the columns (or lines) you want to replace.

To copy individual columns or lines, press the REPLACE FIELDS soft key (see 4.4 Enhanced File Management for the TNC 426, TNC 430).

Pocket table for tool changer

For automatic tool changing you need the pocket table TOOL_P.TCH. The TNC 426, TNC 430 with the NC software 280 474-xx manages more than one pocket table with any file names. To activate a specific pocket table for program run you must select it in the file management of a Program Run mode of operation (status M).



Editing a pocket table in a Program Run operating mode:

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

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

EDIT OFF/ON
- ▶ Set the EDIT soft key to ON

Selecting a pocket table in the Programming and Editing operating mode (only TNC 426, TNC 430 with NC software 280 474-xx):

PGM MGT
- ▶ Calling the file manager
- ▶ To select the file type, press the SELECT TYPE soft key.
- ▶ To show files of the type .TCH, press the soft key TCH FILES (second soft-key row).
- ▶ Select a file or enter a new file name. Conclude your entry with the ENT key or SELECT soft key.

You can enter the information below on a tool into a pocket table

Editing functions for pocket tables	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Select previous page in table	PAGE ⇧
Select next page in table	PAGE ⇩
Reset pocket table	RESET POCKET TABLE
Go to the beginning of the next line	NEXT LINE
Reset tool number column T	RESET COLUMN T

Abbr.	Input	Dialog
P	Pocket number of the tool in the tool magazine	–
T	Tool number	Tool number ?
ST	Special tool with a large radius requiring several pockets in the tool magazine. If your special tool takes up pockets in front of and behind its actual pocket, these additional pockets need to be locked (status L).	Special tool Yes = ENT / No = NO ENT TNC 410: Special tool?
F	Fixed tool number. The tool is always returned to the same pocket.	Fixed pocket Yes = ENT / No = NO ENT TNC 410: Fixed pocket?
L	Locked pocket	Pocket locked Yes = ENT / No = NO ENT TNC 410: Pocket locked?
PLC	Information on this **tool pocket** that is to be sent to the PLC	PLC status?
TNAME	Display of the tool name from TOOL.T (only NC software 280 474-xx)	–

Calling tool data

A tool call in the machining program is triggered with the function T.

Example:

T **1**

Press T on the alphabetic keyboard.

Enter the tool number or the tool name: The tool must already be defined in a G99 block or in the tool table. To call a tool by the tool name, enter the name in quotation marks. The tool name always refers to the entry in the active tool table TOOL .T (not TNC 410).
In addition, for the TNC 426, TNC 430 with NC software 280 474-xx:
If you wish to call a tool with other compensation values, enter also the index you defined in the tool table after the decimal point.

Only TNC 426, TNC 430 with NC software 280 474-xx and TNC 410:

ENT

`Tool length oversize?`

0.5 Delta value for the tool length

ENT

`Tool radius oversize?`

0.5 Delta value for tool radius

G **17** Select spindle axis: e.g. Z-axis

S **2500** Select rotational speed and end the block with the END key

The program blocks window will display the following line:

```
N20 T5 G17 S2500 *
```

or

```
N20 T5.2 DL+0.5 DR+0.5 G17 S2500 *
```

Tool preselection with tool tables
When you use tool tables, enter a G51 block to preselect the next tool to be selected. Simply enter the tool number or a corresponding Q parameter, or type the tool name in quotation marks (not TNC 410).

Tool change

> The tool change function can vary depending on the individual machine tool. Refer to your machine tool manual.

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 T0 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 11.4 "Program Run").
- ▶ Change the tool.
- ▶ Resume the program run (see section 11.4 "Program Run").

Automatic tool change
If your machine tool has automatic tool changing capability, the program run is not interrupted. When the TNC reaches a tool call with T, it replaces the inserted tool by another from the tool magazine.

Automatic tool change if the tool life expires: M101

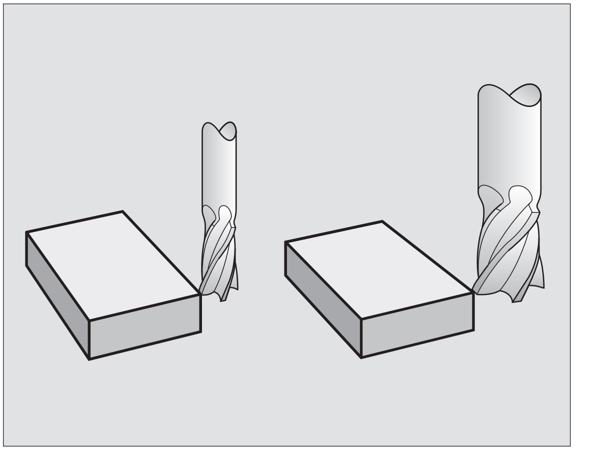
> This function can vary depending on the individual machine tool. Refer to your machine tool manual.

The TNC automatically changes the tool if the tool life TIME1 expires during program run. To use this miscellaneous function, activate M101 at the beginning of the program. M101 is reset with M102.

The tool is not always changed immediately, but, depending on the workload of the control, a few NC blocks later.

Prerequisites for standard NC blocks with radius compensation G40, G41, G42
The radius of the replacement tool must be the same as that of the original tool. If the radii are not equal, the TNC displays an error message and does not replace the tool. This error message can be suppressed with M107 (not TNC 410).

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. The TNC accounts for the compensation value in up to five axes (TNC 410: four axes) including the rotary axes.

Tool length compensation

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

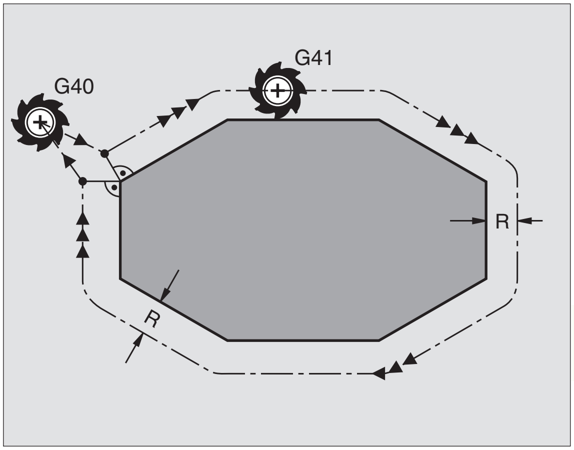
> If you cancel a positive length compensation with 0, the distance between tool and workpiece will be reduced.
>
> After a tool call, the path of the tool in the tool axis, as entered in the part program, is adjusted by the difference between the length of the previous tool and that of the new one.

For tool length compensation, the TNC takes the delta values from both the T block and the tool table into account.

Compensation value = $L + DL_T + DL_{TAB}$, where

L	is the tool length L from the G99 block or tool Table
DL_T	is the oversize for length DL in the T block (not taken into account by the position display)
DL_{TAB}	is the oversize for length DL in the tool table

Tool radius compensation



The NC block for programming a tool movement contains:

- G41 or G42 for radius compensation
- G43 or G44, for radius compensation with axis-parallel traverse
- G40, if there is no radius compensation

Radius compensation becomes effective as soon as a tool is called and is moved in the working plane with G41 or G42.

> The TNC automatically cancels radius compensation if you:
>
> - Program a positioning block with G40
> - Program a call with %...
> - Select a new program

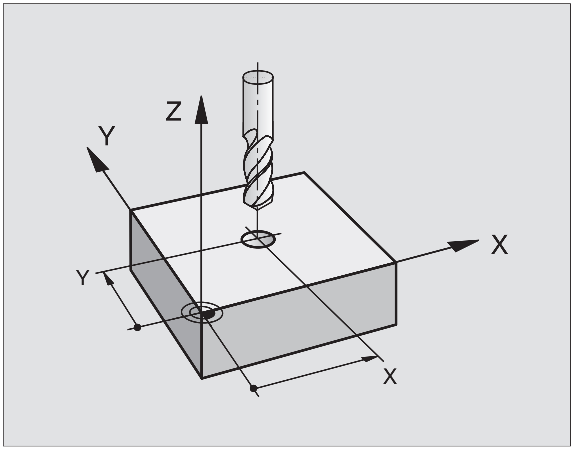
For tool radius compensation, the TNC takes the delta values from both the T block and the tool table into account.

Compensation value = $R + DR_T + DR_{TAB}$, where

R	is the tool radius R from the G99 block or tool table
DR_T	is the oversize for radius DR in the T block (not taken into account by the position display)
DR_{TAB}	is the oversize for radius DR in the tool table

Contouring without radius compensation: G40
The tool center moves in the working plane to the programmed path or coordinates.

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

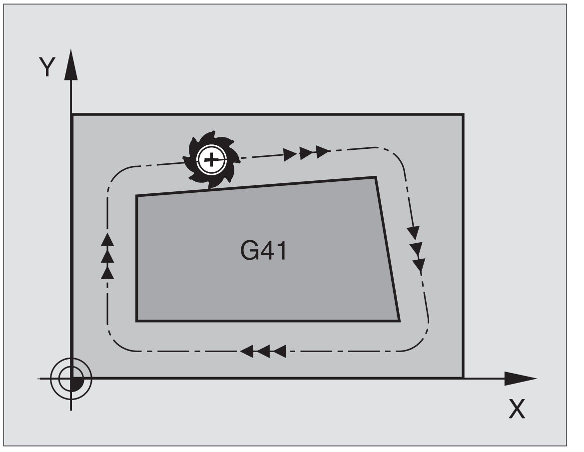


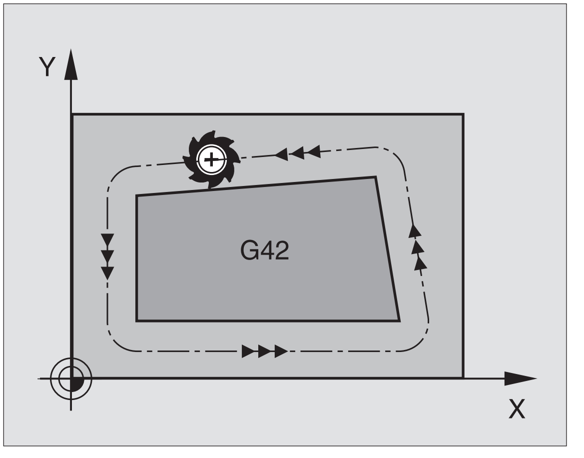
Contouring with radius compensation: G41 and G42

G41 The tool moves to the left of the programmed contour

G42 The tool moves to the right 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





Between two program blocks with different radius compensations (G41 and G42) you must program at least one block without radius compensation (that is, with G40).

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

You can also activate the radius compensation for secondary axes in the working plane. Program the secondary axes too in each following block, since otherwise the TNC will execute the radius compensation in the principal axis again.

Whenever radius compensation is activated with G41/ G42 or canceled with G40, 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.

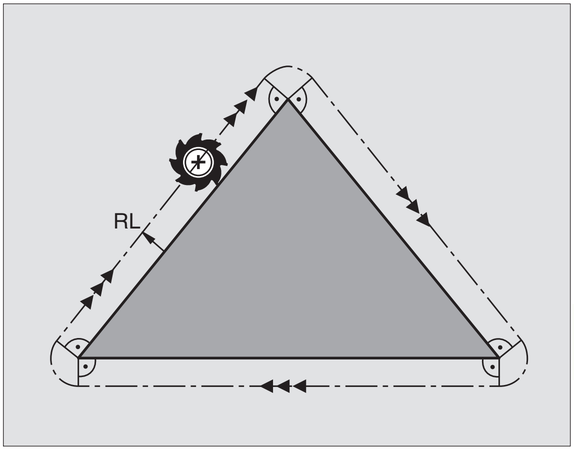
Entering radius compensation

Radius compensation is entered in a G01 block:

G 41	To select tool movement to the left of the contour, select function G41, or
G 42	To select tool movement to the right of the contour, select function G42, or
G 40	To select tool movement without radius compensation or to cancel radius compensation, select function G40.
END	To terminate the block, press the END key.

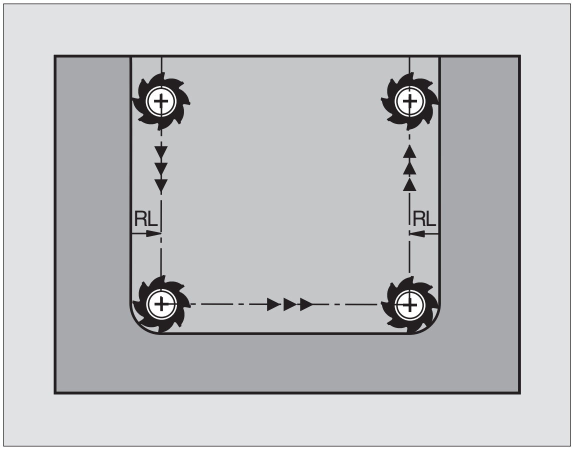
Radius compensation: Machining corners

Outside corners
If you program radius compensation, the TNC moves the tool 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 functions M90 and M112.See "7.4 Miscellaneous Functions for Contouring Behavior."

6

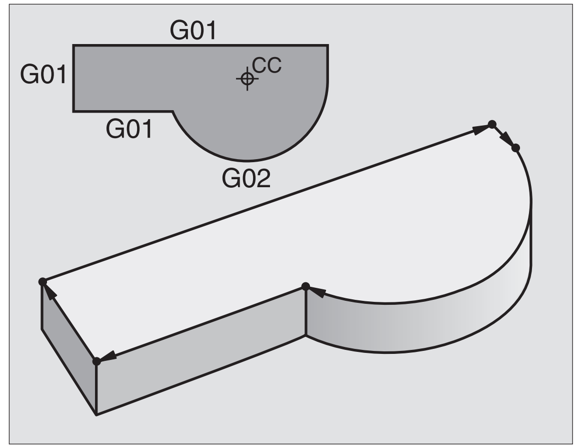
Programming:

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**.



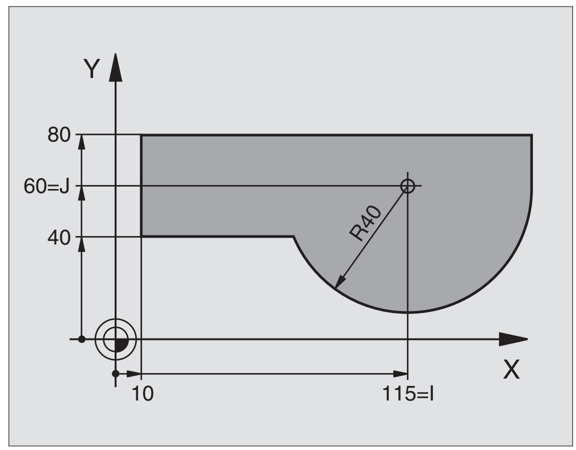
Miscellaneous functions M

With the TNC's miscellaneous functions you can affect

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

Subprograms and program section repeats

If a machining sequence occurs several times in a program, you can save time and reduce the chance of programming errors by entering the sequence once and then defining it as a subprogram or program section repeat. If you wish to execute a specific program section only under certain conditions, you also define this machining sequence as a subprogram. In addition, you can have a part program call a separate program for execution.



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

Programming with Q parameters

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

In addition, parametric programming enables you to measure with the 3-D touch probe during program run.

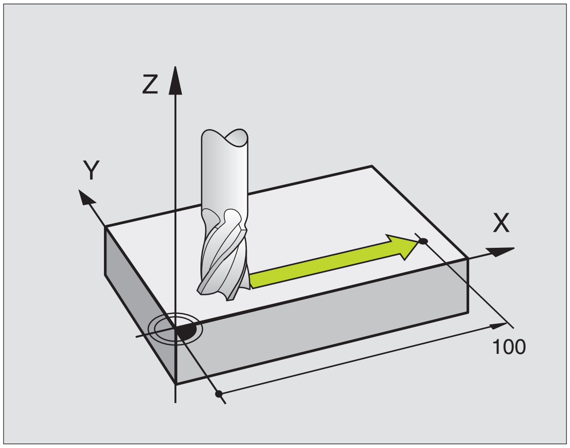
Programming with Q parameters is described in Chapter 10.

6.2 Fundamentals of Path Functions

Programming tool movements for workpiece machining

You create a part program by programming the path functions for the individual contour elements in sequence. You 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.

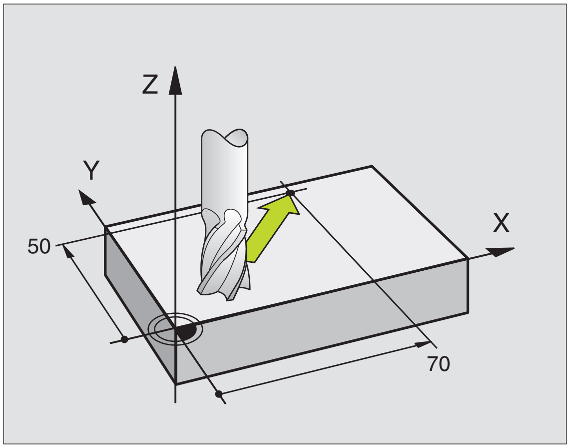
Depending on the individual machine tool, the part program is executed by movement of either the tool or the machine table on which the workpiece is clamped. Nevertheless, you always program path contours as if the tool moves and the workpiece remains stationary.

Example:

```
N50 G00 X+100 *
```

N50	Block number
G00	Path function "straight line at rapid traverse"
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.



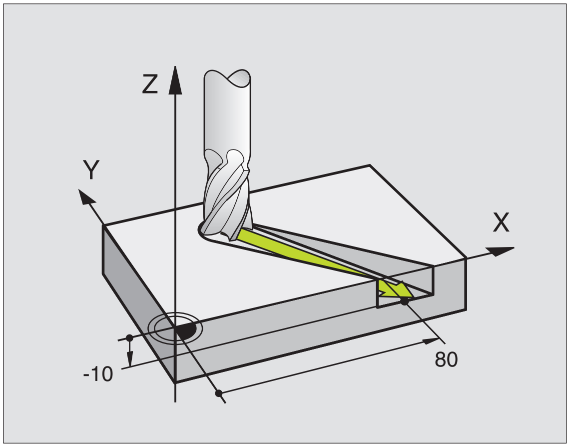
Movement in the main planes

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

Example:

```
N50 G00 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:

```
N50 G01 X+80 Y+0 Z-10 F1000 *
```

See figure at lower right.

Entering more than three coordinates (not TNC 410)

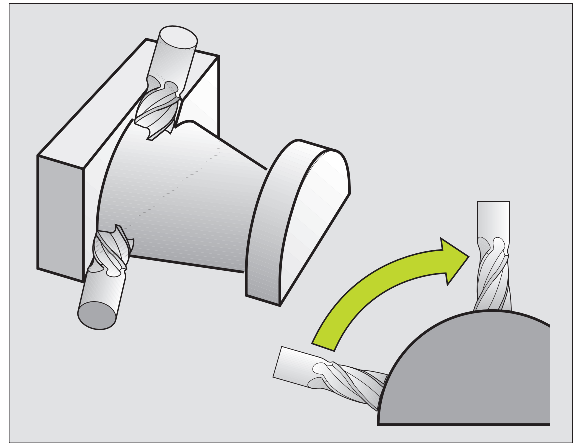
The TNC can control up to five axes simultaneously (for example, three linear and two rotary axes).

Such programs are too complex to program at the machine, however, and are usually created with a CAD system.

Example:

```
N50 G01 G40 X+20 Y+10 Z+2 A+15 C+6 F100 M3 *
```

The TNC graphics cannot simulate movements in more than three axes.



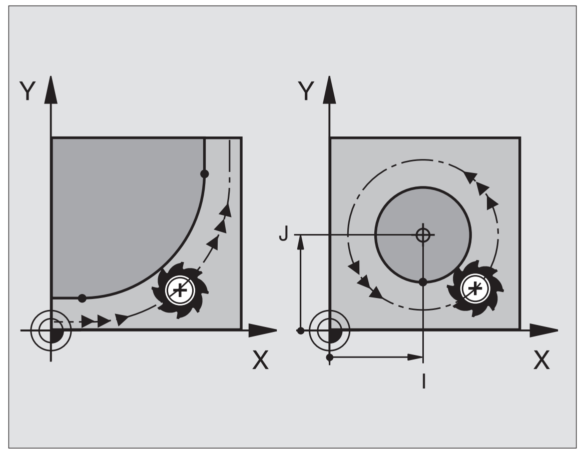
Circles and circular arcs

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

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	Circle center
Z (G17)	**XY**, also UV, XV, UY	**IJ**
Y (G18)	**ZX**, also WU, ZU, WX	**KI**
X (G19)	**YZ**, also VW, YW, VZ	**JK**

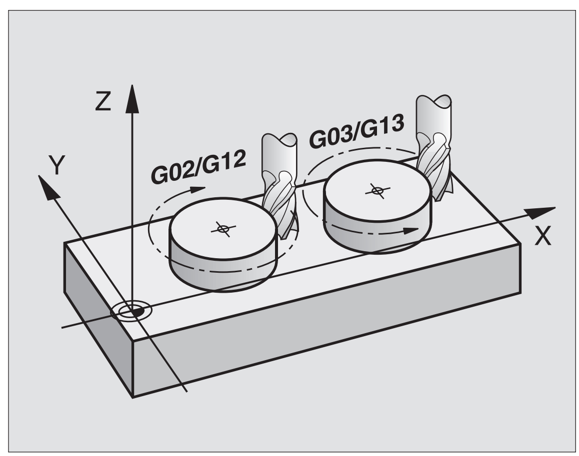
On the TNC 426, TNC 430 you can program circles that do not lie parallel to a main plane by using the function for tilting the working plane (see section 8.9) or Q parameters (see Chapter 10).



Direction of rotation for circular movements

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

Clockwise direction of rotation: GO2/G12
Counterclockwise direction of rotation: G03/G13



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.

Straight-line block, see „6.4 Path Contours _ Polar Coordinates".

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

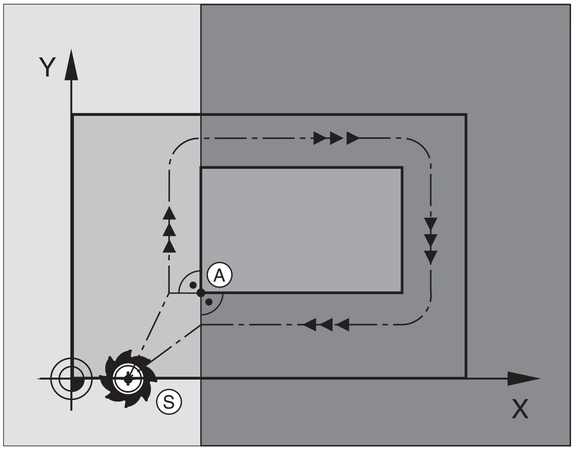
6.3 Contour Approach and Departure

Starting point and end point

The tool approaches the first contour point from the starting point. The starting point must be:

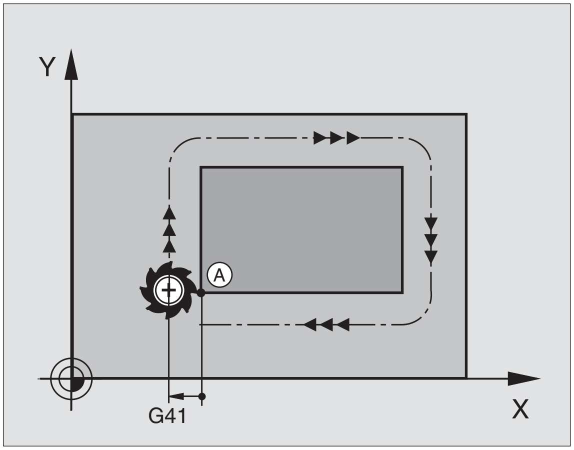
- Programmed without radius compensation
- Approachable without danger of collision
- Close to the first contour point

In order to make sure the contour will not be damaged, the optimal starting point should lie on the extended tool path for machining the first contour element.

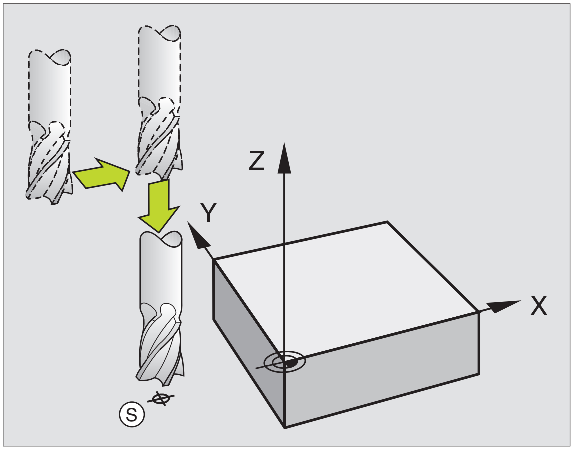


Example
Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.

First contour point
You need to program a radius compensation for the tool movement to the first contour point.



Approaching the starting point in the spindle axis
The tool must be moved to the working depth in the spindle axis. If danger of collision exists, approach the starting point in the spindle axis separately.

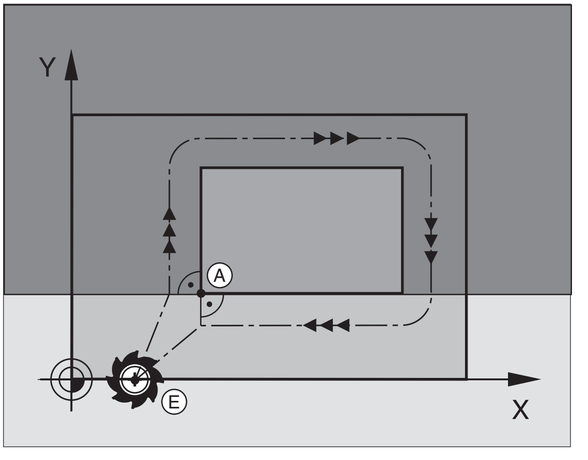


Example NC blocks

```
N30 G00 G40 X+20 Y+30 *
N40 Z-10 *
```

End point
The end point should be selected so that it is:

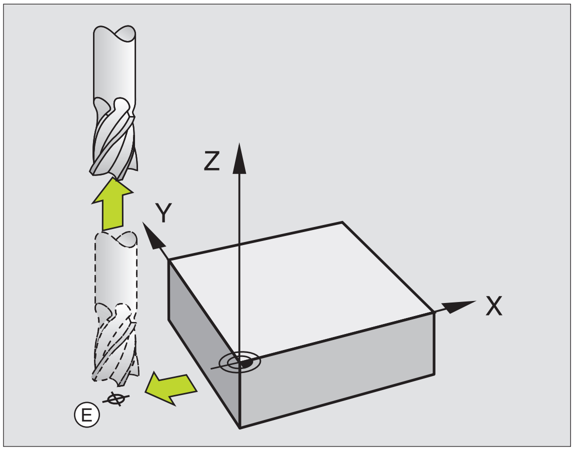
- Approachable without danger of collision
- Near to the last contour point
- In order to make sure the contour will not be damaged, the optimal ending point should lie on the extended tool path for machining the last contour element.



Example:
Figure at center right: If you set the ending point in the dark gray area, the contour will be damaged when the end point is approached.

Depart the end point in the spindle axis:

Program the departure from the end point in the spindle axis separately. See figure at lower right.



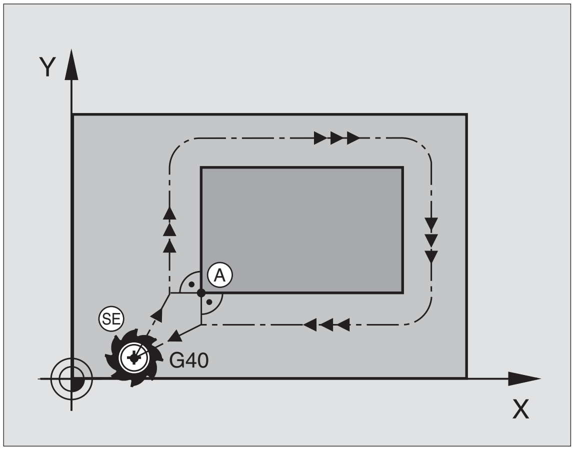
Example NC blocks

```
N50 G00 G40 X+60 Y+70 *
N60 Z+250 *
```

Common starting and end points
Do not program any radius compensation if the starting point and end point are the same.

In order to make sure the contour will not be damaged, the optimal starting point should lie between the extended tool paths for machining the first and last contour elements.

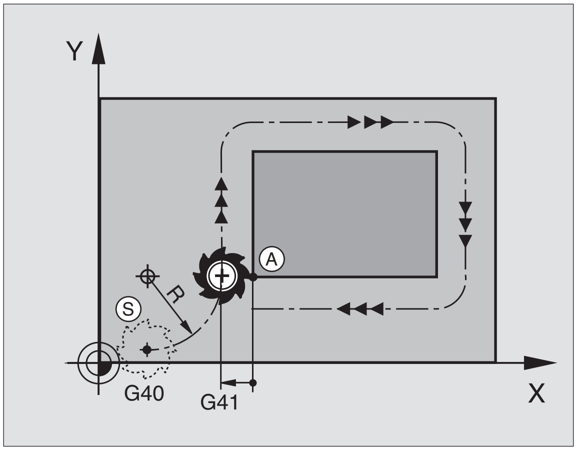
Example
Figure at upper right: If you set the starting point in the dark gray area, the contour will be damaged when the first contour element is approached.



Tangential approach and departure

With G26 (figure at center right), you can program a tangential approach to the workpiece, and with G27 (figure at lower right) a tangential departure. In this way you can avoid dwell marks.

Starting point and end point
The starting point and the end point lie outside the workpiece, close to the first and last contour points. They are to be programmed without radius compensation.

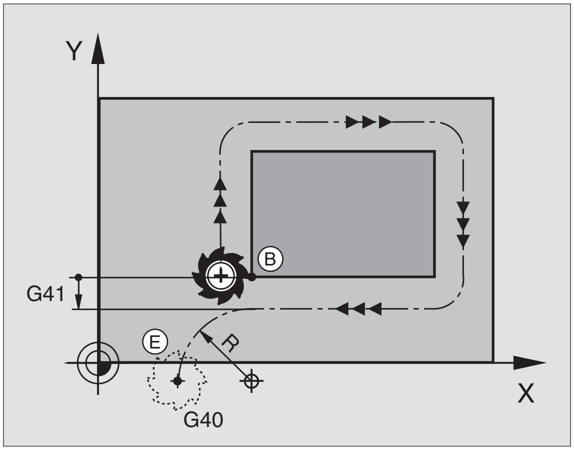


Approach:

▶ Enter G26 after the block in which the first contour element is programmed: This will be the first block with radius compensation G41/G42

Departure:

▶ Enter G27 after the block in which the last contour element is programmed: This will be the last block with radius compensation G41/G42



> Select the radius for G26 and G27 so that it is possible to execute the circular path between the starting point and the first contour point, as well as the last contour point and the end point.

Example NC blocks

`N50 G00 G40 G90 X-30 Y+50 *`	Starting position
`N60 G01 G41 X+0 Y+50 F350 *`	First contour point
`N70 G26 R5 *`	Tangential approach with radius R = 5 mm
`...`	
`PROGRAM CONTOUR BLOCKS`	
`...`	
`N210 X+0 Y+50 *`	Last contour point
`N220 G27 R5 *`	Tangential departure with radius R = 5 mm
`N230 G00 G40 X-30 Y+50 *`	End point

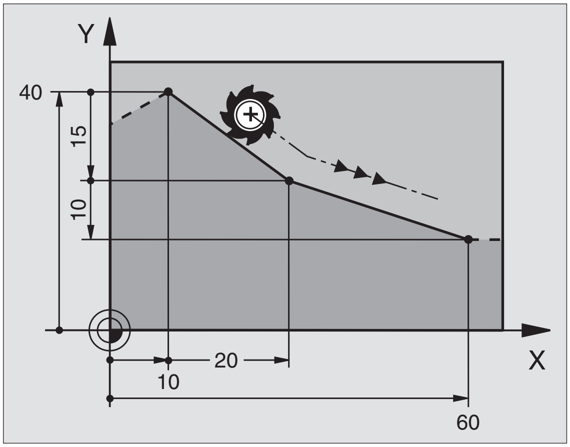
6.4 Path Contours — Cartesian Coordinates

Overview of path functions

Tool movement	Function	Required input
Straight line at rapid traverse Straight line with feed rate F	G00 G01	Coordinates of the end points of the straight line
Chamfer between two straight lines	G24	Length of chamfer R
–	I, J, K	Coordinates of the circle center
Circular path in clockwise direction Circular path in counterclockwise direction	G02 G03	Coordinates of the arc end point in connection with I, J, K or additional circular radius R
Circular path corresponding to active direction of rotation	G05	Coordinates of the arc end point and circular radius R
Circular path with tangential connection to previous contour element	G06	Coordinates of the arc end point
Circular path with tangential connection to previous and following contour element	G25	Rounding-off radius R

Straight line at rapid traverse G00
Straight line with feed rate G01 F . . .

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



G 1

▶ Enter the coordinates of the end point.

Further entries, if necessary:

▶ Radius compensation G40/G41/G42

▶ Feed rate F

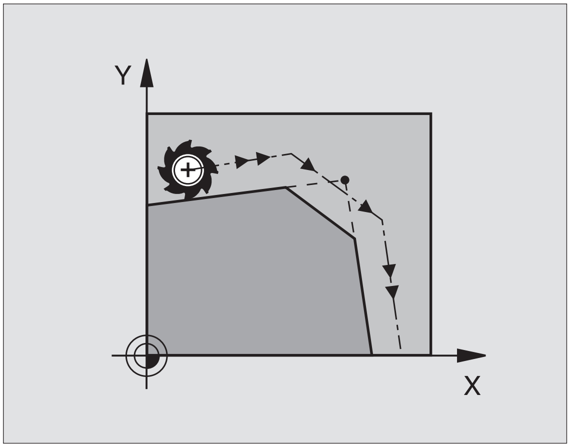
▶ Miscellaneous function M

Example NC blocks

```
N70 G01 G41 X+10 Y+40 F200 M3 *
N80 G91 X+20 Y-15 *
N90 G90 X+60 G91 Y-10 *
```

Inserting a chamfer 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 G24 block must be in the same working plane.
- The radius compensation before and after the G24 block must be the same.
- An inside chamfer must be large enough to accommodate the current tool.

G 24

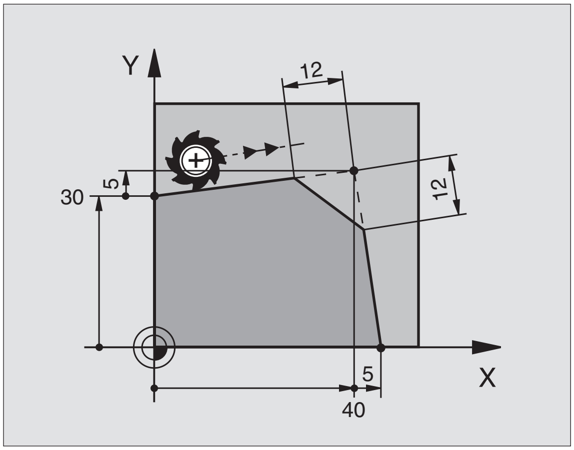
▶ Confirm your entry with the ENT key.

▶ Chamfer side length: Enter the length of the chamfer

▶ Feed rate F (effective in G24 block only)

Example NC blocks

```
N70 G01 G41 X+0 Y+30 F300 M3 *
N80 X+40 G91 Y+5 *
N90 G24 R12 *
N100 X+5 G90 Y+0 *
```



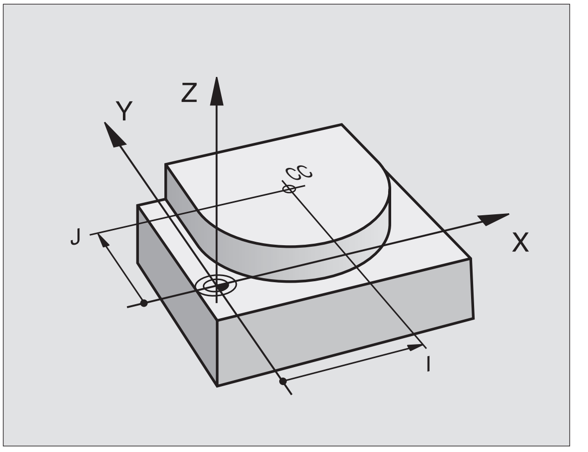
☞ You cannot start a contour with a G24 block!

A chamfer is possible only in the working plane.

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

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

Circle center I, J



You can define a circle center for circles that are programmed with the functions G02/G03. 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-capture key

I J ▶ Enter the coordinates for the circle center
If you want to use the last programmed position, enter G29.

Example NC blocks

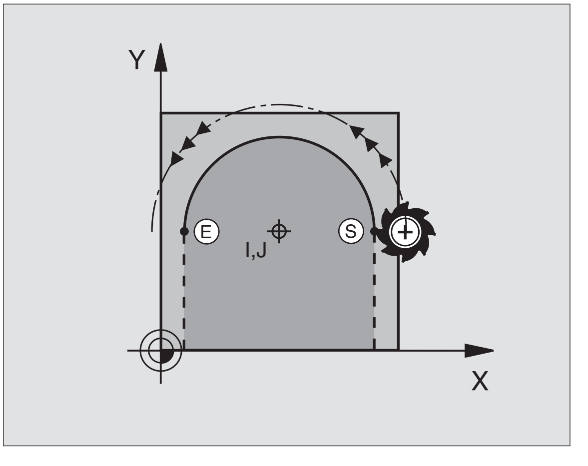
```
N50 I+25 J+25 *
```

Duration of effect
The circle center definition remains in effect until a new circle center is programmed. You can also define a circle center for the secondary axes U, V and W.

Enter incremental values for the circle center I, J
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 I and J is to define a position as a circle center — the tool does not move to the position.
>
> The circle center is also the pole for polar coordinates.
>
> If you wish to define the pole in parallel axes, first press the key I (J) on the ASCII keyboard, and then the orange axis key for the corresponding parallel axis.

Circular path G02/G03/G05 around the circle center I, J



Before programming a circular arc, you must first enter the circle center I, J. The last programmed tool position will be the starting point of the arc.

Direction
- In clockwise direction: G02
- In counterclockwise direction: G03
- Without direction indication: G05
  The TNC traverses the circular arc with the last programmed direction of rotation.

▶ Move the tool to the circle starting point.

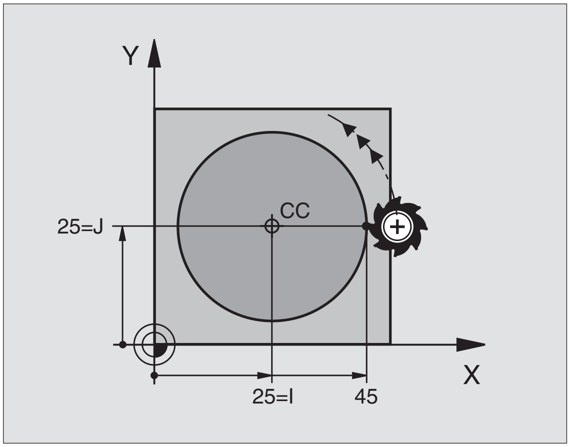
I J ▶ Enter the coordinates of the circle center.

G 3 ▶ Enter the coordinates of the arc end point.

Further entries, if necessary:

▶ Feed rate F

▶ Miscellaneous function M



Example NC blocks

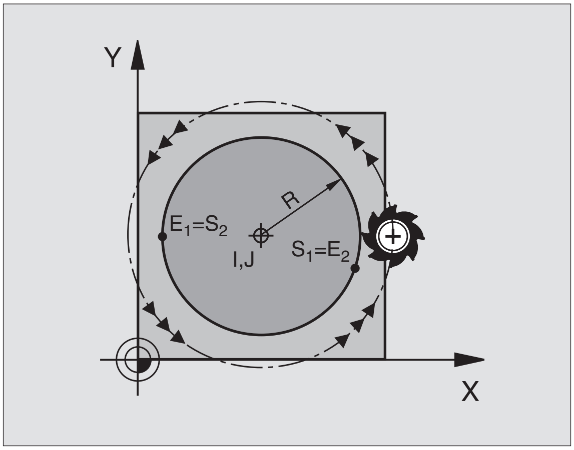
```
N50 I+25 J+25 *
N60 G01 G42 X+45 Y+25 F200 M3 *
N70 G03 X+45 Y+25 *
```

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 (selected with MP7431, not for TNC 410)

Circular path G02/G03/G05 with defined radius

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



Direction

- In clockwise direction: G02
- In counterclockwise direction: G03
- Without direction indication: G05
  The TNC traverses the circular arc with the last programmed direction of rotation.

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

G 3 ▶ Enter the coordinates of the arc end point.

▶ Radius R
Note: The algebraic sign determines the size of the arc.

Further entries, if necessary:

▶ Feed rate F

▶ Miscellaneous function M

Full circle
For a full circle, program two 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 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:

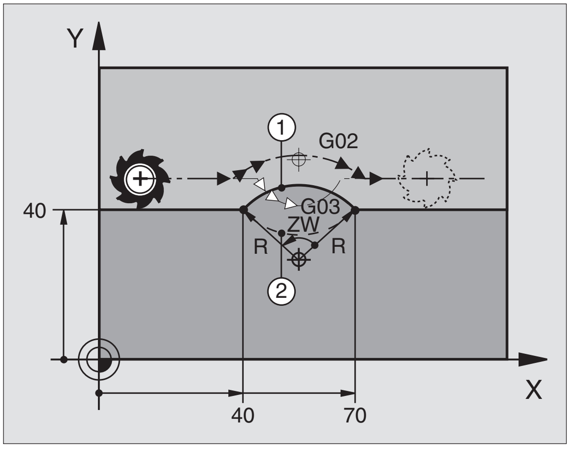
Smaller arc: CCA<180°
Enter the radius with a positive sign R>0

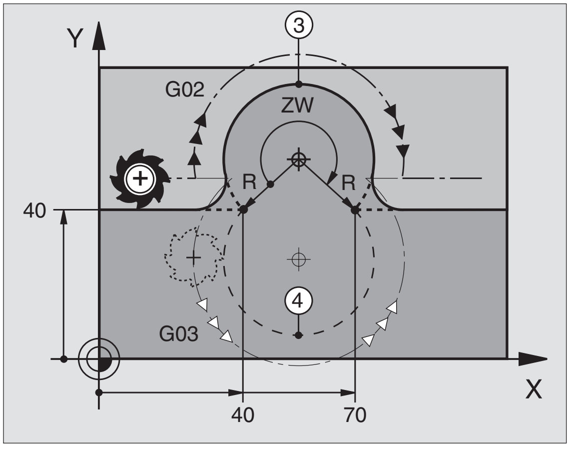
Larger arc: CCA>180°
Enter the radius with a negative sign R<0

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

Convex: Direction of rotation G02 (with radius compensation G41)

Concave: Direction of rotation G03 (with radius compensation G41)





Example NC blocks
See figure at right.

```
N100 G01 G41 X+40 Y+40 F200 M3 *
N110 G02 X+70 Y+40 R+20 * (arc 1)
```
or

```
N110 G03 X+70 Y+40 R+20 * (arc 2)
```
or

```
N110 G02 X+70 Y+40 R-20 * (arc 3)
```
or

```
N110 G03 X+70 Y+40 R-20 * (arc 4)
```

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

The maximum radius is 99 999 mm
(TNC 410: 9999 mm).

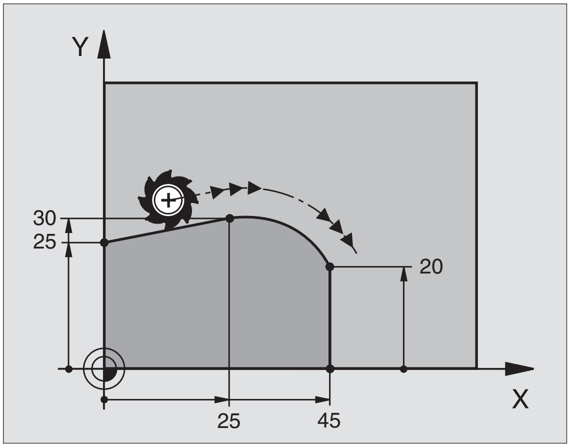
You can also enter rotary axes A, B and C.

Circular path G06 with tangential approach

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 G06 block. This requires at least two positioning blocks.



G 6

▶ Enter the coordinates of the arc end point.

Further entries, if necessary:

▶ Feed rate F

▶ Miscellaneous function M

Example NC blocks

```
N70 G01 G41 X+0 Y+25 F300 M3 *
N80 X+25 Y+30 *
N90 G06 X+45 Y+20 *
N100 G01 Y+0 *
```

A tangential arc is a two-dimensional operation: the coordinates in the G06 block and in the contour element preceding it must be in the same plane of the arc.

Rounding corners G25

The G25 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.

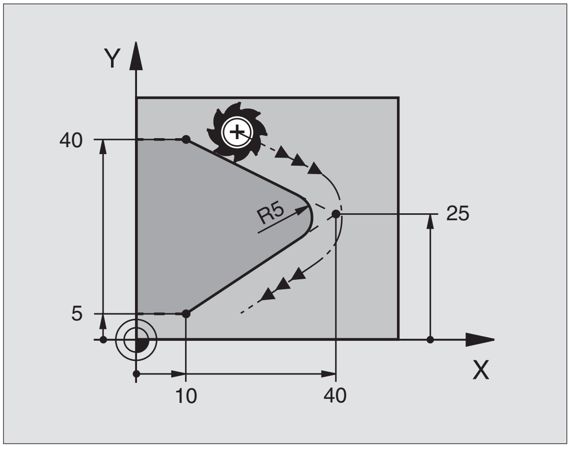
The rounding arc must be large enough to accommodate the tool.

G 25

▶ Confirm your entry with the ENT key.

▶ Rounding radius: Enter the radius of the circular arc

▶ Feed rate for rounding the corner.



Example NC blocks

```
N50 G01 G41 X+10 Y+40 F300 M3 *
N60 X+40 Y+25 *
N70 G25 R5 F100 *
N80 X+10 Y+5 *
```

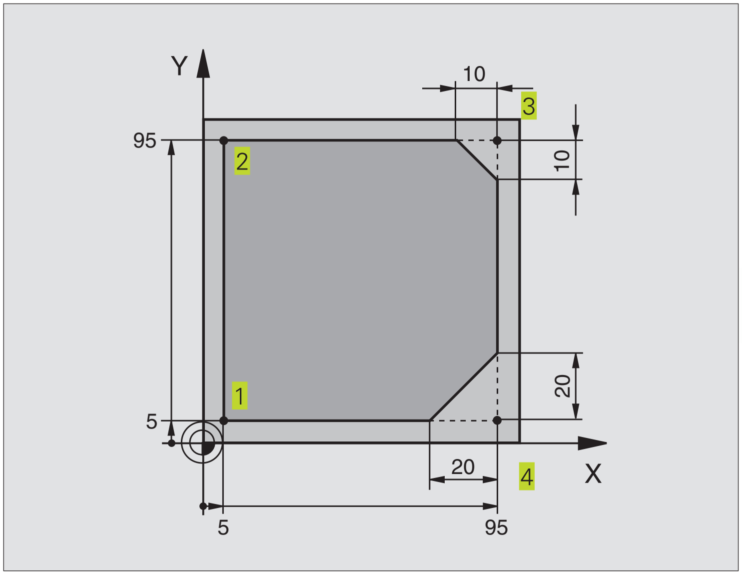
In the preceding and subsequent contour elements, both coordinates must lie in the plane of the rounding arc. If you machine the contour without tool-radius compensation, you must program both coordinates in the working plane.

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

A feed rate programmed in the G25 block is effective only in that block. After the G25 block, the previous feed rate becomes effective again.

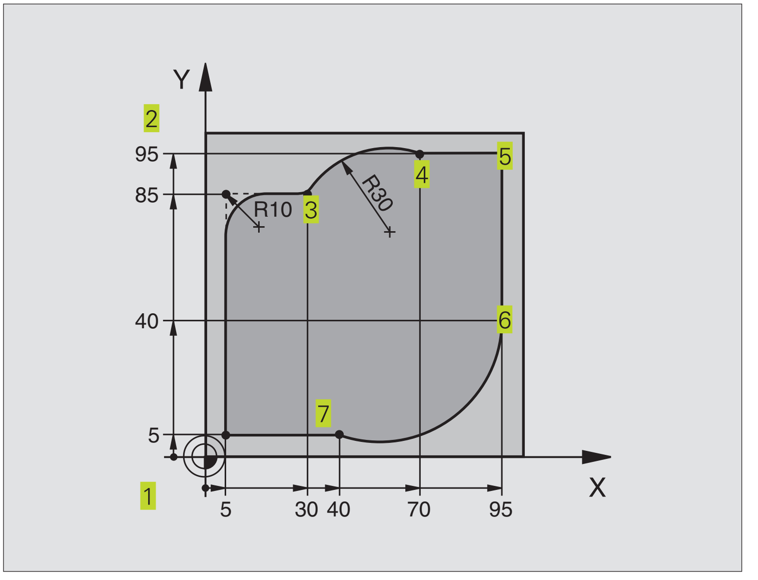
You can also use a G25 block for a tangential contour approach (see Section „6.3 Contour Approach and Departure“).

Example: Linear movements and chamfers with Cartesian coordinates



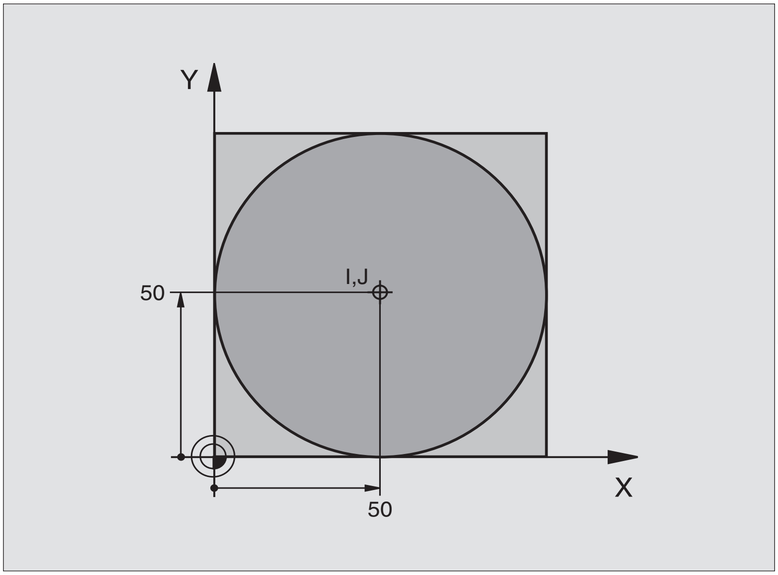
%LINEAR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define blank form for graphic workpiece simulation
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define tool in the program
N40 T1 G17 S4000 *	Call tool in the spindle axis and with the spindle speed S
N50 G00 G40 G90 Z+250 *	Retract tool in the spindle axis at rapid traverse
N60 X-10 Y-10 *	Pre-position the tool
N70 G01 Z-5 F1000 M3 *	Move to working depth at feed rate F = 1000 mm/min
N80 G01 G41 X+5 Y+5 F300 *	Approach the contour at point 1, activate radius compensation G41
N90 G26 R5 F150 *	Tangential approach
N100 Y+95 *	Move to point 2
N110 X+95 *	Point 3: first straight line for corner 3
N120 G24 R10 *	Program chamfer with length 10 mm
N130 Y+5 *	Point 4: 2nd straight line for corner 3, 1st straight line for corner 4
N140 G24 R20 *	Program chamfer with length 20 mm
N150 X+5 *	Move to last contour point 1, second straight line for corner 4
N160 G27 R5 F500 *	Tangential departure
N170 G40 X-20 Y-20 F1000 *	Retract tool in the working plane, cancel radius compensation
N180 G00 Z+250 M2 *	Retract tool in the spindle axis, end of program
N999999 %LINEAR G71 *	

Example: Circular movements with Cartesian coordinates



`%CIRCULAR G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define blank form for graphic workpiece simulation
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+10 *`	Define tool in the program
`N40 T1 G17 S4000 *`	Call tool in the spindle axis and with the spindle speed S
`N50 G00 G40 G90 Z+250 *`	Retract tool in the spindle axis at rapid traverse
`N60 X-10 Y-10 *`	Pre-position the tool
`N70 G01 Z-5 F1000 M3 *`	Move to working depth at feed rate F = 1000 mm/min
`N80 G01 G41 X+5 Y+5 F300 *`	Approach the contour at point 1, activate radius compensation G41
`N90 G26 R5 F150 *`	Tangential approach
`N100 Y+85 *`	Point 2: first straight line for corner 2
`N110 G25 R10 *`	Insert radius with R = 10 mm
`N120 X+30 *`	Move to point 3: starting point of the arc with G02
`N130 G02 X+70 Y+95 R+30 *`	Move to point 4: end point of the arc with G02, radius 30 mm
`N140 G01 X+95 *`	Move to point 5
`N150 Y+40 *`	Move to point 6
`N160 G06 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
`N170 G01 X+5 *`	Move to last contour point 1
`N180 G27 R5 F500 *`	Tangential departure
`N190 G40 X-20 Y-20 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N200 G00 Z+250 M2 *`	Retract tool in the spindle axis, end of program
`N999999 %CIRCULAR G71 *`	

Example: Full circle with Cartesian coordinates



`%C-CC G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+12,5 *`	Define the tool
`N40 T1 G17 S3150 *`	Call the tool
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 I+50 J+50 *`	Define the circle center
`N70 X-40 Y+50 *`	Pre-position the tool
`N80 G01 Z-5 F1000 M3 *`	Move to working depth
`N90 G01 G41 X+0 Y+50 F300 *`	Approach starting point, radius compensation G41
`N100 G26 R5 F150 *`	Tangential approach
`N110 G02 X+0 *`	Move to the circle end point (= circle starting point)
`N120 G27 R5 F500 *`	Tangential departure
`N130 G01 G40 X-40 Y-50 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N140 G00 Z+250 M2 *`	Retract tool in the spindle axis, end of program
`N999999 %C-CC G71 *`	

6.5 Path Contours—Polar Coordinates

With polar coordinate you can define a position in terms of its angle H and its distance R relative to a previously defined pole I, J. 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

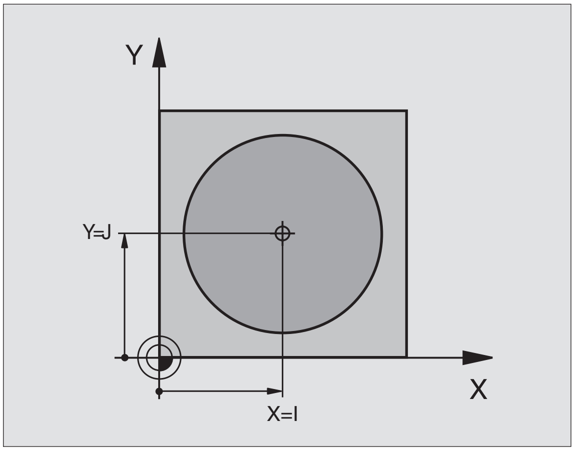
Tool movement	Function	Required input
Straight line at rapid traverse end point Straight line at feed rate F	G10 G11	Polar radius, polar angle of the straight-line
Circular path in clockwise direction Circular path in counterclockwise direction Circular path corresponding to active direction	G12 G13 G15	Polar angle of the circle end point
Circular path with tangential connection to previous contour element	G16	Polar radius, polar angle of the circle end point

Zero point for polar coordinates: pole I, J

You can set the pole I, J at any point in the machining program, before indicating points in polar coordinates. Set the pole in the same way as you would program the circle center.

I J

▶ Enter Cartesian coordinates for the pole, or if you want to use the last programmed position, enter G29.



Straight line at rapid traverse G10
Straight line with feed rate G11 F . . .

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

G 11

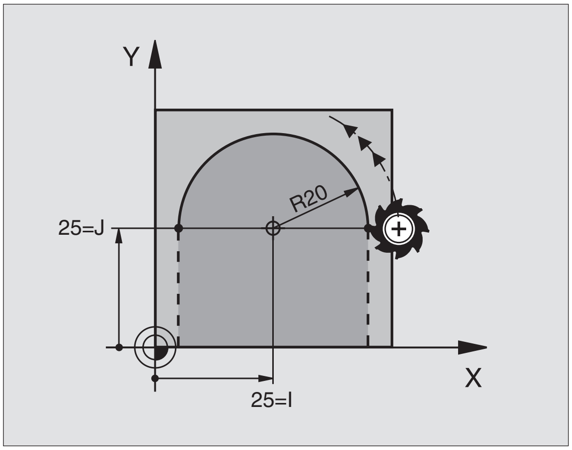
- Polar coordinates radius R: Enter distance from the straight line end point to the pole I, J
- Polar-coordinates angle H: Angular position of the straight-line end point between –360° and +360°

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

Example NC blocks

```
N120 I+45 J+25 *
N130 G11 G42 R+30 H+0 F300 M3 *
N140 H+60 *
N150 G91 H+60 *
N160 G90 H+180 *
```

Circular path G12/G13/G15 around pole I, J



The polar coordinate radius R is also the radius of the arc. It is defined by the distance from the starting point to the pole I, J. The last programmed tool position before the G12, G13 or G15 block is the starting point of the arc.

Direction

- In clockwise direction: G12
- In counterclockwise direction: G13
- Without direction indication: G15
  The TNC traverses the circular arc with the last programmed direction of rotation.

G 13

- Polar coordinates angle H: Angular position of thearc end point between –5400° and +5400°

Example NC blocks

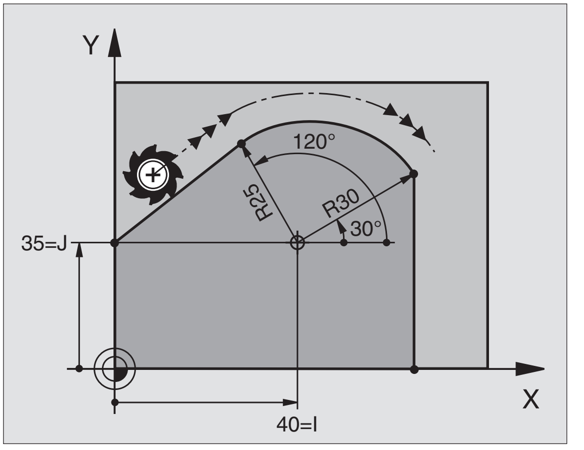
```
N180 I+25 J+25 *
N190 G11 G42 R+20 H+0 F250 M3 *
N200 G13 H+180 *
```

Circular path G16 with tangential approach

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

G 16

- ▶ Polar coordinates radius R: Distance from the arc end point to the pole I, J
- ▶ Polar coordinates angle H: Angular position of the arc end point.

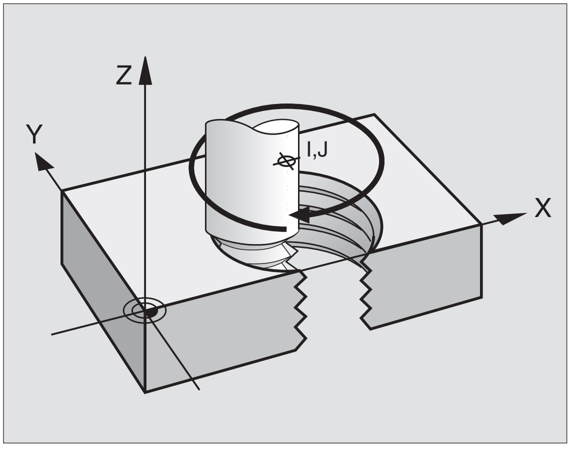


Example NC blocks

```
N120 I+40 J+35 *
N130 G01 G41 X+0 Y+35 F250 M3 *
N140 G11 R+25 H+120 *
N150 G16 R+30 H+30 *
N160 G01 Y+0 *
```

The pole I, J is **not** the center of the contour arc!

Helical interpolation



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

A helix is programmed only in polar coordinates.

Application

- Large-diameter internal and external threads
- Lubrication grooves

Calculating the helix

To program a helix, you must enter the total angle through which the tool is to move on the helix in incremental dimensions, and the total height of the helix.

For calculating a helix that is to be cut in 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

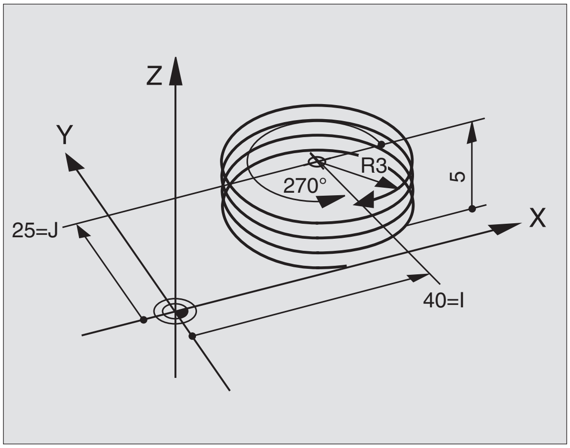
The table below illustrates in which way the shape of the helix is determined by the work direction, direction of rotation and radius compensation.

Internal thread	Work direct.	Direct. of rot.	Radius comp.
Right-handed	Z+	G13	G41
Left-handed	Z+	G12	G42
Right-handed	Z–	G12	G42
Left-handed	Z–	G13	G41
External thread			
Right-handed	Z+	G13	G42
Left-handed	Z+	G12	G41
Right-handed	Z–	G12	G41
Left-handed	Z–	G13	G42

Programming a helix

Always enter the same algebraic sign for the direction of rotation and the incremental total angle G91 H. The tool may otherwise move in a wrong path and damage the contour.

For the total angle G91 H, 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").



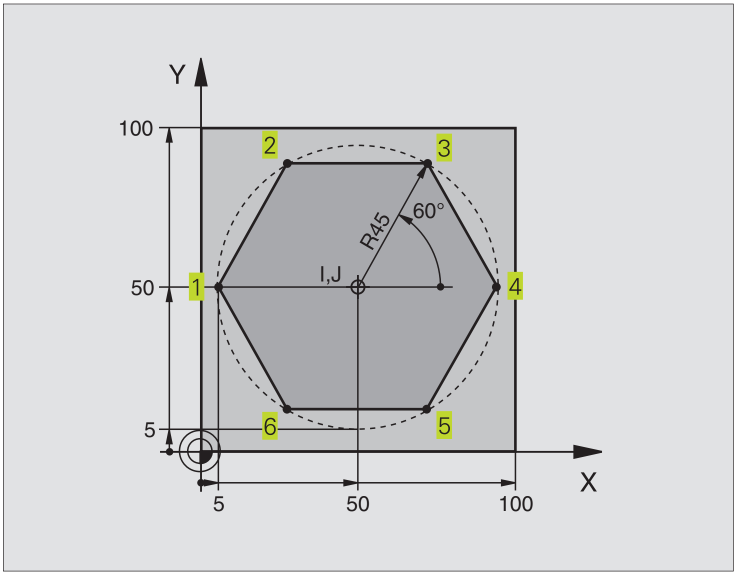
G 12

- Polar coordinates angle H: Enter the total angle (G91) of tool traverse along the helix in incremental dimensions. **After entering the angle, identify the tool axis with an axis selection key.**
- Enter the coordinate for the height of the helix in incremental dimensions.
- Enter radius compensation G40/G41/G42

Example NC blocks

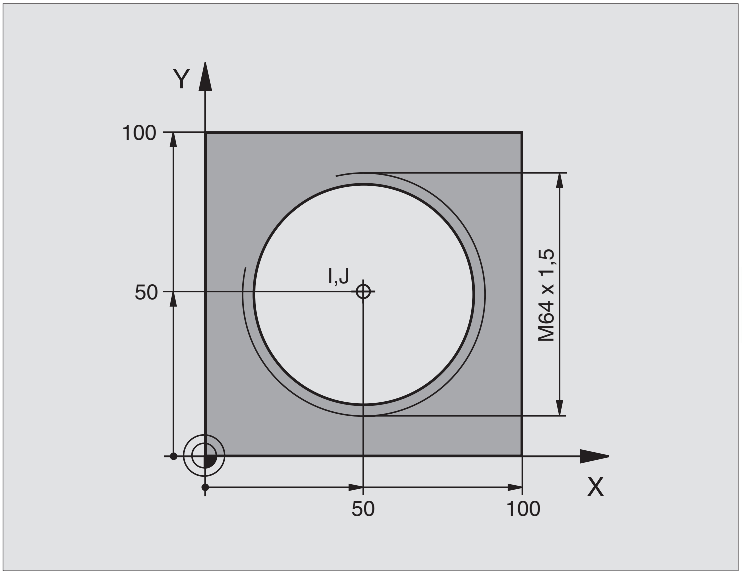
```
N120 I+40 J+25 *
N130 G01 Z+0 F100 M3 *
N140 G11 G41 R+3 H+270 *
N150 G12 G91 H-1800 Z+5 F+50 *
```

Example: Linear movement with polar coordinates



`%LINEARPO G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+7,5 *`	Define the tool
`N40 T1 G17 S4000 *`	Call the tool
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 I+50 J+50 *`	Define the datum for polar coordinates
`N70 G10 R+60 H+180 *`	Pre-position the tool
`N80 G01 Z-5 F1000 M3 *`	Move to working depth
`N90 G11 G41 R+45 H+180 F250 *`	Approach the contour at point 1
`N110 G26 R5 *`	Tangential approach
`N120 H+120 *`	Move to point 2
`N130 H+60 *`	Move to point 3
`N140 H+0 *`	Move to point 4
`N150 H-60 *`	Move to point 5
`N160 H-120 *`	Move to point 6
`N170 H+180 *`	Move to point 1
`N180 G27 R5 F500 *`	Tangential departure
`N190 G40 R+60 H+180 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N200 G00 Z+250 M2 *`	Retract tool in the spindle axis, end of program
`N999999 %LINEARPO G71 *`	

Example: Helix



`%HELIX G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+5 *`	Define the tool
`N40 T1 G17 S1400 *`	Call the tool
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 X+50 Y+50 *`	Pre-position the tool
`N70 G29 *`	Transfer the last programmed position as the pole
`N80 G01 Z-12,75 F1000 M3 *`	Move to working depth
`N90 G11 G41 R+32 H+180 F250 *`	Approach first contour point
`N100 G26 R2 *`	Tangential approach
`N110 G13 G91 H+3240 Z+13.5 F200 *`	Helical interpolation
`N120 G27 R2 F500 *`	Tangential departure
`N170 G01 G40 G90 X+50 Y+50 F1000 *`	Retract tool in the working plane, cancel radius compensation
`N180 G00 Z+250 M2 *`	Retract tool in the spindle axis, end of program

To cut a thread with more than 16 revolutions

`...N80 G01 Z-12,75 F1000 M3 *`	
`N90 G11 G41 H+180 R+32 F250 *`	
`N100 G26 R2 *`	
`N110 G98 L1 *`	Identify beginning of program section repeat
`N120 G12 G91 H+360 Z+1,5 F200 *`	Enter pitch directly as incremental Z value
`N130 L1,24 *`	Program the number of repeats (thread revolutions)
`N999999 %HELIX G71 *`	

7

Programming:

Miscellaneous functions

7.1 Entering Miscellaneous Functions M

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

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

> The machine tool builder may add some M functions that are not described in this User's Manual. Your machine manual provides more detailed information.

You can enter a miscellaneous function M in a positioning block or as a separate block.

You usually enter only the number of the M function. With certain miscellaneous functions, the TNC asks you to enter parameters after you press the ENT key.

In the operating modes Manual and Electronic Handwheel, you enter the miscellaneous functions with the soft key M.

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

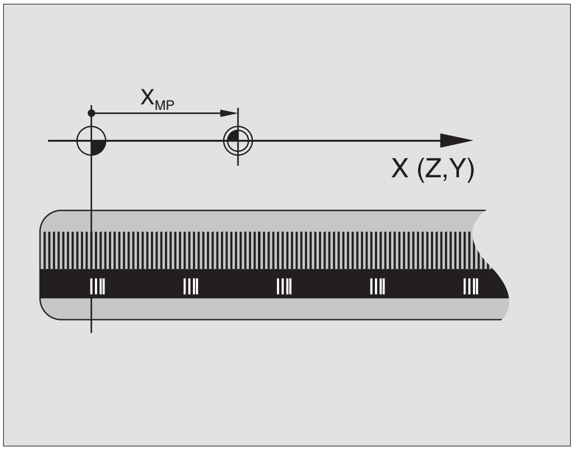
M functions come into effect in the block in which they are called. Unless the M function is only effective blockwise, it is canceled in a subsequent block or at the end of the program. Some M functions are effective only in the block in which they are called.

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.

M91 and M92 are not effective in a tilted working plane. If you program these M functions in a tilted plane, the TNC will display an error message.

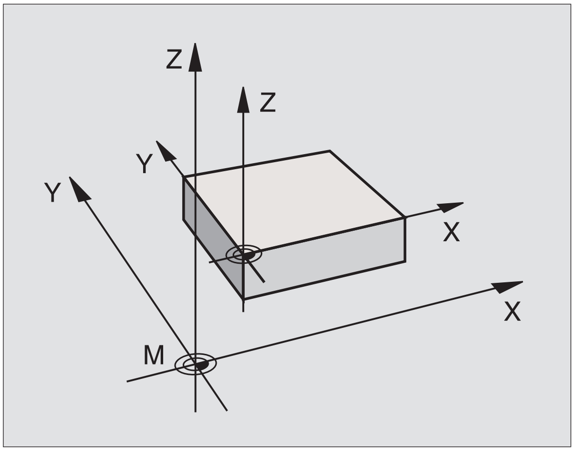
Effect
M91 and M92 are effective only in the blocks in which they are programmed with M91 or M92.

M91 and M92 become effective at the start of block.

Workpiece datum
If you want the coordinates to always be referenced to the machine datum, you can inhibit datum setting for one or more axes (see machine parameter 7295).

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

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



To activate the datum that was last set: M104 (only TNC 426, TNC 430 with NC software 280 474-xx)

When processing pallet tables, the TNC may overwrite your most recently entered datum with values from the pallet table. With M104 you can reactivate the original datum.

Effect
M104 is effective only in the blocks in which it is programmed.

M104 becomes effective at the end of block.

Moving to positions in an untilted coordinate system with a tilted working plane: M130 (not with TNC 410)

Standard behavior with a tilted working plane
The TNC places the coordinates in the positioning blocks in the tilted coordinate system.

Behavior with M130
The TNC places coordinates in **straight line blocks** in the untilted coordinate system

The TNC then positions the (tilted) tool to the programmed coordinates of the untilted system.

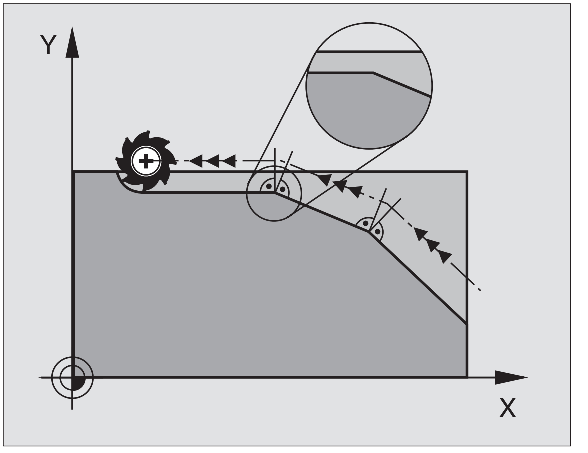
Effect
M130 functions only in straight-line blocks without tool radius compensationand in blocks in which M130 is programmed.

7.4 Miscellaneous Functions for Contouring Behavior

Smoothing corners: M90

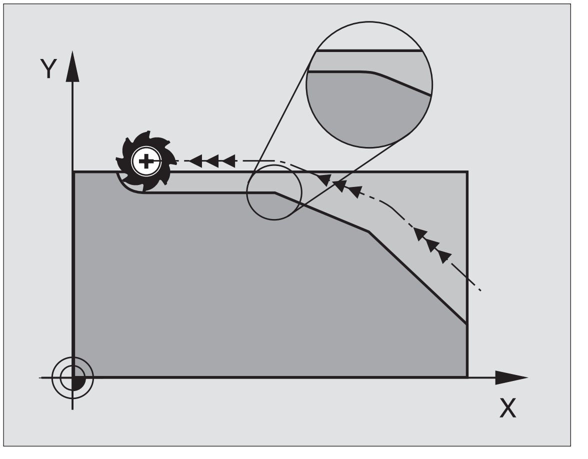
Standard behavior
The TNC stops the tool briefly in positioning blocks without tool radius compensation. This is called an accurate stop.

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.

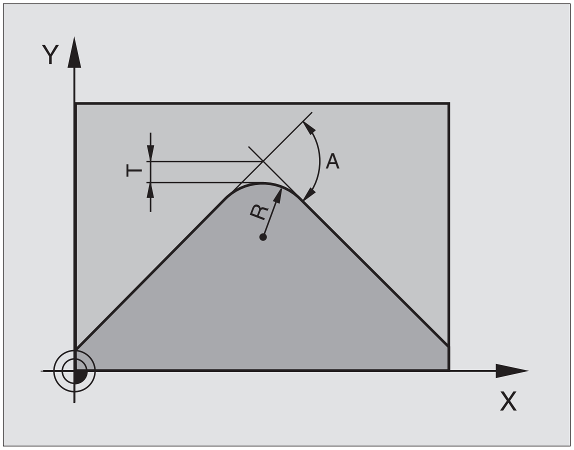


Effect
M90 is effective only in the blocks in which it is programmed with M90.

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). Not with TNC 426 or TNC 430.

Entering contour transitions between two contour elements: M112 (not TNC 426, TNC 430)



Standard behavior
The TNC stops briefly for all changes in direction that are greater than the limit angle defined in MP7460(exact stop).

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

Behavior with M112

> You can adjust the effect of M112 by redefining machine parameters.

The TNC inserts a selectable contour transition between **any contour elements (compensated and uncompensated),** in the plane or in three dimensions :

- Tangential circle: MP7415.0 = 0
  an acceleration jump results from the change in the curvature at the connection points.
- Third-degree polynomial (cubic spline): MP7415.0 = 1
  There is no velocity jump at the connection points.
- Fifth-degree polynomial: MP7415.0 = 2
  There is no acceleration jump at the connection points.
- Seventh-degree polynomial: MP7415.0 = 3 (standard setting)
  There is no jump in the rate of acceleration change.

Permissible contour deviation E
With the tolerance value T you define the distance by which the milled contour can deviate from the programmed contour. If you do not enter a tolerance value, the TNC calculates the most exact contour transition possible at the programmed feed rate.

Limit angle H
If you enter a limit angle A, the TNC smoothens only those contour transitions whose angle of directional change is greater than the programmed limit angle. If you enter a limit angle = 0, the TNC moves the tool at a constant acceleration also over tangential transitions. Input range: 0° to 90°

Entering M112 in a positioning block
If you press the soft key M112 in a positioning block (in answer to the "Miscellaneous function?" prompt), the TNC then continues the dialog by asking you for the permissible tolerance T and the limit angle A.

You can also define E and H through Q parameters. See Chapter 10 "Programming: Q Parameters."

Effect
M112 is effective during operation with velocity feedforward as well as with servo lag.

M112 becomes effective at the start of block.

To cancel M112, enter M113.

Example NC block

```
N50 G01 G40 X+123.723 Y+25.491 F800 M112 E0.01 H10 *
```

Contour filter: M124 (not TNC 426, TNC 430)

Standard behavior
The TNC includes all available points in its calculation of a contour transition between contour elements.

Behavior with M124

> You can adjust the effect of M124 by redefining machine parameters.

The TNC filters contour elements with small point spacing and inserts a transitional contour.

Shape of contour transition

- Tangential circle: MP7415.0 = 0
  an acceleration jump results from the change in the curvature at the connection points.
- Third-degree polynomial (cubic spline): MP7415.0 = 1
  There is no velocity jump at the connection points.
- Fifth-degree polynomial: MP7415.0 = 2
  There is no acceleration jump at the connection points.
- Seventh-degree polynomial: MP7415.0 = 3 (standard setting)
  There is no jump in the rate of acceleration change.

Rounding of contour transitions

- Do not round the contour transition: MP7415.1 = 0
  execute the contour transition as defined in MP7415.0 (standard contour transition: 7th-degree polynomial)
- Round the contour transition: MP7415.1 = 1
  Execute the contour transition so that the straight line segments remaining between the contour transitions are also rounded.

Minimum length E of a contour element
With parameter E you define the length up to which the TNC should filter contour elements out. If you have defined a permissible contour deviation in M112, the TNC will respect it. If you do not enter a maximum contour deviation, the TNC calculates the most exact contour transition possible without reducing the programmed feed rate.

Programming M124
If in a positioning block (with the dialog "Miscellaneous function") you press the soft key M124, the TNC then continues the dialog for this block and asks for the tolerance value E.

You can also define E through Q parameters. See Chapter 10 "Programming: Q Parameters"

Effect
M124 becomes effective at the start of block. Like M112, M124 is reset with M113.

Example NC block

```
N50 G01 G40 X+123.723 Y+25.491 F800 M124 E0.01 *
```

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 center right.

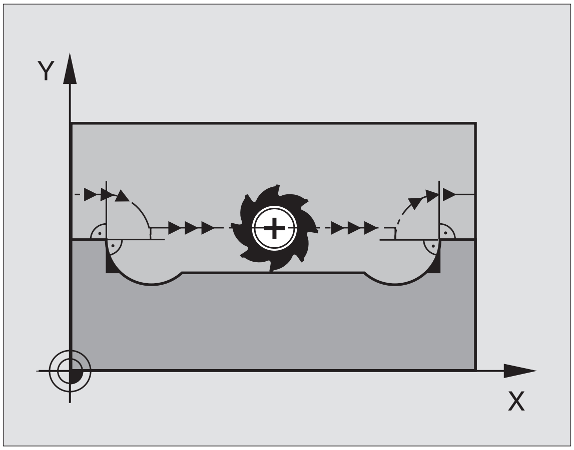
In such cases the TNC interrupts program run and generates the error message "Tool radius too large."

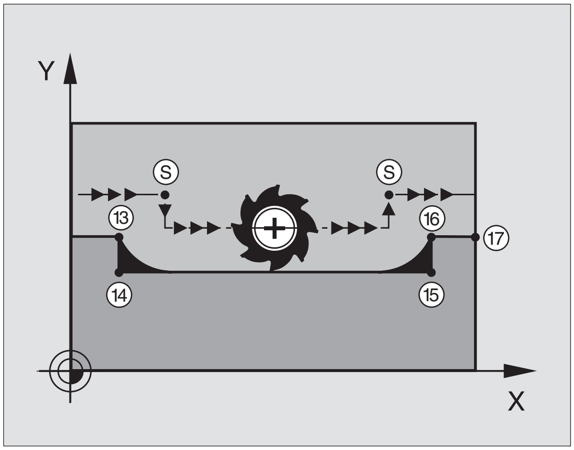
Behavior with M97
The TNC calculates the intersection of the contour elements — as at inside corners — and moves the tool over this point. See figure at lower right.

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.





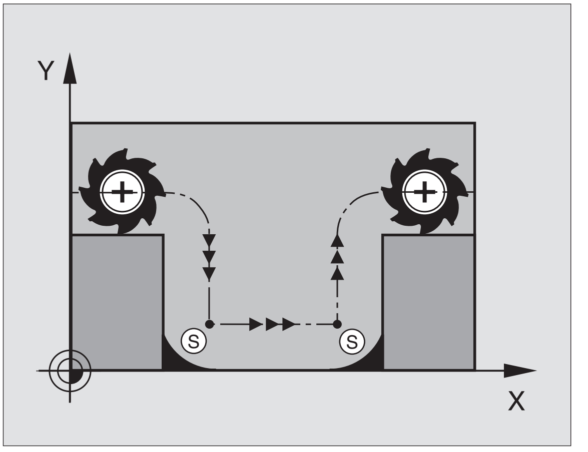
Example NC blocks

`N50 G99 G01 ... R+20 *`	Large tool radius
`...`	
`N130 X ... Y ... F .. M97 *`	Move to contour point 13
`N140 G91 Y-0.5 .... F.. *`	Machine small contour step 13 to 14
`N150 X+100 ... *`	Move to contour point 15
`N160 Y+0.5 ... F.. M97 *`	Machine small contour step 15 to 16
`N170 G90 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).



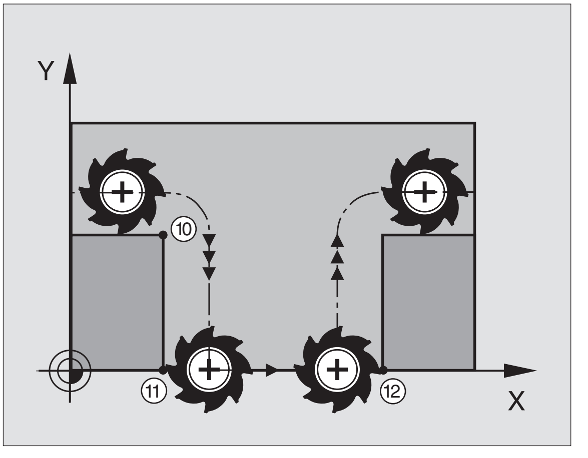
Behavior with M98
With the miscellaneous function M98, the TNC temporarily suspends radius compensation to ensure that both corners are completely machined (see figure at lower right).

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:

```
N100 G01 G41 X ... Y... F *
N110 X...  G91 Y... M98 *
N120 X+ ... *
```



Feed rate factor for plunging movements: M103

Standard behavior
The TNC moves the tool at the last programmed feed rate, regardless of the direction of traverse.

Behavior with M103
The TNC reduces the feed rate when the tool moves in the negative direction of the tool axis. The feed rate for plunging FZMAX is calculated from the last programmed feed rate FPROG and a factor F%:

FZMAX = FPROG x F%

Programming M103
If you enter M103 in a positioning block, the TNC continues the dialog by asking you the factor F.

Effect
M103 becomes effective at the start of block.
To cancel M103, program M103 once again **without a factor**.

Example NC blocks
The feed rate for plunging is to be 20% of the feed rate in the plane.

...	Actual contouring feed rate (mm/min):
N170 G01 G41 X+20 Y+20 F500 M103 F20 *	500
N180 Y+50 *	500
N190 G91 Z–2.5 *	100
N200 Y+5 Z–5	141
N210 X+50	500
N220 G90 Z+5	500

M103 is activated with machine parameter 7440; see section 14.1 "General User Parameters."

Feed rate in micrometers per spindle revolution: M136 (only TNC 426, TNC 430 with NC software 280 474-xx)

Standard behavior
The TNC moves the tool at the programmed feed rate F in mm/min.

Behavior with M136
With M136, the TNC does not move the tool in mm/min, but rather at the programmed feed rate F in microns per spindle revolution. If you change the spindle speed by using the spindle override, the TNC changes the feed rate accordingly.

Effect
M136 becomes effective at the start of block.

You can cancel M136 by programming M137.

Feed rate at circular arcs: M109/M110/M111

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

Behavior at circular arcs with M109
The TNC adjusts the feed rate for circular arcs at inside and outside contours such that the feed rate at the tool cutting edge remains constant.

Behavior at circular arcs with M110
The TNC keeps the feed rate constant for circular arcs at inside contours only. At outside contours, the feed rate is not adjusted.

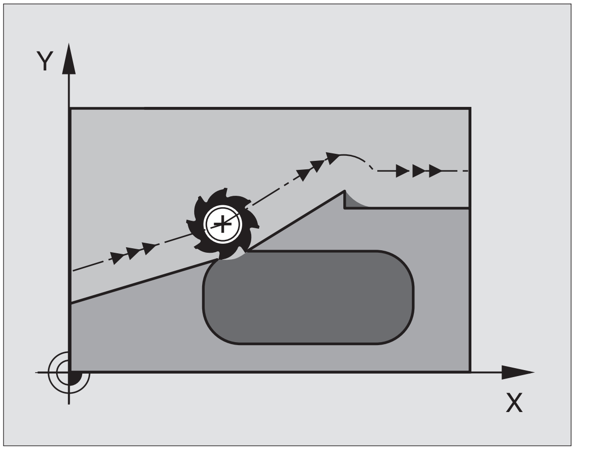
M110 is also effective for the inside machining of circular arcs using contour cycles.

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

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

Standard behavior
If the tool radius is larger than the contour step that is to be machined with radius compensation, the TNC interrupts program run and generates an error message. Although you can use M97 to inhibit the error message (see "Machining small contour steps: M97"), this will result in dwell marks and will also move the corner.

If the programmed contour contains undercut features, the tool may damage the contour. —See figure at right.



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

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

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

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

- radius compensation is canceled, or
- M120 LA0 is programmed, or
- M120 is programmed without LA.
- Call another program with %...

M120 becomes effective at the start of block.

Constraints (only for TNC 426, TNC 430)
- After an external or internal stop, you can only re-enter the contour with the function RESTORE POS. AT N.
- If you are using the path functions G25 and G24, the blocks before and after G25 or CHF must contain only coordinates of the working plane.

Superimposing handwheel positioning during program run: M118 (not TNC 410)

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

Behavior with M118
M118 permits manual corrections by handwheel during program run. You can use this miscellaneous function by entering axis-specific values X, Y and Z (in mm) behind M118.

Programming M118

If you enter M118 in a positioning block, the TNC continues the dialog for this block by asking you the axis-specific values. The coordinates are entered with the orange axis direction buttons or the ASCII keyboard.

Effect
Cancel handwheel positioning by programming M118 once again without X, Y and Z.

M118 becomes effective at the start of block.

Example NC block
You wish to be able to use the handwheel during program run to move the tool in the working plane X/Y by ±1 mm of the programmed value:

```
G01 G41 X+0 Y+38,5 F125 M118 X1 Y1
```

M118 is always effective in the original coordinate system, even if the working plane is tilted!

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

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

7.5 Miscellaneous Functions for Rotary Axes

Feed rate in mm/min on rotary axes A, B, C: M116 (not TNC 410)

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

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

Feed rate in mm/min on rotary axes with M116

> The machine geometry must be entered in machine parameters 7510 ff. by the machine tool builder.

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

Effect
M116 is effective in the working plane.
You can cancel M116 with M117; M116 is also canceled at the end of the program.

M116 becomes effective at the start of block.

Shorter-path traverse of rotary axes: M126

Standard behavior
The standard behavior of the TNC while positioning rotary axes whose display has been reduced to values less than 360° is dependent on machine parameter 7682. In machine parameter 7682 is set whether the TNC should consider the difference between nominal and actual position, or whether the TNC should always (even without M126) choose the shortest path traverse to the programmed position. The table at upper right shows examples.

Behavior with M126
With M126, the TNC will move the axis on the shorter path of traverse if you reduce display of a rotary axis to a value less than 360°. The table at lower right shows examples.

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

Standard behavior of the TNC

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

Behavior with M126

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

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

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

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

Behavior with M94
At the start of block, the TNC first reduces the current angular value to a value less than 360° and then moves the tool to the programmed value. If several rotary axes are active, M94 will reduce the display of all rotary axes. To have the TNC reduce the display for a specific rotary axis only, enter the axis after M94.

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

```
N50 M94 *
```

Additionally on the TNC 426 and TNC 430:
To reduce display of the C axis only

```
N50 M94 C *
```

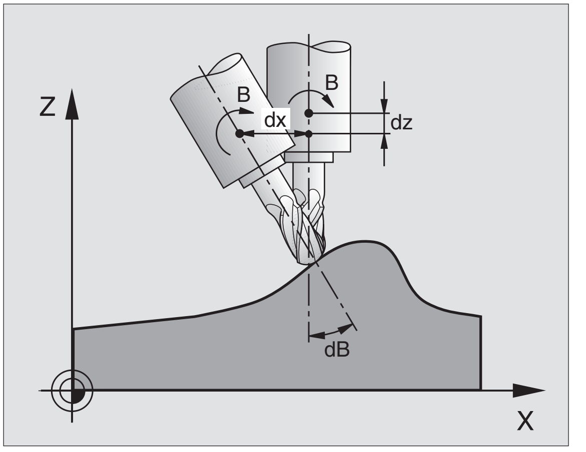
To reduce display of all active rotary axes and then move the tool in the C axis to the programmed value:

```
N50 G00 C+180 M94 *
```

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

M94 becomes effective at the start of block.

Automatic compensation of machine geometry when working with tilted axes: M114 (not TNC 410)



Standard behavior
The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated by a postprocessor (see figure at top right) and traversed in a positioning block. As the machine geometry is also relevant, the NC program must be calculated separately for each machine tool.

Behavior with M114
If the position of a controlled tilted axis changes in the program, the TNC automatically compensates the tool offset by a 3-D length compensation. As the geometry of the individual machine tools is set in machine parameters, the TNC also compensates machine-specific offsets automatically. Programs only need to be calculated by the postprocessor once, even if they are being run on different machines with TNC control.

If your machine tool does not have controlled tilted axes (head tilted manually or positioned by the PLC), you can enter the current valid swivel head position after M114 (e.g. M114 B+45, Q parameters permitted).

The radius compensation must be calculated by a CAD system or by a postprocessor. A programmed radius compensation RL/RR will result in an error message.

If the tool length compensation is calculated by the TNC, the programmed feed rate refers to the point of the tool. Otherwise it refers to the tool datum.

If you machine tool is equipped with a swivel head that can be tilted under program control, you can interrupt program run and change the position of the tilted axis, for example with the handwheel.

With the RESTORE POS. AT N function, you can then resume program run at the block at which the part program was interrupted. If M114 is active, the TNC automatically calculates the new position of the tilted axis.

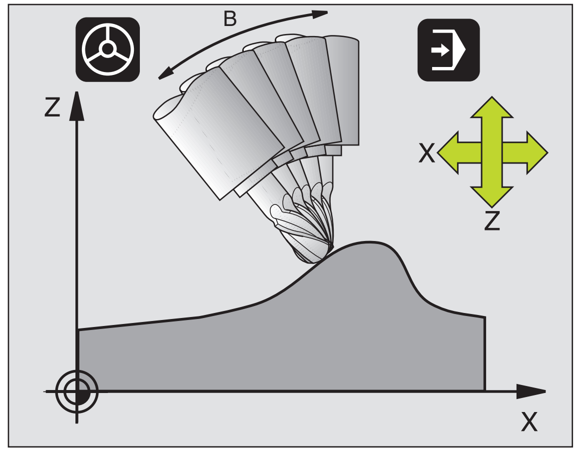
If you wish to use the handwheel to change the position of the tilted axis during program run, use M118 in conjunction with M128.

Effect
M114 becomes effective at the start of block, M115 at the end of block. M114 is not effective when tool radius compensation is active.

To cancel M114, enter M115. At the end of program, M114 is automatically canceled.

The machine geometry must be entered in machine parameters 7510 ff. by the machine tool builder.

Maintaining the position of the tool tip when positioning with tilted axes (TCPM*): M128



Standard behavior

The TNC moves the tool to the positions given in the part program. If the position of a tilted axis changes in the program, the resulting offset in the linear axes must be calculated and traversed in a positioning block (see figure on the left with M114).

Behavior with M128

If the position of a controlled tilted axis changes in the program, the position of the tool tip to the workpiece remains the same.

If you wish to use the handwheel to change the position of the tilted axis during program run, use M118 in conjunction with M128. Handwheel positioning in a machine-based coordinate is possible when M128 is active.

> For tilted axes with Hirth coupling: Do not change the position of the tilted axis after retracting the tool. Otherwise you might damage the contour.

After M128 you can program another feed rate, at which the TNC will carry out the compensation movements in the linear axes. If you program no feed rate here, or if you program a larger feed rate than is defined in machine parameters 7471, the feed rate from machine parameter 7471 will be effective.

> Reset M128 before positioning with M91 or M92 and before a T block.
>
> To avoid contour gouging you must use only spherical cutters with M128.
>
> The tool length must refer to the spherical center of the tool tip.
>
> The TNC does not adjust the active radius compensation in accordance with the new position of the tilted axis. The result is an error which is dependent on the angular position of the rotary axis.
>
> If M128 is active, the TNC shows in the status display the following symbol: an

*) **TCPM** = Tool Center Point Management

M128 on tilting tables

If you program a tilting table movement while M128 is active, the TNC rotates the coordinate system accordingly. If for example you rotate the C axis by 90° and then program a movement in the X axis, the TNC executes the movement in the machine axis Y.

The TNC also transforms the defined datum, which has been shifted by the movement of the rotary table.

Effect

M128 becomes effective at the start of block, M129 at the end of block. M128 is also effective in the manual operating modes and remains active even after a change of mode. The feed rate for the compensation movement will be effective until you program a new feed rate or until you reset M128 with M129.

To cancel M128, enter M129. The TNC also resets M128 if you select a new program in a program run operating mode.

> The machine geometry must be entered in machine parameters 7510 ff. by the machine tool builder.

Example NC block

Moving at 1000 mm/min to compensate a radius.

```
L X+0 Y+38.5 RL F125 M128 F1000
```

Exact stop at corners with nontangential transitions: M134

Standard behavior
The standard behavior of the TNC during positioning with rotary axes is to insert a transitional element in nontangential contour transitions. The contour of the transitional element depends on the acceleration, the rate of acceleration (jerk), and the defined tolerance for contour deviation.

> With the machine parameters 7440 you can change the standard behavior of the TNC so that M134 becomes active automatically whenever a program is selected (see section 14.1 "General User Parameters").

Behavior with M134
The moves the tool during positioning with rotary axes so as to perform an exact stop at nontangential contour transitions.

Effect
M134 becomes effective at the start of block, M135 at the end of block.

You can reset M134 with M135. The TNC also resets M134 if you select a new program in a program run operating mode.

Selection of tilting axes: M138 (only TNC 426, TNC 430 with NC software 280 474-xx)

Standard behavior
The TNC performs M114 and M128, and tilts the working plane, only in those axes for which the machine tool builder has set the appropriate machine parameters.

Behavior with M138
The TNC performs the above functions only in those tilting axes that you have defined using M138.

Effect
M138 becomes effective at the start of block.

You can reset M138 by reprogramming it without entering any axes.

Example NC block
Perform the above-mentioned functions only in the tilting axis C:

```
L Z+100 R0 FMAX M138 C
```

7.6 Miscellaneous Functions for Laser Cutting Machines (not TNC 410)

The TNC can control the cutting efficiency of a laser by transferring voltage values through the S-analog output. You can influence laser efficiency during program run through the miscellaneous functions M200 to M204.

Entering miscellaneous functions for laser cutting machines
If you enter an M function for laser cutting machines in a positioning block, the TNC continues the dialog by asking you the required parameters for the programmed function.

All miscellaneous functions for laser cutting machines become effective at the start of block.

Output the programmed voltage directly: M200

The TNC outputs the value programmed after M200 as the voltage V.

Input range: 0 to 9.999 V

Effect
M200 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of distance: M201

M201 outputs the voltage in dependence on the distance to be covered. The TNC increases or decreases the current voltage linearly to the value programmed for V.

Input range: 0 to 9.999 V

Effect
M201 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of speed: M202

The TNC outputs the voltage as a function of speed. In the machine parameters, the machine tool builder defines up to three characteristic curves FNR in which specific feed rates are assigned to specific voltages. Use miscellaneous function M202 to select the curve FNR from which the TNC is to determine the output voltage.

Input range: 1 to 3

Effect
M202 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (time-dependent ramp): M203

The TNC outputs the voltage V as a function of the time TIME. The TNC increases or decreases the current voltage linearly to the value programmed for V within the time programmed for TIME.

Input range
Voltage V: 0 to 9.999 Volt
TIME: 0 to 1.999 seconds

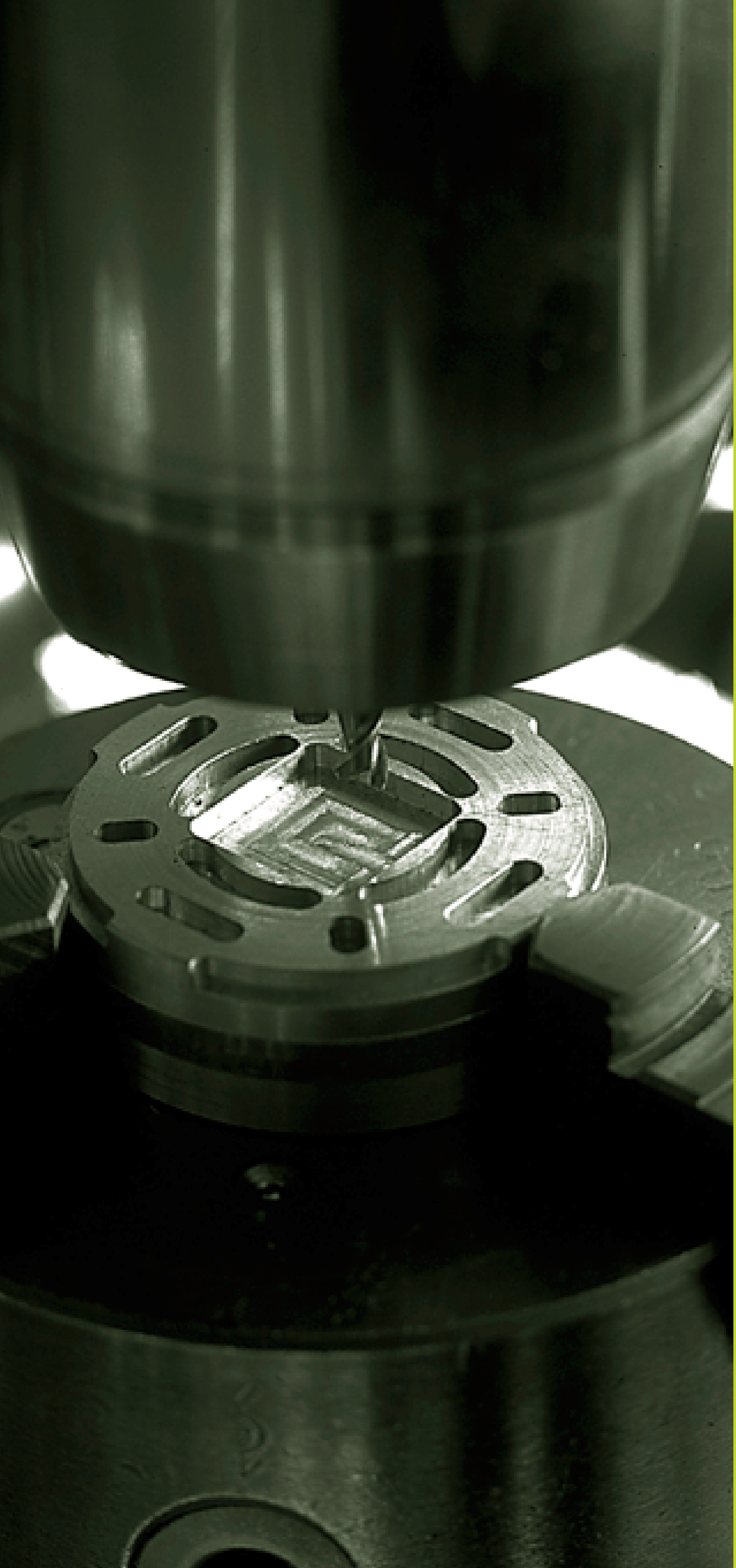
Effect
M203 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.

Output voltage as a function of time (time-dependent pulse): M204

The TNC outputs a programmed voltage as a pulse with a programmed duration TIME.

Input range
Voltage V: 0 to 9.999 Volt
TIME: 0 to 1.999 seconds

Effect
M204 remains in effect until a new voltage is output through M200, M201, M202, M203 or M204.



8

Programming:

Cycles

8.1 General Information on 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 lists the various cycle groups.

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

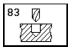
Defining a cycle

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 the desired cycle, for example G83 PECKING. 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

▶ 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

Example NC block

```
N50 G83 P01 +2 P02 -30 P03 +5 P04 1 P05 150 *
```

In order to be able to run cycles G83 to G86, G74 to G78 and G56 to G59 on older TNC models, you must program an additional negative sign before the values for safety clearance and plunging depth.

Group of Cycles	Soft key
Cycles for pecking, reaming, boring, back boring and thread cutting	DRILLING
Cycles for milling pockets, studs and slots	POCKETS/ STUDS/ SLOTS
Cycles for producing hole patterns, such as circular or linear patterns	PATTERN
SL (Subcontour List) cycles, which allow the contour-parallel machining of relatively complex contours consisting of several overlapping subcontours, cylinder surface interpolation (not TNC 410)	SL CYCLES
Cycles for face milling of flat or twisted surfaces	MULTIPASS MILLING
Coordinate transformation cycles which enable datum shift, rotation, mirror image, enlarging and reducing for various contours	COORD. TRANSF.
Special cycles such as dwell time, program call and oriented spindle stop Tolerance (not TNC 410)	SPECIAL CYCLES

Calling the Cycle

Prerequisites

The following data must always be programmed before a cycle call

- G30/G31 for graphic display (needed only for test graphics)
- Tool call
- Direction of spindle rotation (M functions M3/M4)
- Define cycle

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
- SL cycle CONTOUR GEOMETRY
- SL cycle CONTOUR DATA (not TNC 410)
- Cycle G62 TOLERANCE (not TNC 410)
- Coordinate transformation cycles
- Cycle G04 DWELL TIME

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 G79:

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
- G79 or
- a new cycle

Working with the secondary axes U/V/W

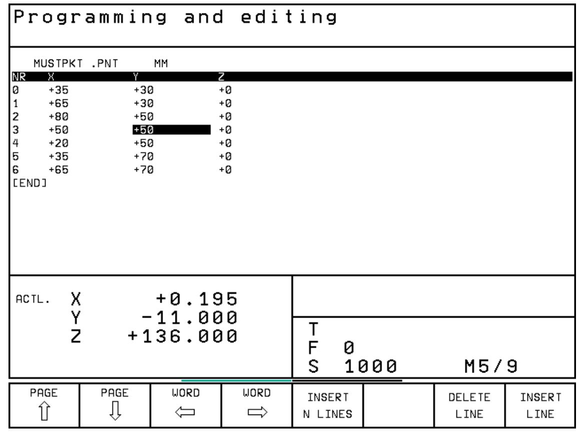
The TNC performs infeed movements in the axis that was defined in the TOOL CALL block as the spindle axis. It performs movements in the working plane only in the principle axes X, Y or Z. Exceptions:

- you program secondary axes for the side lengths in cycles G74 SLOT MILLING and G75/G76 POCKET MILLING
- You program secondary axes in the contour geometry subprogram of an SL cycle.

8.2 Point Tables (only TNC 410)

You should create a point table whenever you want to run a cycle, or several cycles in sequence, on an irregular point pattern.

If you are using drilling cycles, the coordinates of the working plane in the point table represent the hole centers. If you are using milling cycles, the coordinates of the working plane in the point table represent the starting-point coordinates of the respective cycle (e.g. center-point coordinates of a circular pocket). Coordinates in the spindle axis correspond to the coordinate of the workpiece surface.



Creating a point table

Select the Programming and Editing mode of operation.

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

File name =

NEW ENT — Enter the name of the point table and confirm your entry with the ENT key.

MM INCH — If necessary, switch to inches as unit of measure: Press the MM/INCH soft key.

.PNT — Press the .PNT soft key to select the point table file type.

Selecting point tables in the program

Select the Programming and Editing mode of operation.

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

POINT TABLE — Press the POINT TABLE soft key.

Enter the name of the point table and confirm your entry with the END key.

Example NC block:

```
N72 %:PAT: "NAMES"*
```

Calling a cycle in connection with point tables

Before programming, note the following:

With G79 PAT the TNC runs the points table that you last defined (even if you have defined the point table in a program that was nested with %.

The TNC uses the coordinate in the spindle axis as the clearance height for the cycle call.

If you want the TNC to call the last defined fixed cycle at the points defined in a point table, then program the cycle call with G79 PAT:

CYCL CALL

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

The TNC moves the tool back to the safe height over each successive starting point (safe height = the spindle axis coordinate for cycle call). To use this procedure also for the cycles number 200 and greater, you must define the 2nd set-up clearance (Q204)as 0.

If you want to move at reduced feed rate when pre-positioning in the spindle axis, use the miscellaneous function M103 (see section "7.4 Miscellaneous Functions for Contouring Behavior").

Effect of the point tables with Cycles G83, G84 and G74 to G78
The TNC interprets the points of the working plane as coordinates of the hole centers. The coordinate of the spindle axis defines the upper surface of the workpiece, so the TNC can pre-position automatically (first in the working plane, then in the spindle axis).

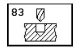
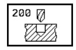
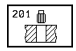
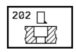
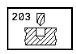
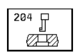
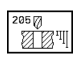
Effect of the point tables with SL cycles and Cycle G39
The TNC interprets the points as an additional datum shift.

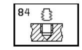
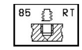
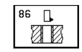
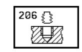
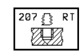
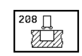
Effect of the point tables with Cycles G200 to G204
The TNC interprets the points of the working plane as coordinates of the hole centers. If you want to use the coordinate defined in the point table for the spindle axis as the starting point coordinate, you must define the workpiece surface coordinate (Q203) as 0 (see the example in section "8.3 Drilling Cycles").

Effect of the point tables with Cycles G210 to G215
The TNC interprets the points as an additional datum shift. If you want to use the points defined in the point table as starting-point coordinates, you must define the starting points and the workpiece surface coordinate (Q203) in the respective milling cycle as 0 (see the example in section "8.4 Cycles for Milling Pockets, Studs and Slots").

8.3 Drilling Cycles

The TNC offers 9 (or 13) cycles for all types of drilling operations:

Cycle	Soft key
G83 PECKING Without automatic pre-positioning	
G200 DRILLING With automatic pre-positioning, 2nd set-up clearance	
G201 REAMING With automatic pre-positioning, 2nd set-up clearance	
G202 BORING With automatic pre-positioning, 2nd set-up clearance	
G203 UNIVERSAL DRILLING With automatic pre-positioning, 2nd setup clearance, chip breaking, and decrement	
G204 BACK BORING With automatic pre-positioning, 2nd set-up clearance	
G205 UNIVERSAL PECKING **(only with TNC 426 and TNC 430 with NC software 280 474-xx)** With automatic pre-positioning, 2nd setup clearance, chip breaking, advanced stop distance	

Cycle	Soft key
G84 TAPPING With a floating tap holder	
G85 RIGID TAPPING Without a floating tap holder	
G86 THREAD CUTTING (not in TNC 410)	
G206 TAPPING NEW **(only with TNC 426, TNC 430 with NC software 280 474-xx)** with floating tap holder automatic pre-positioning, 2nd setup clearance	
G207 RIGID TAPPING NEW **(only with the TNC 426, TNC 430 with NC software 280 474-xx)** Without floating tap holder, with automatic pre-positioning, 2nd setup clearance	
G208 BORE MILLING **(only with the TNC 426, TNC 430 with NC software 280 474-xx)** With automatic pre-positioning, 2nd set-up clearance	

PECKING (Cycle G83)

1 The tool drills from the current position to the first plunging depth at the programmed feed rate F.

2 When it reaches the first plunging depth, the tool retracts in rapid traverse to the starting position and advances again to the first plunging depth minus the advanced stop distance t.

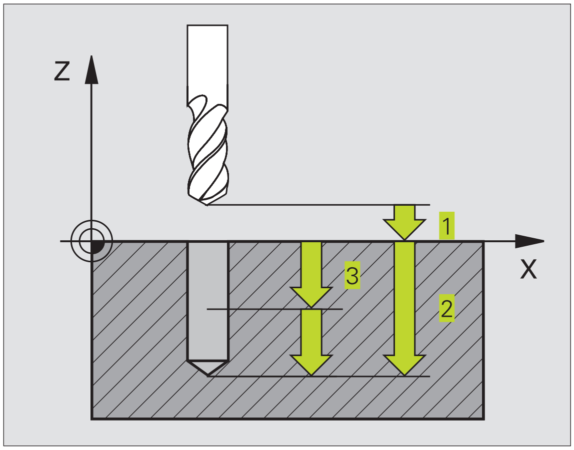
3 The advanced stop distance is automatically calculated by the control:
- At a total hole depth of up to 30 mm: t = 0.6 mm
- At a total hole depth exceeding 30 mm: t = hole depth / 50

Maximum advanced stop distance: 7 mm

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

5 The TNC repeats this process (1 to 4) until the programmed 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 for chip breaking.



Example NC block:

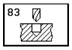
```
N10 G83 P01  2 P02 -20  5 P03  0 P04 500*
```

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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.



- Setup 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)
- Plunging depth 3 (incremental value): Infeed per cut. The tool will drill to the total hole depth in one movement if:
  - The plunging depth is equal to the total hole depth
  - The plunging depth is greater than the total hole depth

  The total hole depth does not have to be a multiple of the plunging 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 G200)

1 The TNC positions the tool in the tool axis at rapid traverse to the set-up clearance above the workpiece surface.

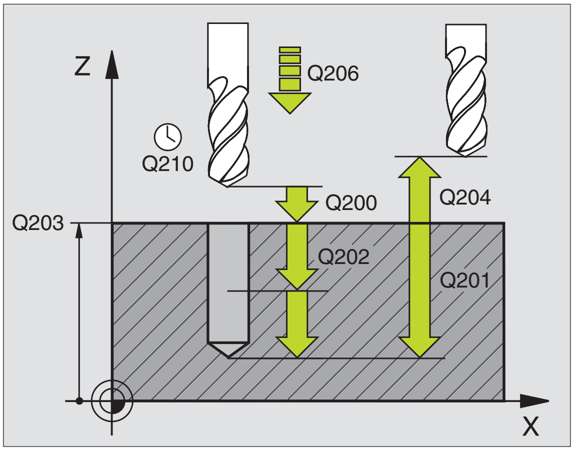
2 The tool drills to the first plunging depth at the programmed feed rate F.

3 The TNC returns the tool at rapid traverse to the setup clearance, dwells there (if a dwell time was entered), and then moves at rapid traverse to the setup clearance above the first plunging depth.

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

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

6 At the hole bottom, the tool path is retraced to set-up clearance or, if programmed, to the 2nd set-up clearance in rapid traverse.



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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

Example NC block:

```
N70  G200 Q200=2 Q201=-20 Q206=150
          Q202=5 Q210=0 Q203=+0 Q204=50*
```

- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min
- Plunging depth Q202 (incremental value): Infeed per cut The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

  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 value): Coordinate of the workpiece surface

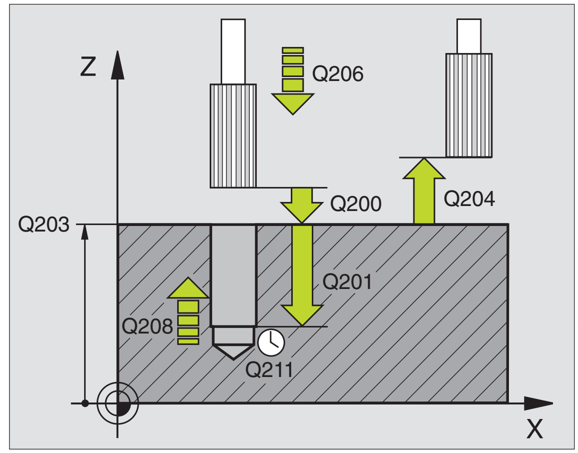
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

The TNC 426, TNC 430 with NC software 280 474-xx also provides:

- Dwell time at depth Q211: time in seconds that the tool remains at the hole bottom

REAMING (Cycle G201)

1. The TNC positions the tool in the tool axis at rapid traverse to the input 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 rapid traverse.



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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

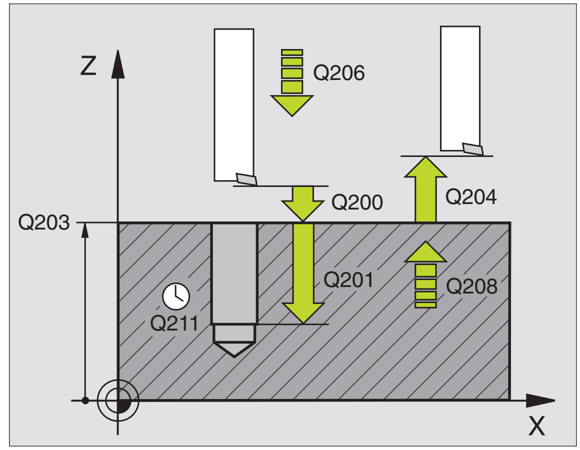
- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole
- Feed rate for plunging Q206: Traversing speed of the tool during reaming in mm/min
- Dwell time at depth Q211: Time in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

Example NC block:

```
N80  G201 Q200=2 Q201=-20 Q206=150
          Q211=0.25 Q208=500 Q203=+0
          Q204=50*
```

BORING (Cycle G202)

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 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 TNC moves the tool at the retraction feed rate to the set-up clearance and then, if entered, to the 2nd set-up clearance at rapid traverse. If Q214=0 the tool point remains on the wall of the hole.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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

After the cycle is completed, the TNC restores the coolant and spindle conditions that were active before the cycle call.

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

Example NC block:

```
N90  G202 Q200=2 Q201=-20 Q206=150
          Q211=0.5 Q208=500 Q203=+0 Q204=50
          Q214=1*
```

- 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.

The TNC 426, TNC 430 with NC software 280 474-xx also provides:

- Angle for spindle orientation Q336 (absolute): angle at which the TNC positions the tool before retracting it.

UNIVERSAL DRILLING (Cycle G203)

1 The TNC positions the tool in the tool axis at rapid traverse to the input set-up clearance above the workpiece surface.

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

3 If you have programmed chip breaking, the tool then retracts by the setup clearance. If you are working without chip breaking, the tool retracts at the RETRACTION FEED RATE to setup clearance, remains there — if programmed — for the entered dwell time, and advances again in rapid traverse to the setup clearance above the first plunging depth.

4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.

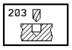
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

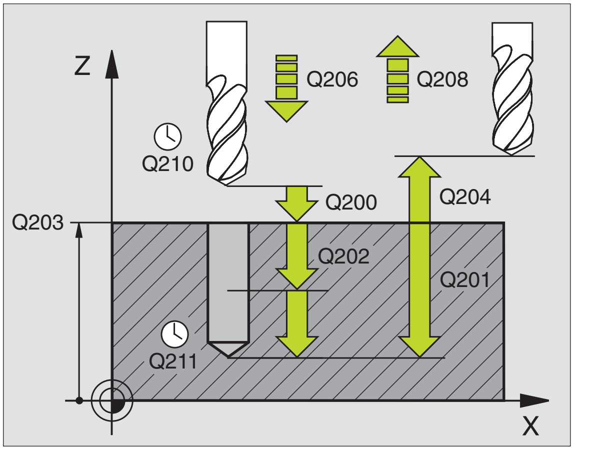
6 The tool remains at the hole bottom — if programmed — for the entered DWELL TIME to cut free, and then retracts to set-up clearance at the retraction feed rate. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

The algebraic sign for the cycle parameter TOTAL HOLE 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 TNC will go to depth in one movement if:
  - The plunging depth is equal to the depth
  - The plunging depth is greater than the depth

  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 value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Decrement Q212 (incremental value): Value by which the TNC decreases the plunging depth 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 rapid traverse.

The TNC 426, TNC 430 with NC software 280 474-xx also provides:

- Retraction rate for chip breaking Q256 (incremental): value by which the TNC retracts the tool during chip breaking

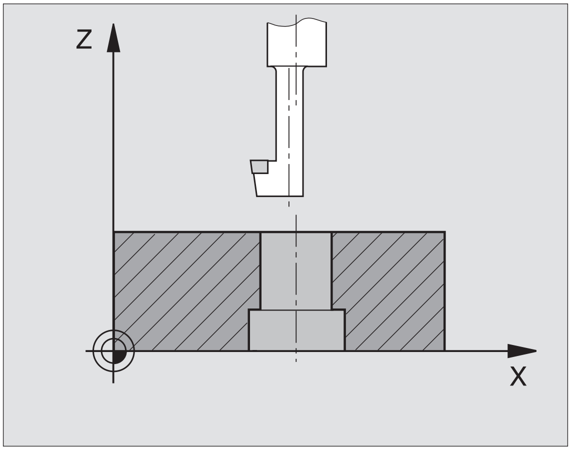
Example NC block:

```
N10  G203 Q200=2 Q201=-20 Q206=150
          Q202=5 Q210=0 Q203=+0 Q204=50
          Q212=0.2 Q213=3 Q205=3 Q211=0.25
          Q208=500*
```

BACK BORING (Cycle G204)

Machine and TNC must be specially prepared by the machine tool builder to perform back boring.

Special boring bars for upward cutting are required for this cycle.



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

1 The TNC positions the tool in the tool axis at rapid traverse to set-up clearance above the workpiece surface.

2 The TNC then orients the spindle to the 0° position with an oriented spindle stop, and displaces the tool by the off-center distance.

3 The tool is then plunged into the already bored hole at the feed rate for pre-positioning until the tooth has reached set-up clearance on the underside of the workpiece.

4 The TNC then centers the tool again over the bore hole, switches on the spindle and the coolant and moves at the feed rate for boring to the depth of bore.

5 If a dwell time is entered, the tool will pause at the top of the bore hole and will then be retracted from the hole again. The TNC carries out another oriented spindle stop and the tool is once again displaced by the off-center distance.

6 The tool then retracts to set-up clearance at the pre-positioning feed rate, and from there — if programmed — to the 2nd set-up clearance in rapid traverse.

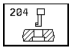
Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

The algebraic sign for the cycle parameter depth determines the working direction. Note: A positive sign bores in the direction of the positive spindle axis.

The entered tool length is the total length to the underside of the boring bar and not just to the tooth.

When calculating the starting point for boring, the TNC considers the tooth length of the boring bar and the thickness of the material.



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

1: Displace tool in the negative main axis direction

2: Displace tool in the negative secondary axis direction

3: Displace tool in the positive main axis direction

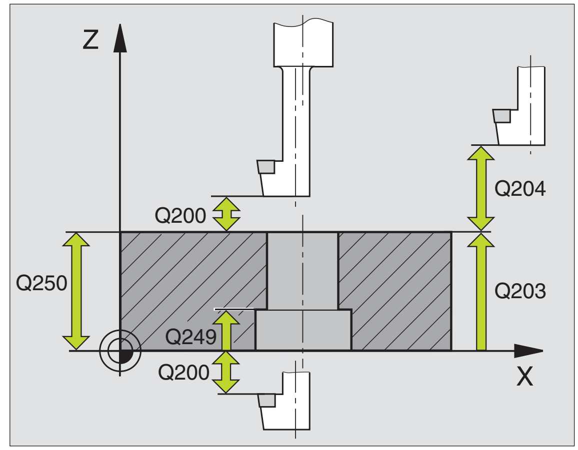
4: Displace tool in the positive secondary axis direction

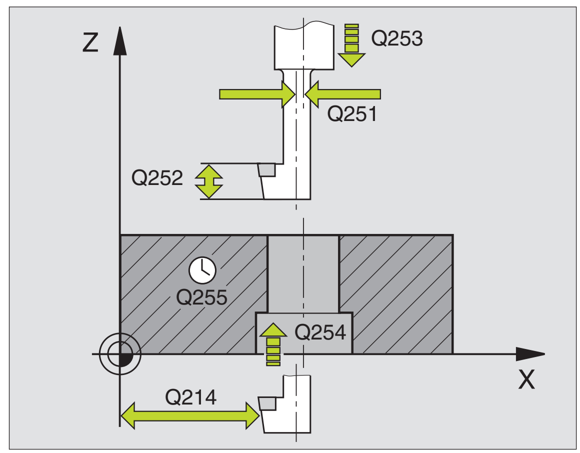
Danger of collision!

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 can plunge into the hole without danger of collision.

The TNC 426, TNC 430 with NC software 280 474-xx also provides:

- Angle for spindle orientation Q336 (absolute): angle at which the TNC positions the tool before retracting it.





Example NC block:

```
N11  G204 Q200=2 Q249=+5 Q250=20
          Q251=3.5 Q252=15 Q253=750 Q254=200
          Q255=0 Q203=+0 Q204=50 Q214=1*
```

UNIVERSAL PECKING (Cycle G205, only with the TNC 426, TNC 430 with NC software 280 474-xx)

1 The TNC positions the tool in the tool axis at rapid traverse to the input set-up clearance above the workpiece surface.

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

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

4 The tool then advances with another infeed at the programmed feed rate. If programmed, the plunging depth is decreased after each infeed by the decrement.

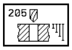
5 The TNC repeats this process (2 to 4) until the programmed total hole depth is reached.

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 rapid traverse.

Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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

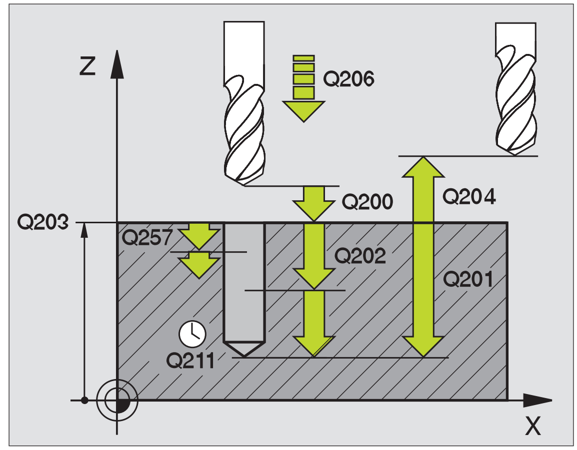


- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole (tip of drill taper)
- Feed rate for plunging Q206: Traversing speed of the tool during drilling in mm/min
- Plunging depth Q202 (incremental value): Infeed per cut The TNC will go to depth in one movement if:
  - the plunging depth is equal to the depth
  - the plunging depth is greater than the depth

  The depth does not have to be a multiple of the plunging depth.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Decrement Q212 (incremental value): Value by which the TNC decreases the plunging depth Q201.
- Minimum plunging depth Q205 (incremental value): If you have entered a decrement, the TNC limits the plunging depth to the value entered with Q205.
- Upper advanced stop distance Q258 (incremental): setup clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the first plunging depth
- Lower advanced stop distance Q259 (incremental): setup clearance for rapid traverse positioning when the TNC moves the tool again to the current plunging depth after retraction from the hole; value for the last plunging depth

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

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



Example NC block:

```
N12  G205 Q200=2 Q201=-80 Q206=150
          Q202=15 Q203=+0 Q204=50 Q212=0.5
          Q205=3 Q258=0.5 Q259=1 Q257=5
          Q256=0.2 Q211=0.25*
```

BORE MILLING (Cycle G208, only with the TNC 426, TNC 430 with NC software 280 474-xx)

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

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

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

4 The TNC then positions the tool at the center of the hole again

5 Finally the TNC returns to the setup clearance at rapid traverse. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in rapid traverse.

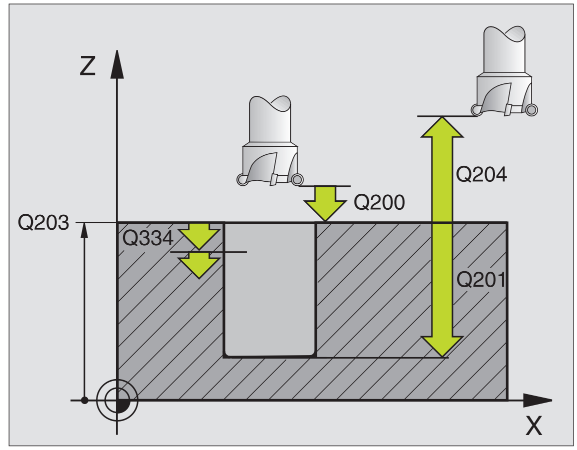
Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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

If you have entered the bore hole diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.

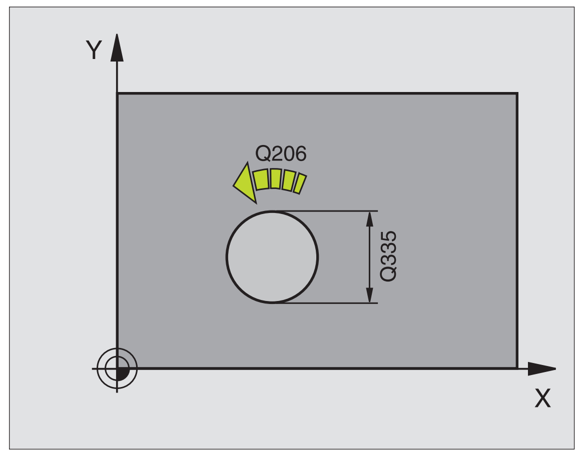
- Set-up clearance Q200 (incremental value): Distance between tool lower edge and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of hole
- Feed rate for plunging Q206: Traversing speed of the tool during helical drilling in mm/min
- Infeed per helix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



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

To prevent the infeed within safe limits, enter the max. plunge angle of the tool in the tool table, column ANGLE (see section 5.2 "Tool Data"). The TNC then calculates automatically the max. infeed permitted and changes your entered value accordingly.

- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Nominal diameter Q335 (absolute): Bore-hole diameter. If you have entered the nominal diameter to be the same as the tool diameter, the TNC will bore directly to the entered depth without any helical interpolation.



Example NC block:

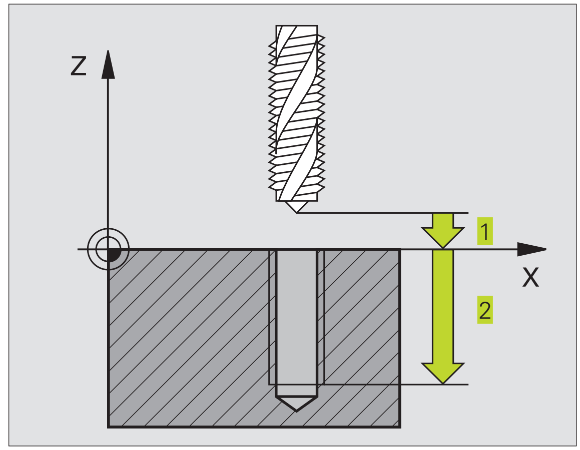
```
N12  G208 Q200=2 Q201=-80 Q206=150
     Q334=1.5 Q203=+0 Q204=50 Q335=25*
```

TAPPING with a floating tap holder (Cycle G84)

1 The tool drills to the total hole depth in one movement

2 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the starting position at the end of the DWELL TIME.

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



Before programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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.

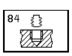
A floating tap holder is required for tapping. It must compensate the tolerances between feed rate and spindle speed during the tapping process.

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.

Example NC block:

```
N13 G84 P01 2 P02 -20 P03 0 P04 100*
```



- Setup 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

Retract tool if program is interrupted (not TNC 410)
If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.

TAPPING NEW with floating tap holder (Cycle G206, only with TNC 426, TNC 430 with NC software 280 474-xx)

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.

2 The tool drills to the total hole depth in one movement

3 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the DWELL TIME. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in FMAX.

4 At the set-up clearance, the direction of spindle rotation reverses once again.

Before programming, note the following:

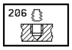
Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

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).

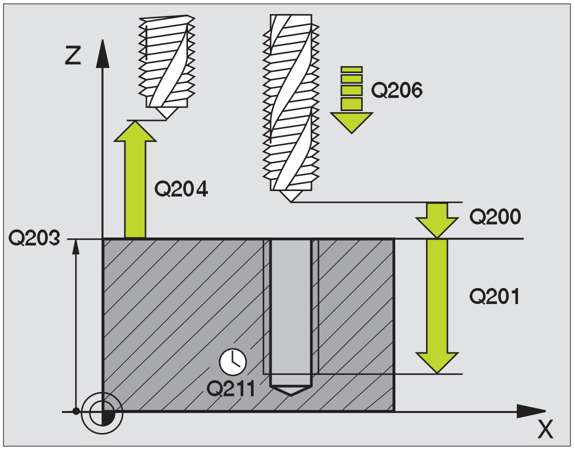
For tapping right-hand threads activate the spindle with M3, for left-hand threads use M4.



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

Retracting after a program interruption
If you interrupt program run during tapping with the machine stop button, the TNC will display a soft key with which you can retract the tool.



Example NC block:

```
N25  G206 Q200=2 Q201=-20 Q206=150
       Q211=0.25 Q203=+0 Q204=50*
```

RIGID TAPPING (Cycle G85)

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 programming, note the following:

Program a positioning block for the starting point (hole center) in the working plane with RADIUS COMPENSATION G40.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

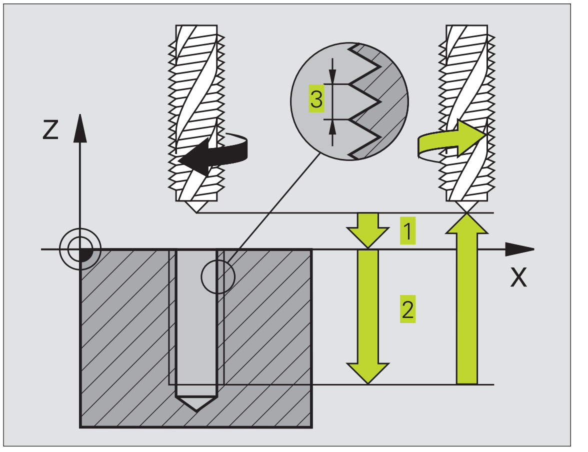
The algebraic sign for the parameter total hole depth determines the working direction.

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.

- ▶ Setup 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

Retract tool if program is interrupted (not TNC 410)
If you interrupt program run during tapping with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.



Example NC blocks:

```
N18 G85 P01 2 P02 -20 P03 +1*
```

RIGID TAPPING NEW (Cycle G207, only with the TNC 426, TNC 430 with NC software 280 474-xx)

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.

The advantages of rigid tapping over tapping with a floating tap holder are described under Cycle 85.

1 The TNC positions the tool in the tool axis at rapid traverse FMAX to the programmed set-up clearance above the workpiece surface.

2 The tool drills to the total hole depth in one movement

3 Once the tool has reached the total hole depth, the direction of spindle rotation is reversed and the tool is retracted to the set-up clearance at the end of the DWELL TIME. If you have entered a 2nd set-up clearance, the tool subsequently moves to that position in FMAX.

4 The TNC stops the spindle turning at set-up clearance

Before programming, note the following:

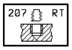
Program a positioning block for the starting point (hole center) in the working plane with radius compensation G40.

The algebraic sign for the parameter total hole depth determines the working direction.

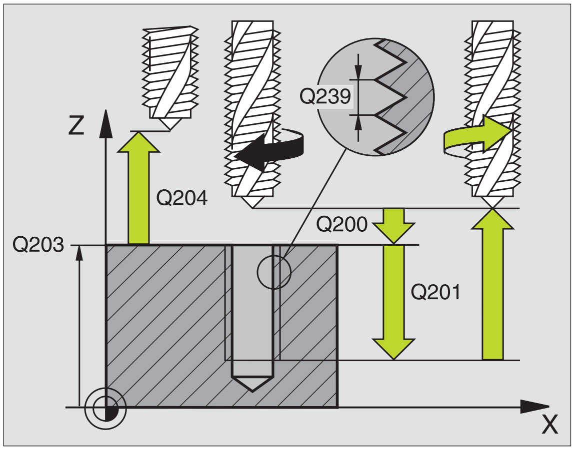
The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during tapping, the feed rate is automatically adjusted.

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).



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



Retracting after a program interruption
If you interrupt program run during thread cutting with the machine stop button, the TNC will display the soft key MANUAL OPERATION. If you press the MANUAL OPERATION key, you can retract the tool under program control. Simply press the positive axis direction button of the active tool axis.

Example NC blocks:

```
N26  G207 Q200=2 Q201=-20 Q239=+1
         Q203=+0 Q204=50*
```

THREAD CUTTING (Cycle G86, not TNC 410)

Machine and control must be specially prepared by the machine tool builder to enable thread cutting.

Cycle G86 THREAD CUTTING is performed by means of spindle control. The tool moves with the active spindle speed from its current position to the entered depth. As soon as it reaches the end of thread, spindle rotation is stopped. Tool approach and departure must be programmed separately. The most convenient way to do this is by using OEM cycles. The machine tool builder can give you further information.

Before programming, note the following:

The TNC calculates the feed rate from the spindle speed. If the spindle speed override is used during thread cutting, the feed rate is automatically adjusted.

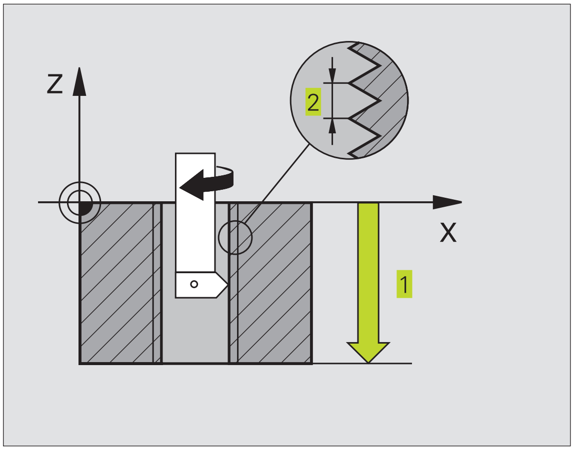
The feed-rate override knob is disabled.

The TNC automatically activates and deactivates spindle rotation. Do not program M3 or M4 before cycle call.

- Total hole depth 1: Distance between current tool position and end of thread.

  The algebraic sign for the total hole depth determines the working direction (a negative value means a negative working direction in the tool axis)

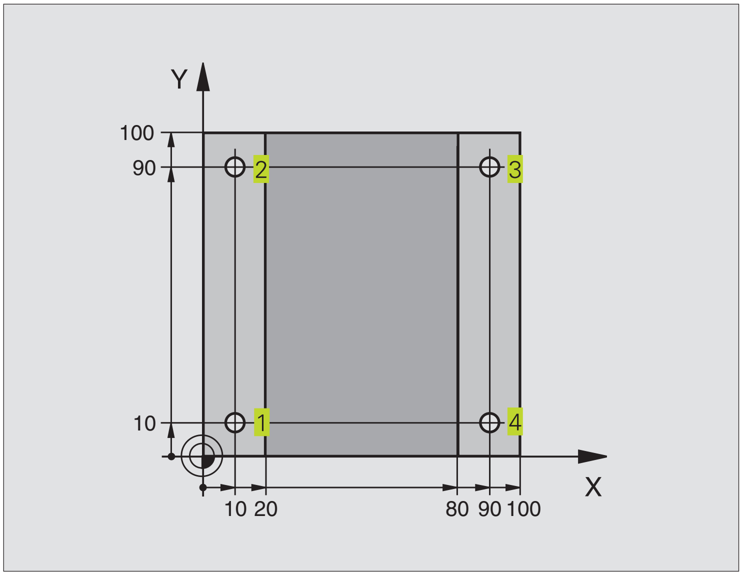
- PITCH 2:
  Pitch of the thread. The algebraic sign differentiates between right-hand and left-hand threads:
  + = right-hand thread (M3 with negative depth)
  – = left-hand thread (M4 with negative depth)



Example NC blocks:

```
N22  G86 P01 -20 P02 +1*
```

Example: Drilling cycles

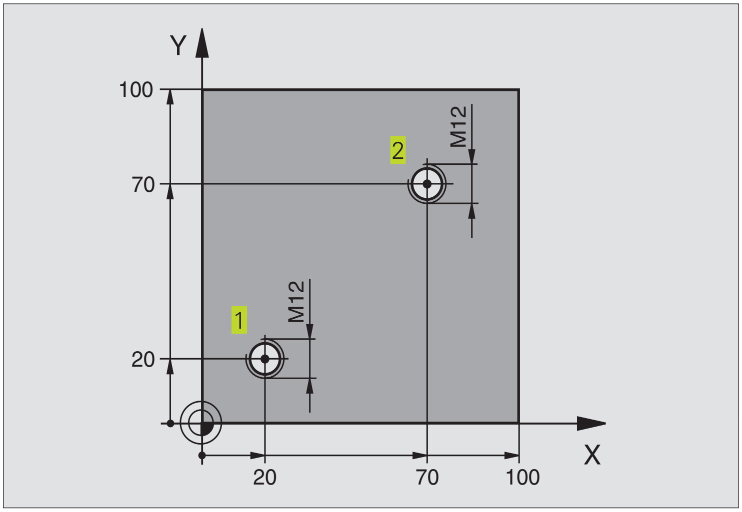


%C200 G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+3 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G200 Q200=2 Q201=-15 Q206=250	Define cycle
Q202=5 Q210=0 Q203=0 Q204=50 *	
N70 X+10 Y+10 M3 *	Approach hole 1, spindle ON
N80 Z-8 M99 *	Pre-position in the spindle axis, cycle call
N90 Y+90 M99 *	Approach hole 2, call cycle
N100 Z+20 *	Retract in the spindle axis
N110 X+90 *	Approach hole 3
N120 Z-8 M99 *	Pre-position in the spindle axis, cycle call
N130 Y+10 M99 *	Approach hole 4, call cycle
N140 G00 Z+250 M2 *	Retract in the tool axis, end program
N999999 %C200 G71 *	

Example: Drilling cycles

Program sequence

- Program the drilling cycle in the main program
- Program the machining operation in a subprogram (see section 9 “Programming: Subprograms and Program Section Repeats”)



`%C18 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-20 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+6 *`	Define the tool
`N40 T1 G17 S4500 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 G86 P01 +30 P02 -1,75 *`	Define THREAD CUTTING cycle
`N70 X+20 Y+20 *`	Approach hole 1
`N80 L1,0 *`	Call subprogram 1
`N90 X+70 Y+70 *`	Approach hole 2
`N100 L1,0 *`	Call subprogram 1
`N110 G00 Z+250 M2 *`	Retract tool, end of main program
`N120 G98 L1 *`	Subprogram 1: Thread cutting
`N130 G36 S0 *`	Orient spindle (makes it possible to cut repeatedly)
`N140 G01 G91 X-2 F1000 *`	Tool offset to prevent collision during tool infeed (dependent on core diameter and tool)
`N150 G90 Z-30 *`	Move to starting depth
`N160 G91 X+2 *`	Reset the tool to hole center
`N170 G79 *`	Calling the Cycle
`N180 G90 Z+5 *`	Retract tool
`N190 G98 L0 *`	End of subprogram 1
`N999999 %C18 G71 *`	

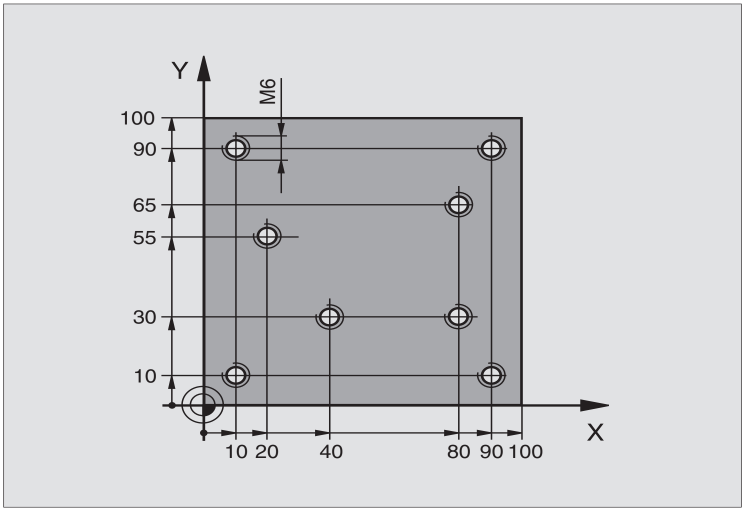
Example: Calling drilling cycles in connection with point tables (only with TNC 410)

Program sequence

- Centering
- Drilling
- Tapping M6

The drill hole coordinates are stored in the point table TAB1.PNT (see next page) and are called by the TNC with G79 PAT.

The tool radii are selected so that all work steps can be seen in the test graphics.



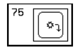
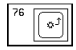
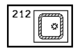
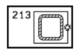
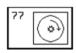
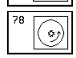
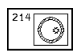
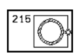
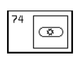
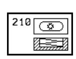
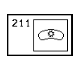
`%1 G71*`	
`N10 G30 G17 X+0 Y+0 Z-20*`	Define the workpiece blank
`N20 G31 X+100 Y+100 Z+0*`	
`N30 G99 1 L+0 R+4*`	Tool definition of center drill
`N40 G99 2 L+0 R+2.4*`	Tool definition: drill
`N50 G99 3 L+0 R+3*`	Tool definition of tap
`N60 T1 G17 S5000*`	Tool call of centering drill
`N70 G01 G40 Z+10 F5000*`	Move tool to clearance height (Enter a value for F.
	The TNC positions to the clearance height after every cycle)
`N80 %:PAT: "TAB1"*`	Defining point tables
`N90 G200 Q200=2 Q201=-2 Q206=150 Q202=2`	Cycle definition: Centering
`Q210=0 Q203=+0 Q204=0*`	The value 0 must be entered with Q203 and Q204.
`N100 G79 "PAT" F5000 M3*`	Cycle call in connection with point table TAB1.PNT
	Feed rate between points: 5000 mm/min
`N110 G00 G40 Z+100 M6*`	Retract the tool, change the tool

`N120 T2 G17 S5000*`	Call the drilling tool
`N130 G01 G40 Z+10 F5000*`	Move tool to clearance height (enter a value for F)
`N140 G200 Q200=2 Q201=-25 Q206=150 Q202=5`	Cycle definition: drilling
`Q210=0 Q203=+0 Q204=0*`	The value 0 must be entered with Q203 and Q204.
`N150 G79 "PAT" F5000 M3*`	Cycle call in connection with point table TAB1.PNT
`N160 G00 G40 Z+100 M6*`	Retract the tool, change the tool
`N170 T3 G17 S200*`	Tool call for tap
`N180 G00 G40 Z+50*`	Move tool to clearance height
`N190 G84 P01 +2 P02 -15 P030 P04 150*`	Cycle definition for tapping
`N200 G79 "PAT" F5000 M3*`	Cycle call in connection with point table TAB1.PNT
`N210 G00 G40 Z+100 M2*`	Retract in the tool axis, end program
`N99999 %1 G71*`	

Point tableTAB1.PNT

```
     TAB1   .PNT         MM
NR   X            Y            Z
0    +10          +10          +0
1    +40          +30          +0
2    +90          +10          +0
3    +80          +30          +0
4    +80          +65          +0
5    +90          +90          +0
6    +10          +90          +0
7    +20          +55          +0
[END]
```

8.4 Cycles for milling pockets, studs and slots

Cycle	Soft key
G75/G76 POCKET MILLING (rectangular) Roughing cycle without automatic pre-positioning G75: In clockwise direction G76: Counterclockwise	 
G212 POCKET FINISHING (rectangular) Finishing cycle with automatic pre-positioning, and 2nd set-up clearance	
G213 STUD FINISHING (rectangular) Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
G77/G78 CIRCULAR POCKET MILLING Roughing cycle without automatic pre-positioning G77: In clockwise direction G78: Counterclockwise	 
G214 CIRCULAR POCKET FINISHING Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
G215 CIRCULAR STUD FINISHING Finishing cycle with automatic pre-positioning and 2nd set-up clearance	
G74 SLOT MILLING Roughing/finishing cycle without automatic pre-positioning, vertical downfeed	
G210 SLOT WITH RECIPROCATING PLUNGE-CUT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	
G211 CIRCULAR SLOT Roughing/finishing cycle with automatic pre-positioning and reciprocating plunge-cut	

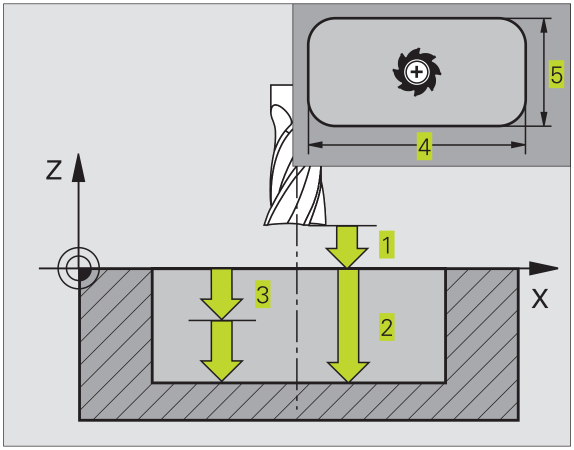
POCKET MILLING (Cycles G75, G76)

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

2 The cutter begins milling in the positive axis direction of the longer side (on square pockets, always starting in the positive Y direction) and then roughs out the pocket from the inside out.

3 This process (1 to 2) is repeated until the depth is reached.

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



Before programming, note the following:

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION G40.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

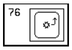
The algebraic sign for the depth parameter determines the working direction.

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

The following condition must be met for the second line length:
2nd side length greater than [(2 x rounding-off radius) + stepover factor k].

Direction of rotation during rough-out

- In clockwise direction: G75
- Counterclockwise: G76



- ▶ Setup 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
- ▶ Plunging depth 3 (incremental value): Infeed per cut. The tool will advance to the depth in one movement if:
  - the plunging depth equals the depth
  - the plunging depth is greater than the depth
- ▶ Feed rate for plunging: Traversing speed of the tool during penetration
- ▶ 1st side length 4: Pocket length, parallel to the main axis of the working plane
- ▶ 2nd side length 5: Pocket width
- ▶ Feed rate F: Traversing speed of the tool in the working plane
- ▶ Rounding off radius: Radius for the pocket corners. If Radius = 0 is entered, the pocket corners will be rounded with the radius of the cutter.

Example NC blocks:

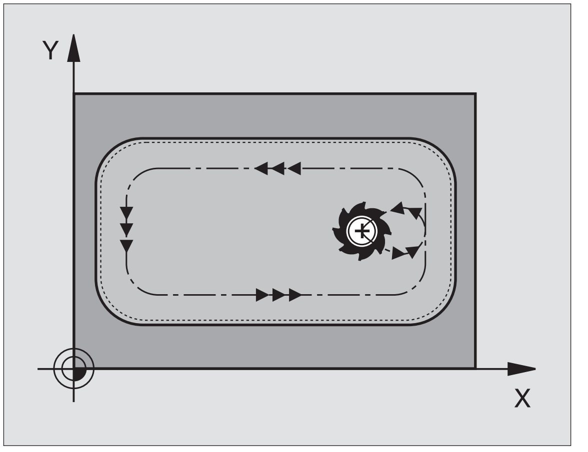
```
N27  G75 P01  2 P02 -20 P03  5 P04 100
     P05 X+80 P06 Y+60 P07 275 P08 5*
...
N35  G76 P01  2 P02 -20 P03  5 P04 100
     P05 X+80 P06 Y+60 P07 275 P08 5*
```

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 G212)



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 subsequently to the center of the pocket.
2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the allowance and tool radius into account for calculating the starting point. If necessary, the TNC plunge-cuts into the pocket center.
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse 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).

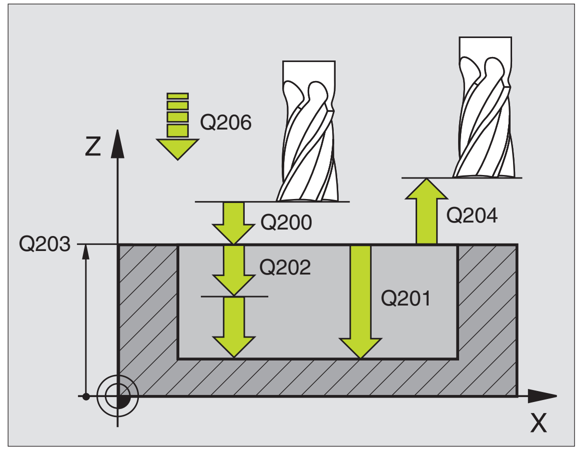
Before programming, note the following:

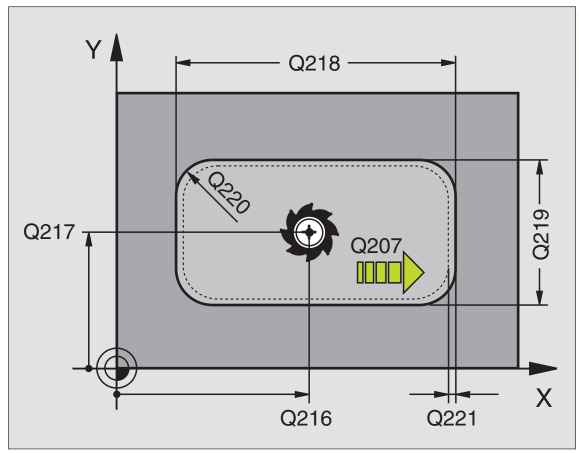
The algebraic sign for the depth parameter determines the working direction.

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 value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the pocket in the 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 main 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.





Example NC block:

```
N34  G212 Q200=2 Q201=-20 Q206=150
          Q202=5 Q207=500 Q203=+0 Q204=50
          Q216=+50 Q217=+50 Q218=80 Q219=60
          Q220=5 Q221=0*
```

STUD FINISHING (Cycle G213)

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.

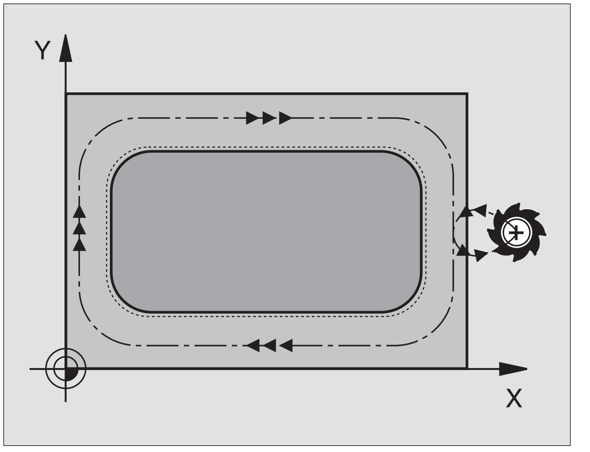
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

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.

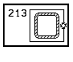
7 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or — if programmed — to the 2nd set-up clearance, and finally to the center of the stud (end position = starting position).



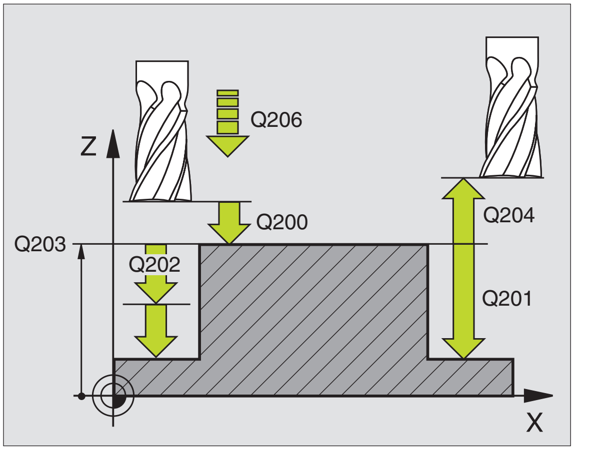
Before programming, note the following:

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 for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- Plunging depth Q202 (incremental value): Infeed per cut Enter a value greater than 0.
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface



Example NC block:

```
N35  G213 Q200=2 Q201=-20 Q206=150
     Q202=5 Q207=500 Q203=+0 Q204=50
     Q216=+50 Q217=+50 Q218=80 Q219=60
     Q220=5 Q221=0*
```

▶ 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 main axis of the working plane

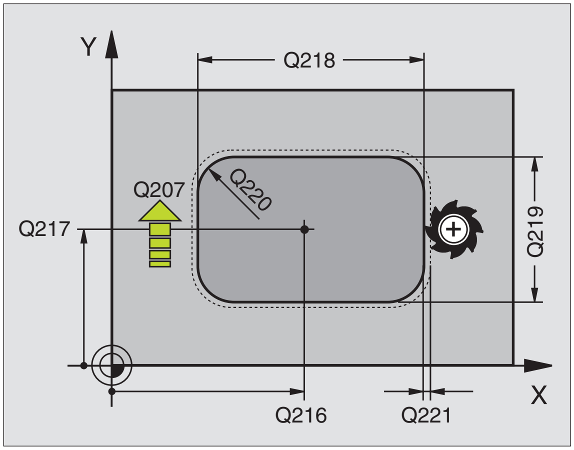
▶ Center in 2nd axis Q217 (absolute value): Center of the stud in the secondary axis of the working plane

▶ First side length Q218 (incremental value): Stud length, parallel to the main axis of the working plane

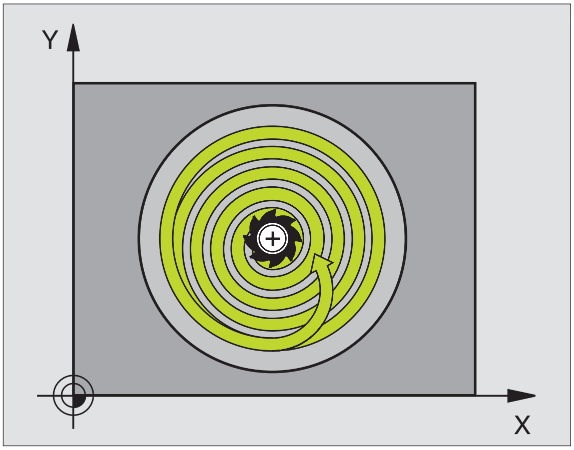
▶ Second side length Q219 (incremental value): Stud length, parallel to the secondary axis of the working plane

▶ 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.



CIRCULAR POCKET MILLING (Cycles G77, G78)



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

2 The tool subsequently follows a spiral path at the feed rate F — see figure at right. For calculating the stepover factor k, see Cycle G75/G76 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 programming, note the following:

Program a positioning block for the starting point (pocket center) in the working plane with RADIUS COMPENSATION G40.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

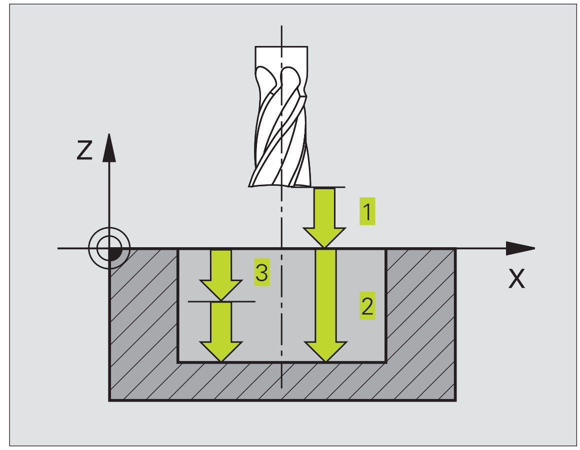
The algebraic sign for the depth parameter determines the working direction.

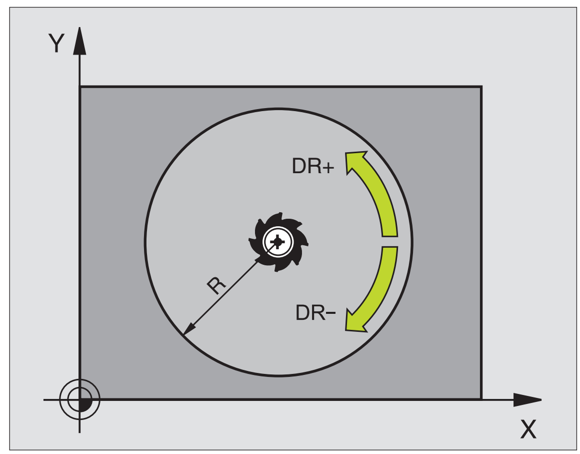
This cycle requires a center-cut end mill (ISO 1641), or pilot drilling at the pocket center.

Direction of rotation during rough-out

- In clockwise direction: G77
- In counterclockwise direction: G78

- Setup 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
- Plunging depth 3 (incremental value):
  Infeed per cut. The tool will advance to the depth in one movement if:
  n the plunging depth equals the depth
  n the plunging depth is greater than the depth
- Feed rate for plunging: Traversing speed of the tool during penetration
- Circular radius: Radius of the circular pocket
- Feed rate F: Traversing speed of the tool in the working plane





Example NC blocks:

```
N36  G77 P01  2 P02 -20 P03  5 P04 100
      P05 40 P06 250*
...
N48  G78 P01  2 P02 -20 P03  5 P04 100
      P05 40 P06 250*
```

CIRCULAR POCKET FINISHING (Cycle G214)

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 subsequently to the center of the pocket.

2 From the pocket center, the tool moves in the working plane to the starting point for machining. The TNC takes the workpiece blank diameter and tool radius into account for calculating the starting point. If you enter a workpiece blank diameter of 0, the TNC plunge-cuts into the pocket center.

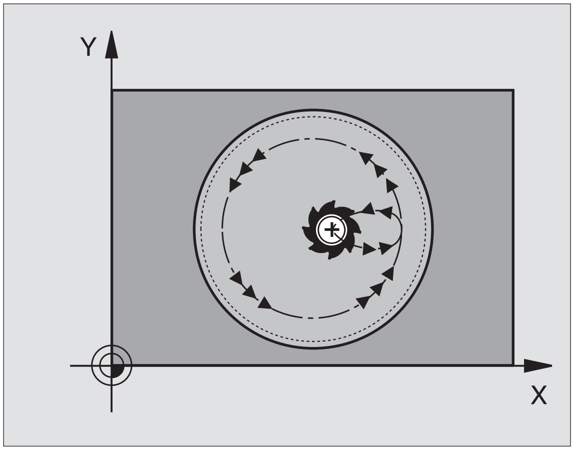
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

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.

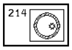
7 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or — if programmed — to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).



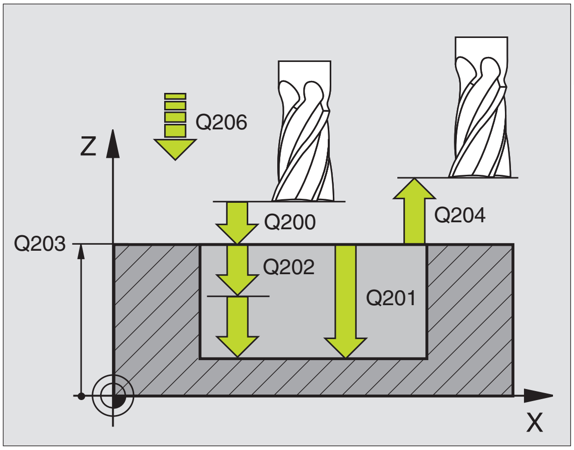
Before programming, note the following:

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.



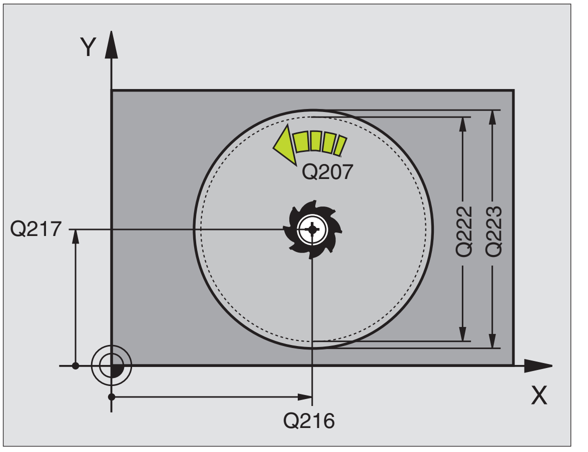
- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of pocket
- Feed rate for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting 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
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.



Example NC block:

```
N42  G214 Q200=2 Q201=-20 Q206=150
     Q202=5 Q207=500 Q203=+0 Q204=50
     Q216=+50 Q217=+50 Q222=79 Q223=80*
```

- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the pocket in the 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.
- 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 G215)

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 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.

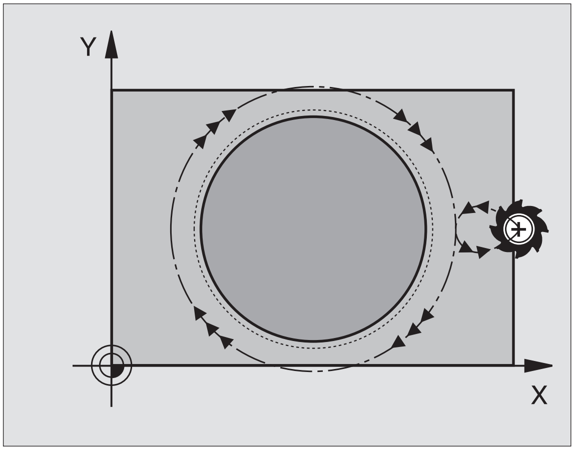
3 If the tool is at the 2nd set-up clearance, it moves in rapid traverse to set-up clearance, and from there advances to the first plunging depth at the feed rate for plunging.

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.

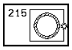
7 At the end of the cycle, the TNC retracts the tool in rapid traverse to set-up clearance, or - if programmed - to the 2nd set-up clearance, and finally to the center of the pocket (end position = starting position).



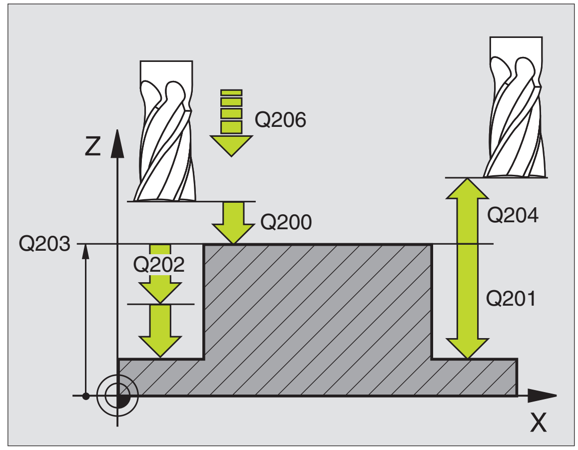
Before programming, note the following:

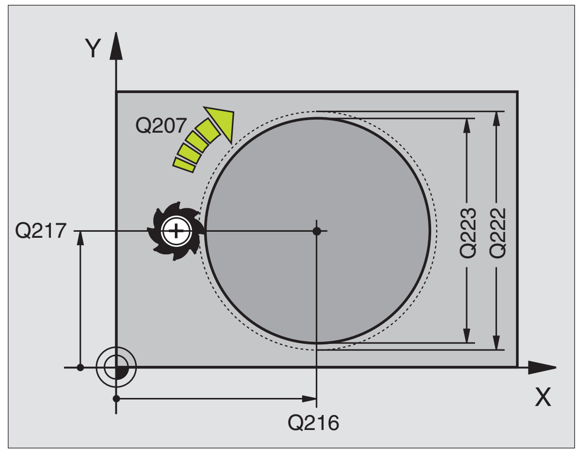
The algebraic sign for the depth parameter determines the working direction.

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 for plunging Q206: Traversing speed of the tool in mm/min when moving to depth. If you are plunge-cutting into the material, enter a low value; if you have already cleared the stud, enter a higher feed rate.
- Plunging depth Q202 (incremental value): Infeed per cut; enter a value greater than 0.
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.
- Center in 1st axis Q216 (absolute value): Center of the stud in the main 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.

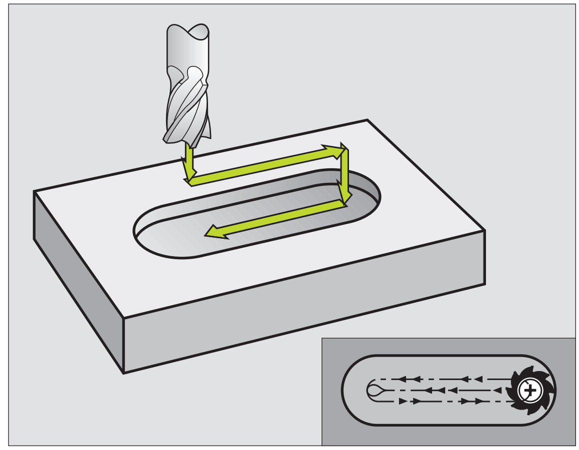




Example NC block:

```
N43  G215 Q200=2 Q201=-20 Q206=150
          Q202=5 Q207=500 Q203=+0 Q204=50
          Q216=+50 Q217=+50 Q222=81 Q223=80*
```

SLOT MILLING (Cycle G74)



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.

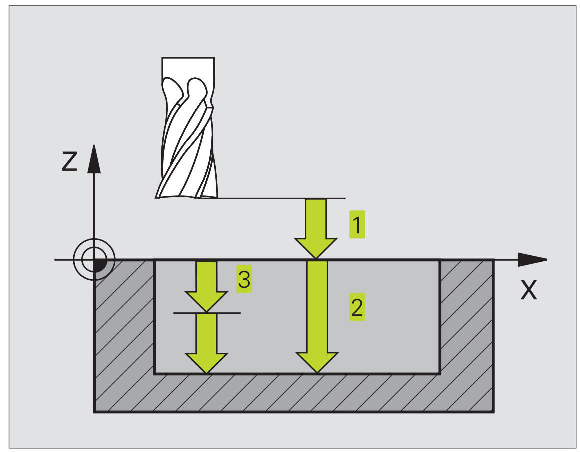
This process is repeated until the programmed milling depth is reached.

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 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 programming, note the following:

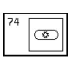
Program a positioning block for the starting point in the working plane — to the center of the slot (second side length) and, within the slot, offset by the tool radius — with RADIUS COMPENSATION G40.

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

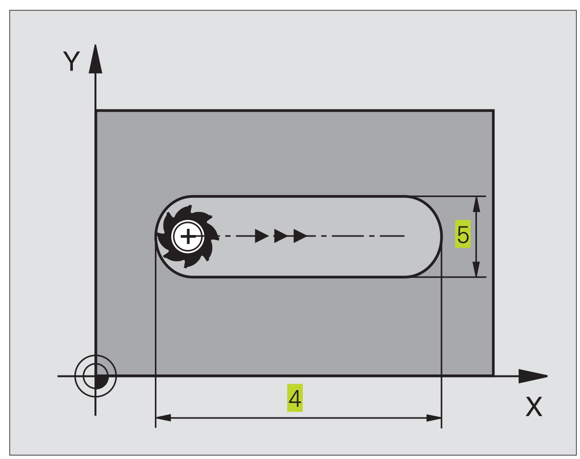
The algebraic sign for the depth parameter determines the working direction.

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.



- Setup 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
- Plunging depth 3 (incremental value): Infeed per cut; the TNC will advance to the depth in one movement if:
  - the plunging depth equals the depth
  - the plunging depth is greater than the depth
- Feed rate for plunging: Traversing speed of the tool during penetration



- 1st side length 4: Slot length; specify the sign to determine the first milling direction
- 2nd side length 5: Slot width
- Feed rate F: Traversing speed of the tool in the working plane

Example NC block:

```
N44  G74 P01  2 P02 -20 P03  5 P04 100
       P05 X+80 P06 Y12 P07 275*
```

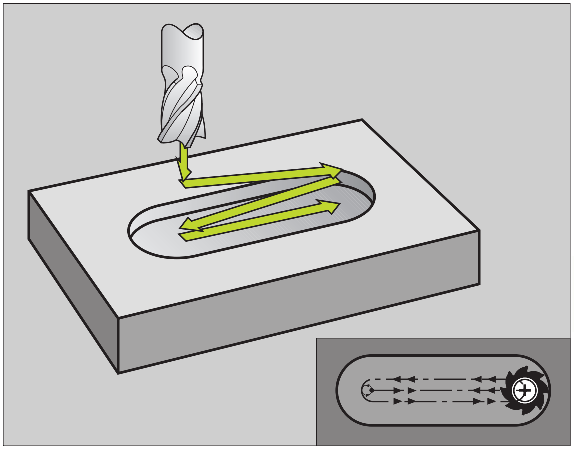
SLOT with reciprocating plunge-cut (Cycle G210)

Before 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.

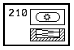
4 At the milling depth, the TNC moves the tool for the purpose of face milling to the other end of the slot and then back to the center of the slot.

Finishing process

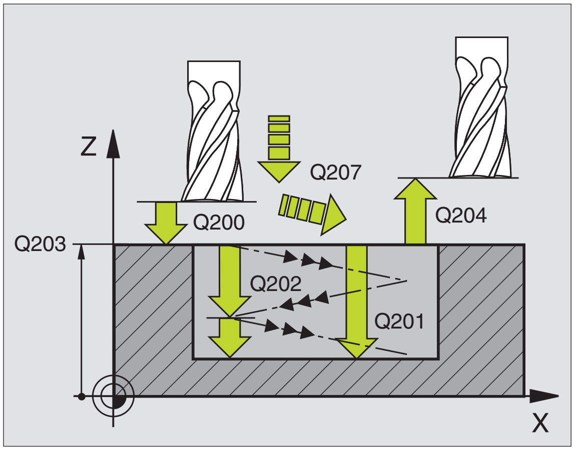
5 The TNC advances the tool from the slot center tangentially to the contour of the finished part. The tool subsequently climb mills the contour (with M3).

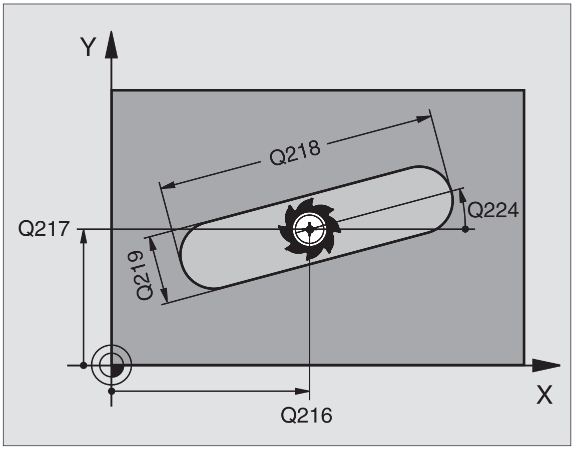
6 When the tool reaches the end of the contour, it departs the contour tangentially and returns to the center of the slot.

7 At the end of the cycle, the tool is retracted in rapid traverse 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 main 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.





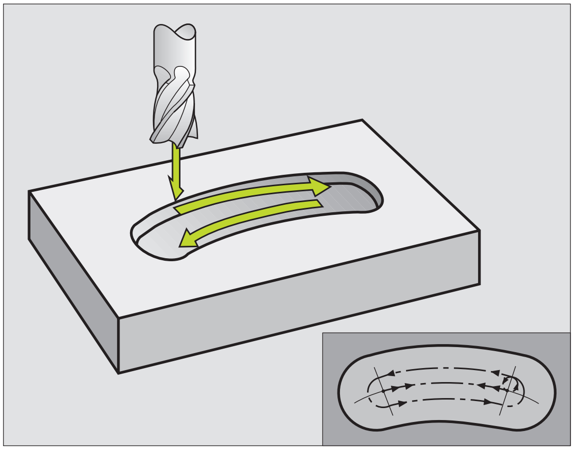
Example NC block:

```
N51  G210 Q200=2 Q201=-20 Q207=500
          Q202=5 Q215=0 Q203=+0 Q204=50
          Q216=+50 Q217=+50 Q218=80 Q219=12
          Q224=+15*
```

CIRCULAR SLOT with reciprocating plunge-cut (Cycle G211)

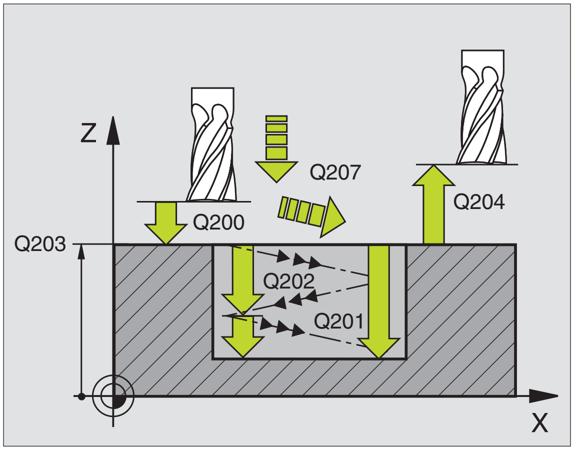
Roughing process

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 to set-up clearance and - if programmed - to the 2nd set-up clearance.

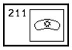


Before programming, note the following:

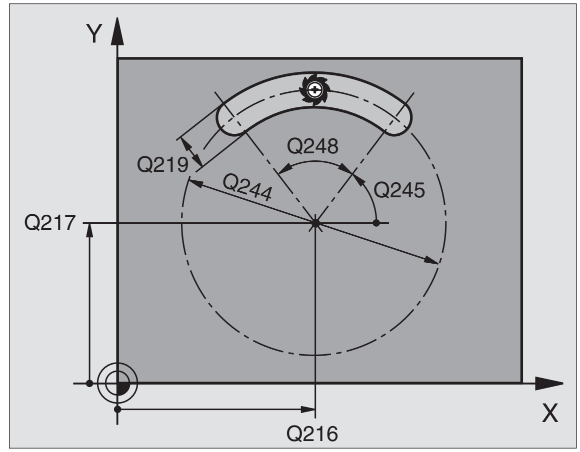
The algebraic sign for the depth parameter determines the working direction.

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.



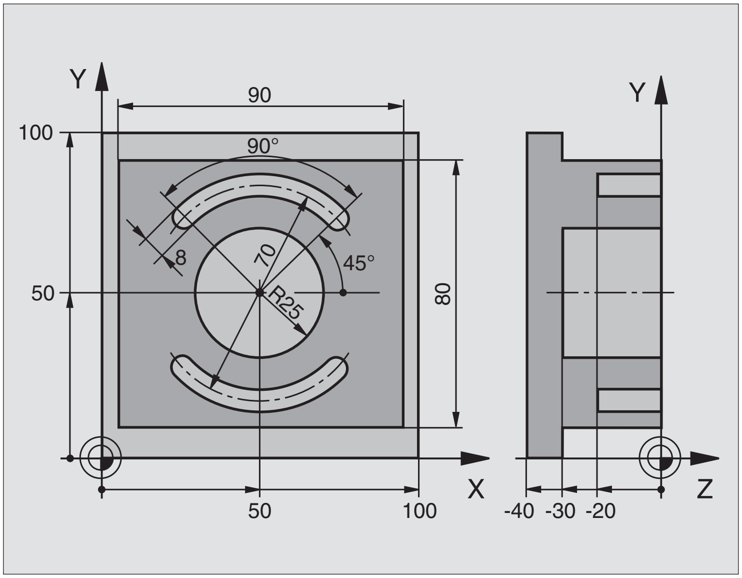
- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface.
- Depth Q201 (incremental value): Distance between workpiece surface and bottom of slot
- Feed rate for milling Q207: Traversing speed of the tool in mm/min while milling.
- Plunging depth Q202 (incremental 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 main 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 NC block:

```
N52  G211 Q200=2 Q201=-20 Q207=500
          Q202=5 Q215=0 Q203=+0 Q204=50
          Q216=+50 Q217=+50 Q244=80 Q219=12
          Q245=+45 Q248=90*
```

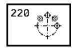
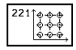
Example: Milling pockets, studs and slots



`%C210 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+6 *`	Define the tool for roughing/finishing
`N40 G99 T2 L+0 R+3 *`	Define slotting mill
`N50 T1 G17 S3500 *`	Call tool for roughing/finishing
`N60 G00 G40 G90 Z+250 *`	Retract the tool
`N70 G213 Q200=2 Q201=-30 Q206=250 Q202=5`	Define cycle for machining the contour outside
`Q207=250 Q203=+0 Q204=20 Q216=+50`	
`Q217=+50 Q218+90 Q219=80 Q220=0 Q221=5 *`	
`N80 G79 M03 *`	Call cycle for machining the contour outside
`N90 G78 P01 2 P02 -30 P03 5 P04 250 P05 25`	Define CIRCULAR POCKET MILLING cycle
`P06 400 *`	
`N100 G00 G40 X+50 Y+50 *`	
`N110 Z+2 M99 *`	Call CIRCULAR POCKET MILLING cycle
`N120 Z+250 M06 *`	Tool change
`N130 T2 G17 S5000 *`	Call slotting mill
`N140 G211 Q200=2 Q201=-20 Q207=250`	Define cycle for slot 1
`Q202=5 Q215=0 Q203=+0 Q204=100`	
`Q216=+50 Q217=+50 Q244=70 Q219=8`	
`Q245=+45 Q248=90 *`	
`N150 G79 M03 *`	Call cycle for slot 1
`N160 D00 Q245 P01 +225 *`	New starting angle for slot 2
`N170 G79 *`	Call cycle for slot 2
`N180 G00 Z+250 M02 *`	Retract in the tool axis, end program
`N999999 %C210 G71 *`	

8.5 Cycles for Machining Hole Patterns

The TNC provides two cycles for machining hole patterns:

Cycle	Soft key
G220 CIRCULAR PATTERN	
G221 LINEAR PATTERN	

You can combine Cycle G220 and Cycle G221 with the following fixed cycles:

Cycle G83	PECKING
Cycle G84	TAPPING with a floating tap holder
Cycle G74	SLOT MILLING
Cycle G75/G76	POCKET MILLING
Cycle G77/G78	CIRCULAR POCKET MILLING
Cycle G85	RIGID TAPPING
Cycle G86	THREAD CUTTING
Cycle G200	DRILLING
Cycle G201	REAMING
Cycle G202	BORING
Cycle G203	UNIVERSAL MILLING CYCLE
Cycle G204	BACK BORING
Cycle G212	POCKET FINISHING
Cycle G213	STUD FINISHING
Cycle G214	CIRCULAR POCKET FINISHING
Cycle G215	CIRCULAR STUD FINISHING

The TNC 426, TNC 430 with NC software 280 474-xx also provides:

Cycle G205	UNIVERSAL PECKING
Cycle G206	TAPPING NEW with a floating tap holder
Cycle G207	RIGID TAPPING NEW without a floating tap holder
Cycle G208	BORE MILLING

CIRCULAR PATTERN (Cycle 220)

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

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.

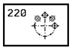
3 The tool then approaches the starting point for the next machining operation on a straight line at set-up clearance (or 2nd set-up clearance).

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

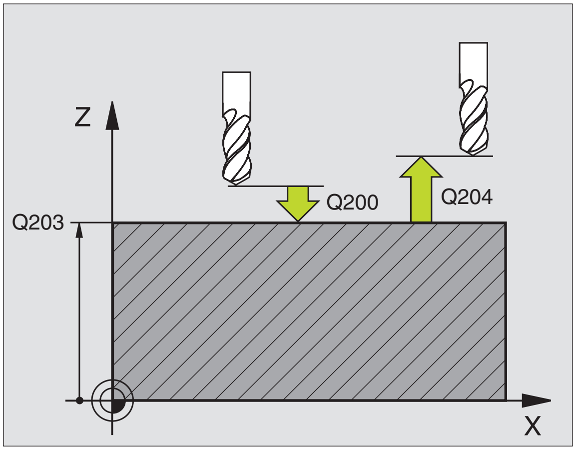
Before programming, note the following:

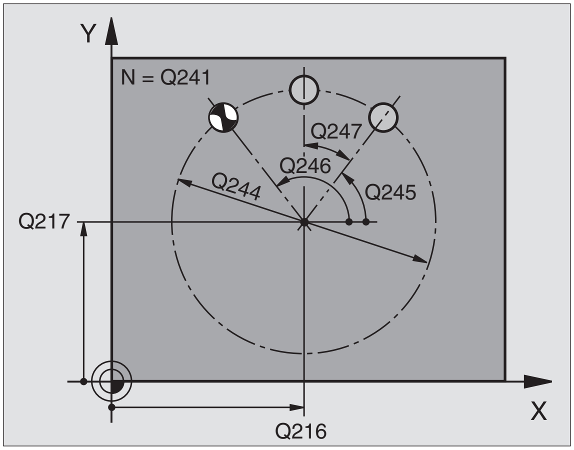
Cycle G220 is DEF active, which means that Cycle G220 automatically calls the last defined fixed cycle.

If you combine Cycle G220 with one of the fixed cycles G200 to G208 and G212 to G215, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G220 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 (does not apply to complete circles). Do not enter the same value for the stopping angle and starting angle. If you enter the stopping angle greater than the starting angle, machining will be carried out counterclockwise; otherwise, machining will be clockwise.
- ANGLE STEP Q247 (incremental): Angle between two machining operations on a pitch circle. If you enter an ANGLE STEP of 0, the TNC will calculate the ANGLE STEP from the STARTING and STOPPING ANGLES and the number of pattern repetitions. If you enter a value other than 0, the TNC will not take the STOPPING ANGLE into account. The sign for the ANGLE STEP determines the working direction (- = clockwise).





Example NC block:

```
N53  G220 Q216=+50 Q217=+50 Q244=80
     Q245=+0 Q246=+360 Q247=+0 Q241=8
     Q200=2 Q203=+0 Q204=50*
```

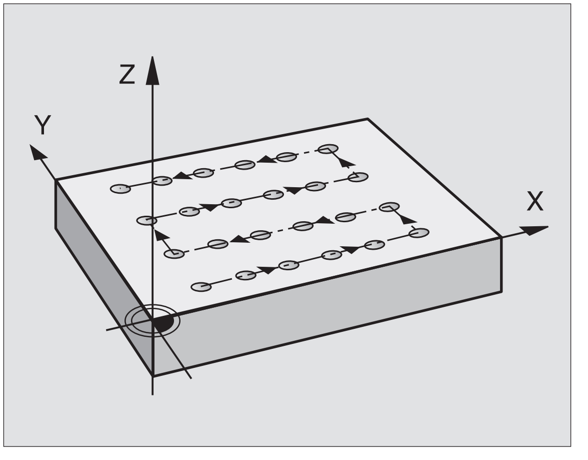
- Number of repetitions Q241: Number of machining operations on a pitch circle
- Set-up clearance Q200 (incremental value): Distance between tool tip and workpiece surface. Enter a positive value.
- Workpiece surface coordinate Q203 (absolute value): Coordinate of the workpiece surface
- 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.

The TNC 426, TNC 430 with NC software 280 474-xx also provides:

- Traversing to clearance height Q301: definition of how the tool is to move between machining processes:

0: Move to set-up clearance
1: Move to 2nd set-up clearance

LINEAR PATTERN (Cycle 221)



Before programming, note the following:

Cycle G221 is DEF active, which means that Cycle G221 automatically calls the last defined fixed cycle.

If you combine Cycle G220 with one of the fixed cycles G200 to G208 and G212 to G215, the set-up clearance, workpiece surface and 2nd set-up clearance that you defined in Cycle G220 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.

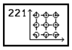
5 The tool subsequently moves to the last point on the second line where it carries out the machining operation.

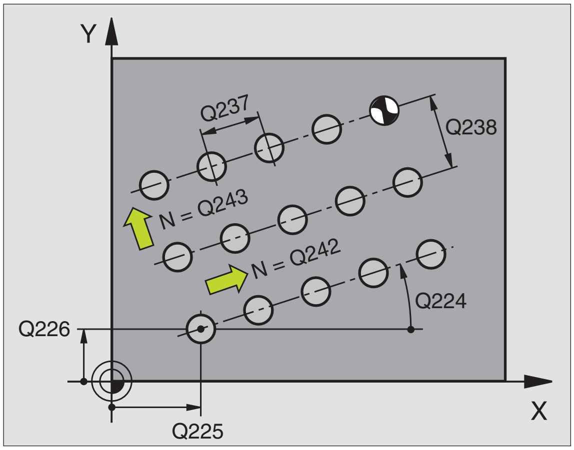
6 From this position, the tool approaches the starting point for the next machining operation in the negative main axis direction.

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

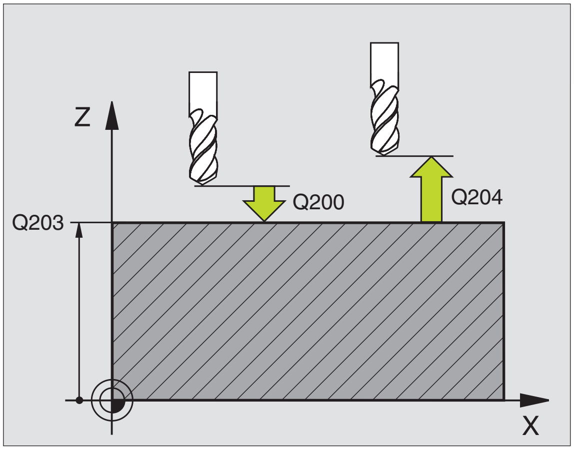
8 The tool then moves to the starting point of the next line.

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 value): Coordinate of the workpiece surface
- ▶ 2nd set-up clearance Q204 (incremental value): Coordinate in the tool axis at which no collision between tool and workpiece (clamping devices) can occur.



The TNC 426, TNC 430 with NC software 280 474-xx also provides:

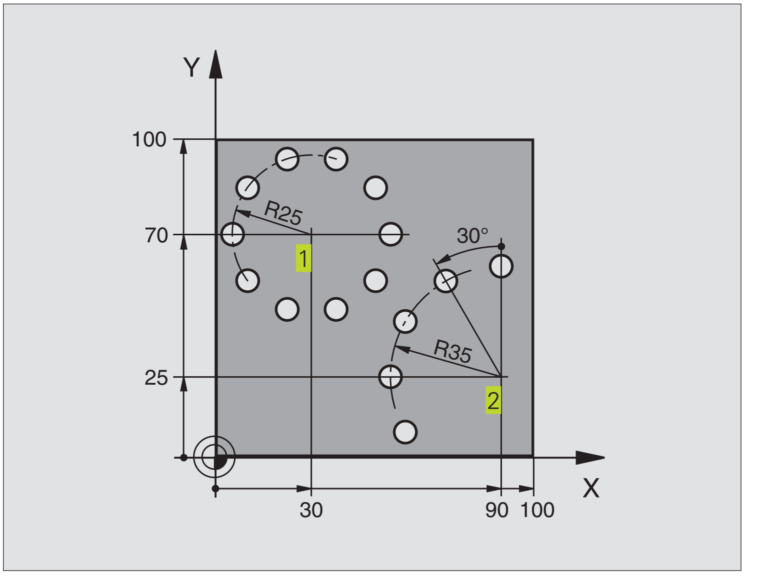
- ▶ Traversing to clearance height Q301: definition of how the tool is to move between machining processes:

0: Move to set-up clearance
1: Move to 2nd set-up clearance

Example NC block:

```
N54  G221 Q225=+15 Q226=+15 Q237=+10
          Q238=+8 Q242=6 Q243=4 Q224=+15
          Q200=2 Q203=+0 Q204=50*
```

Example: Circular hole patterns



`%BOHRB G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+3 *`	Define the tool
`N40 T1 G17 S3500 *`	Tool call
`N50 G00 G40 G90 Z+250 M03 *`	Retract the tool
`N60 G200 Q200=2 Q201=-15 Q206=250`	Cycle definition: drilling
`     Q202=4 Q210=0 Q203=+0 Q204=0 *`	
`N70 G220 Q216=+30 Q217=+70 Q244=50`	Cycle definition: circular hole pattern 1
`     Q245=+0 Q246=+360 Q247=+0 Q241=10`	
`     Q200=2 Q203=+0 Q204=100 *`	
`N80  G220 Q216=+90 Q217=+25 Q244=70`	Cycle definition: circular hole pattern 2
`     Q245=+90 Q246=+360 Q247=+30 Q241=5`	
`     Q200=2 Q203=+0 Q204=100 *`	
`N90 G00 G40 Z+250 M02 *`	Retract tool, end of program
`N999999 %BOHRB G71`	

8.6 SL Cycles Group I

SL cycles allow the contour-oriented machining of complex contours.

Characteristics of the contour

- A contour can be composed of several overlapping subcontours (up to 12 subcontours are possible). Islands and pockets can form a subcontour.
- The subcontour list (subprogram numbers) is entered in Cycle G37 CONTOUR GEOMETRY. The TNC calculates the contour from the subcontours.
- The individual subcontours are defined in subprograms.
- The memory capacity for programming an SL cycle is limited. All subprograms together can contain, for example, up to 128 straight-line blocks.

Characteristics of the subprograms

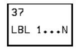
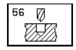
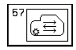
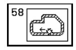
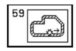
- Coordinate transformations are allowed.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation G42.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation G41.
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. Parallel axes are allowed.

Characteristics of the fixed cycles

> **TNC 410:**
>
> With MP7420.0 and MP7420.1 you define how the TNC should move the tool during area clearance (see "14.1 General User Parameters").

- The TNC automatically positions the tool to the starting position in the machining plane before a cycle. In the spindle axis the tool must be pre-positioned to set-up clearance.
- Each level of infeed depth is roughed-out axis-parallel or at a preset angle (angle defined in Cycle G57). In the standard setting, islands are traversed at safety clearance In MP7420.1 you can also define that the TNC should rough-out individual pockets separately, plunging only once for each pocket.
- The TNC takes the entered finishing allowance (cycle G57) into consideration

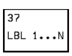
Overview of SL cycles

Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	
G56 PILOT DRILLING (optional)	
G57 ROUGH-OUT (essential)	
G58/G59 CONTOUR MILLING (optional) G58: In clockwise direction G59: In counterclockwise direction	 

CONTOUR GEOMETRY (Cycle G37)

All subprograms that are superimposed to define the contour are listed in Cycle G37 CONTOUR GEOMETRY (see figure bottom right).

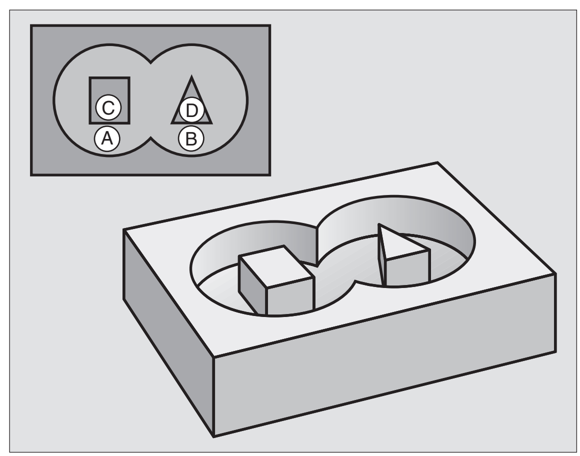
> Before programming, note the following:
>
> Cycle G37 is DEF active which means that it becomes effective as soon as it is defined in the part program.
>
> You can list up to 12 subprograms (subcontours) in Cycle G37.



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

Program structure: Working with SL cycles

```
%SL G71 *
...
N12 G37 P01 ...
...
N16 G56 P01 ...
N17 G79 *
...
N18 G57 P01 ...
N19 G79 *
...
N26 G59 P01 ...
N27 G79 *
...
N50 G00 G40 G90 Z+250 M2 *
N51 G98 L1 *
...
N60 G98 L0 *
N61 G98 L2 *
...
N62 G98 L0 *
...
N999999 %SL G71 *
```



Example NC block:

```
N54  G37 P01 1 P02 5 P03 7*
```

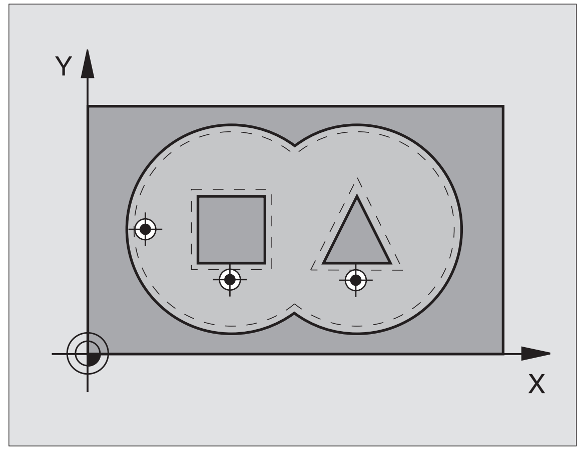
PILOT DRILLING (Cycle G56)

Process

Same as Cycle G83 Pecking

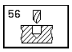
Application

Cycle G56 is for PILOT DRILLING of the cutter infeed points. It accounts for the finishing allowance. The cutter infeed points also serve as starting points for roughing.



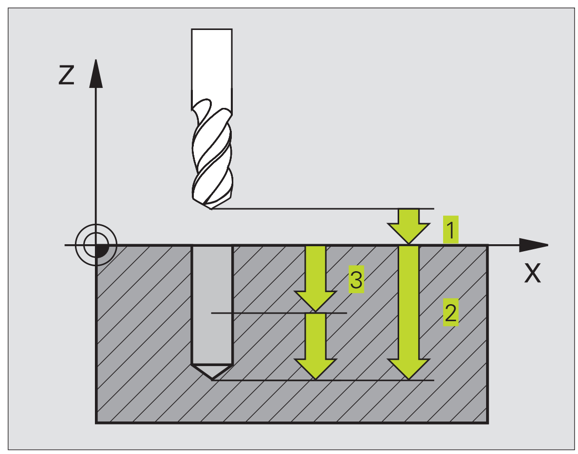
Before programming, note the following:

Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).



- Setup 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)
- Plunging depth 3 (incremental value): Infeed per cut. The TNC will go to depth in one movement if:
  - The plunging depth is equal to the total hole depth
  - The plunging depth is greater than the total hole depth

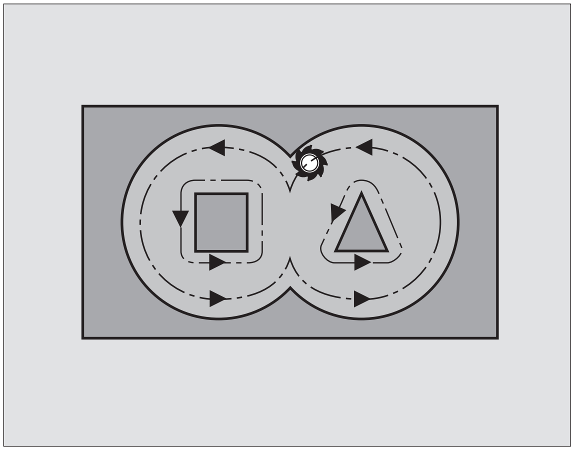
  The total hole depth does not have to be a multiple of the plunging depth.
- Feed rate for plunging: Traversing speed in mm/min for drilling
- Finishing allowance: Allowance in the machining plane



Example NC block:

```
N54 G56 P01 2 P02 -15 P03 5 P04 250
    P05 +0.5*
```

ROUGH-OUT (Cycle G57)



Process

1 The TNC positions the tool in the working plane above the first cutting point, taking the finishing allowance into consideration.

2 The TNC moves the tool at the feed rate for plunging to the first plunging depth.

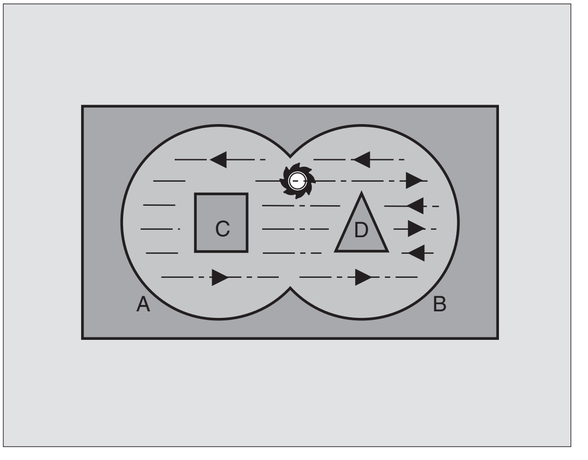
The contour is fully rough-milled (see figure at top right):

1 The tool mills the first subcontour at the programmed feed rate, taking the finishing allowance in the machining plane into consideration

2 Further depths and further subcontours are milled by the TNC in the same way

3 The TNC moves the tool in the spindle axis to the set-up clearance and then positions it above the first cutter infeed point in the machining plane.

Rough out pocket (see figure at center right):



1 After reaching the first plunging depth, the tool mills the contour at the programmed feed rate paraxially or at the entered roughing angle.

2 The island contours (here: C/D) are traversed at set-up clearance

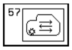
3 This process is repeated until the entered milling depth is reached.

Before programming, note the following:

With MP7420.0 and MP7420.1 you define how the TNC should machine the contour (see section "14.1 General User Parameters").

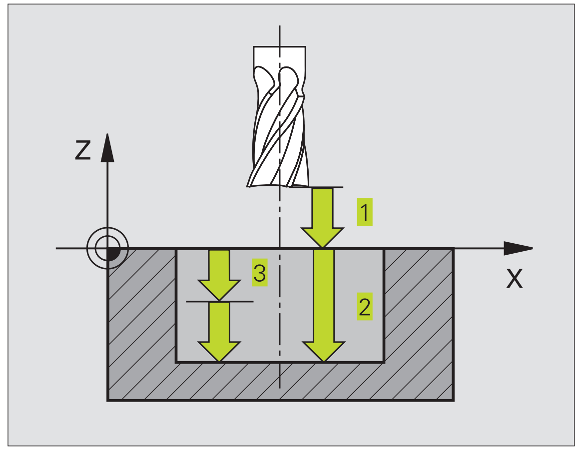
Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle G56.



- Setup clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- Milling depth 2 (incremental value): Distance between workpiece surface and pocket floor
- Plunging depth 3 (incremental value): Infeed per cut. The tool will advance to the total hole depth in one movement if:
  - the plunging depth equals the total hole depth
  - the plunging depth is greater than the total hole depth

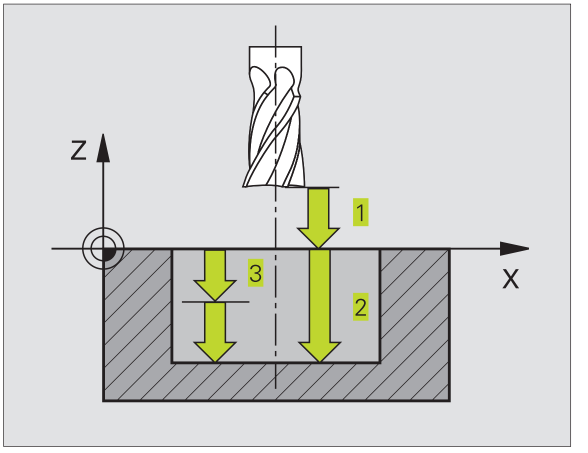
  The milling depth does not have to be a multiple of the plunging depth.
- Feed rate for plunging: Traversing speed of the tool in mm/min during penetration
- Finishing allowance: Allowance in the machining plane
- Rough-out angle: Direction of the roughing-out movement The rough-out angle is referenced to the main axis of the machining plane. Enter the angle so that the cuts can be as long as possible
- Feed rate: Feed rate for milling in mm/min



Example NC block:

```
N54 G57 P01 2 P02 -15 P03 5 P04 250
    P05 +0.5 P06+30 P07 500*
```

CONTOUR MILLING (Cycle G58/G59)



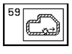
Application
Cycle G58/G59 CONTOUR MILLING serves for finishing the contour pocket.

> **Before programming, note the following:**
>
> Program a positioning block for the starting point in the tool axis (set-up clearance above the workpiece surface).

Direction of rotation during contour milling

- In clockwise direction: G58
- In counterclockwise direction: G59

The TNC finishes each subcontour separately, even at several infeed depths.



- ▶ Setup clearance 1 (incremental value): Distance between tool tip (at starting position) and workpiece surface
- ▶ Milling depth 2 (incremental value): Distance between workpiece surface and pocket floor
- ▶ Plunging depth 3 (incremental value): Infeed per cut. The TNC will go to depth in one movement if:
  - The plunging depth equals the milling depth
  - The plunging depth is greater than the milling depth

  The milling depth does not have to be a multiple of the plunging depth.
- ▶ Feed rate for plunging: Traversing speed of the tool in mm/min during penetration
- ▶ Feed rate: Feed rate for milling in mm/min

Example NC blocks:

```
N54  G58 P01  2 P02 -15 P03  5 P04 250
         P05 500*
...
N71  G59 P01  2 P02 -15 P03  5 P04 250
         P05 500*
```

8.7 SL Cycles Group II (not TNC 410)

SL cycles allow the contour-oriented machining of complex contours and achieve a particularly high degree of surface finish.

Characteristics of the contour

- A contour can be composed of several overlapping subcontours (up to 12 subcontours are possible). Islands and pockets can form a subcontour.
- The subcontour list (subprogram numbers) is entered in Cycle G37 CONTOUR GEOMETRY. The TNC calculates the contour from the subcontours.
- The individual subcontours are defined in subprograms.
- The memory capacity for programming an SL cycle is limited. All subprograms together can contain, for example, up to 128 straight-line blocks.

Characteristics of the subprograms

- Coordinate transformations are allowed.
- The TNC ignores feed rates F and miscellaneous functions M.
- The TNC recognizes a pocket if the tool path lies inside the contour, for example if you machine the contour clockwise with radius compensation G42.
- The TNC recognizes an island if the tool path lies outside the contour, for example if you machine the contour clockwise with radius compensation G41.
- The subprograms must not contain tool axis coordinates.
- The working plane is defined in the first coordinate block of the subprogram. The secondary axes U,V,W are permitted.

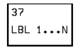
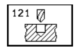
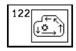
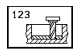
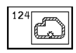
Characteristics of the fixed cycles

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

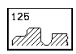
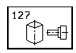
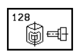
> ☞ With MP7420 you can determine where the tool is positioned at the end of Cycles G121 to G124.

The machining data (such as milling depth, finishing allowance and setup clearance) are entered as CONTOUR DATA in Cycle 120.

Overview of SL cycles

Cycle	Soft key
G37 CONTOUR GEOMETRY (essential)	
G120 CONTOUR DATA (essential)	
G121 PILOT DRILLING (optional)	
G122 ROUGH-OUT (essential)	
G123 FLOOR FINISHING (optional)	
G124 SIDE FINISHING (optional)	

Enhanced cycles:

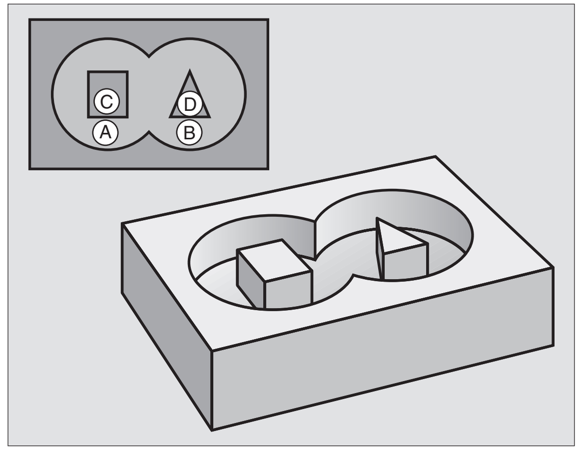
Cycle	Soft key
G125 CONTOUR TRAIN	
G127 CYLINDER SURFACE	
G128 CYLINDER SURFACE Slot milling	

Program structure: Working with SL cycles

```
%SL2 G71 *
...
N120 G37 ... *
N130 G120... *
...
N160 G121 ... *
N170 G79 *
...
N180 G122 ... *
N190 G79 *
...
N220 G123 ... *
N230 G79 *
...
N260 G124 ... *
N270 G79 *
...
N500 G00 G40 Z+250 M2 *
N510 G98 L1 *
...
N550 G98 L0 *
N560 G98 L2 *
...
N600 G98 L0 *
...
N99999 %SL2 G71 *
```

CONTOUR GEOMETRY (Cycle G37)

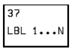
All subprograms that are superimposed to define the contour are listed in Cycle G37 CONTOUR GEOMETRY.



Before programming, note the following:

Cycle G37 is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can list up to 12 subprograms (subcontours) in Cycle G37.



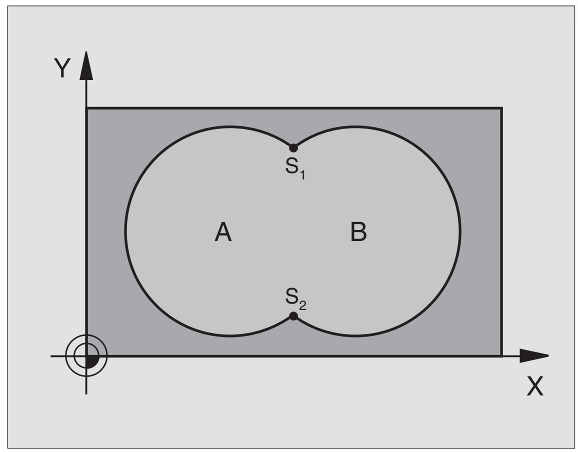
- Label numbers for the contour: Enter all label numbers for the individual subprograms that are to be superimposed to define the contour. Confirm every label number with the ENT key. When you have entered all numbers, conclude entry with the END key.

Example NC block:

```
N120  G37 P01 1 P02 5 P03 7*
```

Overlapping contours

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



Subprograms: Overlapping pockets

The subsequent programming examples are contour subprograms that are called by Cycle G37 CONTOUR GEOMETRY in a main program.

Pockets *A* and *B* overlap.

The TNC calculates the points of intersection S_1 and S_2 (they do not have to be programmed).

The pockets are programmed as full circles.

Subprogram 1: Left pocket

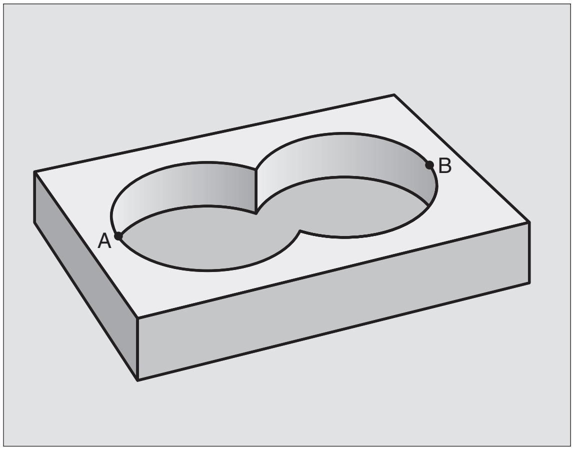
```
N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50  *
N550 G98 L0 *
```

Subprogram 2: Right pocket

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

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

- The surfaces *A* and *B* must be pockets.
- The first pocket (in Cycle G37) must start outside the second pocket.



Surface A:

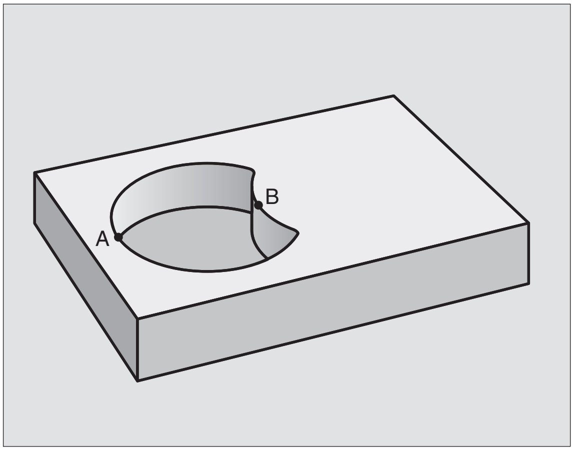
```
N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50  *
N550 G98 L0 *
```

Surface B:

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

Area of exclusion
Surface *A* is to be machined without the portion overlapped by *B*

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



Surface A:

```
N510 G98 L1 *
N520 G01 G42 X+10 Y+50 *
N530 I+35 J+50 *
N540 G02 X+10 Y+50  *
N550 G98 L0 *
```

Surface B:

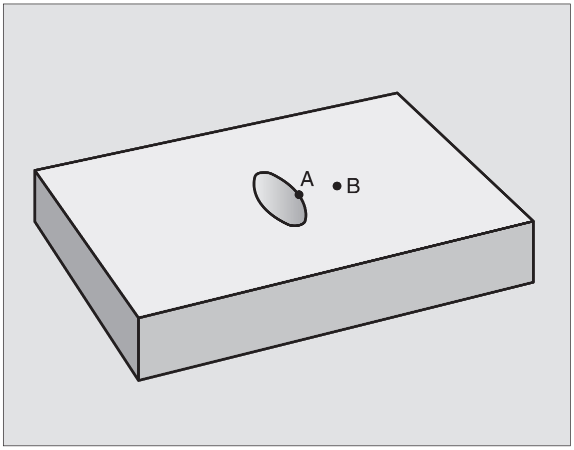
```
N560 G98 L2 *
N570 G01 G41 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

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

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

Surface A:

```
N510 G98 L1 *
N520 G01 G42 X+60 Y+50 *
N530 I+35 J+50 *
N540 G02 X+60 Y+50 *
N550 G98 L0 *
```

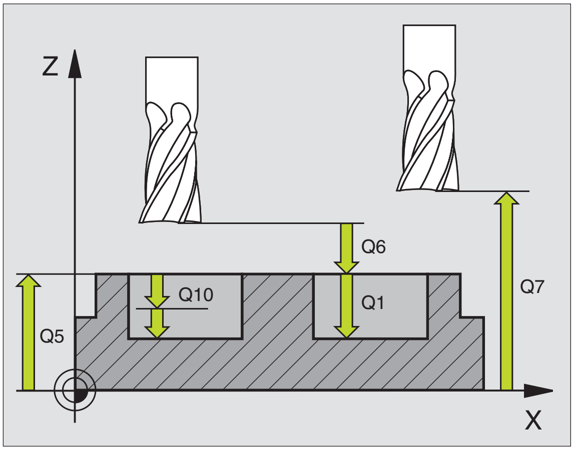


Surface B:

```
N560 G98 L2 *
N570 G01 G42 X+90 Y+50 *
N580 I+65 J+50  *
N590 G02 X+90 Y+50 *
N600 G98 L0 *
```

CONTOUR DATA (Cycle G120)

Machining data for the subprograms describing the subcontours are entered in Cycle G120.



Before programming, note the following:

Cycle G120 is DEF active which means that it becomes effective as soon as it is defined in the part program.

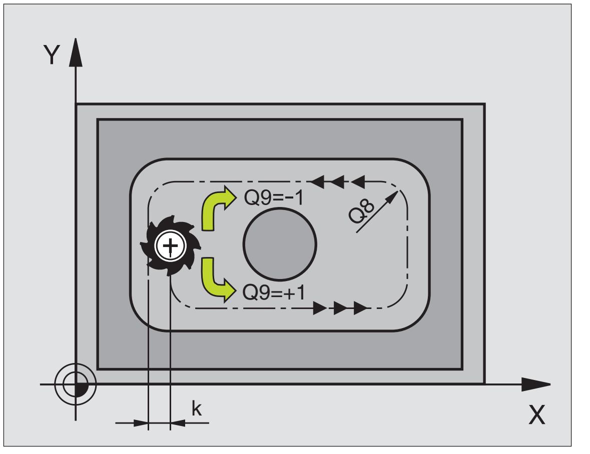
The algebraic sign for the depth parameter determines the working direction.

The machining data entered in Cycle G120 are valid for Cycles G121 to G124.

If you are using the SL cycles in Q parameter programs, the cycle parameters Q1 to Q19 cannot be used as program parameters.

120 CONTOUR DATA

- ▶ Milling depth Q1 (incremental value): Distance between workpiece surface and pocket floor
- ▶ Path overlap factor Q2: Q2 x tool radius = stepover factor k
- ▶ Finishing allowance for side Q3 (incremental value): Finishing allowance in the working plane
- ▶ Finishing allowance for floor Q4 (incremental value): Finishing allowance in the tool axis
- ▶ Workpiece surface coordinate Q5 (absolute value): Absolute coordinate of the workpiece surface



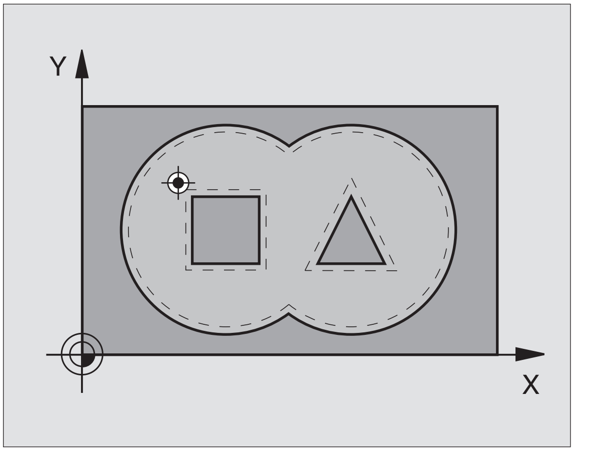
- Set-up clearance Q6 (incremental value): Distance between tool tip and workpiece surface
- Clearance height Q7 (absolute value): Absolute height at which the tool cannot collide with the workpiece (for intermediate positioning and retraction at the end of the cycle)
- Inside corner radius Q8: Inside „corner" rounding radius; entered value is referenced to the tool midpoint path
- Direction of rotation ? Clockwise = -1 Q9: Machining direction for pockets
  - Clockwise (Q9 = -1 up-cut milling for pocket and island)
  - Counterclockwise (Q9 = +1 climb milling for pocket and island)

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

Example NC block:

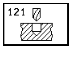
```
N57  G120 Q1=-20 Q2=1 Q3=+0.2 Q4=+0.1 Q5=+0 Q6=+2
     Q7=+50 Q8=0.5 Q9=+1*
```

PILOT DRILLING (Cycle G121)



Process
Same as Cycle G83 Pecking (see section 8.3 "Drilling Cycles").

Application
Cycle G121 is for PILOT DRILLING of the cutter infeed points. It accounts for the allowance for side and the allowance for floor as well as the radius of the rough-out tool. The cutter infeed points also serve as starting points for roughing.

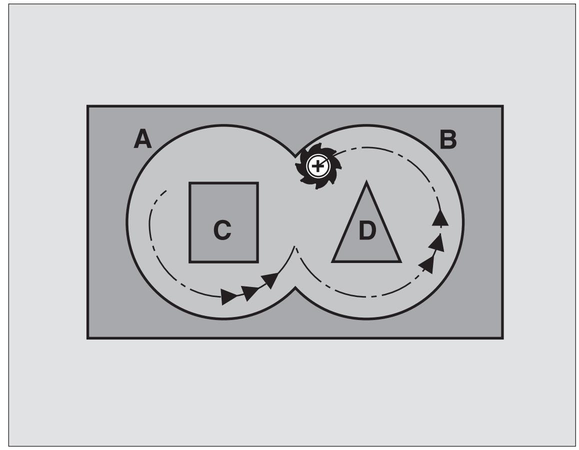


- Plunging depth Q10 (incremental value): Dimension by which the tool drills in each infeed (negative sign for negative working direction)
- Feed rate for plunging Q11: Traversing speed of the tool in mm/min during drilling
- Rough-out tool number Q13: Tool number of the roughing mill

Example NC block:

```
N58  G121 Q10=+5 Q11=100 Q13=1*
```

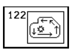
ROUGH-OUT (Cycle G122)



1. The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.
2. In the first plunging depth, the tool mills the contour from inside outward at the milling feed rate.
3. First the island contours (*C* and *D* in the figure at right) are rough-milled until the pocket contour (*A*, *B*) is approached.
4. Then the pocket contour is rough-milled and the tool is retracted to the clearance height.

Before programming, note the following:

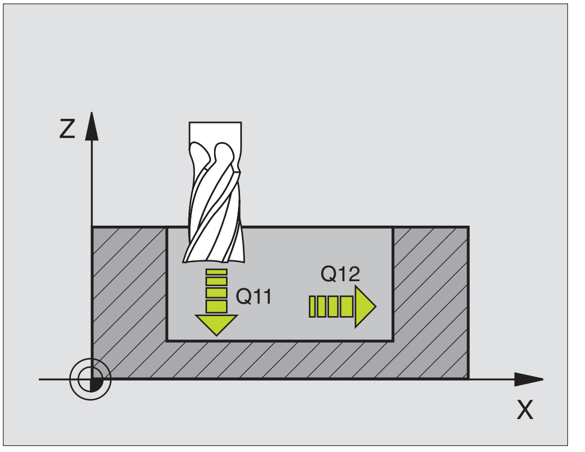
This cycle requires a center-cut end mill (ISO 1641) or pilot drilling with Cycle G121.



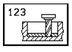
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed
- Feed rate for plunging Q11: Traversing speed of the tool in mm/min during penetration
- Feed rate for milling Q12: Traversing speed for milling in mm/min
- Coarse roughing tool number Q18: Number of the tool with which the TNC has already coarse-roughed the contour. If the contour has not been coarse-roughed, enter zero. If you enter a value other than 0, the TNC will only rough-out the portion that could not be machined with the coarse roughing tool.
  If the portion that is to be fine-roughed cannot be approached from the side, the TNC will mill in a reciprocating plunge-cut. For this purpose, you must enter the tooth length LCUTS and the maximum plunge angle ANGLE of the tool in the tool table TOOL.T (see section 5.2 "Tool Data"). The TNC will otherwise generate an error message.
- Reciprocation feed rate Q19: Traversing speed of the tool in mm/min during reciprocating plunge-cut

FLOOR FINISHING (Cycle G123)

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



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



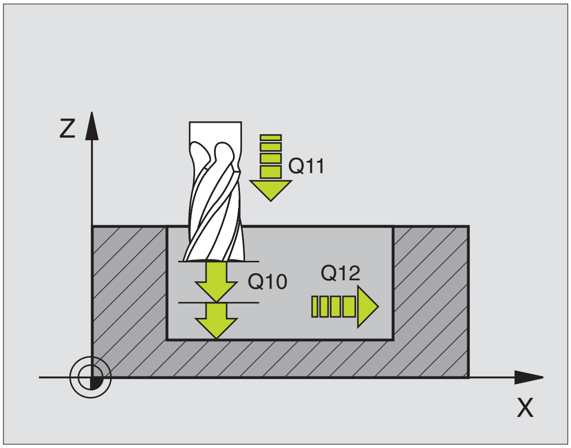
- Feed rate for plunging: Traversing speed of the tool during penetration
- Feed rate for milling Q12: Traversing speed for milling

Example NC block:

```
N60  G123 Q11=100 Q12=350*
```

SIDE FINISHING (Cycle G124)

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



Before programming, note the following:

The sum of allowance for side (Q14) and the radius of the finish mill must be smaller than the sum of allowance for side (Q3, Cycle G120) and the radius of the rough mill.

This calculation also holds if you run Cycle G124 without having roughed out with Cycle G122; in this case, enter "0" for the radius of the rough mill.

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

- Direction of rotation ? Clockwise = –1 Q9: Direction of machining:
  +1: Counterclockwise
  –1: Clockwise
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed
- Feed rate for plunging Q11: Traversing speed of the tool during penetration
- Feed rate for milling Q12: Traversing speed for milling
- Finishing allowance for side Q14 (incremental value): Enter the allowed material for several finish-milling operations. If you enter Q14 = 0, the remaining finishing allowance will be cleared.

Example NC block:

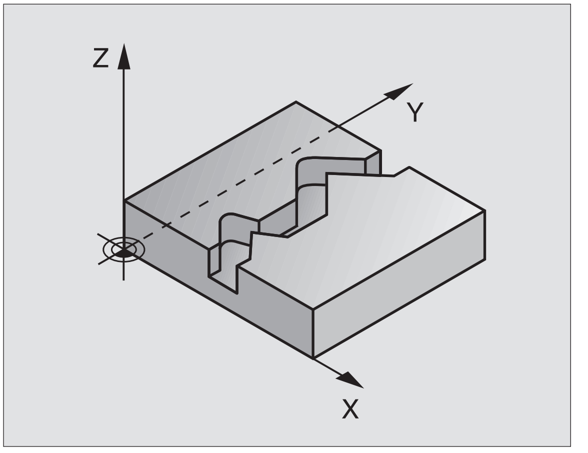
```
N61  G124 Q9=+1 Q10=+5 Q11=100 Q12=350 Q14=+0*
```

CONTOUR TRAIN (Cycle G125)

In conjunction with Cycle G37 CONTOUR GEOMETRY, this cycle facilitates the machining of open contours (where the starting point of the contour is not the same as its end point).

Cycle G125 CONTOUR TRAIN offers considerable advantages over machining an open contour using positioning blocks:

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



Before programming, note the following:

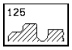
The algebraic sign for the depth parameter determines the working direction.

The TNC takes only the first label of Cycle G37 CONTOUR GEOMETRY into account.

The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle.

Cycle G120 CONTOUR DATA is not required.

Positions that are programmed in incremental dimensions immediately after Cycle G125 are referenced to the position of the tool at the end of the cycle.



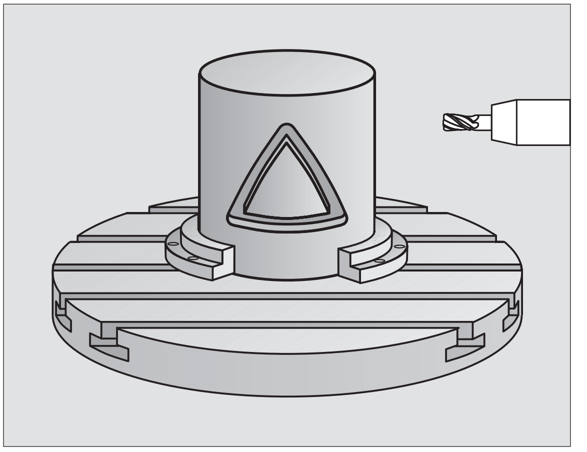
- Milling depth Q1 (incremental value): Distance between workpiece surface and contour floor
- Finishing allowance for side Q3 (incremental value): Finishing allowance in the working plane
- Workpiece surface coordinate Q5 (absolute value): Absolute coordinate of the workpiece surface referenced to the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

Example NC block:

```
N62  G125 Q1=-20 Q3=+0 Q5=+0 Q7=+50 Q10=+5 Q11=100
     Q12=350 Q15=+1*
```

CYLINDER SURFACE (Cycle G127)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle G127.

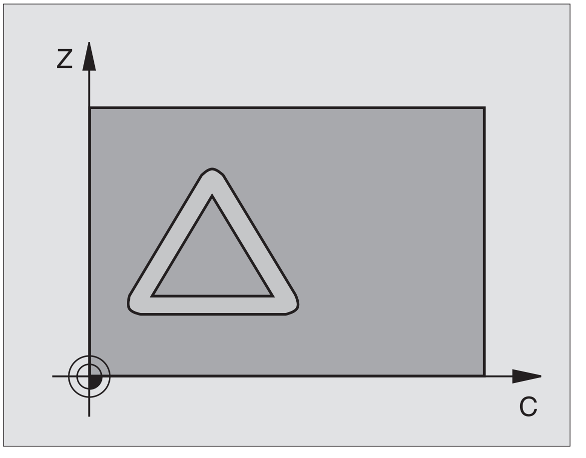


This cycle enables you to program a contour in two dimensions and then roll it onto a cylindrical surface for 3-D machining. The programmed contour is traversed with G40 or with G41/G42.

The contour is described in a subprogram identified in Cycle G37 CONTOUR GEOMETRY.

The subprogram contains coordinates in a rotary axis and in its parallel axis. The rotary axis C, for example, is parallel to the Z axis. The available path functions are G1, G11, G24, G25 and G2/G3/G12/G13 with R.

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



1 The TNC positions the tool over the cutter infeed point, taking the allowance for side into account.

2 At the first plunging depth, the tool mills along the programmed contour at the milling feed rate Q12.

3 At the end of the contour, the TNC returns the tool to the setup clearance and returns to the point of penetration;

4 Steps 1 to 3 are repeated until the programmed milling depth Q1 is reached.

5 Then the tool moves to the setup clearance.

Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle.

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

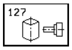
This cycle requires a center-cut end mill (ISO 1641).

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message „Contour programming error“ is output, set MP 810.x = 0.



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

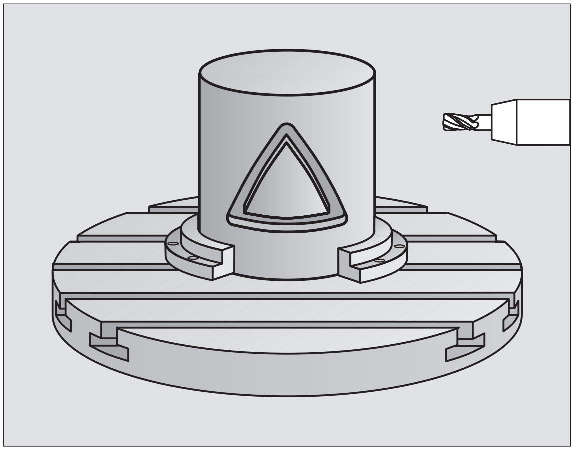
Example NC block:

```
N63  G127 Q1=-8 Q3=+0 Q6=+0 Q10=+3 Q11=100 Q12=350
          Q16=25 Q17=0*
```

CYLINDER SURFACE slot milling (Cycle G128, only in TNC 426, TNC 430 with NC software 280 474-xx)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle G128.

This cycle enables you to program a guide notch in two dimensions and then transfer it onto a cylindrical surface. Unlike Cycle G127, with this cycle the TNC adjusts the tool so that, with radius compensation active, the walls of the slot are always parallel. Program the center-line path of the contour.



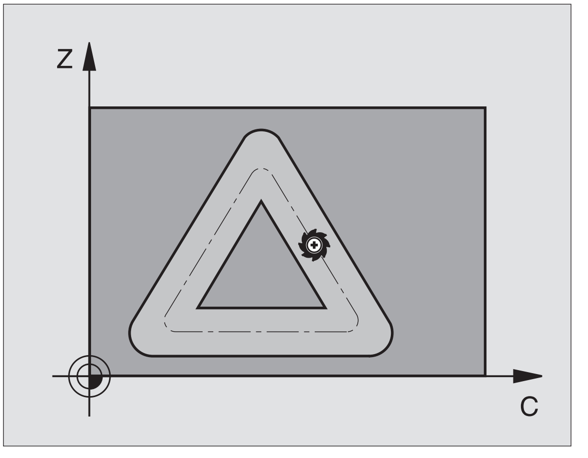
1 The TNC positions the tool over the cutter infeed point.

2 At the first plunging depth, the tool mills along the programmed slot wall at the milling feed rate Q12 while respecting the finishing allowance for the side.

3 At the end of the contour, the TNC moves the tool to the opposite wall and returns to the infeed point.

4 Steps 2 to 3 are repeated until the programmed milling depth Q1 is reached.

5 Then the tool moves to the setup clearance.



Before programming, note the following:

The memory capacity for programming an SL cycle is limited. For example, you can program up to 128 straight-line blocks in one SL cycle.

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

This cycle requires a center-cut end mill (ISO 1641).

If the cutter diameter is smaller than half the slot width, you may want to run Cycle G127 with an R0 tool radius for roughing.

The cylinder must be set up centered on the rotary table.

The tool axis must be perpendicular to the rotary table. If this is not the case, the TNC will generate an error message.

This cycle can also be used in a tilted working plane.

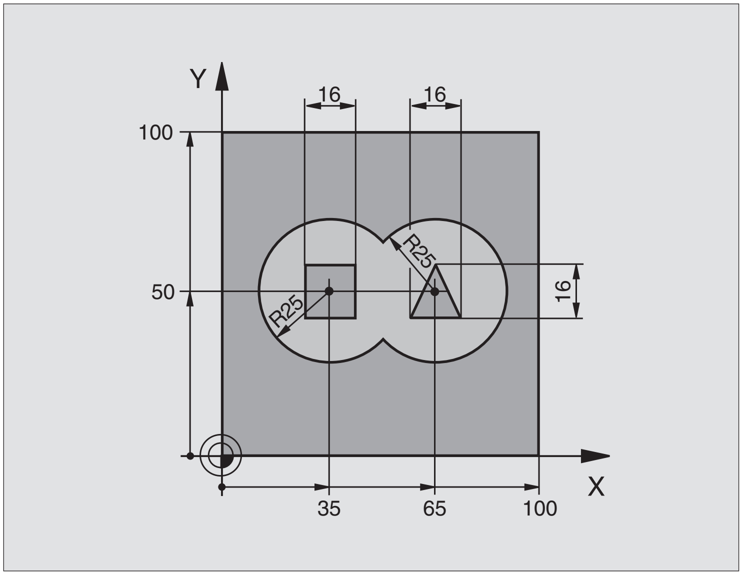
The TNC checks whether the compensated and non-compensated tool paths lie within the display range of the rotary axis, which is defined in Machine Parameter 810.x. If the error message „Contour programming error" is output, set MP 810.x = 0.

- Milling depth Q1 (incremental value): Distance between the cylindrical surface and the floor of the contour
- Finishing allowance for Q3 (incremental value): Finishing allowance in the plane of the unrolled cylindrical surface. This allowance is effective in the direction of the radius compensation.
- Set-up clearance Q6 (incremental value): Distance between the tool tip and the cylinder surface
- Plunging depth Q10 (incremental value): Dimension by which the tool plunges in each infeed
- Feed rate for plunging Q11: Traversing speed of the tool in the tool axis
- Feed rate for milling Q12: Traversing speed of the tool in the working plane
- Radius Q16: Radius of the cylinder on which the contour is to be machined
- Dimension type ? Q17: The dimensions for the rotary axis of the subprogram are given either in degrees (0) or in mm/inches (1)
- Slot width Q20: Width of the slot to be machined

Example NC block:

```
N63  G128 Q1=-8 Q3=+0 Q6=+0 Q10=+3 Q11=100 Q12=350
     Q16=25 Q17=0 Q20=12*
```

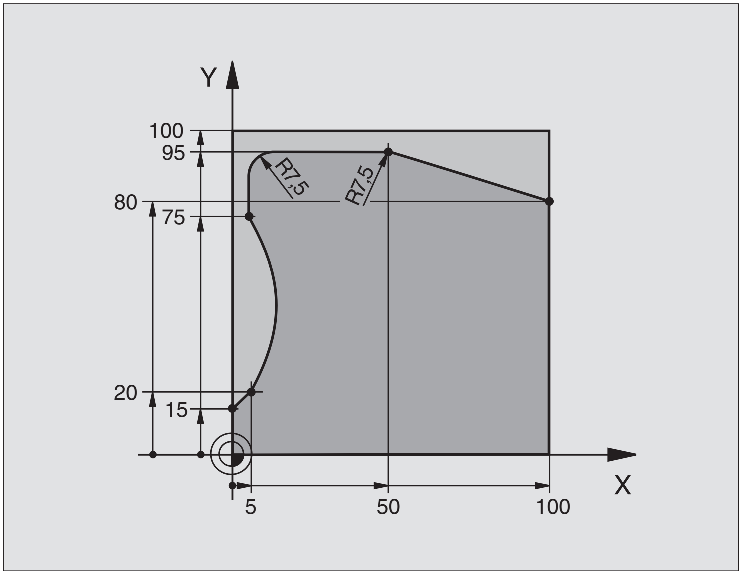
Example: Pilot drilling, roughing-out and finishing overlapping contours



%C21 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+6 *	Tool definition: drill
N40 G99 T2 L+0 R+6 *	Define the tool for roughing/finishing
N50 T1 G17 S4000 *	Call the drilling tool
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 P02 2 P03 3 P04 4 *	Define contour subprogram
N80 G120 Q1=-20 Q2=1 Q3=+0.5 Q4=+0.5	Define general machining parameters
Q5=+0 Q6=+2 Q7=+100 Q8=+0.1 Q9=-1 *	
N90 G121 Q10=+5 Q11=250 Q13=2 *	Cycle definition: PILOT DRILLING
N100 G79 M3 *	Cycle call: PILOT DRILLING
N110 Z+250 M6 *	Tool change
N120 T2 G17 S3000 *	Call tool for roughing/finishing
N130 G122 Q10=+5 Q11=100 Q12=350 *	Cycle definition: ROUGH-OUT
N140 G79 M3 *	Cycle call: ROUGH-OUT
N150 G123 Q11=100 Q12=200 *	Cycle definition: FLOOR FINISHING
N160 G79 *	Cycle call: FLOOR FINISHING
N170 G124 Q9=+1 Q10=+5 Q11=100 Q12=400	Cycle definition: SIDE FINISHING
Q14=+0 *	
N180 G79 *	Cycle call: SIDE FINISHING
N190 G00 Z+250 M2 *	Retract in the tool axis, end program
N200 G98 L1 *	Contour subprogram 1: left pocket

N210 I+35 J+50 *	
N220 G01 G42 X+10 Y+50 *	
N230 G02 X+10 *	
N240 G98 L0 *	
N250 G98 L2 *	Contour subprogram 2: right pocket
N260 I+65 J+50 *	
N270 G01 G42 X+90 Y+50 *	
N280 G02 X+90 *	
N290 G98 L0 *	
N300 G98 L3 *	Contour subprogram 3: square left island
N310 G01 G41 X+27 Y+50 *	
N320 Y+58 *	
N330 X+43 *	
N340 Y+42 *	
N350 X+27 *	
N360 G98 L0 *	
N370 G98 L4 *	Contour subprogram 4: triangular right island
N380 G01 G41 X+65 Y+42 *	
N390 X+57 *	
N400 X+65 Y+58 *	
N410 X+73 Y+42 *	
N420 G98 L0 *	
N999999 %C21 G71 *	

Example: Contour train

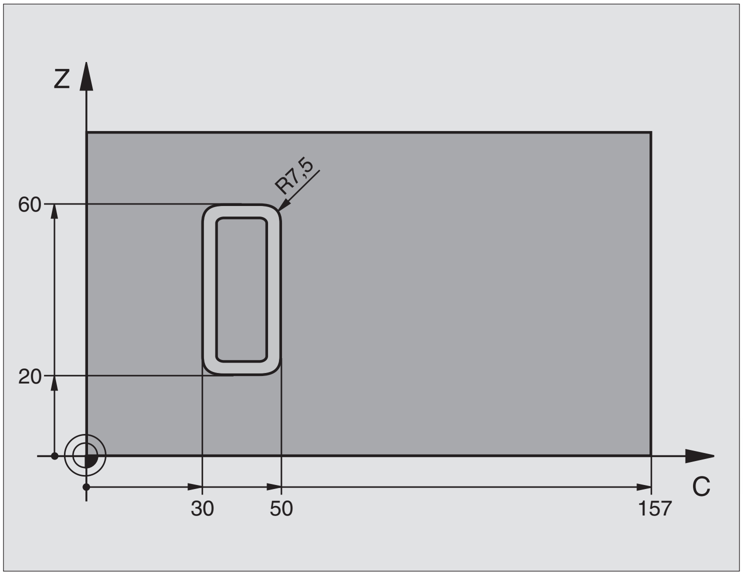


%C25 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	Define the workpiece blank
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+10 *	Define the tool
N50 T1 G17 S2000 *	Tool call
N60 G00 G40 G90 Z+250 *	Retract the tool
N70 G37 P01 1 *	Define contour subprogram
N80 G125 Q1=-20 Q3=+0 Q5=+0 Q7=+250	Define machining parameters
Q10=+5 Q11=100 Q12=200 Q15=+1 *	
N90 G79 M3 *	Call the cycle
N100 G00 G90 Z+250 M2 *	Retract in the tool axis, end program
N110 G98 L1 *	Contour subprogram
N120 G01 G41 X+0 Y+15 *	
N130 X+5 Y+20 *	
N140 G06 X+5 Y+75 *	
N150 G01 Y+95 *	
N160 G25 R7.5 *	
N170 X+50 *	
N180 G25 R7.5 *	
N190 X+100 Y+80 *	
N200 G98 L0 *	
N999999 %C25 G71 *	

Example: Cylinder surface

Cylinder centered on rotary table.

Datum at center of rotary table

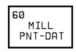
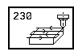
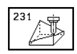


`%C27 G71 *`	
`N10 G99 T1 L+0 R+3.5 *`	Define the tool
`N20 T1 G18 S2000 *`	Call tool, tool axis is Y
`N30 G00 G40 G90 Y+250 *`	Retract the tool
`N40 G37 P01 1 *`	Define contour subprogram
`N50 G127 Q1=-7 Q3=+0 Q6=+2 Q10=+4`	Define machining parameters
`Q11=100 Q12=250 Q16=25 *`	
`N60 C+0 M3 *`	Pre-position rotary table
`N70 G79 *`	Call the cycle
`N80 G00 G90 Y+250 M2 *`	Retract in the tool axis, end program
`N90 G98 L1 *`	Contour subprogram
`N100 G01 G41 C+91.72 Z+20 *`	Data for the rotary axis are entered in degrees
`N110 C+114.65 Z+20 *`	Drawing dimensions are converted from mm to degrees
`N120 G25 R7.5 *`	(157 mm = 360°)
`N130 G91 Z+40 *`	
`N140 G90 G25 R7.5 *`	
`N150 G91 C-45.86 *`	
`N160 G90 G25 R7.5 *`	
`N170 Z+20 *`	
`N180 G25 R7.5 *`	
`N190 C+91.72 *`	
`N200 G98 L0 *`	
`N999999 %C27 G71 *`	

8.8 Cycles for Face Milling

The TNC offers four cycles for machining surfaces with the following characteristics:

- Created by digitizing or with a CAD/CAM system
- Flat, rectangular surfaces
- Flat, oblique-angled surfaces
- Surfaces that are inclined in any way
- Twisted surfaces

Cycle	Soft key
G60 RUN DIGITIZED DATA For multipass milling of digitized surface data in several infeeds (not TNC 410)	
G230 MULTIPASS MILLING For flat rectangular surfaces	
G231 RULED SURFACE For oblique, inclined or twisted surfaces	

RUN DIGITIZED DATA (Cycle G60, not TNC 410)

1 From the current position, the TNC positions the tool in rapid traverse in the tool axis to the set-up clearance above the MAX point that you have programmed in the cycle.

2 The tool then moves at rapid traverse in the working plane to the MIN point you have programmed in the cycle.

3 From this point, the tool advances to the first contour point at the feed rate for plunging.

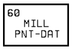
4 The TNC subsequently processes all points that are stored in the digitizing data file at the feed rate for milling. If necessary, the TNC retracts the tool between machining operations to set-up clearance if specific areas are to be left unmachined.

5 At the end of the cycle, the tool is retracted in rapid traverse to set-up clearance.

Before programming, note the following:

You can use Cycle G60 to run digitizing data and PNT files.

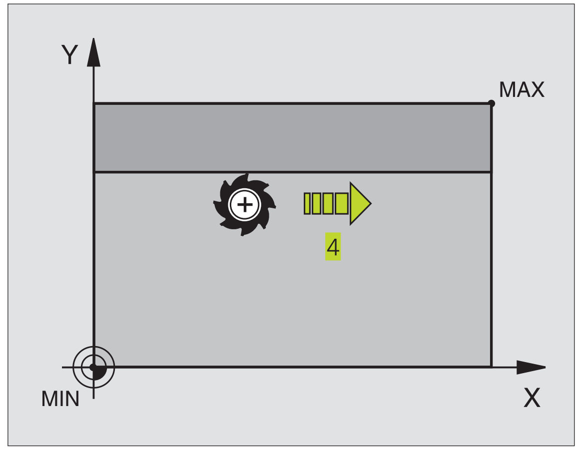
If you want to run PNT files in which no tool axis coordinate is programmed, the milling depth is derived from the programmed MIN point in the tool axis.

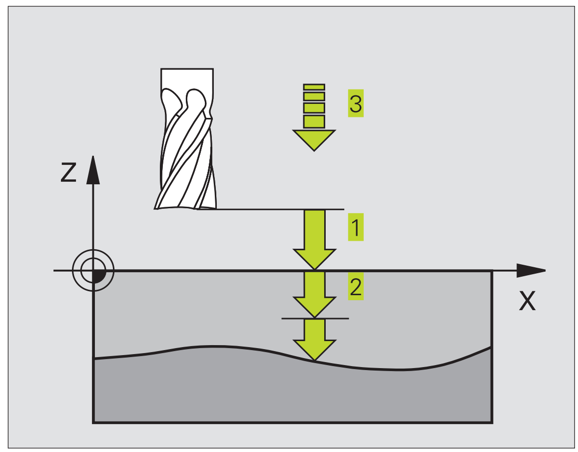


- PGM Name digitizing data: Enter the name of the file in which the digitizing data is stored. If the file is not stored in the current directory, enter the complete path. If you wish to run a point table, you must also enter the extension .PNT
- Min. point range: Lowest coordinates (X, Y and Z coordinates) in the range to be milled
- Max. point range: Highest coordinates (X, Y and Z coordinates) in the range to be milled
- Setup clearance 1 (incremental value): Distance between tool tip and workpiece surface for tool movements in rapid traverse
- Plunging depth 2 (incremental value): Dimension by which the tool is advanced in each infeed
- Feed rate for plunging 3: Traversing speed of the tool in mm/min during penetration
- Feed rate for milling 4: Traversing speed of the tool in mm/min while milling
- Miscellaneous function M: Optional entry of a miscellaneous function, for example M13

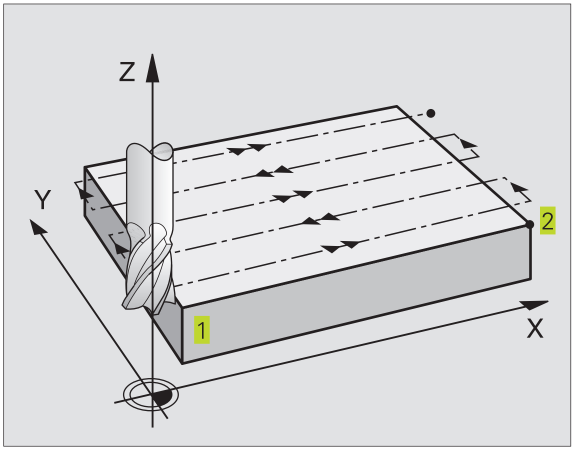
Example NC block:

```
N64  G60 P01 BSP.I P02 X+0 P03 Y+0 P04 Z-20
     P05 X+100 P06 Y+100 P07 Z+0 P08  2 P09 +5
     P10 100 P11 350 P12 M13*
```





MULTIPASS MILLING (Cycle G230)



1 From the current position, the TNC positions the tool at 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 at rapid traverse 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 direction of the first axis.

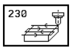
6 Multipass milling is repeated until the programmed surface has been completed.

7 At the end of the cycle, the tool is retracted in rapid traverse to set-up clearance.

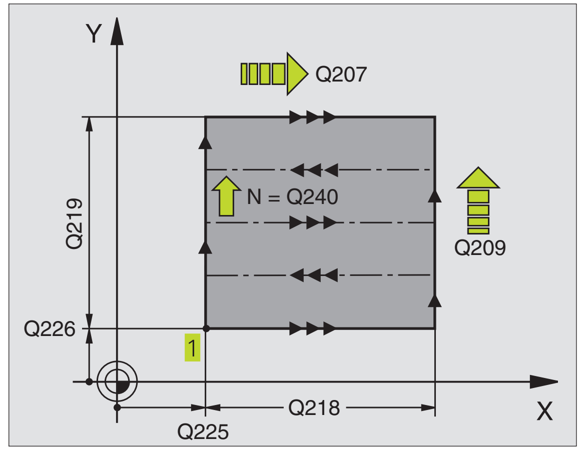
Before programming, note the following:

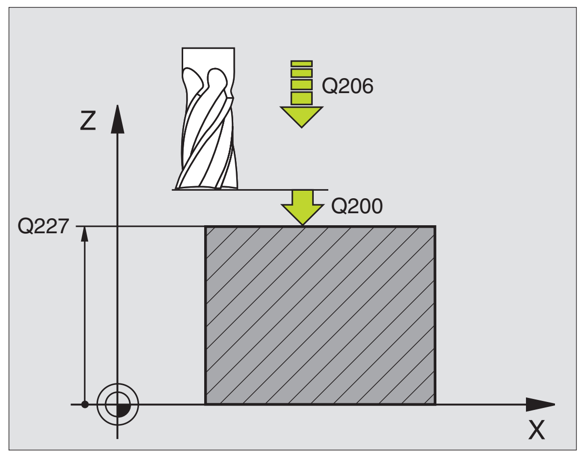
From the current position, the TNC positions the tool at the starting point 1, first in the working plane and then in the tool axis.

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.





Example NC block:

```
N71  G230 Q225=+10 Q226=+12 Q227=+2.5
          Q218=150 Q219=75 Q240=25 Q206=150
          Q207=500 Q209=200 Q200=2*
```

RULED SURFACE (Cycle 231)

1 From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1.

2 The tool subsequently advances to the stopping point 2 at the feed rate for milling.

3 From this point, the tool moves in rapid traverse by the tool diameter in the positive tool axis direction, and then back to starting point 1.

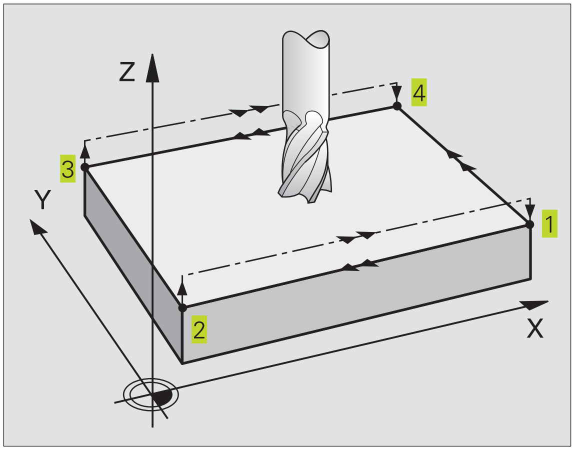
4 At the starting position 1 the TNC moves the tool back to the the last traversed Z value.

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.

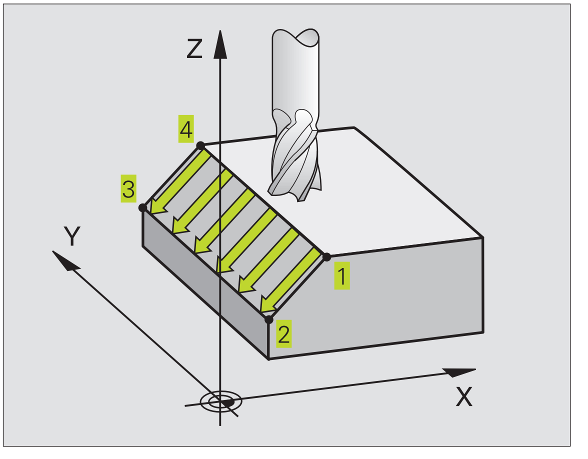


Cutting motion

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 point 1 / 2 to point 3 / 4 . You can position point 1 in any corner of the surface to be machined.

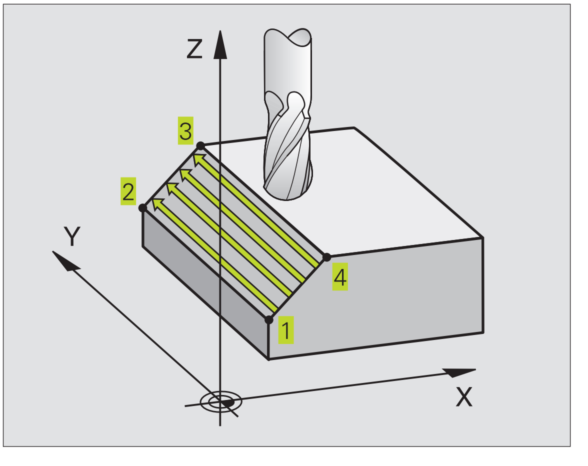
If you are using an end mill for the machining operation, you can optimize the surface finish in the following ways

- 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.

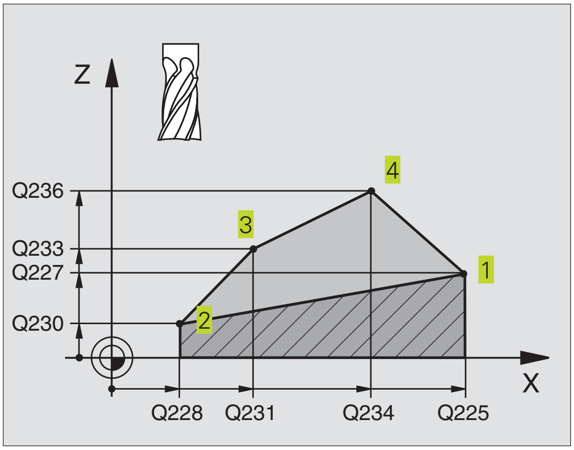


Before programming, note the following:

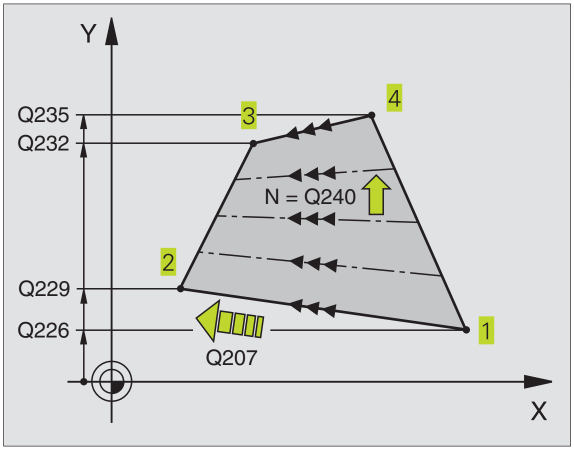
From the current position, the TNC positions the tool in a linear 3-D movement to the starting point 1. Pre-position the tool in such a way that no collision between tool and clamping devices can occur.

The TNC moves the tool with radius compensation G40 to the programmed positions.

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



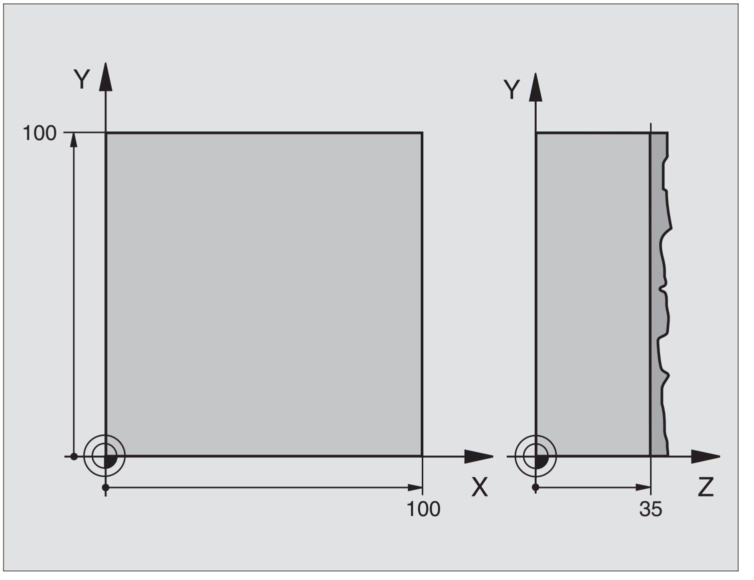
- Starting point in 1st axis Q225 (absolute value): Starting point 1 of the surface to be face milled in the working plane
- Starting point in 2nd axis Q226 (absolute value): Starting point coordinate 1 of the surface to be face-milled in the secondary axis of the working plane
- Starting point in 3rd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



Example NC block:

```
N72  G231 Q225=+0 Q226=+5 Q227=-2
     Q228=+100 Q229=+15 Q230=+5 Q231=+15
     Q232=+125 Q233=+25 Q234=+85 Q235=+95
     Q236=+35 Q240=40 Q207=500*
```

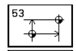
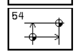
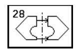
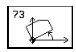
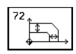
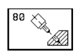
Example: Multipass milling



`%C230 G71`	
`N10 G30 G17 X+0 Y+0 Z+0 *`	Define the workpiece blank
`N20 G31 G90 X+100 Y+100 Z+40 *`	
`N30 G99 T1 L+0 R+5 *`	Define the tool
`N40 T1 G17 S3500 *`	Tool call
`N50 G00 G40 G90 Z+250 *`	Retract the tool
`N60 G230 Q225=+0 Q226=+0 Q227=+35`	Cycle definition: MULTIPASS MILLING
`Q218=100 Q219=100 Q240=25 Q206=250`	
`Q207=400 Q209=150 Q200=2 *`	
`N70 X-25 Y+0 M03 *`	Pre-position near the starting point
`N80 G79 *`	Call the cycle
`N90 G00 G40 Z+250 M02 *`	Retract in the tool axis, end program
`N999999 %C230 G71 *`	

8.9 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
G53/G54 DATUM SHIFT For shifting contours directly within the program or with datum tables	 
G28 MIRROR IMAGE For mirroring contours	
G73 ROTATION For rotating contours in the working plane	
G72 SCALING FACTOR For increasing or reducing the size of contours	
G80 WORKING PLANE For executing machining operations in a tilted coordinate system on machines with swivel heads and/or tilting tables (not TNC 410)	

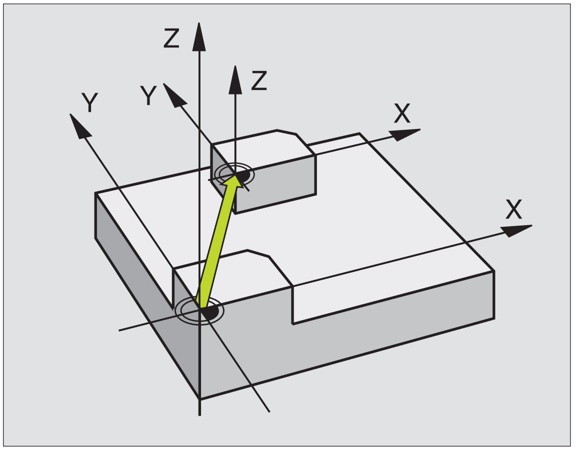
Effect of coordinate transformations
A coordinate transformation becomes effective as soon as it is defined — it is not called. It remains in effect until it is changed or canceled.

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 N999999 %... block (depending on machine parameter 7300)
- Select a new program

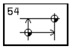
DATUM SHIFT (Cycle G54)

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.

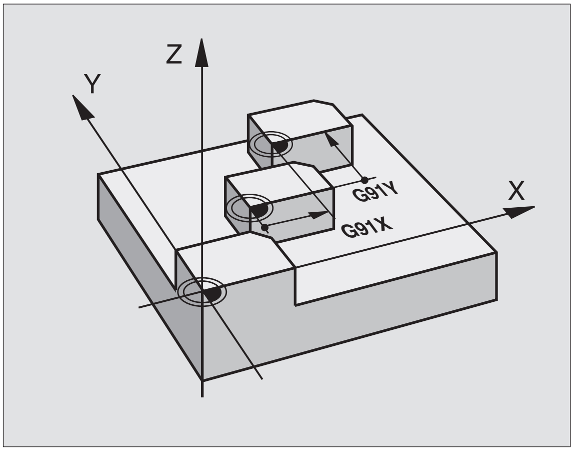


▶ Datum shift: Enter the coordinates of the new datum. Absolute values are referenced to the manually set workpiece datum. Incremental values are always referenced to the datum which was last valid — this can be a datum which has already been shifted.

Additionally withTNC 410:

REF

▶ REF: Press the REF soft key 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.

Graphics (notTNC 410)

If you program a new workpiece blank after a datum shift, you can use machine parameter 7310 to determine whether the blank is referenced to the current datum or to the original datum. Referencing a new BLK FORM to the current datum enables you to display each part in a program in which several pallets are machined.

Status Displays

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

Example NC blocks:

```
N72  G54 G90 X+25 Y-12.5 Z100*
```

or

```
N72  G54 G90 REF X+25 Y-12.5 Z100*
```

DATUM SHIFT with datum tables (Cycle G53)

Datums from a datum table can be referenced either to the current datum or to the machine datum (depending on machine parameter 7475).

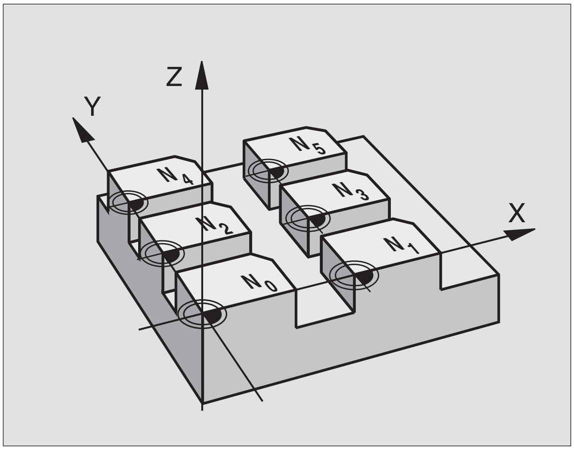
The coordinate values from datum tables are only effective with absolute coordinate values.

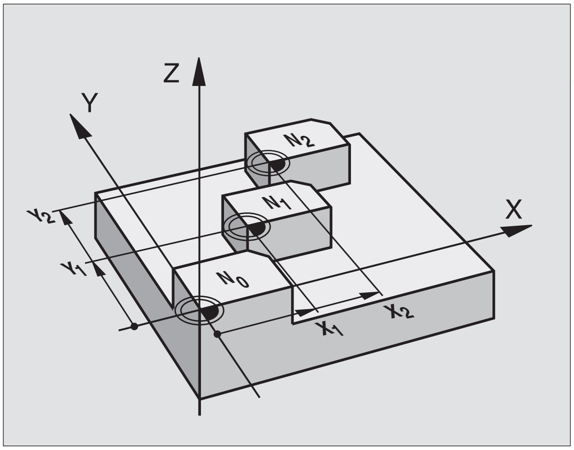
For the TNC 426, TNC 430:

If you are using the interactive programming graphics with the datum tables, you must select the desired datum table (status S) in the Test Run mode of operation before starting the programming graphics.

New lines can only be inserted at the end of the table.

If you are working with only one datum table, be sure to activate the correct datum in the program run modes of operation.





Application

Datum tables are used for

- frequently recurring machining sequences at various locations on the workpiece
- frequent use of the same datum shift

Within a program, you can either program datum points directly in the cycle definition or call them from a datum table.

▶ Datum shift: Enter the number of the datum from the datum table or a Q parameter. If you enter a Q parameter, the TNC activates the datum number found in the Q parameter. To activate the datum table, see the instructions given later in this chapter.

Cancellation

- Call a datum shift to the coordinates X=0; Y=0 etc. from a datum table.
- Execute a datum shift to the coordinates X=0; Y=0 etc. directly via cycle definition.

Status Displays

If datums in the table are referenced to the machine datum, then:

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

Example NC block:

```
N72  G53 #12*
```

Editing a datum table with the TNC 410

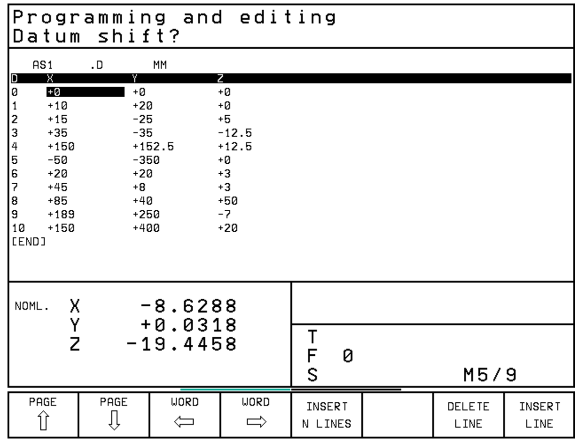
Select the datum table in the PROGRAMMING AND EDITING mode of operation.

PGM MGT

▶ To call the file manager, press the PGM MGT key — see also section 4 "File Management" for more information.

▶ To select an already existing datum table, move the highlight to the desired table and confirm with the ENT key

▶ To open a new datum table, enter a new file name and confirm with the ENT key. Press the „.D" soft key to open the table



Editing a datum table with TNC 426, TNC 430

Select the datum table in the PROGRAMMING AND EDITING mode of operation.

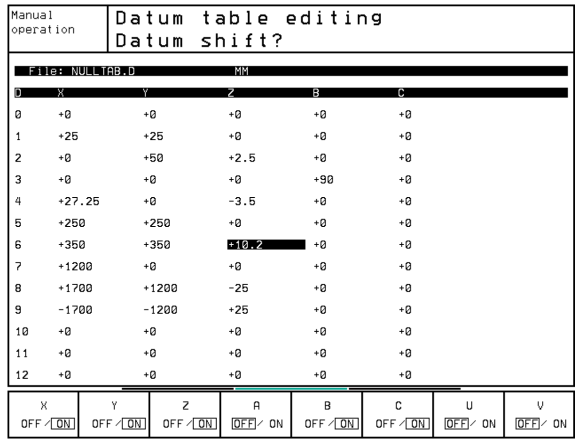
PGM MGT

▶ To call the file manager, press the PGM MGT key — see also section 4 "File Management" for more information.

▶ Display the datum tables: Press the soft keys SELECT TYPE and SHOW .D

▶ Select the desired table or enter a new file name.

▶ Edit the file. The soft-key row comprises the following functions for editing:



Function	Soft key
Select beginning of table	BEGIN ⇧
Select end of table	END ⇩
Go to the previous page	PAGE ⇧
Go to the next page	PAGE ⇩
Insert line	INSERT LINE
Delete line	DELETE LINE
Confirm the entered line and go to the beginning of the next line (not TNC 410)	NEXT LINE
Insert a certain number of lines	APPEND N LINES
Move the highlight one column to the left (not TNC 410)	WORD ⇦
Move the highlight one column to the right (not TNC 410)	WORD ⇨

With the function „Actual position capture“ the TNC stores the position of that axis in the header of the table which is above the marked value (not TNC 410).

Configuring datum tables (notTNC 410)
On the second and third soft-key rows you can define for each datum table the axes for which you wish to set the datums. In the standard setting all of the axes are active. If you wish to exclude an axis, set the corresponding soft key to OFF. The TNC then deletes that column from the datum table.

To leave a datum table
Select a different type of file in file management and choose the desired file.

Activating a datum table for program run (TNC 410)
With the TNC 410 you must use the function %:TAB: in the NC program to select the datum table from which the TNC is to take the datums:

PGM CALL

- To select the functions for program call, press the PGM CALL key.
- Press the DATUM TABLE soft key.
- Enter the name of the datum table, then confirm with the END key.

Example NC block:

```
N72 %:TAB: "NAMES"*
```

Activating a datum table for program run (TNC 426, TNC 430)
With the TNC 426, TNC 430 you must manually activate the datum table in a program run mode of operation:

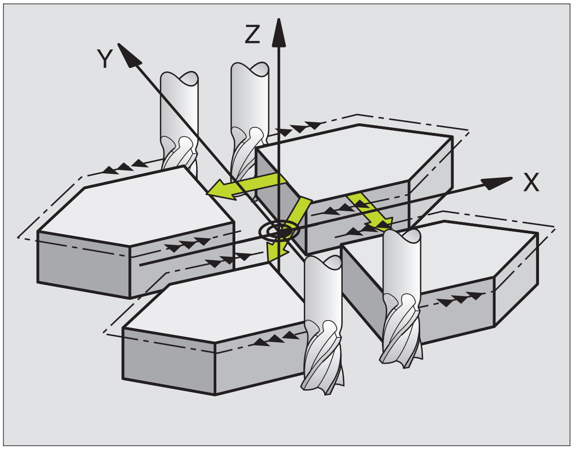
- Select a program run mode, e.g., Program Run, Full Sequence

PGM MGT

- To call the file manager, press the PGM MGT key — see also section 4 “File Management” for more information.
- To select an already existing datum table, move the highlight to the desired table and confirm with the ENT key. The TNC indicates with M the selected table in the status field.

MIRROR IMAGE (Cycle G28)

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

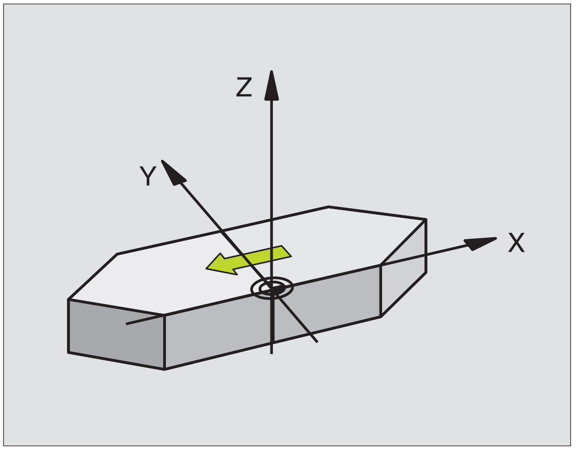


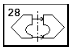
Effect
The MIRROR IMAGE cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active mirrored axes are shown in the additional status display.

- If you mirror only one axis, the machining direction of the tool is reversed (except in fixed cycles).
- If you mirror two axes, the machining direction remains the same.

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

- If the datum lies on the contour to be mirrored, the element simply flips over — 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.



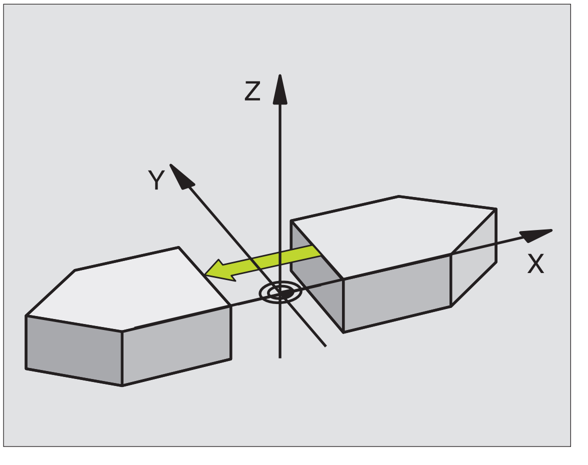


▶ Mirror image axis ?: Enter the axis to be mirrored. You can mirror all axes, including rotary axes, except for the spindle axis and its auxiliary axes.

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

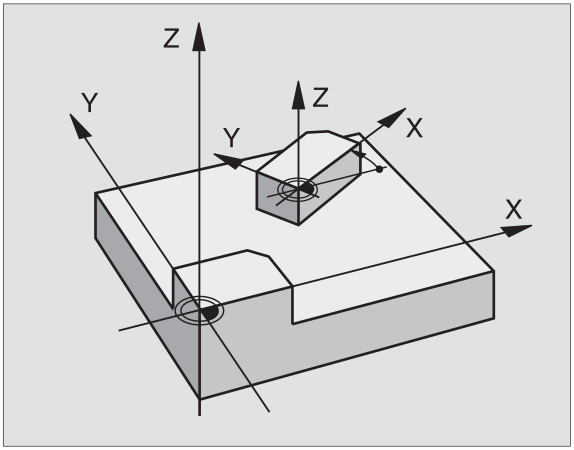
Example NC block:

```
N72 G28 X Y*
```



ROTATION (Cycle G73)

The TNC can rotate the coordinate system about the active datum in the working plane within a program.



Effect

The ROTATION cycle becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active rotation angle is shown in the additional status display.

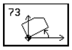
Reference axis for the rotation angle:

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

Before programming, note the following:

An active radius compensation is canceled by defining Cycle G73 and must therefore be reprogrammed, if necessary.

After defining Cycle G73, you must move both axes of the working plane to activate rotation for all axes.



- Rotation: Enter the rotation angle H in degrees (°). Input range: –360° to +360° (absolute G90 before H or incremental G91 before H).

Cancellation

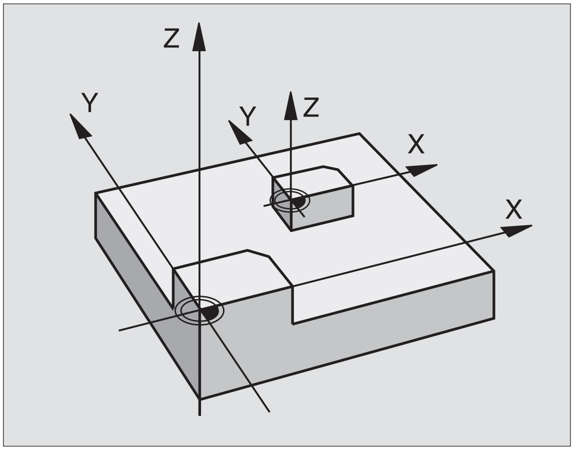
Program the G73 ROTATION cycle once again with a rotation angle of 0°.

Example NC block:

```
N72  G73 G90 H+25*
```

SCALING FACTOR (Cycle G72)

The TNC can increase or reduce the size of contours within a program, enabling you to program shrinkage and oversize allowances.



Effect

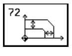
The SCALING FACTOR becomes effective as soon as it is defined in the program. It is also effective in the Positioning with MDI mode of operation. The active scaling factor is shown in the additional status display.

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 F. The TNC multiplies the coordinates and radii by the F factor (as described under "Activation" above)

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

Reduction F less than 1 (down to 0.000 001)

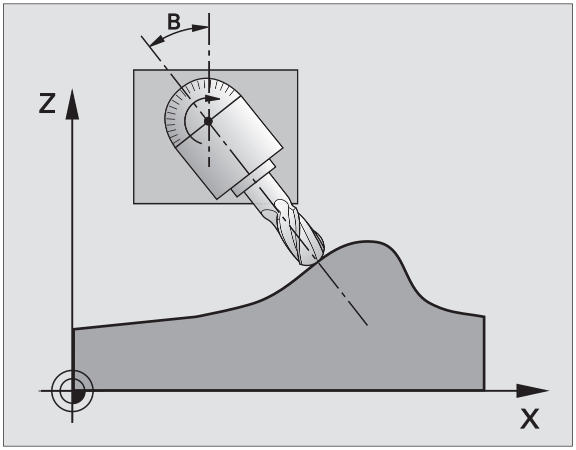
Cancellation

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

Example NC block:

```
N72  G72 F0.980000*
```

WORKING PLANE (Cycle G80, not TNC 410)



The functions for tilting the working plane are interfaced to the TNC and the machine tool by the machine tool builder. With some swivel heads and tilting tables, the machine tool builder determines whether the entered angles are interpreted as coordinates of the tilt axes or as mathematical angles of a tilted plane. Your machine manual provides more detailed information.

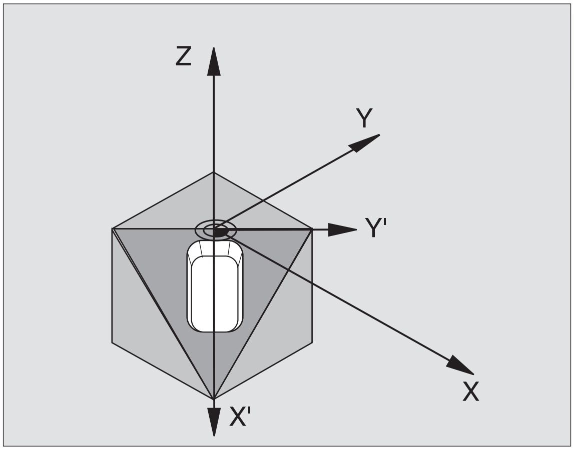
The working plane is always tilted around the active datum.

The fundamentals of this TNC function are described in section 2.5 "Tilting the Working Plane." It is important that you read through this section thoroughly.

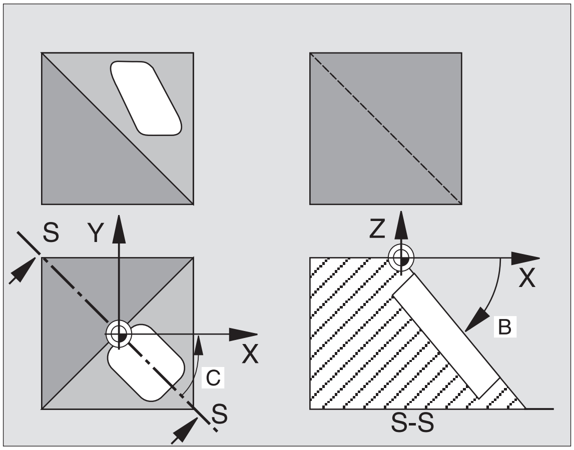
Effect

In Cycle G80 you define the position of the working plane - i.e. the position of the tool axis referenced to the machine coordinate system - by entering tilt angles. There are two ways to determine the position of the working plane:

- Enter the slope of the tilted axes directly (see figure at upper right)
- Describe the position of the working plane using up to 3 rotations (spatial angle) of the **machine-referenced** coordinate system. The required spatial angle can be calculated by cutting a perpendicular line through the tilted working plane and considering it from the axis around which you wish to tilt (see figures at center right and at bottom right). With two spatial angles, every tool position in space can be defined exactly.



Note that the position of the tilted coordinate system, and therefore also all movement in the tilted system, are dependent on your description of the tilted plane.



If you program the position of the working plane via spatial angles, the TNC will calculate the required angle positions of the tilted axes automatically and will store these in the parameters Q120 (A-axis) to Q122 (C-axis). If two solutions are possible, the TNC will choose the shorter path from the zero position of the rotary axes.

The axes are always rotated in the same sequence for calculating the tilt of the plane: The TNC first rotates the A axis, then the B axis, and finally the C axis.

Cycle G80 becomes effective as soon as it is defined in the program. As soon as you move an axis in the tilted system, the compensation for this specific axis is activated. You have to move all axes to activate compensation for all axes.

If you set the function TILTING program run to ACTIVE in the Manual Operation mode (see section 2.5 "Tilting the Working Plane"), the angular value entered in this menu is overwritten by Cycle G80 WORKING PLANE.

- Tilt axis and tilt angle: The axes of rotation together with the associated tilt angles. The rotary axes A, B and C are programmed using soft keys.

If the TNC automatically positions the rotary axes, you can enter the following parameters

- Feed rate ? F=: Traverse speed of the rotary axis during automatic positioning
- Set-up clearance ? (incremental): the TNC positions the tilting head so that the position that results from the extension of the tool by the set-up clearance does not change relative to the workpiece.

Cancellation

To cancel the tilt angle, redefine the WORKING PLANE cycle and enter an angular value of 0° for all axes of rotation. You must then program the WORKING PLANE cycle once again by answering the dialog question with the NO ENT key to disable the function.

Positioning an axis of rotation

> The machine tool builder determines whether Cycle G80 positions the axes of rotation automatically or whether they must be pre-positioned in the program. Your machine manual provides more detailed information.

If the axes are positioned automatically in Cycle G80:

- The TNC can position only controlled axes
- In order for the tilted axes to be positioned, you must enter a feed rate and a set-up clearance in addition to the tilting angles, during cycle definition.
- You can use only preset tools (with the full tool length defined in the G99 block or in the tool table)
- The position of the tool tip as referenced to the workpiece surface remains nearly unchanged after tilting
- The TNC tilts the working plane at the last programmed feed rate. The maximum feed rate that can be reached depends on the complexity of the swivel head or tilting table.

Example NC blocks

`N50 G00 G40 Z+100 *`	
`N60 X+25 Y+10 *`	
`N70 G01 A+15 F1000 *`	Positioning an axis of rotation
`N80 G80 A+15 *`	Define the angle for calculation of the compensation
`N90 G00 G40 Z+80 *`	Activate compensation for the tool axis
`N100 X-7.5 Y-10 *`	Activate compensation for the working plane

Position display in the tilted system

On activation of Cycle G80, the displayed positions (ACTL. and NOML.) and the datum indicated in the additional status display are referenced to the tilted coordinate system. The positions displayed immediately after cycle definition may not be the same as the coordinates of the last programmed position before Cycle G80.

Workspace monitoring

The TNC monitors only those axes in the tilted coordinate system that are moved. If any of the software limit switches is traversed the TNC will display an error message.

Positioning in a tilted coordinate system

With the miscellaneous function M130 you can move the tool, while the coordinate system is tilted, to positions that are referenced to the non-tilted coordinate system (see section "7.3 Miscellaneous Functions for Coordinate Data").

Combining coordinate transformation cycles

When combining coordinate transformation cycles, always make sure the working plane is swiveled around the active datum. You can program a datum shift before activating Cycle G80. In this case, you are shifting the "machine-based coordinate system."

If you program a datum shift after having activated Cycle G80, you are shifting the "tilted coordinate system."

Important: When resetting the cycles, use the reverse sequence used for defining the them:

1. Activate datum shift
2. Activate tilting function
3. Activate rotation

...
Machining
...

1. Reset rotation
2. Reset tilting function
3. Reset datum shift

Automatic workpiece measurement in the tilted system

The cycle G55 enables you to have the TNC measure a workpiece in a tilted system automatically. The TNC stores the measured data in Q parameters for further processing (for example, for printout).

Procedure for working with Cycle G80 WORKING PLANE

1 Write the program

- Define the tool (not required, when TOOL.T is active), and enter the full tool length.
- Call the tool.
- Retract the tool in the tool axis to a position where there is no danger of collision with the workpiece (clamping devices) during tilting.
- If required, position the tilt axis or axes with a G01 block to the appropriate angular value(s) (depending on a machine parameter).
- Activate datum shift if required.
- Define Cycle G80 WORKING PLANE; enter the angular values for the tilt axes.
- Traverse all main axes (X, Y, Z) to activate compensation.
- Write the program as if the machining process were to be executed in a non-tilted plane.
- Reset Cycle G80 WORKING PLANE; program 0° for all tilt axes.
- Disable the WORKING PLANE function; redefine Cycle G80 and answer the dialog question with "NO ENT."
- Reset datum shift if required.
- Position the tilt axes to the 0° position if required.

2 Clamp the workpiece

3 Preparations in the operating mode
Positioning with MDI
Pre-position the tilt axis/axes to the corresponding angular value(s) for setting the datum. The angular value depends on the selected reference plane on the workpiece.

4 Preparations in the operating mode
Manual Operation
Use the 3D-ROT soft key to set the function TILT WORKING PLANE to ACTIVE in the Manual Operation mode. Enter the angular values for the tilt axes into the menu if the axes are not controlled.

If the axes are not controlled, the angular values entered in the menu must correspond to the actual position(s) of the tilted axis or axes, respectively. The TNC will otherwise calculate a wrong datum.

5 Setting the datum

- Manually by touching the workpiece with the tool in the non-tilted coordinate system (see section 2.4 "Setting the Datum Without a 3-D Touch Probe")
- Automatically by using a HEIDENHAIN 3-D touch probe (see section 12.3 "Setting the Datum with a 3-D Touch Probe")

6 Start the part program in the operating mode Program Run, Full Sequence

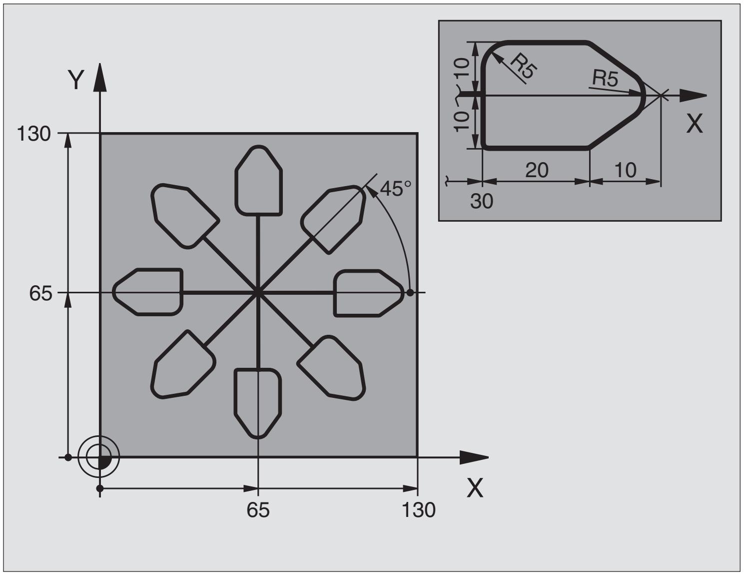
7 Manual Operation mode

Use the 3D-ROT soft key to set the function TILT WORKING PLANE to INACTIVE. Enter an angular value of 0° for each axis in the menu (see section 2.5 "Tilting the Working Plane").

Example: Coordinate transformation cycles

Program sequence

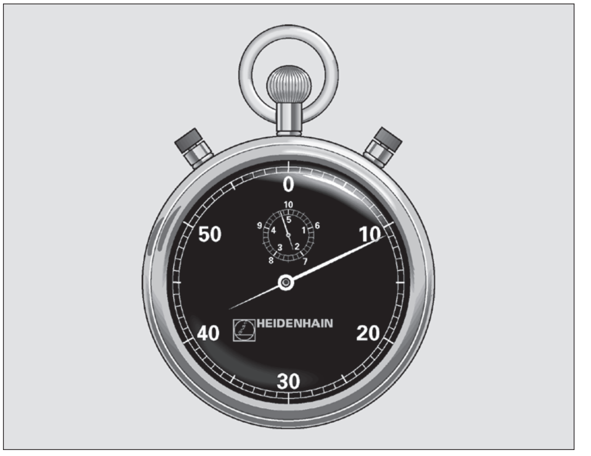
- Program the coordinate transformations in the main program
- Program the machining operation in subprogram 1 (see section 9 "Programming: Subprograms and Program Section Repeats")



%KOUMR G71 *	
N10 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N20 G31 G90 X+130 Y+130 Z+0 *	
N30 G99 T1 L+0 R+1 *	Define the tool
N40 T1 G17 S4500 *	Tool call
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G54 X+65 Y+65 *	Shift datum to center
N70 L1.0 *	Call milling operation
N80 G98 L10 *	Set label for program section repeat
N90 G73 G91 H+45 *	Rotate by 45° (incremental)
N100 L1,0 *	Call milling operation
N110 L10.6 *	Return jump to LBL 10; execute the milling operation six times
N120 G73 G90 H+0 *	Reset the rotation
N130 G54 X+0 Y+0 *	Reset the datum shift
N140 G00 Z+250 M2 *	Retract in the tool axis, end program

`N150 G98 L1 *`	Subprogram 1:
`N160 G00 G40 X+0 Y+0 *`	Define milling operation
`N170 Z+2 M3 *`	
`N180 G01 Z-5 F200 *`	
`N190 G41 X+30 *`	
`N200 G91 Y+10 *`	
`N210 G25 R5 *`	
`N220 X+20 *`	
`N230 X+10 Y-10 *`	
`N240 G25 R5 *`	
`N250 X-10 Y-10 *`	
`N260 X-20 *`	
`N270 Y+10 *`	
`N280 G40 G90 X+0 Y+0 *`	
`N290 G00 Z+20 *`	
`N300 G98 L0 *`	
`N999999 %KOUMR G71 *`	

8.10 Special Cycles

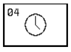


DWELL TIME (Cycle G04)

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.



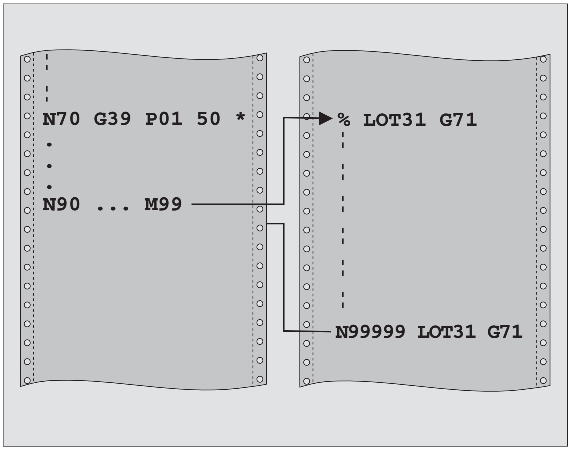
- Dwell time in seconds: Enter the dwell time in seconds

  Input range 0 to 3600 s (1 hour) in 0.001 s steps

Example NC block:

```
N72  G04 F1.5*
```

PROGRAM CALL (Cycle G39)



Routines that you have programmed (such as special drilling cycles or geometrical modules) can be written as main programs and then called like fixed cycles.

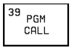
Before programming, note the following:

If you want to define a conversational dialog program to be a cycle, enter the file type .H behind the program name.

For the TNC 426, TNC 430:

If the program you are defining to be a cycle is located in the same directory as the program you are calling it from, you need only to enter the program name.

If the program you are defining to be a cycle is not located in the same directory as the program you are calling it from, you must enter the complete path (for example \CONV35\FK1\50.I).



▶ Program name: Enter the name of the program you want to call and, if necessary, the directory it is located in.

The program is called with
- G79 (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.

Example NC blocks

N550 G39 P01 50 *	Program 50 is a cycle"
N560 G00 X+20 Y+50 M99 *	Call program 50

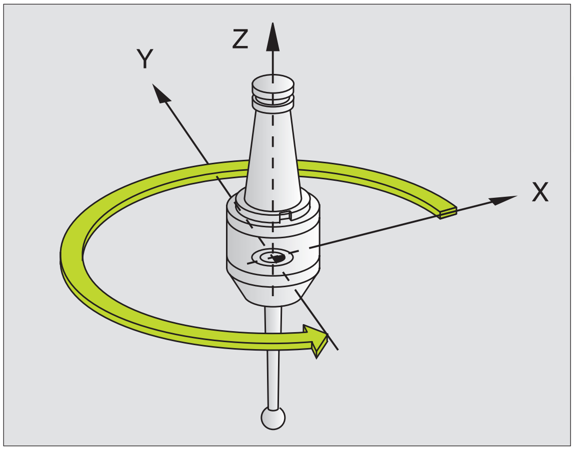
ORIENTED SPINDLE STOP (Cycle G36)

The TNC and the machine tool must be specially prepared by the machine tool builder for the use of Cycle G36.

The control can control the machine tool spindle and rotate it to a given angular position.

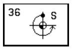
Oriented spindle stops are required for

- Tool changing systems with a defined tool change position
- Orientation of the transmitter/receiver window of HEIDENHAIN 3-D touch probes with infrared transmission



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

If you program M19 without having defined Cycle G36, 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	0.001°

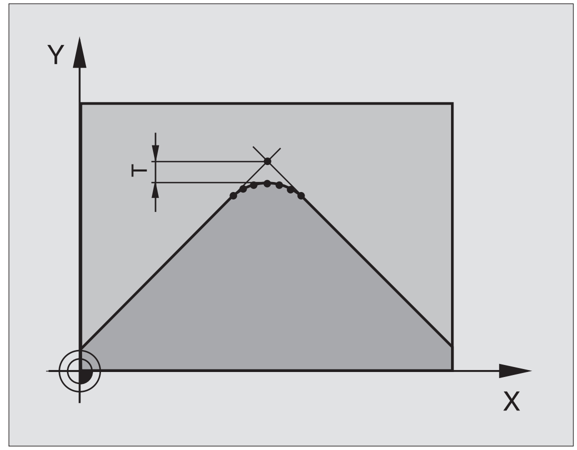
Example NC block:

```
N72  G36 S25*
```

TOLERANCE (Cycle G62, not TNC 410)

The TNC automatically smoothens the contour between two path elements (whether compensated or not). The tool has constant contact with the workpiece surface. If necessary, the TNC automatically reduces the programmed feed rate so that the program can be machined at the fastest possible speed without "jerk". As a result the surface quality is improved and the machine is protected.

A contour deviation results from the smoothing out. The size of this deviation (tolerance value) is set in a machine parameter by the machine manufacturer. You can change the pre-set tolerance value with Cycle G62 (see figure at top right).



Fast contour milling is adapted to suit both the TNC and your machine by the machine manufacturer. Your machine manual provides more detailed information.

Before programming, note the following:

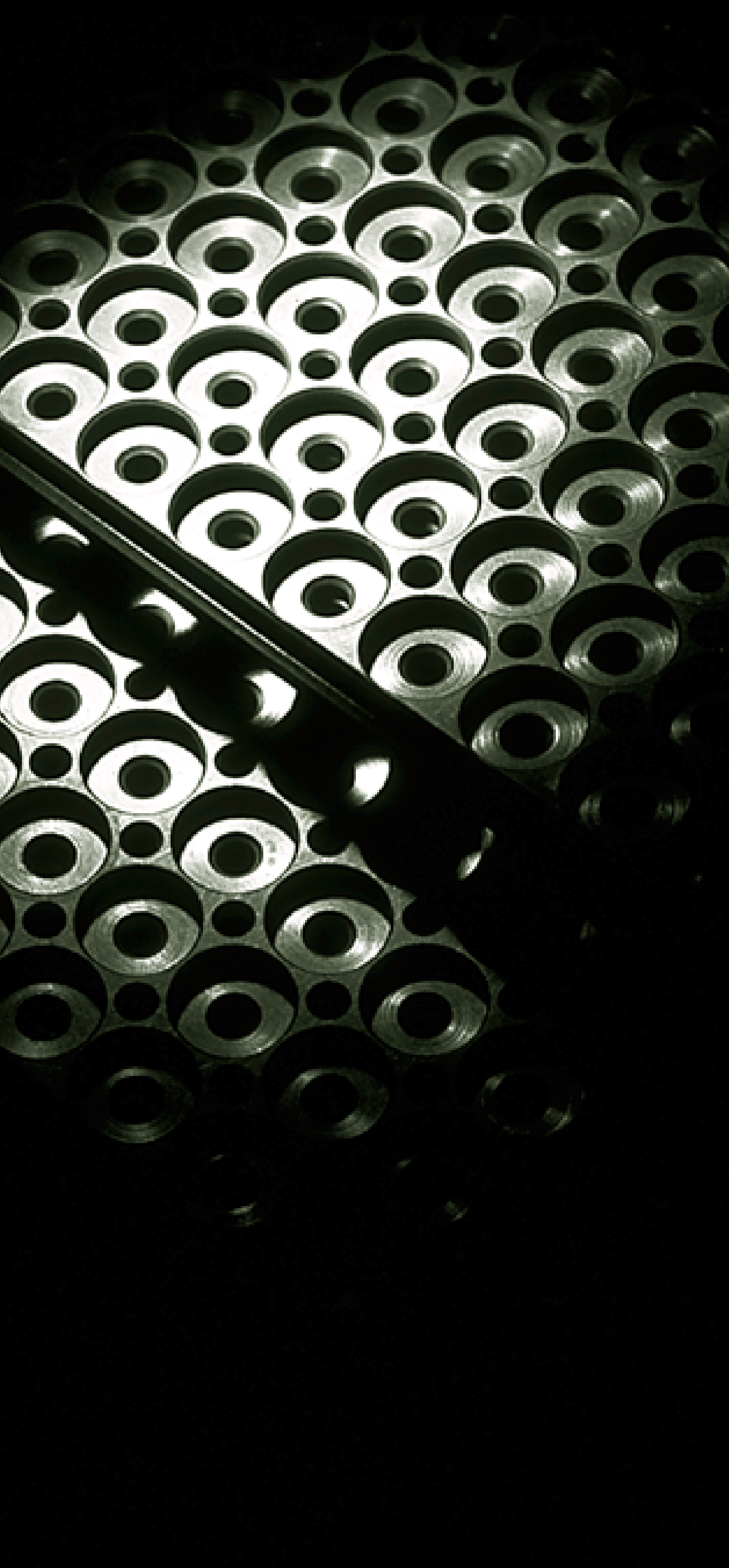
Cycle G62 is DEF active which means that it becomes effective as soon as it is defined in the part program.

You can reset Cycle G62 by defining it again and confirming the dialog question after the TOLERANCE VALUE with NO ENT.

- Tolerance value for corner rounding: Permissible contour deviation in mm

Example NC block:

```
N72  G62 T0.05*
```



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 G98 labels.

A label is identified by a number between 1 and 254. Each label can be set only once with G98 in a program.

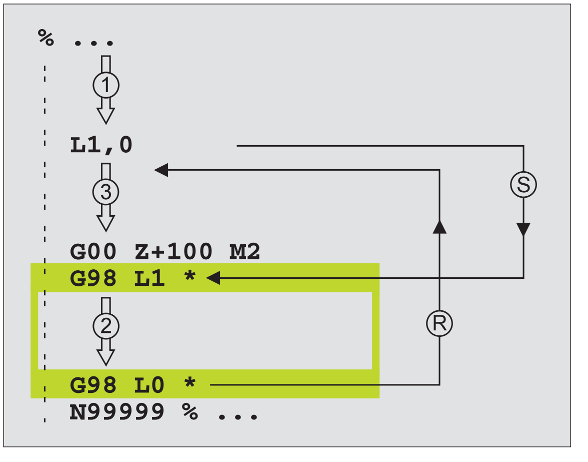
If a label is set more than once, the TNC sends an error message at the end of the G98 block.

For the TNC 426, TNC 430:

With very long programs, you can limit the number of blocks to be checked for repeated labels with MP7229.

Label 0 (G98 L0) 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 Ln.0. n can be any label number
2 The subprogram is then executed from beginning to end. The subprogram end is marked with G98 L0.
3 The TNC then resumes the part program from the block after the subprogram call Ln.0.

Programming notes
- A main program can contain up to 254 subprograms.
- You can call subprograms in any sequence and as often as desired.
- A subprogram cannot call itself.
- Write subprograms at the end of the main program (behind the block with 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

G 98

- ▶ Mark the beginning of the subprogram by selecting function G98.
- ▶ Press the "L" key on the alphabetic keyboard and enter a subprogram number
- ▶ Mark the end of the subprogram by selecting function G98 again and entering "L0".

Calling a subprogram

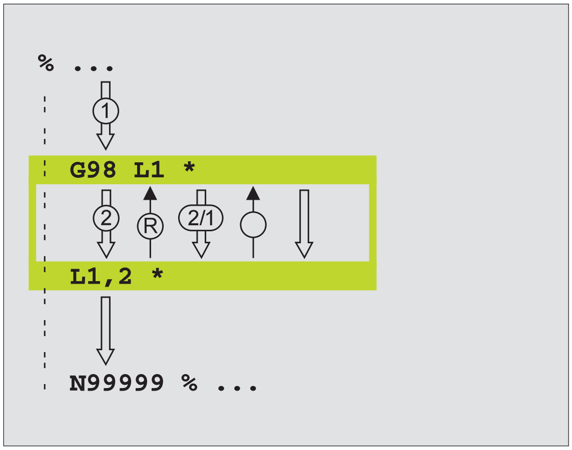
L

- ▶ To call a subprogram, press the L key.
- ▶ Enter the label number for the program you are calling and "**.0**".

L0.0 is not permitted, as it corresponds to the program end call.

9.3 Program Section Repeats

Program section repeats begin with the marker G98 Ln. n can be any label number. A program section repeat ends with Ln.m, where m is the number of repeats.



Operating sequence

1 The TNC executes the part program up to the end of the program section (L1.2)

2 Then the program section between the called label and the label call L1.2 is repeated the number of times entered after the decimal point.

3 The TNC then resumes the part program after the last repetition

Programming notes

- You can repeat a program section up to 65 534 times in succession.
- The total number of times the program section is executed is always one more than the programmed number of repeats.

Programming a program section repeat

G 98

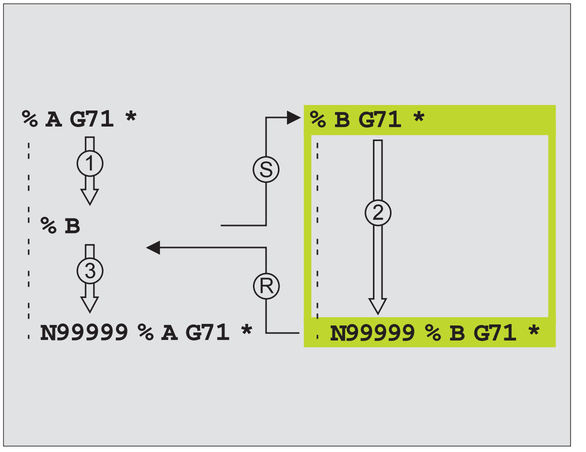
- ▶ Mark the beginning of the subprogram by selecting function G98.
- ▶ Press the "L" key and enter a label number for the program section to be repeated.

Calling a program section repeat

L

- ▶ Press the L key. Enter the label number for the program section to be repeated, and the number of repetitions after the comma.

9.4 Program as Subprogram



1 The TNC executes the part program up to the block in which another program is called with %.

2 Then the other program is run from beginning to end.

3 The TNC then resumes the first (calling) part program with the block behind the program call.

Programming notes

- No labels are needed to call any program as a subprogram.
- The called program must not contain the miscellaneous functions M2 or M30.
- The called program may not contain a call with % into the calling program.

Calling any program as a subprogram

%

▶ To call the program, press the % key and enter the name of the program you wish to call

Function	Soft key
Calling a program in conversational dialog format	.H
Calling a program in ISO format	.I
Call externally saved program (only TNC 410)	EXT
Convert %EXT block to % INT (calling internally stored program (only TNC 410)	INT
Call program type defined by the MOD function „Program input“ (only TNC 410)	VOREINST.

You can also call a program with Cycle G39.

If you want to call a conversational dialog program,
enter the file type .H behind the program name.

For the TNC 426, TNC 430:

You need only enter the program name if the program you want to call is located in the same directory as the program you are calling it from.

If the called program is not located in the same directory as the program you are calling it from, you must enter the complete path, e.g. TNC:\VZW35\ROUGH\PGM1.I

9.5 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
- Maximum nesting depth for calling main programs: 4
- You can nest program section repeats as often as desired

Subprogram within a subprogram

Example NC blocks

`%UPGMS G71 *`	
`...`	
`N170 L1.0 *`	Subprogram at label G98 L1 is called
`...`	
`N350 G00 G40 Z+100 M2 *`	Last program block of the
	main program (with M2)
`N360 G98 I1 *`	Beginning of subprogram 1
`...`	
`N390 L2.0 *`	Subprogram at label G98 L2 is called.
`...`	
`N450 G98 L0 *`	End of subprogram 1
`N460 G98 L2 *`	Beginning of subprogram 2
`...`	
`N620 G98 L0 *`	End of subprogram 2
`N999999 %UPGMS G71*`	

Program execution

1st step: Main program UPGMS is executed up to block N170.

2nd step: Subprogram 1 is called, and executed up to block N390.

3rd step: Subprogram 2 is called, and executed up to block N620. 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 N400 up to block N450. End of subprogram 1 and return jump to the main program UPGMS.

5th step: Main program UPGMS is executed from block N180 up to block N350. Return jump to block 1 and end of program.

Repeating program section repeats

Example NC blocks

`%REPS G71 *`	
`...`	
`N150 G98 L1 *`	Beginning of program section repeat 1
`...`	
`N200 G98 L2 *`	Beginning of program section repeat 2
`...`	
`N270 L2.2 *`	The program section between this block and G98 L2
`...`	(block N200) is repeated twice.
`N350 L1.1 *`	The program section between this block and G98 L1
`...`	(block N150) is repeated once.
`N999999 %REPS G71 *`	

Program execution

1st step: Main program REPS is executed up to block N270.

2nd step: Program section between block N270 and block N200 is repeated twice.

3rd step: Main program REPS is executed from block N280 to block N350.

4th step: Program section between block N350 and block N150 is repeated once (including the program section repeat between N200 and block N270).

5th step: Main program REPS is executed from block N360 to block N999 999 (end of program).

Repeating a subprogram

Example NC blocks

`%UPGREP G71 *`	
`...`	
`N100 G98 L1 *`	Beginning of the program section repeat
`N110 L2.0 *`	Subprogram call
`N120 L1.2 *`	The program section between this block and G98 L1
`...`	(block N100) is repeated twice.
`N190 G00 G40 Z+100 M2 *`	Last program block of the main program with M2
`N200 G98 L2 *`	Beginning of subprogram
`...`	
`N280 G98 L0 *`	End of subprogram
`N999999 %UPGREP G71 *`	

Program execution

1st step: Main program UPGREP is executed up to block N110.

2nd step: Subprogram 2 is called and executed.

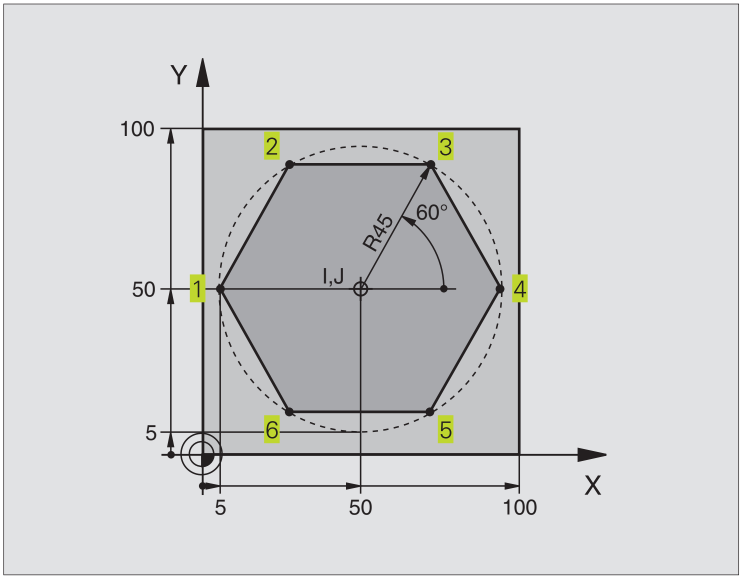
3rd step: Program section between block N120 and block N100 is repeated twice. Subprogram 2 is called twice.

4th step: Main program UPGREP is executed from block N130 to block N190. End of program.

Example: Milling a contour in several infeeds

Program sequence

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

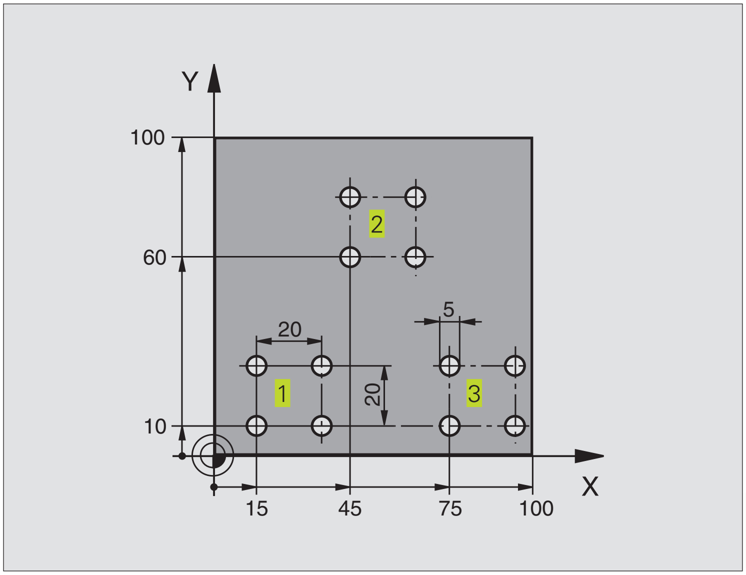


%PGMWDH G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+7,5 *	Define the tool
N40 T1 G17 S4000 *	Call the tool
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 I+50 J+50 *	Set pole
N70 G10 R+60 H+180 *	Pre-position in the working plane
N80 G01 Z+0 F1000 M3 *	Pre-position to the workpiece surface
N90 G98 L1 *	Set label for program section repeat
N100 G91 Z-3 *	Infeed depth in incremental values (in the open)
N110 G11 G41 G90 R+45 H+180 F250 *	First contour point
N120 G26 R5 *	Approach contour
N130 H+120 *	Contour
N140 H+60 *	
N150 H+0 *	
N160 H-60 *	
N170 H-120 *	
N180 H+180 *	
N190 G27 R5 F500 *	Depart contour
N200 G40 R+60 H+180 F1000 *	Retract tool
N210 L1.9 *	Return jump to LBL 1; section is repeated a total of 9 times
N220 G00 Z+250 M2 *	Retract in the tool axis, end program
N999999 %PGMWDH G71 *	

Example: Groups of holes

Program sequence

- Approach the groups of holes in the main program
- Call the group of holes (subprogram 1)
- Program the group of holes only once in subprogram 1

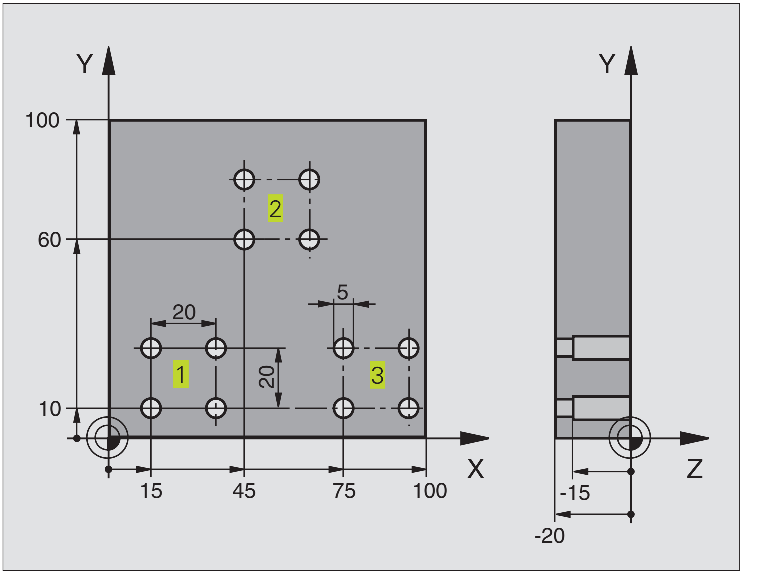


%UP1 G71 *	
N10 G30 G17 X+0 Y+0 Z-40 *	
N20 G31 G90 X+100 Y+100 Z+0 *	
N30 G99 T1 L+0 R+2.5 *	Define the tool
N40 T1 G17 S5000 *	Call the tool
N50 G00 G40 G90 Z+250 *	Retract the tool
N60 G83 P01 +2 P02 -30 P03 +5 P04 0	Cycle definition: drilling
P05 300 *	
N70 X+15 Y+10 M3 *	Approach starting point for 1st set of bore holes, spindle ON
N80 L1,0 *	Call the subprogram for the group
N90 X+45 Y+60 *	Move to starting point for group 2
N100 L1,0 *	Call the subprogram for the group
N110 X+75 Y+10 *	Move to starting point for group 3
N120 L1.0 *	Call the subprogram for the group
N130 G00 Z+250 M2 *	End of main program
N140 G98 L1 *	Beginning of subprogram 1: Group of holes
N150 Z+2 M99 *	1st hole; move to setup clearance in Z, call cycle
N160 G91 X+20 M99 *	Move to 2nd hole, call cycle
N170 Y+20 M99 *	Move to 3rd hole, call cycle
N180 X-20 G90 M99 *	Move to 4th hole, call cycle
N190 G98 L0 *	End of subprogram 1
N999999 %UP1 G71 *	

Example: Groups of holes with several tools

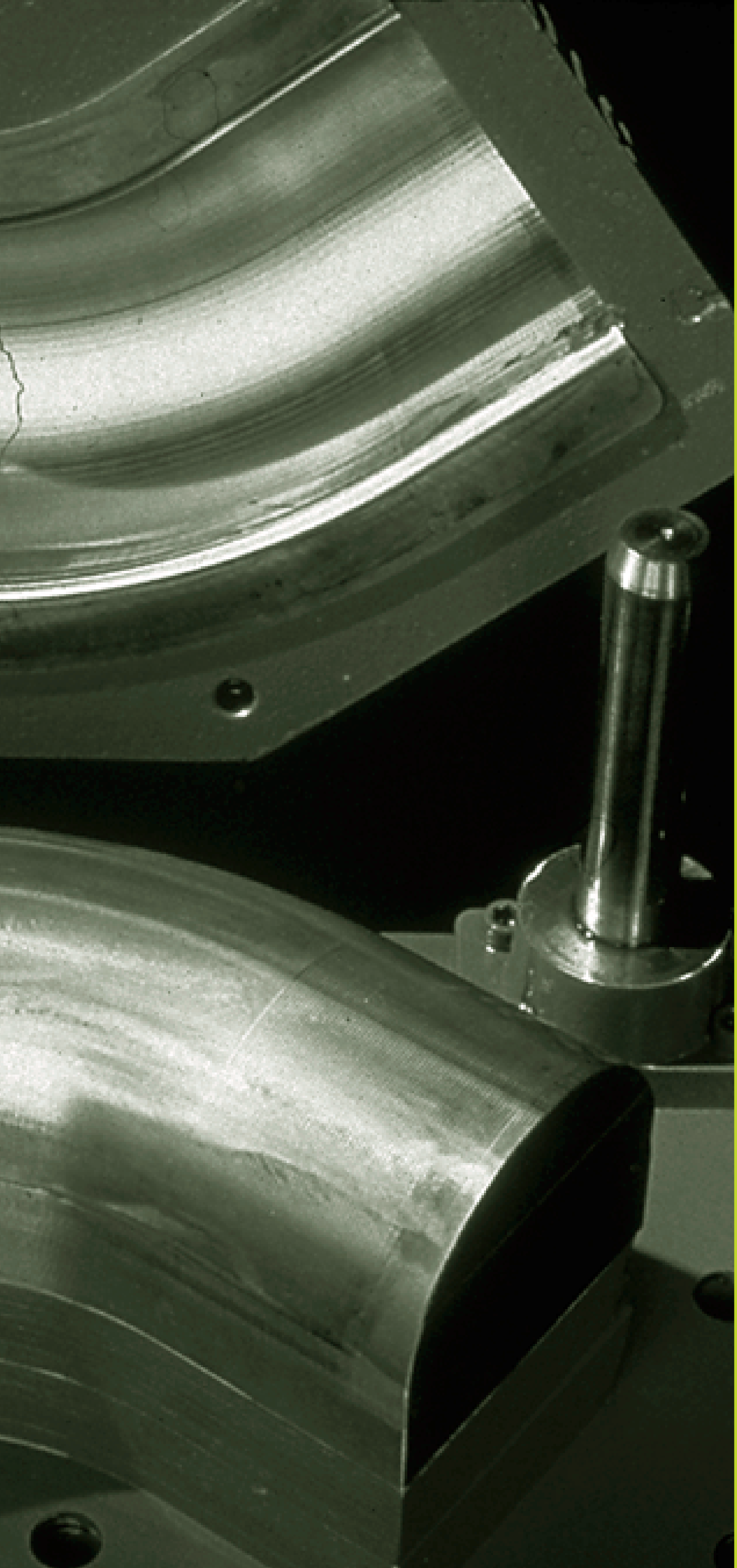
Program sequence

- Program the fixed cycles in the main program
- Call the entire hole pattern (subprogram 1)
- Approach the groups of holes in subprogram 1, call group of holes (subprogram 2)
- Program the group of holes only once in subprogram 2



`%UP2 G71 *`	
`N10 G30 G17 X+0 Y+0 Z-40 *`	
`N20 G31 G90 X+100 Y+100 Z+0 *`	
`N30 G99 T1 L+0 R+4 *`	Define tool: center drill
`N40 G99 T2 L+0 R+3 *`	Tool definition: drill
`N50 G99 T3 L+0 R+3.5 *`	Tool definition of tap
`N60 T1 G17 S5000 *`	Call tool: center drill
`N70 G00 G40 G90 Z+250 *`	Retract the tool
`N80 G83 P01 +2 P02 -3 P03 +3 P04 0`	Cycle definition: Centering
`     P05 250 *`	
`N90 L1.0 *`	Call subprogram 1 for the entire hole pattern
`N100 G00 Z+250 M6 *`	Tool change
`N110 T2 G17 S4000 *`	Call the drilling tool
`N120 G83 P01 +2 P02 -25 P03 +5 P04 0`	Cycle definition: drilling
`     P05 250 *`	
`N130 L1.0 *`	Call subprogram 1 for the entire hole pattern
`N140 G00 Z+250 M6 *`	Tool change
`N150 T3 G17 S500 *`	Tool call for tap
`N160 G84 P01 +2 P02 -15 P03 0 P04 500 *`	Cycle definition for tapping
`N170 L1.0 *`	Call subprogram 1 for the entire hole pattern
`N180 G00 Z+250 M2 *`	End of main program

`N190 G98 L1 *`	Beginning of subprogram 1: Entire hole pattern
`N200 G00 G40 G90 X+15 Y+10 M3 *`	Move to starting point for group 1
`N210 L2.0 *`	Call subprogram 2 for the group
`N220 X+45 Y+60 *`	Move to starting point for group 2
`N230 L2.0 *`	Call subprogram 2 for the group
`N240 X+75 Y+10 *`	Move to starting point for group 3
`N250 L2.0 *`	Call subprogram 2 for the group
`N260 G98 L0 *`	End of subprogram 1
`N270 G98 L2 *`	Beginning of subprogram 2: Group of holes
`N280 Z+2 M99 *`	1st hole with active fixed cycle
`N290 G91 X+20 M99 *`	Move to 2nd hole, call cycle
`N300 Y+20 M99 *`	Move to 3rd hole, call cycle
`N310 X-20 G90 M99 *`	Move to 4th hole, call cycle
`N320 G98 L0 *`	End of subprogram 2
`N999999 %UP2 G71 *`	

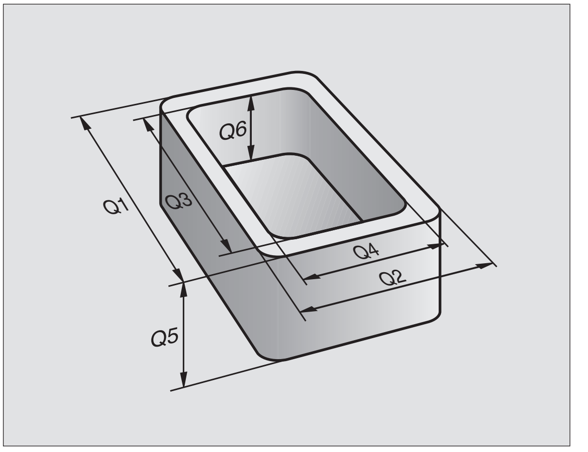


10

Programming:

Q Parameters

10.1 Principle and Overview



You can program an entire family of parts in a single part program. You do this by entering variables called *Q parameters* instead of fixed numerical values.

Q parameters can represent information such as:

- Coordinate values
- Feed rates
- RPM
- Cycle data

Q parameters also enable you to program contours that are defined through mathematical functions. You can also use Q parameters to make the execution of machining steps depend on logical conditions.

Q parameters are designated by the letter Q and a number between 0 and 399. They are grouped according to three ranges:

Meaning	Range
Freely applicable parameters, global for all programs in the TNC memory.	Q0 to Q99
Parameters for special TNC functions	Q100 to Q199
Parameters that are primarily used for cycles, global for all programs in the TNC memory	Q200 to Q399 (TNC 410: to Q299)

Programming notes
You can mix Q parameters and fixed numerical values within a program.

Q parameters can be assigned numerical values between –99 999.9999 and +99 999.9999.

Some Q parameters are always assigned the same data by the TNC. For example, Q108 is always assigned the current tool radius. For further information, see section "10.9 Preassigned Q Parameters."

If you are using the parameters Q1 to Q99 in OEM cycles, define via MP7251 whether the parameters are only to be used locally in the OEM cycles, or may be used globally.

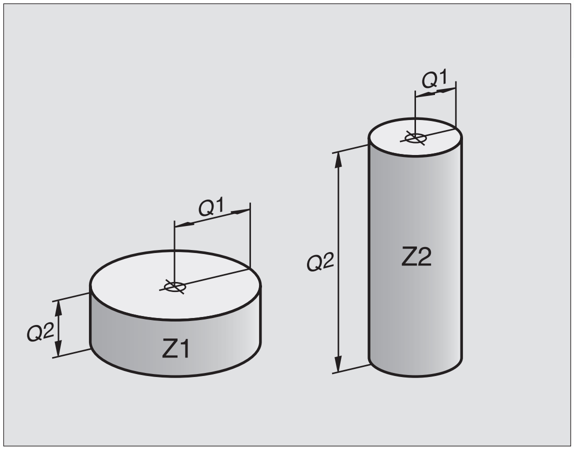
Calling Q parameter functions
TNC 426, TNC 430: Press the PARAMETER soft key while you are entering a part program.

TNC 410: Press the „Q" key (to be found among the keys for value input and axis selection, beneath the –/+ key).

The TNC then displays the following soft keys:

Function group	Soft key
Basic arithmetic (assign, add, subtract, multiply, divide, square root)	BASIC ARITHM.
Trigonometric functions	TRIGO-NOMETRY
If/then conditions, jumps	JUMP
Other functions	DIVERSE FUNCTION
Entering Formulas Directly	FORMULA

10.2 Part Families — Q Parameters in Place of Numerical Values



The Q parameter function D0: ASSIGN assigns numerical values to Q parameters. This enables you to use variables in the program instead of fixed numerical values.

Example NC blocks

```
N150 D00 Q10 P01 +25 *         ASSIGN:
...                            Q10 contains the value 25
N250 G00 X +Q10 *              corresponds to G00 X +25
```

You need write only one program for a whole family of parts, entering the characteristic dimensions as Q parameters.

To program a particular part, you then assign the appropriate values to the individual Q parameters.

Example
Cylinder with Q parameters

Cylinder radius	R	=	Q1
Cylinder height	H	=	Q2
Cylinder Z1	Q1	=	+30
	Q2	=	+10
Cylinder Z2	Q1	=	+10
	Q2	=	+50

10.3 Describing Contours Through Mathematical Functions

The Q parameters listed below enable you to program basic mathematical functions in a part program:

▶ To select the mathematical functions: Press the BASIC ARITHMETIC soft key. The TNC then displays the following soft keys:

Function	Soft key
D00: ASSIGN e.g. D00 Q5 P01 +60 * Assigns a numerical value.	D0 X = V
D01: ADDITION e.g. D01 Q1 P01 –Q2 P02 –5 * Calculates and assigns the sum of two values.	D1 X + V
D02: SUBTRACTION e.g. D02 Q1 P01 +10 P02 +5 * Calculates and assigns the difference of two values.	D2 X – V
D03: MULTIPLICATION e.g. D03 Q2 P01 +3 P02 +3 * Calculates and assigns the product of two values.	D3 X * V
D04: DIVISION e.g. D04 Q4 P01 +8 P02 +Q2 * Calculates and assigns the quotient of two values **Not permitted:** division by 0	D4 X / V
D05: SQUARE ROOT e.g. D05 Q20 P01 4 * Calculates and assigns the square root of a number. **Not permitted:** square root of a negative number	D5 SQRT

At the right of the "=" character you can enter:

- Two numbers
- Two Q parameters
- A number and a Q parameter

The Q parameters and numerical values in the equations can be entered with positive or negative signs.

Example: Programming fundamental operations

Q PARA-METER: To select Q parameter functions, press the Q key or the PARAMETER soft key.

BASIC ARITHM.: To select the mathematical functions: Press the BASIC ARITHMETIC soft key.

D0 X = Y: To select the Q parameter function ASSIGN, press the D0 X = Y soft key.

Parameter number for result?

5 ENT: Enter a parameter number, for example 5.

1st value or parameter ?

10 ENT: Assign a value to Q5, for example 10.

Q PARA-METER: To select Q parameter functions, press the Q key or the PARAMETER soft key.

BASIC ARITHM.: To select the mathematical functions: Press the BASIC ARITHMETIC soft key.

D3 X * Y: To select the Q parameter function MULTIPLICATION, press the D03 X * Y soft key.

Parameter number for result?

12 ENT: Enter a Q parameter number, for example 12.

Multiplicand?

Q5 ENT: Enter Q5 for the first value.

Multiplier?

7 ENT: Enter 7 for the second value.

The TNC displays the following program blocks:

```
N160 D00 Q5 P01 +10 *
N170 D03 Q12 P01 +Q5 P02 +7 *
```

10.4 Trigonometric Functions

Sine, cosine and tangent are terms designating the ratios of sides of right triangles. For a right triangle, the trigonometric functions of the angle a are defined by the following equations:

Sine: $\sin \alpha = a / c$

Cosine: $\cos \alpha = b / c$

Tangent: $\tan \alpha = a / b = \sin \alpha / \cos \alpha$

where

- c is the side opposite the right angle (hypotenuse)
- a is the side opposite the angle (opposite)
- b is the third side (adjacent)

The TNC can find the angle from the tangent

$\alpha = \arctan \alpha = \arctan (a / b) = \arctan (\sin \alpha / \cos \alpha)$

Example:

$a = 10$ mm

$b = 10$ mm

$\alpha = \arctan (a / b) = \arctan 1 = 45°$

Furthermore:

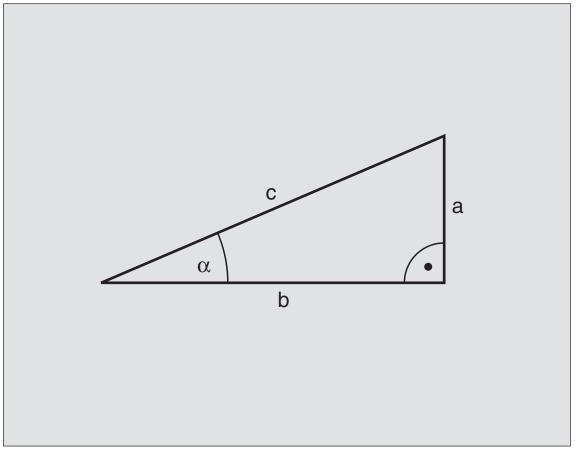
$a^2 + b^2 = c^2$ (where $a^2 = a \times a$)

$c = \sqrt{(a^2 + b^2)}$

Programming trigonometric functions

Press the TRIGONOMETRY soft key to call the trigonometric functions. The TNC then displays the soft keys that are listed in the table at right.

Programming: compare „Example: Programming fundamental operations".



Function	Soft key
D06: SINE Example D06 Q20 P01 –Q5 * Calculate the sine of an angle in degrees (°) and assign it to a parameter.	D6 SIN(X)
D07: COSINE Example D07 Q21 P01 –Q5 * Calculate the cosine of an angle in degrees (°) and assign it to a parameter.	D7 COS(X)
D08: ROOT-SUM OF SQUARES Example D08 Q10 P01 +5 P02 +4 * Calculate and assign length from two values	D8 X LEN Y
D13: ANGLE Example D13 Q20 P01 +10 P02 –Q1 * Calculate the angle from the arc tangent of two sides or from the sine and cosine of the angle (0 < angle < 360°) and assign it to a parameter.	D13 X ANG Y

10.5 If-Then Decisions with Q Parameters

The TNC can make logical If-Then decisions by comparing a Q parameter with another Q parameter or with a numerical value. If the condition is fulfilled, the TNC continues the program at the label that is programmed after the condition (for information on labels, see section 9 "Subprograms and Program Section Repeats"). If it is not fulfilled, the TNC continues with the next block.

To call another program as a subprogram, enter a program call with % after label G98.

Unconditional jumps

An unconditional jump is programmed by entering a conditional jump whose condition is always true. Example:

D0: P01 +10 P02 +10 P03 1 *

Programming If-Then decisions

Press the JUMP soft key to call the if-then conditions. The TNC then displays the following soft keys:

Function	Soft key
D09: IF EQUAL, JUMP Example D09 P01 +Q1 P02 +Q3 P03 5 * If the two values or parameters are equal, jump to the given label.	D9 IF X EQ V GOTO
D10: IF NOT EQUAL, JUMP Example D10 P01 +10 P02 –Q5 P03 10 * If the two values or parameters are not equal, jump to the given label.	D10 IF X NE V GOTO
D11: IF GREATERTHAN, JUMP Example D11 P01 +Q1 P02 +10 P03 5 * If the first parameter or value is greater than the second value or parameter, jump to the given label.	D11 IF X GT V GOTO
D12: IF LESSTHAN, JUMP Example D12 P01 +Q5 P02 +0 P03 1 * If the first value or parameter is less than the second value or parameter, jump to the given label.	D12 IF X LT V GOTO

Abbreviations used:

IF	If
EQU	Equals
NE	Not equal
GT	Greater than
LT	Less than
GOTO	Go to

10.6 Checking and Changing Q Parameters

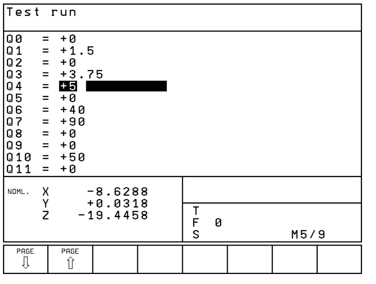
During a program run or test run, you can check or change Q parameters if necessary.

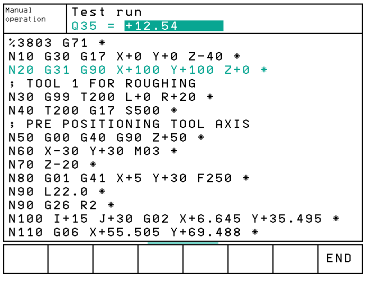
- ▶ If you are in a program run, interrupt it (for example by pressing the machine STOP button and the INTERNAL STOP soft key). If you are doing a test run, interrupt it.

Q

- ▶ To call the Q parameter functions, press the Q key.
- ▶ TNC 426, TNC 430:
  Enter the number of the Q parameter and press the ENT key. The TNC displays the current value of the Q parameter in the dialog line.
- ▶ TNC 410:
  Select desired Q parameter number using arrow keys or PAGE soft keys
- ▶ If you wish to change the value, enter a new value, confirm it with the ENT key and conclude your entry with the END key.

  To leave the value unchanged, terminate the dialog with the END key.





10.7 Additional Functions

Press the DIVERSE FUNCTION soft key to call the additional functions. The TNC then displays the following soft keys:

Function	Soft key
D14:ERROR Output error messages	D14 ERROR=
D15: PRINT Unformatted output of texts or Q parameter values	D15 PRINT
D19:PLC Transfer values to the PLC	D19 PLC=

D14: ERROR
Output error messages

With the function D14: ERROR you can call messages under program control. The messages were preprogrammed by the machine tool builder or by HEIDENHAIN. If the TNC encounters a block with FN 14 during program run, it will interrupt the run and display an error message. The program must then be restarted. The error number are listed in the table below.

Example NC block

The TNC is to display the text stored under error number 254:

```
N180 D14:P01 254 *
```

Range of error numbers	Standard dialog text
0 ... 299	D14: Error code 0 299
300 ... 999	Machine-dependent dialog
1000 ... 1099	Internal error messages (see table at right)

Error code and text	
1000	Spindle ?
1001	Tool axis is missing
1002	Slot width too large
1003	Tool radius too large
1004	Range exceeded
1005	Start position incorrect
1006	ROTATION not permitted
1007	SCALING FACTOR not permitted
1008	MIRRORING not permitted
1009	Datum shift not permitted
1010	Feed rate is missing
1011	Entry value incorrect
1012	Wrong sign programmed
1013	Entered angle not permitted
1014	Touch point inaccessible
1015	Too many points
1016	Contradictory entry
1017	CYCL incomplete
1018	Plane wrongly defined
1019	Wrong axis programmed
1020	Wrong RPM
1021	Radius comp. undefined
1022	Rounding-off undefined
1023	Rounding radius too large
1024	Program start undefined
1025	Excessive subprogramming
1026	Angle reference missing
1027	No fixed cycle defined
1028	Slot width too large
1029	Pocket too small
1030	Q202 not defined
1031	Q205 not defined
1032	Enter Q218 greater than Q219
1033	CYCL 210 not permitted
1034	CYCL 211 not permitted
1035	Q220 too large
1036	Q222 must be greater than Q223
1037	Q244 must be greater than 0
1038	Q245 must not equal Q246
1039	Angle range must be under 360°
1040	Q223 must be greater than Q222
1041	Q214: 0 not permitted

D15: PRINT
Unformatted output of texts or Q parameter values

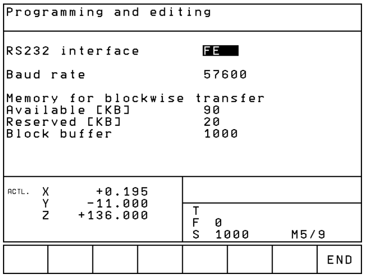
Setting the data interface for the TNC 410:

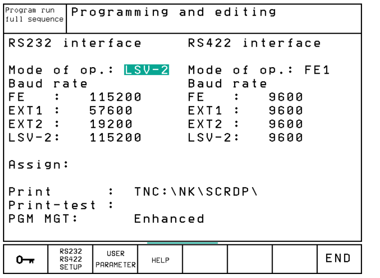
In the menu option RS 232 interface, you must enter where the texts or Q parameters are to be stored.

Setting up the data interface for TNC 426, TNC 430:

In the menu option PRINT or PRINT-TEST, you must enter the path for storing the texts or Q parameters.

See Chapter "13 MOD Functions, Setting the Date Interface."





The function D15: PRINT transfers Q parameter values and error messages through the data interface, for example to a printer. When you save the data in the TNC memory or transfer them to a PC, the TNC stores the data in the file %D15RUN.A (output in program run mode) or in the file %D15SIM.A (output in test run mode).

To output dialog texts and error messages with D15: PRINT "numerical value"
Numerical values from 0 to 99:
Dialog texts for OEM cycles

Numerical values exceeding 100:
PLC error messages

Example: Output of dialog text 20

```
N60 D15:P01 20 *
```

To output dialog texts and error messages with D15: PRINT "Q parameter"
Application example: Recording workpiece measurement.

You can transfer up to six Q parameters and numerical values simultaneously.

Example: Output of dialog text 1 and numerical value for Q1

```
N70 D15 P01 1 P02 Q1 *
```

D19: PLC
Transferring values to the PLC

The function D19: PLC transfers up to two numerical values or Q parameter contents to the PLC.

Increments and units: 0.1 µm or 0.0001°

Example: Transfer the numerical value 10 (which means 1 µm or 0.001°) to the PLC

```
N60 D19 P01 +10 P02 +Q3 *
```

10.8 Entering Formulas Directly

You can enter mathematical formulas that include several operations directly into the part program by soft key.

Entering formulas
Press the FORMULA soft key to call the formula functions. The TNC displays the following soft keys in several soft-key rows:

Mathematical function	Soft key
Addition Example: Q10 = Q1 + Q5	+
Subtraction Example: Q25 = Q7 – Q108	-
Multiplication Example: Q12 = 5 * Q5	*
Division Example: Q25 = Q1 / Q2	/
Open parentheses Example: Q12 = Q1 * (Q2 + Q3)	(
Close parentheses Example: Q12 = Q1 * (Q2 + Q3)	)
Square Example: Q15 = SQ 5	SQ
Square root Example: Q22 = SQRT 25	SQRT
Sine of an angle Example: Q44 = SIN 45	SIN
Cosine of an angle Example: Q45 = COS 45	COS
Tangent of an angle Example: Q46 = TAN 45	TAN

Mathematical function	Soft key
Arc sine Inverse of the sine. Determine the angle from the ratio of the opposite side to the hypotenuse. Example: Q10 = ASIN 0.75	ASIN
Arc cosine Inverse of the cosine. Determine the angle from the ratio of the adjacent side to the hypotenuse. Example: Q11 = ACOS Q40	ACOS
Arc tangent Inverse of the tangent. Determine the angle from the ratio of the opposite to the adjacent side. Example: Q12 = ATAN Q50	ATAN
Powers Example: Q15 = 3^3	^
The constant PI (value = 3.14159) Example Q15 = PI	PI
Natural logarithm (LN) of a number Base 2.7183 Example: Q15 = LN Q11	LN
Logarithm of a number, base 10 Example: Q33 = LOG Q22	LOG
Exponential function, 2.7183^n Example: Q1 = EXP Q12	EXP
Negate (multiplication by -1) Example: Q2 = NEG Q1	NEG
Drop places after the decimal point (form an integer) Example: Q3 = INT Q42	INT
Absolute value Example: Q4 = ABS Q22	ABS
Drop places before the decimal point (form a fraction) Example: Q5 = FRAC Q23	FRAC

Mathematical function	Soft key
Check the sign of a value (not TNC 426, TNC 430) Example Q12 = SGN Q50 If result for Q12 = 1: Q50 >= 0 If result for Q12 = 0: Q50 < 0	SGN

Rules for formulas

Mathematical formulas are programmed according to the following rules:

- **Higher-level operations are performed first**
  (multiplication and division before addition and subtraction)

```
N120 Q1 = 5 * 3 + 2 * 10 = 35 *
```

1st step: 5 ∗ 3 = 15
2nd step: 2 ∗ 10 = 20
3rd step: 15 + 20 = 35

```
N130 Q2 = SQ 10 – 3^3 = 73 *
```

1st step: $10^2 = 100$
2nd step: $3^3 = 27$
3rd step: 100 – 27 = 73

- **Distributive law**
  for calculating with parentheses

  a * (b + c) = a * b + a * c

Programming example

Calculate an angle with arc tangent as opposite side (Q12) and adjacent side (Q13); then store in Q25.

Q PARA-METER	To select Q parameter functions, press the Q key or the PARAMETER soft key.
FORMULA	For formula input, press the FORMULA soft key

`Parameter number for result?`

25 ENT	Enter the parameter number and confirm with ENT
▷ ATAN	Shift the soft-key row and select the arc tangent function.
◁ (	Shift the soft-key row and open parentheses.
Q 12	Enter Q parameter number 12.
/	Select division.
Q 13	Enter Q parameter number 13.
) END	Close parentheses and conclude formula entry.

Example NC block

```
37 Q25 = ATAN (Q12/Q13)
```

10.9 Preassigned Q Parameters

The Q parameters Q100 to Q122 are assigned values by the TNC. These values include:

- Values from the PLC
- Tool and spindle data
- Data on operating status, etc.

Values from the PLC: Q100 to Q107
The TNC uses the parameters Q100 to Q107 to transfer values from the PLC to an NC program.

Active tool radius: Q108
The active value of the tool radius is assigned to Q108. Q108 is calculated from:

- Tool radius R (tool table or G99 block)
- Delta value DR from the tool table

Tool axis: Q109
The value of Q109 depends on the current tool axis:

Tool axis	Parameter value
No tool axis defined	Q109 = –1
X axis	Q109 = 0
Y axis	Q109 = 1
Z axis	Q109 = 2
U axis	Q109 = 6
V axis	Q109 = 7
W axis	Q109 = 8

Spindle status: Q110
The value of Q110 depends on which M function was last programmed for the spindle:

M function	Parameter value
No spindle status defined	Q110 = –1
M03: Spindle ON, clockwise	Q110 = 0
M04: Spindle ON, counterclockwise	Q110 = 1
M05 after M03	Q110 = 2
M05 after M04	Q110 = 3

Coolant on/off: Q111

M function	Parameter value
M08: Coolant ON	Q111 = 1
M09: Coolant OFF	Q111 = 0

Overlap factor: Q112
The overlap factor for pocket milling (MP7430) is assigned to Q112.

Unit of measurement for dimensions in the program: Q113
The value of parameter Q113 specifies whether the highest-level NC program (for nesting with %...) is programmed in millimeters or inches.

Dimensions of the main program	Parameter value
Metric system (mm)	Q113 = 0
Inch system (inches)	Q113 = 1

Tool length: Q114
The current value for the tool length is assigned to Q114.

Coordinates after probing during program run
The parameters Q115 to Q119 contain the coordinates of the spindle position at the moment of contact during programmed measurement with the 3-D touch probe.

The length and radius of the probe tip are not compensated in these coordinates.

Coordinate axis	Parameter
X axis	Q115
Y axis	Q116
Z axis	Q117
IVth axis	Q118
Vth axis (not TNC 410)	Q119

Deviation between actual value and nominal value during automatic tool measurement with the TT 120 (only conversational dialog)

Actual-nominal deviation	Parameter
Tool length	Q115
Tool radius	Q116

Tilting the working plane with mathematical angles: Rotary axis coordinates calculated by the TNC (not TNC 410)

Coordinates	Parameter
A axis	Q120
B axis	Q121
C axis	Q122

Results of measurements with touch probe cycles
(see also Touch Probe Cycles User's Manual)

Measured actual values	Parameter
Center in reference axis	Q151
Center in minor axis	Q152
Diameter	Q153
Length of pocket	Q154
Width of pocket	Q155
Length in the axis selected in the cycle	Q156
Position of the center line	Q157
Angle of the A axis	Q158
Angle of the B axis	Q159
Coordinate of the axis selected in the cycle	Q160

Measured deviation	Parameter
Center in reference axis	Q161
Center in minor axis	Q162
Diameter	Q163
Length of pocket	Q164
Width of pocket	Q165
Measured length	Q166
Position of the center line	Q167

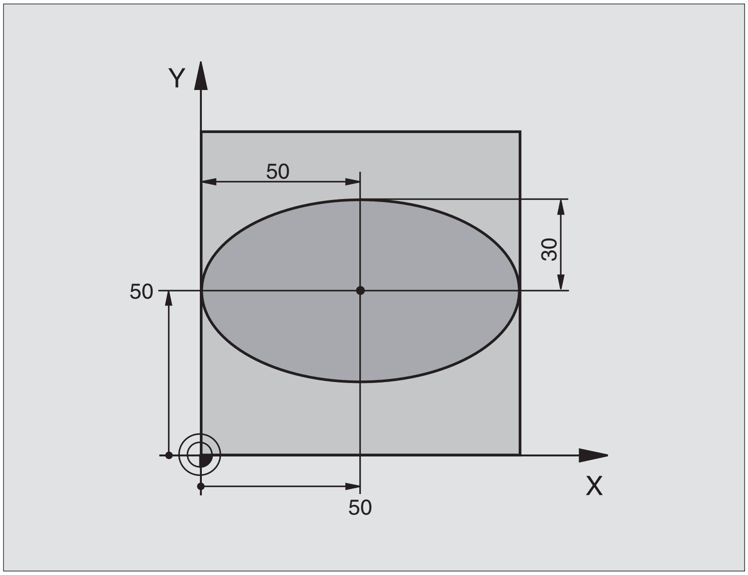
Workpiece status	Parameter
Good	Q180
Re-work	Q181
Scrap	Q182

Example: Ellipse

Program sequence

- The contour of the ellipse is approximated by many short lines (defined in Q7). The more calculating steps you define for the lines, the smoother the curve becomes.
- The machining direction can be altered by changing the entries for the starting and end angles in the plane:

  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is not taken into account.



%ELLIPSIS G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q3 P01 +50 *	Semiaxis in X
N40 D00 Q4 P01 +30 *	Semiaxis in Y
N50 D00 Q5 P01 +0 *	Starting angle in the plane
N60 D00 Q6 P01 +360 *	End angle in the plane
N70 D00 Q7 P01 +40 *	Number of calculating steps
N80 D00 Q8 P01 +30 *	Rotational position of the ellipse
N90 D00 Q9 P01 +5 *	Milling depth
N100 D00 Q10 P01 +100 *	Feed rate for plunging
N110 D00 Q11 P01 +350 *	Feed rate for milling
N120 D00 Q12 P01 +2 *	Setup clearance for pre-positioning
N130 G30 G17 X+0 Y+0 Z-20 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+2.5 *	Define the tool
N160 T1 G17 S4000 *	Call the tool
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 G00 Z+250 M2 *	Retract in the tool axis, end program

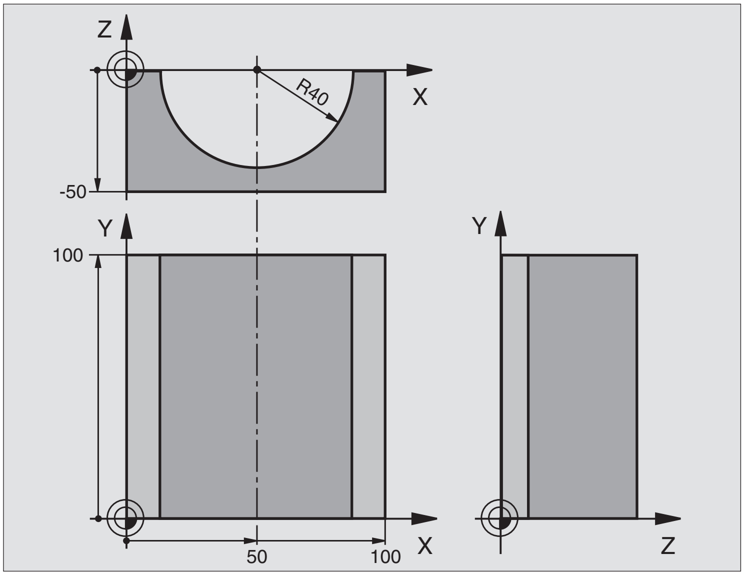
`N200 G98 L10 *`	Subprogram 10: Machining operation
`N210 G54 X+Q1 Y+Q2 *`	Shift datum to center of ellipse
`N220 G73 G90 H+Q8 *`	Account for rotational position in the plane
`N230 Q35 = (Q6 - Q5) / Q7`	Calculate angle increment
`N240 D00 Q36 P01 +Q5 *`	Copy starting angle
`N250 D00 Q37 P01 +0 *`	Set counter
`N260 Q21 = Q3 * COS Q36`	Calculate X coordinate for starting point
`N270 Q22 = Q4 * SIN Q36`	Calculate Y coordinate for starting point
`N280 G00 G40 X+Q21 Y+Q22 M3 *`	Move to starting point in the plane
`N290 Z+Q12 *`	Pre-position in tool axis to setup clearance
`N300 G01 Z-Q9 FQ10 *`	Move to working depth
`N310 G98 L1 *`	
`N320 Q36 = Q36 + Q35`	Update the angle
`N330 Q37 = Q37 + 1`	Update the counter
`N340 Q21 = Q3 * COS Q36`	Calculate the current X coordinate
`N350 Q22 = Q4 * SIN Q36`	Calculate the current Y coordinate
`N360 G01 X+Q21 Y+Q22 FQ11 *`	Move to next point
`N370 D12 P01 +Q37 P02 +Q7 P03 1 *`	Unfinished? If not finished return to label 1
`N380 G73 G90 H+0 *`	Reset the rotation
`N390 G54 X+0 Y+0 *`	Reset the datum shift
`N400 G00 G40 Z+Q12 *`	Move to setup clearance
`N410 G98 L0 *`	End of subprogram
`N999999 %ELLIPSIS G71 *`	

Example: Concave cylinder machined with spherical cutter

Program sequence

- Program functions only with a spherical cutter. The tool length refers to the sphere center.
- The contour of the cylinder is approximated by many short line segments (defined in Q13). The more line segments you define, the smoother the curve becomes.
- The cylinder is milled in longitudinal cuts (here: parallel to the Y axis).
- The machining direction can be altered by changing the entries for the starting and end angles in space:

  Clockwise machining direction:
  starting angle > end angle
  Counterclockwise machining direction:
  starting angle < end angle
- The tool radius is compensated automatically.



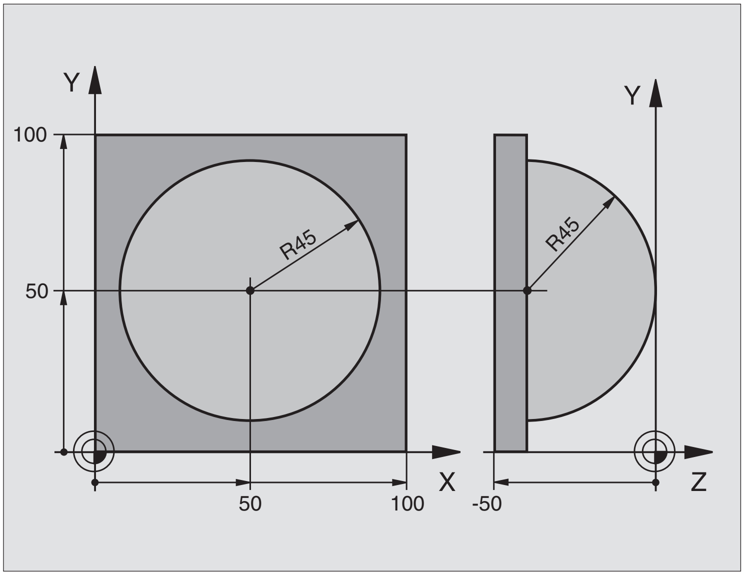
`%CYLIN G71 *`	
`N10 D00 Q1 P01 +50 *`	Center in X axis
`N20 D00 Q2 P01 +0 *`	Center in Y axis
`N30 D00 Q3 P01 +0 *`	Center in Z axis
`N40 D00 Q4 P01 +90 *`	Starting angle in space (Z/X plane)
`N50 D00 Q5 P01 +270 *`	End angle in space (Z/X plane)
`N60 D00 Q6 P01 +40 *`	Radius of the cylinder
`N70 D00 Q7 P01 +100 *`	Length of the cylinder
`N80 D00 Q8 P01 +0 *`	Rotational position in the X/Y plane
`N90 D00 Q10 P01 +5 *`	Allowance for cylinder radius
`N100 D00 Q11 P01 +250 *`	Feed rate for plunging
`N110 D00 Q12 P01 +400 *`	Feed rate for milling
`N120 D00 Q13 P01 +90 *`	Number of cuts
`N130 G30 G17 X+0 Y+0 Z-50 *`	Define the workpiece blank
`N140 G31 G90 X+100 Y+100 Z+0 *`	
`N150 G99 T1 L+0 R+3 *`	Define the tool
`N160 T1 G17 S4000 *`	Call the tool
`N170 G00 G40 G90 Z+250 *`	Retract the tool
`N180 L10.0 *`	Call machining operation
`N190 D00 Q10 P01 +0 *`	Reset allowance
`N200 L10.0 *`	Call machining operation
`N210 G00 G40 Z+250 M2 *`	Retract in the tool axis, end program

`N220 G98 L10 *`	Subprogram 10: Machining operation
`N230 Q16 = Q6 - Q10 - Q108`	Account for allowance and tool, based on the cylinder radius
`N240 D00 Q20 P01 +1 *`	Set counter
`N250 D00 Q24 P01 +Q4 *`	Copy starting angle in space (Z/X plane)
`N260 Q25 = (Q5 - Q4) / Q13`	Calculate angle increment
`N270 G54 X+Q1 Y+Q2 Z+Q3 *`	Shift datum to center of cylinder (X axis)
`N280 G73 G90 H+Q8 *`	Account for rotational position in the plane
`N290 G00 G40 X+0 Y+0 *`	Pre-position in the plane to the cylinder center
`N300 G01 Z+5 F1000 M3 *`	Pre-position in the tool axis
`N310 I+0 K+0 *`	Set pole in the Z/X plane
`N320 G11 R+Q16 H+Q24 FQ11 *`	Move to starting position on cylinder, plunge-cutting obliquely into the material
`N330 G98 L1 *`	
`N340 G01 G40 Y+Q7 FQ11 *`	Longitudinal cut in Y+ direction
`N350 D01 Q20 P01 +Q20 P02 +1 *`	Update the counter
`N360 D01 Q24 P01 +Q24 P02 +Q25 *`	Update solid angle
`N370 D11 P01 +Q20 P02 +Q13 P03 99 *`	Finished? If finished, jump to end.
`N380 G11 R+Q16 H+Q24 FQ12 *`	Move in an approximated „arc" for the next longitudinal cut
`N390 G01 G40 Y+0 FQ11 *`	Longitudinal cut in Y– direction
`N400 D01 Q20 P01 +Q20 P02 +1 *`	Update the counter
`N410 D01 Q24 P01 +Q24 P02 +Q25 *`	Update solid angle
`N420 D12 P01 +Q20 P02 +Q13 P03 1 *`	Unfinished? If not finished return to label 1
`N430 G98 L99 *`	
`N440 G73 G90 H+0 *`	Reset the rotation
`N450 G54 X+0 Y+0 Z+0 *`	Reset the datum shift
`N460 G98 L0 *`	End of subprogram
`N999999 %CYLIN G71 *`	

Example: Convex sphere machined with end mill

Program sequence

- This program requires an end mill.
- The contour of the sphere is approximated by many short lines (in the Z/X plane, defined via Q14). The smaller you define the angle increment, the smoother the curve becomes.
- You can determine the number of contour cuts through the angle increment in the plane (defined in Q18).
- The tool moves upward in three-dimensional cuts.
- The tool radius is compensated automatically.



%BALL G71 *	
N10 D00 Q1 P01 +50 *	Center in X axis
N20 D00 Q2 P01 +50 *	Center in Y axis
N30 D00 Q4 P01 +90 *	Starting angle in space (Z/X plane)
N40 D00 Q5 P01 +0 *	End angle in space (Z/X plane)
N50 D00 Q14 P01 +5 *	Angle increment in space
N60 D00 Q6 P01 +45 *	Radius of the sphere
N70 D00 Q8 P01 +0 *	Starting angle of rotational position in the X/Y plane
N80 D00 Q9 P01 +360 *	End angle of rotational position in the X/Y plane
N90 D00 Q18 P01 +10 *	Angle increment in the X/Y plane for roughing
N100 D00 Q10 P01 +5 *	Allowance in sphere radius for roughing
N110 D00 Q11 P01 +2 *	Setup clearance for pre-positioning in the tool axis
N120 D00 Q12 P01 +350 *	Feed rate for milling
N130 G30 G17 X+0 Y+0 Z-50 *	Define the workpiece blank
N140 G31 G90 X+100 Y+100 Z+0 *	
N150 G99 T1 L+0 R+7,5 *	Define the tool
N160 T1 G17 S4000 *	Call the tool
N170 G00 G40 G90 Z+250 *	Retract the tool
N180 L10.0 *	Call machining operation
N190 D00 Q10 P01 +0 *	Reset allowance
N200 D00 Q18 P01 +5 *	Angle increment in the X/Y plane for finishing
N210 L10.0 *	Call machining operation
N220 G00 G40 Z+250 M2 *	Retract in the tool axis, end program

N230 G98 L10 *	Subprogram 10: Machining operation
N240 D01 Q23 P01 +Q11 P02 +Q6 *	Calculate Z coordinate for pre-positioning
N250 D00 Q24 P01 +Q4 *	Copy starting angle in space (Z/X plane)
N260 D01 Q26 P01 +Q6 P02 +Q108 *	Compensate sphere radius for pre-positioning
N270 D00 Q28 P01 +Q8 *	Copy rotational position in the plane
N280 D01 Q16 P01 +Q6 P02 -Q10 *	Account for allowance in the sphere radius
N290 G54 X+Q1 Y+Q2 Z-Q16 *	Shift datum to center of sphere
N300 G73 G90 H+Q8 *	Account for starting angle of rotational position in the plane
N310 I+0 J+0 *	Set pole in the X/Y plane for pre-positioning
N320 G11 G40 R+Q26 H+Q8 FQ12 *	Pre-position in the plane
N330 G98 L1 *	Pre-position in the tool axis
N340 I+Q108 K+0 *	Set pole in the Z/X plane, offset by the tool radius
N350 G01 Y+0 Z+0 FQ12 *	Move to working depth
N360 G98 L2 *	
N370 G11 G40 R+Q6 H+Q24 FQ12 *	Move upward in an approximated “arc”
N380 D02 Q24 P01 +Q24 P02 +Q14 *	Update solid angle
N390 D11 P01 +Q24 P02 +Q5 P03 2 *	Inquire whether an arc is finished. If not finished, return to label 2.
N400 G11 R+Q6 H+Q5 FQ12 *	Move to the end angle in space
N410 G01 G40 Z+Q23 F1000 *	Retract in the tool axis
N420 G00 G40 X+Q26 *	Pre-position for next arc
N430 D01 Q28 P01 +Q28 P02 +Q18 *	Update rotational position in the plane
N440 D00 Q24 P01 +Q4 *	Reset solid angle
N450 G73 G90 H+Q28 *	Activate new rotational position
N460 D12 P01 +Q28 P02 +Q9 P03 1 *	Unfinished? If not finished, return to label 1
N470 D09 P01 +Q28 P02 +Q9 P03 1 *	
N480 G73 G90 H+0 *	Reset the rotation
N490 G54 X+0 Y+0 Z+0 *	Reset the datum shift
N500 G98 L0 *	End of subprogram
N999999 %BALL G71 *	

11

Test Run
and Program Run

11.1 Graphics

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following three display modes: Using soft keys, select whether you desire:

- Plan view
- Projection in 3 planes
- 3-D view

The TNC graphic depicts the workpiece as if it were being machined with a cylindrical end mill. If a tool table is active, you can also simulate the machining operation with a spherical cutter (not TNC 410). For this purpose, enter R2 = R in the tool table.

The TNC will not show a graphic if

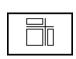
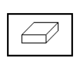
- the current program has no valid blank form definition
- no program is selected

With the TNC 426, TNC 430 you can use machine parameters 7315 to 7317 to have the TNC display a graphic even if no tool axis is defined or moved.

> A graphic simulation is not possible for program sections or programs in which a
>
> - Rotary axis movement
> - Tilted working plane cycle
>
> are defined. In this case, the TNC will display an error message.

Overview of display modes

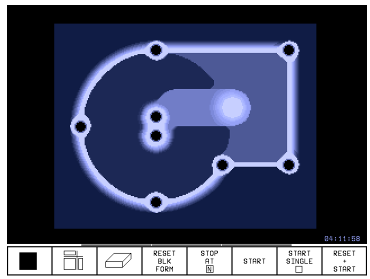
The TNC displays the following soft keys in the Program Run (not TNC 410) and Test Run modes of operation:

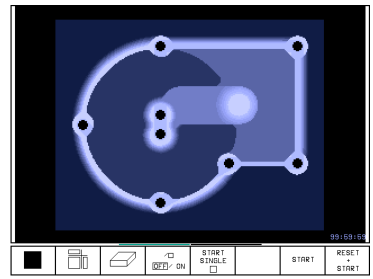
Display mode	Soft key
Plan view	
Projection in 3 planes	
3-D view	

Limitation during Program Run (with TNC 426, TNC 430)

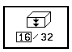
A graphical representation of a running program is not possible if the microprocessor of the TNC is already occupied with complicated machining tasks or if large areas are being machined. Example: Multipass milling over the entire blank form with a large tool. The TNC interrupts the graphics and displays the text ERROR in the graphics window. The machining process is continued, however.

Plan view





- Press the soft key for plan view.

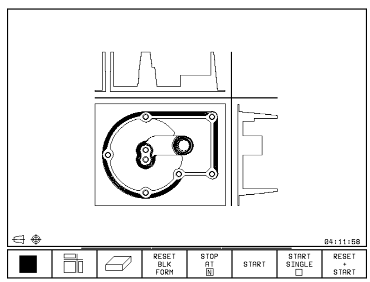


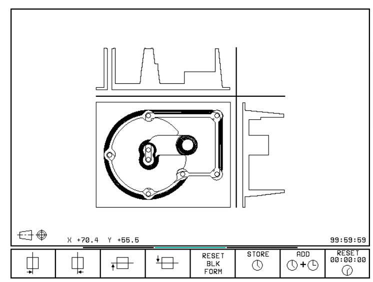
- Select the number of depth levels (after shifting the soft-key row, not TNC 410). You can choose between 16 or 32 shades of depth.

  The deeper the surface, the darker the shade.

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

Projection in 3 planes

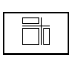




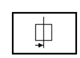
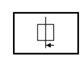
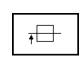
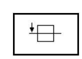
Similar to a workpiece drawing, the part is displayed with a plan view and two sectional planes. A symbol to the lower left indicates whether the display is in first angle or third angle projection according to ISO 6433 (selected with MP7310).

Details can be isolated in this display mode for magnification (see "Magnifying details", not TNC 410).

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 right or left	 
Shift the horizontal sectional plane upwards or downwards	 

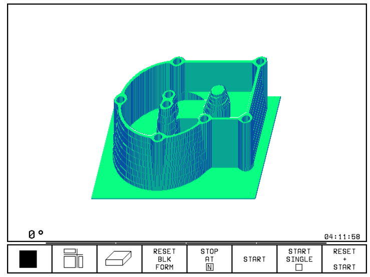
The positions of the sectional planes are visible during shifting.

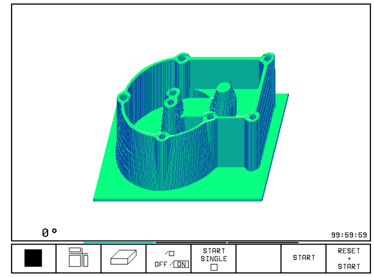
Coordinates of the line of intersection (not TNC 410)

At the bottom of the graphics window, the TNC displays the coordinates of the line of intersection, referenced to the workpiece datum. Only the coordinates of the working plane are shown. This function is activated with machine parameter 7310.

3-D view

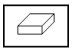
The workpiece is displayed in three dimensions, and can be rotated about the vertical axis.





The workpiece is displayed in three dimensions, and can be rotated about the vertical axis. The shape of the workpiece blank can be depicted by a frame overlay at the beginning of the graphic simulation (not TNC 410).

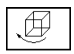
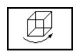
In the Test Run mode of operation you can isolate details for magnification (see “Magnifying details”).



▶ Press the soft key for plan 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 27° steps about the vertical axis	 

Switch the frame overlay display for the workpiece blank on/off (not TNC 410):

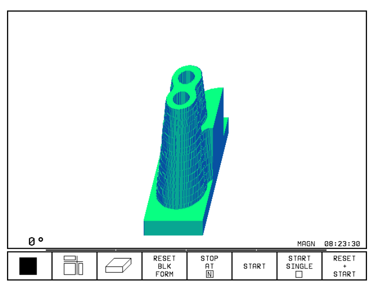
SHOW BLK-FORM

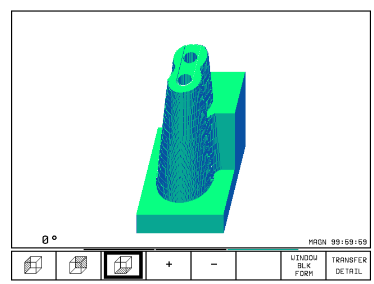
▶ Show the frame overlay with SHOW BLK-FORM

OMIT BLK-FORM

▶ Omit the frame overlay with OMIT BLK FORM

Magnifying details



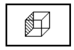
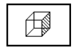
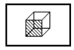
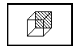
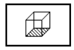
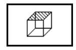


You can magnify details in the Test Run mode of operation in the following display modes, provided that the graphic simulation is stopped:

- Projection in three planes
- 3-D view

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 left/right workpiece surface		
Select the front/back workpiece surface		
Select the top/bottom workpiece surface		
Shift the sectional plane to reduce or magnify the blank form	–	+
Select the isolated detail	TRANSFER DETAIL	

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 and hold the minus or plus soft key, respectively.
- ▶ To select the isolated detail, press the TRANSFER DETAIL soft key
- ▶ Restart the test run by pressing the START soft key (RESET + START returns the workpiece blank to its original state).

Cursor position during detail magnification (not TNC 410)
During detail magnification, the TNC displays the coordinates of the axis that is currently being isolated. The coordinates describe the area determined for magnification. To the left of the slash is the smallest coordinate of the detail (MIN point), to the left is the largest (MAX point).

If a graphic display is magnified, this is indicates with MAGN at the lower right of the graphics window.

If the workpiece blank cannot be further enlarged or reduced, the TNC displays an error message in the graphics window. To clear the error message, enlarge or reduce the workpiece blank.

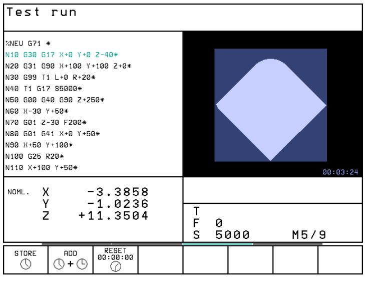
Repeating graphic simulation

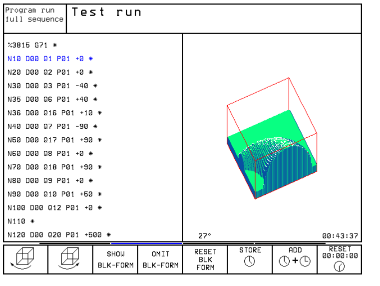
A part program can be graphically simulated as often as desired, either with the complete workpiece or with a detail of it.

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

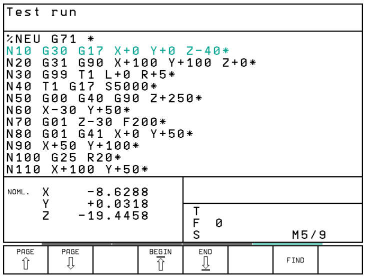
Shift the soft-key rows until the TNC displays the following soft keys with the stopwatch functions:

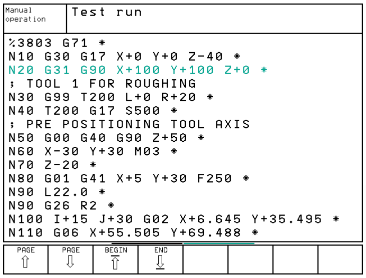
Stopwatch functions	Soft key
Store displayed time	STORE
Display the sum of stored time and displayed time	ADD
Clear displayed time	RESET 00:00:00

The soft keys available to the left of the stopwatch functions depend on the selected screen layout.

11.2 Functions for Program Display in Program Run and Test Run

In the program run modes of operation as well as in the Test Run mode, the TNC provides the following soft keys for displaying a part program in pages:





Function	Soft key
Go back in the program by one screen	PAGE ⇧
Go forward in the program by one screen	PAGE ⇩
Go to the beginning of the program	BEGIN
Go to the end of the program	END

11.3 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:

- 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
- Optional block skip
- Functions for graphic simulation
- Measuring the machining time
- Additional status display

Running a program test

If the central tool file is active, a tool table must be active (status S) to run a program test. Select a tool table via the file manager (PGM MGT) in the Test Run mode of operation.

With the MOD function BLANK IN WORK SPACE, you can activate workspace monitoring for the test run (not TNC 410, see „13 MOD Functions, Showing the Workpiece in the Working Space“).

- ▶ Select the Test Run mode of operation.
- ▶ Call the file manager with the PGM MGT key and select the file you wish to test, or
- ▶ Go to the program beginning: Select line “0” with the GOTO key and confirm you entry with the ENT key.

The TNC then displays the following soft keys:

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

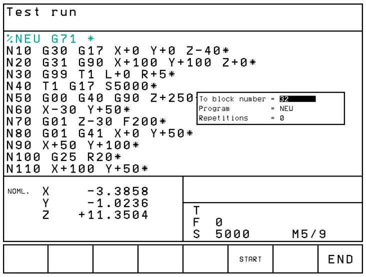
Running a program test up to a certain block

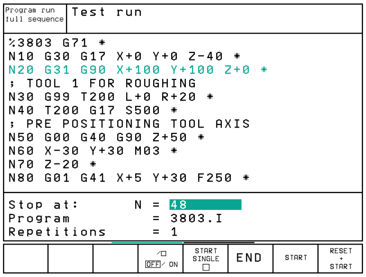
With the STOP AT N function the TNC does a test run up to the block with block number N.

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

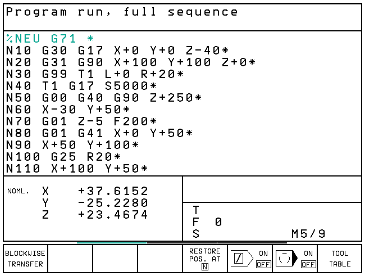
STOP AT N

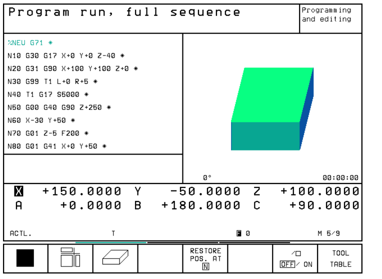
- Stop at N: Enter the block number at which you wish the test to stop.
- Program: If you wish to go into a program that you call with CALL PGM, enter the name of the program containing the block with the selected block number.
- Repetitions: If N is located in a program section repeat, enter the number of repeats that you want to run.
- To test a program section, press the START soft key. The TNC will test the program up to the entered block.





11.4 Program Run





In the Program Run, Full Sequence mode of operation the TNC executes a part program continuously to its end or up to a program stop.

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

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

- Interrupt program run
- Start program run from a certain block
- Optional block skip
- Editing the tool table TOOL.T
- Checking and changing Q parameters
- Superimposing handwheel positioning (not TNC 410)
- Functions for graphic simulation (not TNC 410)
- Additional status display

Running a part program

Preparation

1. Clamp the workpiece to the machine table.
2. Datum setting
3. Select the necessary tables and pallet files (status M).
4. Select the part program (status M).

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

Program Run, Full Sequence

- ▶ Start the part program with the machine START button.

Program Run, Single Block

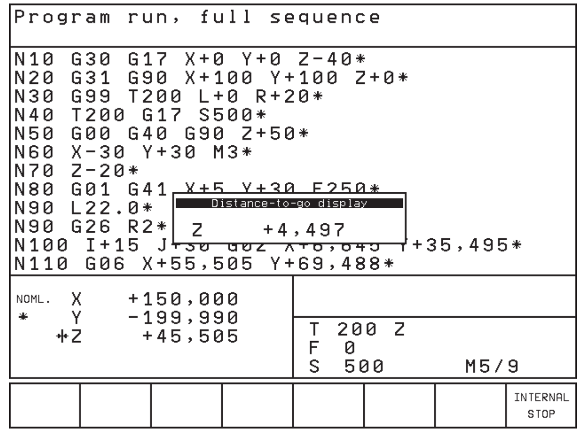
- ▶ Start each block of the part program individually with the machine START button.

Running a part program containing coordinates from non-controlled axes (only TNC 410)

The TNC can also run programs in which you have programmed non-controlled axes.

If the TNC arrives at a block in which you have programmed a non-controlled axis, it stops program run. At the same time it superimposes a window showing the distance-to-go to the target position (see figure at top right). Proceed as follows:

- Move the axis manually to the target position. The TNC constantly updates the distance-to-go window, and always shows the distance remaining to reach the target position.
- Once you have reached the target position, press the NC START key to continue program run. If you press the NC START key before you have arrived at the target position, the TNC will output an error message.



> Machine parameter 1030.x determines how accurately you need to approach the target position (possible input values: 0.001 to 2 mm).
>
> Non-controlled axes must be programmed in separate positioning blocks, otherwise the TNC will output an error message.

Interrupting 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:

- G38
- Miscellaneous function M0, M2 or M30
- Miscellaneous function M6 (determined by the machine tool builder)

To interrupt machining with the machine STOP button:

- ▶ Press the machine 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 INTERNAL 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.

Moving the machine axes during an interruption

You can move the machine axes during an interruption in the same way as in the Manual Operation mode.

TNC 426, TNC 430: Caution - Risk of collision!

If you interrupt program run while the working plane is tilted, you can change from a tilted to a non-tilted coordinate system, and vice versa, by pressing the 3-D ON/OFF soft key.

The functions of the axis direction buttons, the electronic handwheel and the positioning logic for return to contour are then evaluated by the TNC. When retracting the tool make sure the correct coordinate system is active and the angular values of the tilt axes are entered in the 3-D ROT menu.

Example:
Retracting the spindle after tool breakage

- Interrupting machining
- Enable the external direction keys: Press the MANUAL OPERATION soft key.
- Move the axes with the machine axis direction buttons.

Use the function „Returning to the Contour“ (see below) to return to a contour at the point of interruption.

For the TNC 426, TNC 430:

On some machines you may have to press the machine START button after the MANUAL OPERATION soft key to enable the axis direction buttons. Your machine manual provides more detailed information.

Resuming program run after an interruption

> If a program run is interrupted during a fixed cycle, the program must be resumed from the beginning of the cycle. This means that some machining operations will be repeated.
>
> If you interrupted program run during a program section repeat, you can only use GOTO to select other blocks within the program section repeat.

If you interrupt a program run during execution of a subprogram or program section repeat, use the RESTORE POS AT N function to return to the position at which the program run was interrupted.

When a program run is interrupted, the TNC stores:

- The data of the last defined tool
- Active coordinate transformations
- The coordinates of the circle center that was last defined

The stored data are used for returning the tool to the contour after manual machine axis positioning during an interruption (RESTORE POSITION).

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

- The machine STOP button was pressed
- A programmed interruption

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*

▶ Press and hold the END key for two seconds. This induces a TNC system restart.

▶ Remove the cause of the error.

▶ Start again.

If you cannot correct the error, write down the error message and contact your repair service agency.

Mid-program startup (block scan)

The RESTORE POS AT N feature must be enabled and adapted by the machine tool builder. Your machine manual provides more detailed information.

With the RESTORE POS AT N feature (block scan) you can start a part program at any block you desire. The TNC scans the program blocks up to that point. Machining can be graphically simulated.

If you have interrupted a part program with an INTERNAL STOP, the TNC automatically offers the interrupted block N for mid-program startup.

Mid-program startup must not begin in a subprogram.

If the program contains a programmed interruption before the startup block, the block scan is interrupted. Press the machine START button to continue the block scan (TNC 410: press the RESTORE POS. AT N soft key and START).

After a block scan, return the tool to the calculated position with RESTORE POSITION.

For the TNC 426, TNC 430:

All necessary programs, tables and pallet files must be selected in a program run mode of operation (status M).

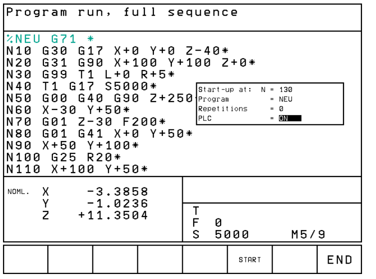
If you are working with nested programs, you can use machine parameter 7680 to define whether the block scan is to begin at block 0 of the main program or at block 0 of the last interrupted program.

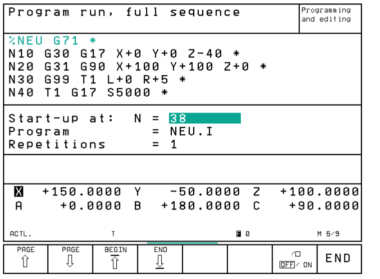
If the working plane is tilted, you can use the 3-D ON/OFF soft key to define whether the TNC is to return to the contour in a tilted or in a non-tilted coordinate system.

- To go to the first block of the current program to start a block scan, enter GOTO "0".
- To select mid-program startup, press the RESTORE POS AT N soft key.

RESTORE POS. AT N

- Start-up at N: Enter the block number N at which the block scan should end.
- Program: Enter the name of the program containing block N.
- Repetitions: If block N is located in a program section repeat, enter the number of repetitions to be calculated in the block scan.
- PLC ON/OFF (not TNC 426, TNC 430): To account for tool calls and miscellaneous functions M: Set the PLC to ON (use the ENT key to switch between ON and OFF). If PLC is set to OFF, the TNC considers only the geometry.
- To start block scan:
  TNC 426, TNC 430: Press the machine START key
  TNC 410: Press the START soft key
- To return to the contour, proceed as described below in "Returning to the contour."





Returning to the contour

With the RESTORE POSITION function, the TNC returns to the workpiece contour in the following situations:

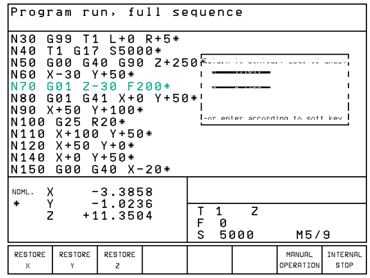
- Return to the contour after the machine axes were moved during a program interruption that was not performed with the INTERNAL STOP function.
- Return to the contour after a block scan with RESTORE POS AT N, for example after an interruption with INTERNAL STOP.
- **For the TNC 426, TNC 430 with NC software 280 474-xx,**
  Depending on the machine, if the position of an axis has changed after the control loop has been opened during a program interruption:

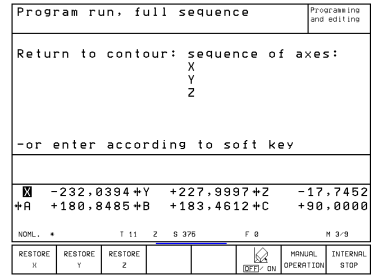
▶ To select a return to contour, press the RESTORE POSITION soft key.

▶ To move the axes in the sequence that the TNC suggests on the screen, press the machine START button.

▶ To move the axes in any sequence, press the soft keys RESTORE X, RESTORE Z, etc., and activate each axis with the machine START key.

▶ To resume machining, press the machine START key.





11.5 Blockwise Transfer: Running Long Programs (not with TNC 426, TNC 430)

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 G99 blocks. If you require more tools then use a tool table.

If the program contains a %... block, the called program must be stored in the TNC memory.

The program may not include:

- Subprograms
- Program section repeats
- Function D15:PRINT

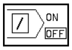
Blockwise program transfer

Configure the data interface with the MOD function (see „13.5 Setting the External Data Interface").

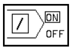
- ▶ Select the Program Run, Full Sequence mode or the Program Run, Single Block mode.
- ▶ Begin blockwise transfer: Press the BLOCKWISE TRANSFER soft key
- ▶ Enter the program name and confirm your entry with the ENT key. The TNC reads in the selected program via data interface
- ▶ Start the part program with the machine START button.

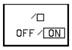
11.6 Optional block skip

In a test run or program run, the TNC can skip over blocks that begin with a slash (/):



▶ To run or test the program with the blocks preceded by a slash, set the soft key to OFF





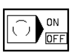
▶ To run or test the program without the blocks preceded by a slash, set the soft key to ON

This function does not work for TOOL DEF blocks.

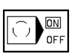
After a power interruption the TNC returns to the most recently selected setting.

11.7 Optional Program Run Interruption (not TNC 426, TNC 430)

The TNC optionally interrupts the program or test run at blocks containing M01. If you use M01 in the Program Run mode, the TNC does not switch off the spindle or coolant.



▶ Do not interrupt program run or test run at blocks containing M01: Set soft key to OFF



▶ Interrupt program run or test run at blocks containing M01: Set soft key to ON

12

3-D Touch Probes

12.1 Touch Probe Cycles in the Manual and Electronic Handwheel

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

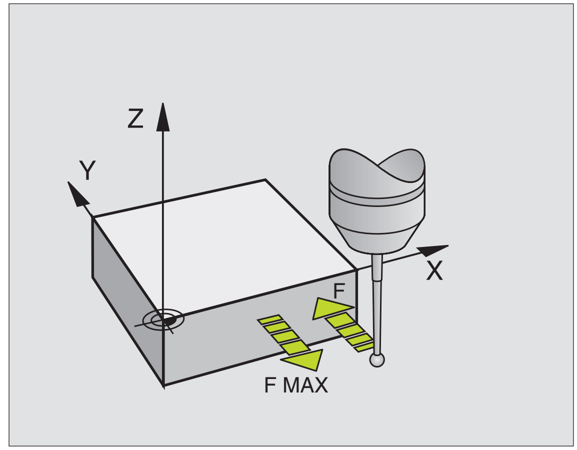
If you are taking measurements during program run, be sure that the tool data (length, radius, axis) can be used from the calibrated data or from the last T block (selected with MP7411).

Note also with TNC 426, TNC 430:

If you are working alternately with a triggering and a measuring touch probe, be sure that

- the correct touch probe is selected with MP 6200
- the measuring and the triggering touch probes are never connected to the control at the same time.

The TNC cannot automatically recognize which probe is actually in the spindle.



After you press the machine 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,
- 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 defines the distance for a triggering touch probe, MP6330 for a measuring touch probe).

To select the touch probe functions:

▶ Select the Manual Operation or Electronic Handwheel mode of operation

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

Function	Soft key
Calibrate the effective length	CAL L
Calibrate the effective radius	CAL R, CAL 3D
Basic rotation	PROBING ROT
Datum setting	PROBING POS
Set the datum at a corner	PROBING P
Set the datum at a circle center	PROBING CC

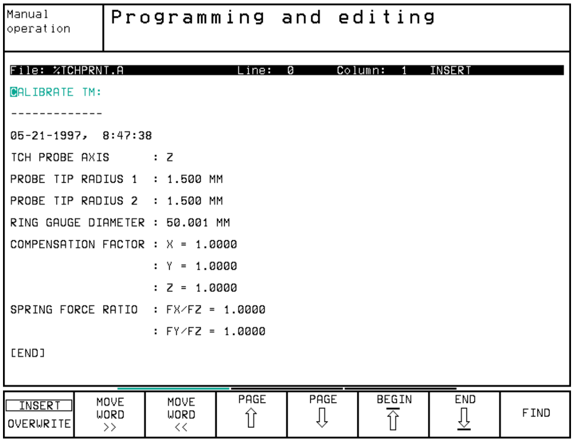
Recording the values measured in a probe cycle (not TNC 410)

The TNC must be specially prepared by the machine tool builder for use of this function. Refer to your machine tool manual.

After executing any selected touch probe cycle, the TNC displays the soft key PRINT. If you press this soft key, the TNC will record the current values determined in the active probe cycle. You can then use the PRINT function in the menu for setting the data interface (see ”13 MOD Functions, Setting the Data Interface”) to define whether the TNC is to

- print the measuring result,
- store the measuring results on the TNC’s hard disk, or
- store the measuring results on a PC.

If you store the measuring results, the TNC creates the ASCII file %TCHPRNT.A (see figure at top right). Unless you define a specific path and interface in the interface configuration menu, the TNC will store the %TCHPRNT file in the main directory TNC:\ .



When you press the PRINT soft key, the %TCHPRNT.A file must not be active in the Programming and Editing mode of operation. The TNC will otherwise display an error message.

The TNC stores the measured data in the %TCHPRNT.A file only. If you execute several touch probe cycles in succession and want to store the resulting measured data, you must make a backup of the contents stored in %TCHPRNT.A between the individual cycles by copying or renaming the file.

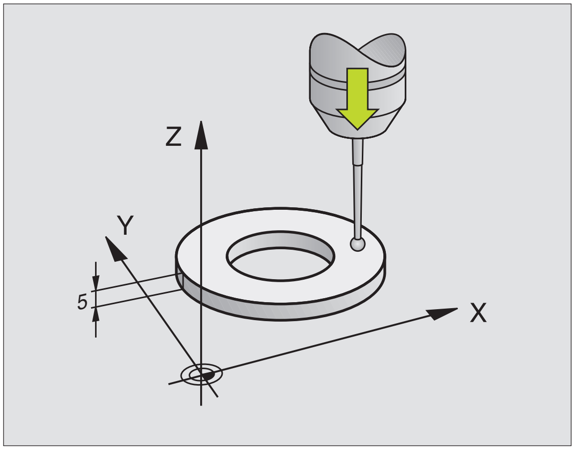
Format and contents of the %TCHPRNT file are preset by the machine tool builder.

Writing the values from probe cycles in datum tables (not TNC 410)

Using the ENTER IN DATUM TABLE soft key, the TNC can write the values measured during a probe cycle in a datum table:

- ▶ Select any probe function
- ▶ Enter the name of the datum table (complete path) in the datum table input box and confirm with ENT
- ▶ Enter the number of the datum in the Datum number = input box and confirm with ENT
- ▶ Press the ENTER IN DATUM TABLE soft key. The TNC writes the data into the indicated datum table.

Calibrating a touch trigger probe



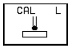
The touch probe must be calibrated:

- during 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.

▶ Enter the tool axis (with the axis key).

▶ Datum: Enter the height of the ring gauge.

▶ The menu items Effective ball radius and Effective length do not require input.

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

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

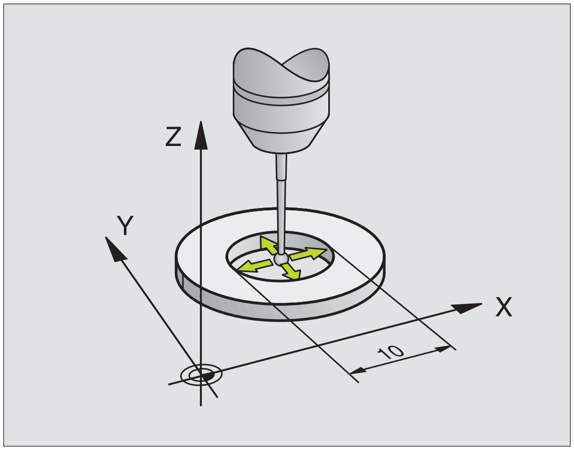
▶ To probe the upper surface of the ring gauge, press the machine START button.

Calibrating the effective radius and compensating center misalignment

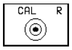
After the touch probe is inserted it normally needs to be exactly aligned with the spindle axis. The misalignment is measured with this calibration function and compensated electronically.

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.

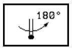


▶ To select the calibration function for the ball-tip radius and the touch probe center misalignment, press the CAL R soft key.

▶ Select the tool axis and enter the radius of the ring gauge.

▶ To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the effective ball-tip radius.

▶ If you want to terminate the calibration function at this point, press the END soft key.

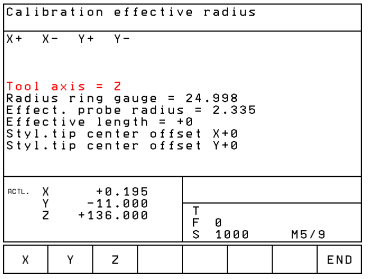


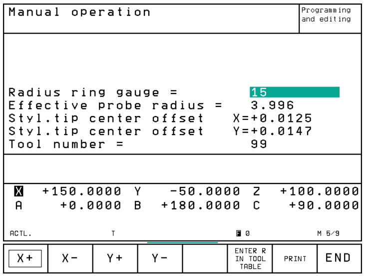
▶ If you want to determine the ball-tip center misalignment, press the 180° soft key. The TNC rotates the touch probe by 180°.

▶ To probe the workpiece, press the machine START button four times. The touch probe contacts a position on the bore in each axis direction and calculates the ball-tip center misalignment.

Displaying calibration values

The TNC stores the effective length and radius, 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.





Storing calibration values in the TOOL.T tool table (not TNC 410)

This function is only available if machine parameter 7411 = 1 is set (Activate touch probe data with tool call T).

If you record measurements during program run, the compensation data for the touch probe can be activated from the tool table via a tool call T. To store the calibration data in the TOOL.T tool table, enter the tool number in the calibration menu (confirm with ENT) and then press the ENTER R IN TOOL TABLE or the ENTER L IN TOOL TABLE soft key.

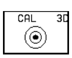
Calibrating a measuring touch probe (not TNC 410)

> If the TNC displays the error message "Stylus already in contact," select the menu for 3-D calibration and then select the RESET 3D soft key.
>
> The measuring touch probe must be calibrated whenever the machine parameters for 3-D touch probes are changed.
>
> The effective length is calibrated in the same way as with triggering touch probes. You must also enter tool radius R2 (corner radius).
>
> With MP6321 you can define whether the TNC should probe to find the stylus center.

The 3-D calibration cycle for measuring touch probes enables you to measure a standard ring gauge fully automatically. (The standard ring gauge is available from HEIDENHAIN). Fix the standard ring gauge to the machine table with fixing clamps.

From the data measured during calibration, the TNC calculates the spring rate of the touch probe, the stylus deflection and the stylus center misalignment. At the end of the calibration cycle, the TNC automatically stores these values in the input menu.

- In the Manual Operation mode, position the touch probe to a position approximately in the center of the standard ring gauge and set it to 180°.

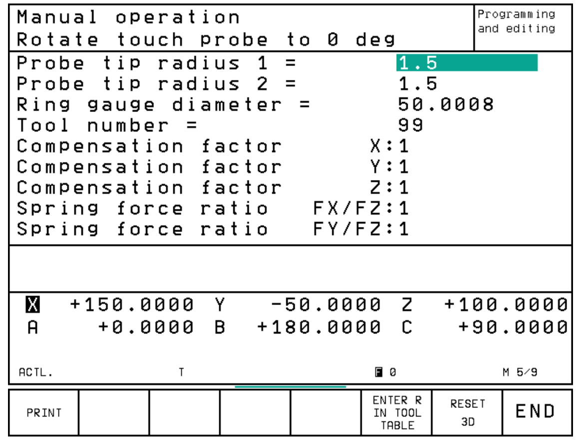


- Select the 3-D calibration cycle. Press the soft key CAL 3D.
- Enter the values for stylus radius 1 and stylus radius 2. Enter the same value for stylus radius 1 and 2 if you are using a stylus with ball tip. Enter different values for stylus radius 1 and 2 if you are using a stylus with a corner radius.
- Diameter ring gauge: The diameter is engraved on the standard ring gauge.
- To start the calibration cycle, press the machine START button: The touch probe measures the standard ring gauge in a programmed sequence of steps.
- Rotate the touch probe to 0° as soon as the TNC asks you to.
- To start the calibration cycle once again to determine center misalignment, press the machine START button. The touch probe again measures the standard ring gauge in a programmed sequence of steps.

Displaying calibration values

The compensation factors and force ratios are stored in the TNC for later use whenever the measuring touch probe is needed.

You can display the stored values on the screen by pressing the 3D CAL soft key.



Storing calibration values in the TOOL.T tool table

This function is only available if machine parameter 7411 = 1 is set (Activate touch probe data with tool call T).

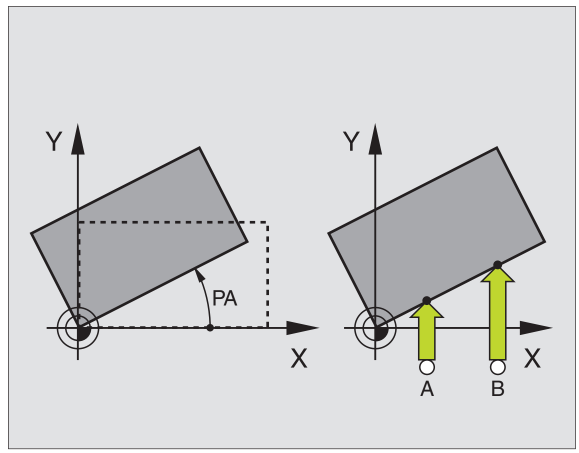
If you conduct measurements during program run, the compensation data for the touch probe can be activated from the tool table via a TOOL CALL. To store the calibration data in the TOOL.T tool table, enter the tool number in the calibration menu (confirm with ENT) and then press the ENTER R IN TOOL TABLE soft key.

The TNC stores the stylus radius 1 in the R column, and the stylus radius 2 in the R2 column.

Compensating workpiece misalignment

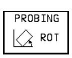
The TNC electronically compensates workpiece misalignment by computing a "basic rotation."

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 center 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.

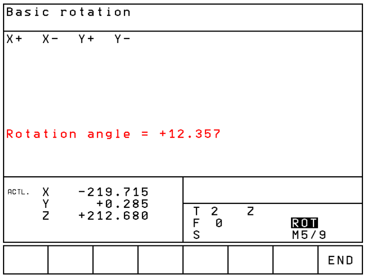


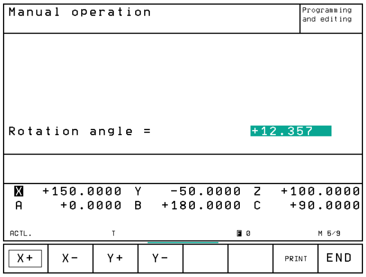
- Select the probing function by pressing the PROBING ROT soft key.
- Position the ball tip at a starting position near the first touch point.
- Select the probe direction perpendicular to the angle reference axis: Select the axis by soft key.
- To probe the workpiece, press the machine START button.
- Position the ball tip at a starting position near the second touch point.
- To probe the workpiece, press the machine START button.

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

Displaying a basic rotation
The angle of the basic rotation appears after ROTATION ANGLE whenever PROBING ROT is selected. The TNC also displays the rotation angle in the additional status display (STATUS POS.) .

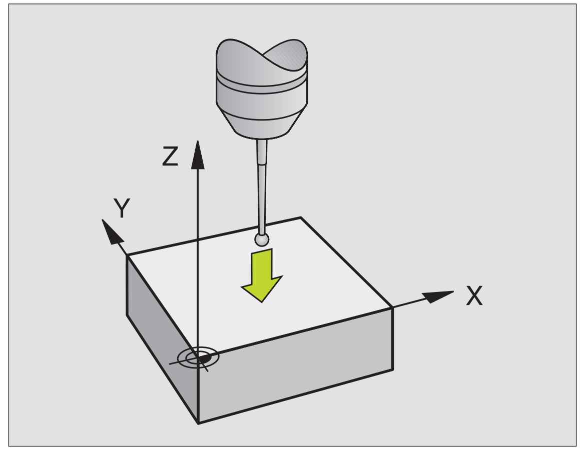
In the status display a symbol is shown for a basic rotation whenever the TNC is moving the axes according to a basic rotation.





To cancel a basic rotation:
- Select the probing function by pressing 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.

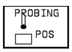
12.2 Setting the Datum with a 3-D Touch Probe



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

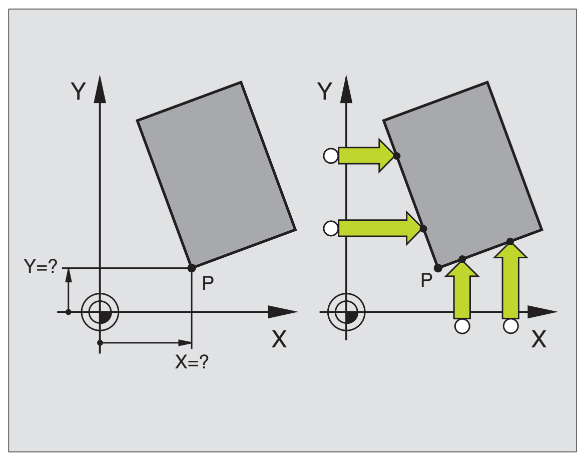
- Datum setting in any axis with PROBING POS
- Defining a corner as datum with PROBING P
- Setting the datum at a circle center with PROBING CC

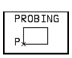
To set the datum in any axis (see figure at upper right)



- ▶ 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 via soft keys.
- ▶ To probe the workpiece, press the machine 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 right)





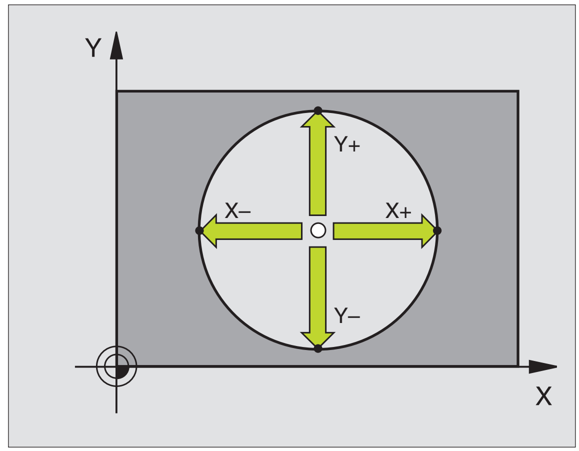
- ▶ Select the touch probe function: Press the PROBING P soft key
- ▶ Touch points of basic rotation ?: Press ENT to transfer the touch point coordinates to memory.
- ▶ Position the touch probe at a starting position near the first touch point of the side that was not probed for basic rotation.
- ▶ Select the probe direction with a soft key.
- ▶ To probe the workpiece, press the machine START button.
- ▶ Position the touch probe near the second touch point on the same side.
- ▶ To probe the workpiece, press the machine 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 ENT 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 coordinates of the datum 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.

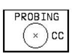


Inside circle

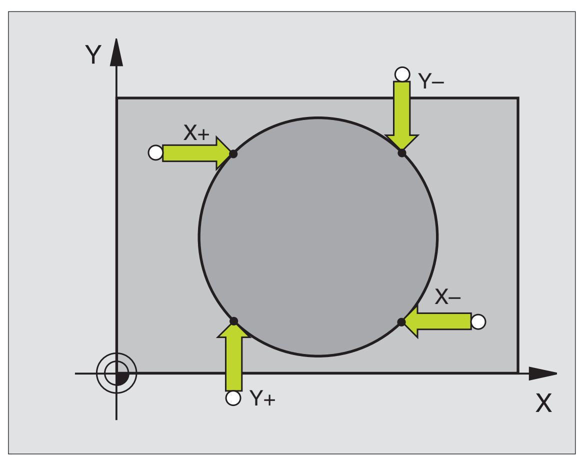
The TNC automatically probes the inside wall in all four coordinate axis directions.

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 machine START button four times. The touch probe touches four points on the inside of the circle.
- If you are probing to find the stylus center (only 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.



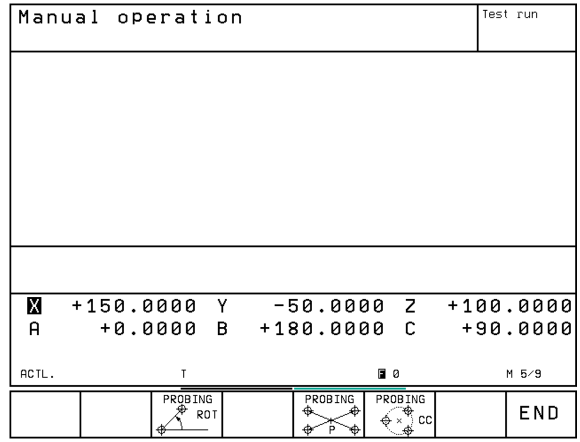
Outside circle

- 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 machine START button.
- Repeat the probing process for the remaining three points.See figure at lower right.
- Enter the coordinates of the datum and confirm your entry with ENT.

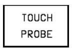
After the probing procedure is completed, the TNC displays the coordinates of the circle center and the circle radius PR.

Setting datums using holes (not TNC 410)

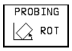
A second soft-key row provides soft keys for using holes or cylindrical studs to set datums.



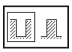
Define whether a hole or stud is to be probed



▶ Select the probing functions with the TOUCH PROBE soft key, shift the soft-key row



▶ Select the probing functions for holes: e.g., press the PROBING ROT soft key.

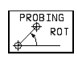
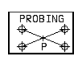
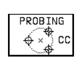


▶ Select holes or cylindrical studs: the selected element appears in a box.

Probing holes
Pre-position the touch probe approximately in the center of the hole. After you have pressed the external START key, the TNC automatically probes four points on the wall of the hole.

Move the touch probe to the next hole and have the TNC repeat the probing procedure until all the holes have been probed to set datums.

Probing cylindrical studs
Position the ball tip at a starting position near the first touch point of the stud. Select the probing direction by soft key and press the machine START button to start probing. Perform the above procedure four times.

Application	Soft key
Basic rotation from 2 holes: The TNC measures the angle between the line connecting the centers of two holes and a nominal position (angle reference axis).	
Basic rotation from 4 holes: The TNC calculates the intersection of the line connecting the first two probed holes with the line connecting the last two probed holes. You need to probe diagonally opposite holes after one another (as shown on the soft key), as otherwise the datum calculated by the TNC will be incorrect	
Circle center from 3 holes: The TNC calculates a circle that intersects the centers of all three holes, and finds the center.	

12.3 Measuring Workpieces with a 3-D Touch Probe

The TNC 426 and TNC 430 offer a number of convenient measuring cycles that enable you to measure workpiece dimensions. These cycles are described in a separate User's Manual. Contact HEIDENHAIN if you would like a copy of the Touch Probe Cycle Manual.

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:

PROBING POS

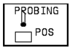
- ▶ To select the touch probe function: Press the PROBING POS soft key
- ▶ Move the touch probe to a starting position near the touch point.
- ▶ Select the probe direction and axis of the coordinate. Use the corresponding soft keys for selection.
- ▶ To probe the workpiece, press the machine START button.

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

Finding the coordinates of a corner in the working plane

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

Measuring workpiece dimensions



PROBING POS

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 probing direction with a soft key.
- ▶ To probe the workpiece, press the machine START button.
- ▶ If you will need the current datum later, write down the value that appears in the Datum display.
- ▶ Datum: Enter “0”.
- ▶ To terminate the dialog, press the END key.
- ▶ Select the 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 the soft keys: Same axis but opposite direction from A.
- ▶ To probe the workpiece, press the machine START button.

The value displayed as Datum is the distance between the two points on the coordinate axis.

To return to the datum that was active before the length measurement:

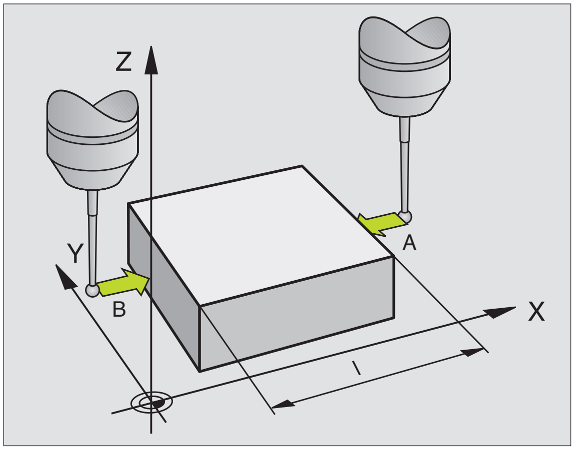
- ▶ To select the touch probe function: Press the PROBING POS soft key.
- ▶ Probe the first touch point again.
- ▶ Set the Datum to the value that you wrote down previously.
- ▶ To terminate the dialog, press the END key.

Measuring angles

You can 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.

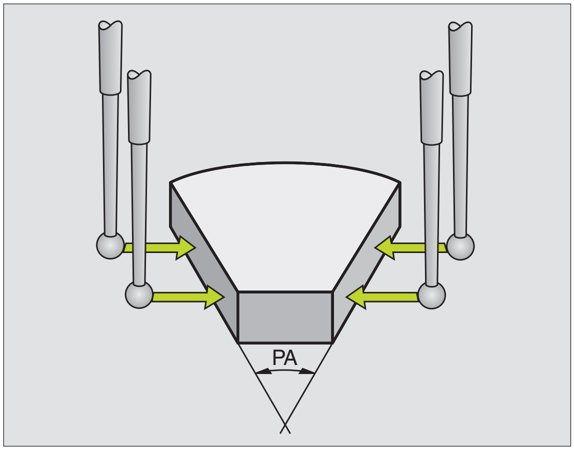
The measured angle is displayed as a value of maximum 90°.



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

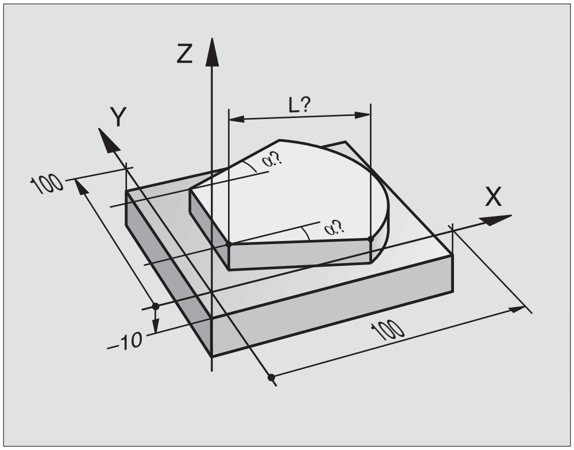
PROBING ROT

- 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:

- Select the probing function by pressing the PROBING ROT soft key.
- Rotation angle: If you will need the current basic rotation later, write down the value that appears under Rotation angle.
- Make a basic rotation 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.



Measuring with the 3-D touch probe during program run

The 3-D touch probe can measure positions on the workpiece while the program is being run — even if the working plane is tilted. Applications:

- Measuring differences in the height of cast surfaces
- Tolerance checking during machining

To program the use of a touch probe, use the G55 function in the Programming and Editing mode of operation. The TNC pre-positions the touch probe to automatically probe the desired position. During probing, the TNC moves the touch probe parallel to the machine axis that was defined in the touch probe cycle. The TNC takes an active basic rotation or rotation only into account for calculating the touch point. The coordinate of the touch point is stored in a Q parameter. The TNC interrupts the probing process if the stylus is not deflected within a certain distance (selectable via MP 6130). Upon contact, the position coordinates of the south pole of the probe tip are also stored in the parameters Q115 to Q119. The values in these parameters do not include the stylus length and radius.

To increase measuring accuracy, you can set in machine parameter 6170 how often you wish the TNC to repeat the probing cycle. If the deviation between the individual measurements exceeds the acceptable range (MP 6171), the TNC outputs an error message.

Pre-position the touch probe manually to avoid a collision when the programmed pre-positioning point is approached.

Be sure to use the tool data (length, radius, axis) either from the calibrated data or from the last G99 block. Selection is made with machine parameter MP7411.

G 55

- Select the probing function and confirm with ENT
- Parameter number for result: Enter the number of the Q parameter to which you want to assign the coordinate.
- Probing axis/Probing direction: Enter the probing axis with the axis selection keys or ASCII keyboard and the algebraic sign for the probing direction. Press ENT to confirm.
- Position value: Use the axis selection keys or the ASCII keyboard to enter all coordinates of the nominal pre-positioning point values for the touch probe.
- To conclude input, press the ENT key.

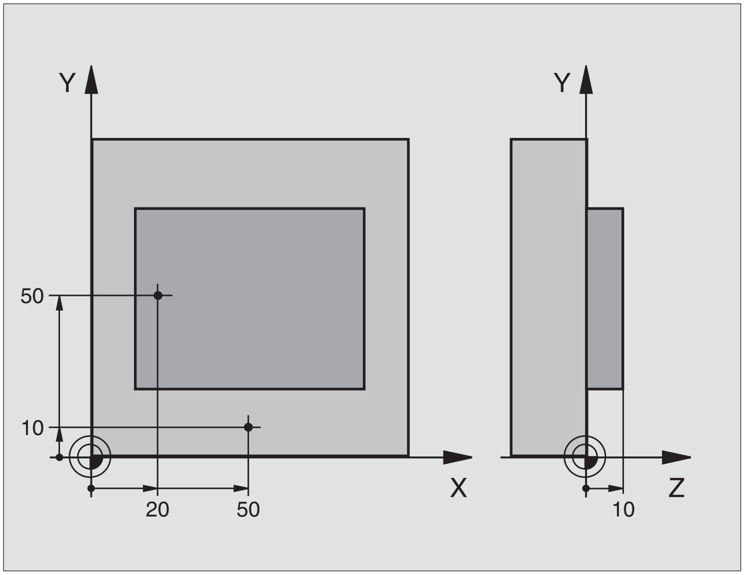
Example NC block

```
N670 G55 P01 Q5 P02 X– X+5 Y+0 Z–5 *
```

Example: Measuring the height of an island on a workpiece

Program sequence

- Assign the program parameters
- Measure height with Cycle G55
- Calculate the height



`%3DPROBE G71 *`	
`N10 D00 Q11 P01 +20 *`	1st touch point: X coordinate
`N20 D00 Q12 P01 +50 *`	1st touch point: Y coordinate
`N30 D00 Q13 P01 +10 *`	1st touch point: Z coordinate
`N40 D00 Q21 P01 +50 *`	2nd touch point: X coordinate
`N50 D00 Q22 P01 +10 *`	2nd touch point: Y coordinate
`N60 D00 Q23 P01 +0 *`	2nd touch point: Z coordinate
`N70 T0 G17 *`	Call touch probe
`N80 G00 G40 G90 Z+250 *`	Retract touch probe
`N90 X+Q11 Y+Q12 *`	Pre-position touch probe
`N100 G55 P01 10 P02 Z-`	Measure upper surface of the workpiece
`X+Q11 Y+Q12 Z+Q13 *`	
`N110 X+Q21 Y+Q22 *`	Pre-position for second measurement
`N120 G55 P01 20 P02 Z- Z+Q23 *`	Measure the depth
`N130 D02 Q1 P01 +Q20 P02 +Q10 *`	Calculate the absolute height of the island
`N140 G38 *`	Interrupt program run: check Q1
`N150 G00 G40 Z+250 M2 *`	Retract in the tool axis, end program
`N999999 %3DPROBE G71 *`	

13

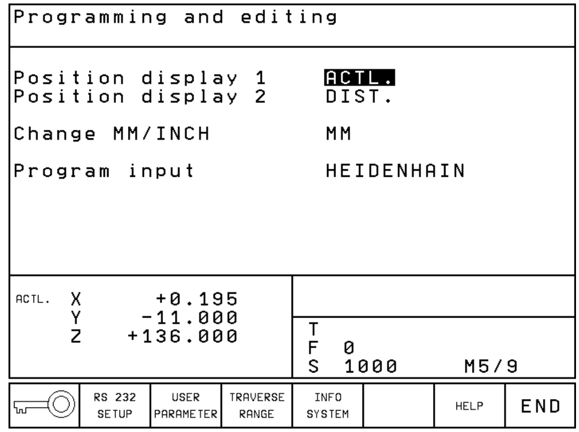
MOD Functions

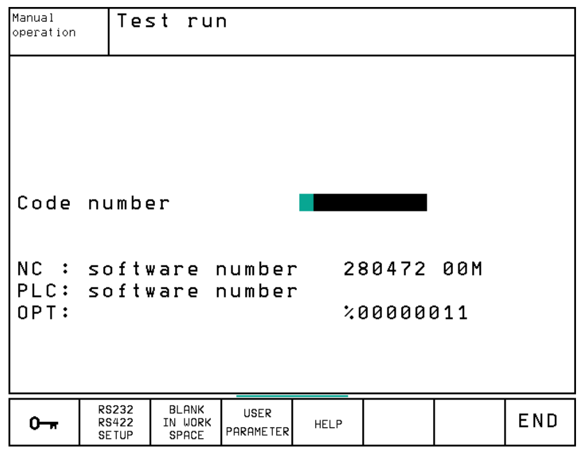
13.1 Selecting, Changing and Exiting the MOD Functions

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

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

▶ To select the MOD functions, press the MOD key. Figure at top right: MOD function on the TNC 410. Figure at center right and on the next page: MOD function on the TNC 426, TNC 430 for test run and in a manual operating mode.





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

There are several possibilities for changing a setting, depending on the function selected:

- Enter a numerical value directly, e.g. when determining traverse range limit
- Change a setting by pressing the ENT key, e.g. when setting program input
- Change a setting via a selection window (not TNC 410): If there is more than one possibility for a particular setting available, you can superimpose a window listing all of the given possibilities by pressing the GOTO key. Select the desired setting directly by pressing the corresponding numerical key (to the left of the colon), or using the arrow keys and then confirming with ENT. If you don't want to change the setting, close the window again with END.

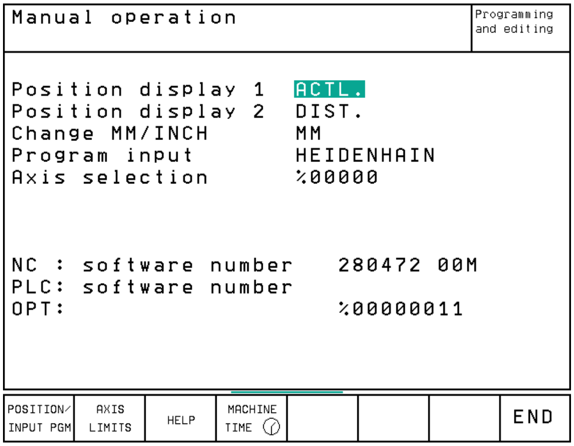
To exit the MOD functions:
▶ Close the MOD functions with the END soft key or key.

Overview of MOD Functions TNC 426, TNC 430
Depending on the selected mode of operation, you can make the following changes:

Programming and Editing:
- Display NC software number
- Display PLC software number
- Enter code number
- Set data interface
- Machine-Specific User Parameters
- HELP files (if provided)

Test Run:
- Display NC software number
- Display PLC software number
- Enter code number
- Setting the Data Interface
- Showing the Workpiece in the Working Space
- Machine-Specific User Parameters
- HELP files (if provided)

In all other modes:
- Display NC software number
- Display PLC software number
- Display code digits for installed options
- Select position display
- Unit of measurement (mm/inches)
- Programming language for MDI
- Select the axes for actual position capture
- Axis traverse limits
- Display datums
- Display operating time
- HELP files (if provided)



13.2 System Information (not TNC 426, TNC 430)

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

- Free program memory
- NC software number
- PLC software numbers are displayed on the TNC screen after the functions have been selected. Directly below them are the code numbers for the installed options (OPT:):
- Options (if present), e.g. digitizing

13.3 Software Numbers and Option Numbers TNC 426, TNC 430

The software numbers of the NC and PLC are displayed in the MOD function opening screen. Directly below them are the code numbers for the installed options (OPT:):

- No option OPT: 00000000
- Option for digitizing with triggering touch probe OPT: 00000001
- Option for digitizing with measuring touch probe OPT: 00000011

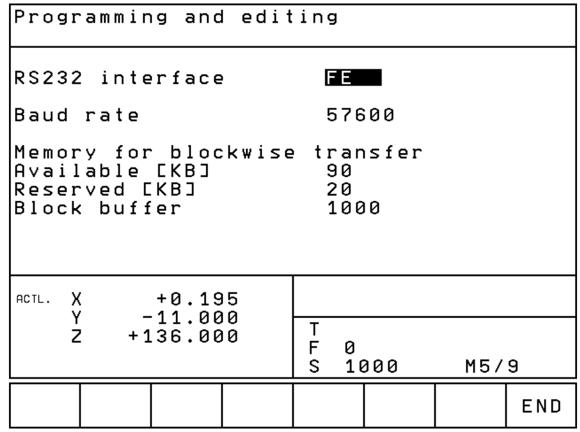
13.4 Code Number

To enter the code number on the TNC 410, press the soft key with the key symbol. The TNC requires a code number for the following functions:

Function	Code number
Select user parameters	123
Enabling special functions for Q parameter programming	555343
Removing file protection (only TNC 410)	86357
Operating hours counter for (only TNC 410): CONTROL ON PROGRAM RUN SPINDLE ON	857282
Configuring an Ethernet card (not TNC 410)	NET123

13.5 Setting the Data Interface for the TNC 410

Press the soft key marked RS 232-SETUP to call a menu 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 TNCremo	EXT1, EXT2
PC with HEIDENHAIN software TNCremo	FE
No data transfer; e.g. digitizing without position value capture, or working without an external device	none

Setting the BAUD RATE

You can set the BAUD-RATE (data transfer speed) from 110 to 115,200 baud. For each operating mode (FE, EXT1 etc.), the TNC stores and individual BAUD RATE.

Creating the memory for blockwise transfer

In order to be able to edit other programs while blockwise execution is in progress, you need to create a memory for blockwise transfer.

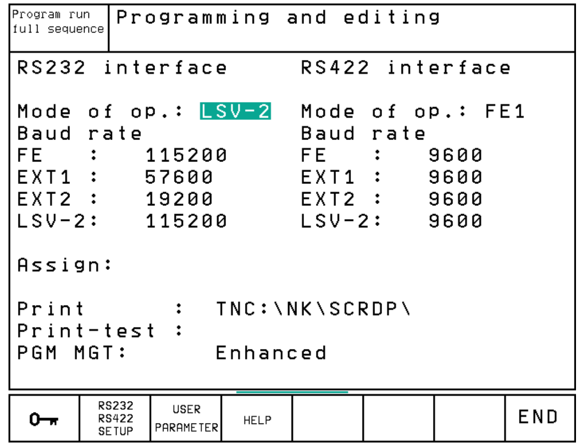
The TNC shows the available free memory space. The reserved memory space should be less than the total free memory space available.

Setting the block buffer

To ensure a continuous program run during blockwise transfer, the TNC needs a certain quantity of blocks stored in program memory.

In the block buffer you define how many NC blocks are read in through the data interface before the TNC begins the program run. The input value for the block buffer depends on the point intervals in the part program. For very small point intervals, enter a large block buffer. For large point intervals, enter a small block buffer. Proposed value: 1000

13.6 Setting Up the Data Interfaces for TNC 426, TNC 430



Press the soft key marked RS 232 / RS 422 SETUP to call a menu for setting the data interfaces:

Setting the RS-232 interface

The mode of operation and baud rates for the RS-232 interface are entered in the upper left of the screen.

Setting the RS-422 interface

The mode of operation and baud rates for the RS-422 interface are entered in the upper right of the screen.

Setting the OPERATING MODE of the external device

The functions "Transfer all files," "Transfer selected file," and "Transfer directory" are not available in the operating modes FE2 and EXT.

Setting the BAUD RATE

You can set the BAUD-RATE (data transfer speed) from 110 to 115,200 baud.

External device	Operating mode	Symbol
HEIDENHAIN floppy disk units FE 401 B FE 401 from prog. no. 230 626 03	 FE1 FE1	
HEIDENHAIN floppy disk unit FE 401 up to prog. no. 230 626 02	FE2	
PC with HEIDENHAIN data transfer Software TNCremo	FE1	
Non-HEIDENHAIN devices such as Punchers, PC without TNCremo	EXT1, EXT2	
PC with HEIDENHAIN software TNCremo for remote operation of the TNC	LSV2	

ASSIGN

This function sets the destination for the transferred data.

Applications:

- Transferring values with Q parameter function D15
- Path on the TNC's hard disk in which the digitized data are stored

The TNC mode of operation determines whether the PRINT or PRINT TEST function is used:

TNC mode of operation	Transfer function
Program Run, Single Block	PRINT
Program Run, Full Sequence	PRINT
Test run	PRINT TEST

You can set PRINT and PRINT TEST as follows:

Function	Path
Output data via RS-232	RS232:\....
Output data via RS-422	RS422:\....
Save data to the TNC's hard disk	TNC:\....
Save data in directory in which the program with D15 or the program with the digitizing cycles is located	- vacant -

File names

Data	Operating mode	File name
Digitizing data	Program Run	Defined in the RANGE cycle
Values with FN15	Program Run	%D15RUN.A
Values with FN15	Test run	%D15SIM.A

13.7 Software for Data Transfer

For transfer of files to and from the TNC, we recommend using the HEIDENHAIN TNCremo data transfer software. With TNCremo, data transfer is possible with all HEIDENHAIN controls via serial interface.

> Please contact your HEIDENHAIN agent if you would like to receive the TNCremo data transfer software for a nominal fee.

System requirements for TNCremo

- AT personal computer or compatible system
- 640 KB working memory
- 1 MB free memory space on your hard disk
- One free serial interface
- Operating system MS-DOS/PC-DOS 3.00 or later, Windows 3.1 or later, OS/2
- A Microsoft-compatible mouse (for ease of operation, not essential)

Installation under Windows

- ▶ Start the SETUP.EXE installation program in the file manager (explorer)
- ▶ Follow the instructions of the setup program

Starting TNCremo under Windows

Windows 3.1, 3.11, NT:

- ▶ Doubleclick on the icon in the program group HEIDENHAIN Applications

Windows 95:

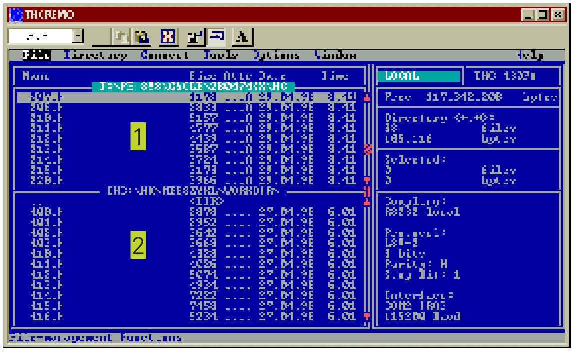
- ▶ Click on <Start>, <Programs>, <HEIDENHAIN Applications>, <TNCremo>

When you start TNCremo for the first time, you will be asked for the type of control you have connected, the interface (COM1 or COM2) and the data transfer speed. Enter the necessary information.

Data transfer between the TNC 410 and TNCremo
Ensure that:

- The TNC is connected to the correct serial port on your PC
- The data transfer speed set on the TNC is the same as that set on TNCremo

Once you have started TNCremo, you will see a list of all of the files that are stored in the active directory on the left of the window Using the menu items <Directory>, <Change>, you can change the active directory or select another directory. To start data transfer at the TNC (see section 4.5 "File Management TNC 410"), select <Connect>, <File server>. TNCremo is now ready to receive data.



Data transfer between TNC 426, TNC 430 and TNCremo
Ensure that:

- The TNC is connected to the correct serial port on your PC
- The data transfer speed set on the TNC for LSV2 operation is the same as that set on TNCremo.

Once you have started TNCremo, you will see a list of all of the files that are stored in the active directory on the left of main window 1 . Using the menu items <Directory>, <Change>, you can change the active directory or select another directory on your PC.

To establish the connection with your TNC, select the items <Connect>, <Link (LSV-2)>. The TNCremo now receives the file and directory structure from the TNC and displays this at the bottom left of the main window (2). To transfer a file from the TNC to the PC, select the file in the TNC window (highlighted with a mouse click) and activate the functions <File> <Transfer>.

To transfer a file from the PC to the TNC, select the file in the PC window and activate the functions <File> <Transfer>.

EndTNCremo
Select the menu items <File>, <Exit>, or press the key combination ALT+X

Refer also to the TNCremo help texts where all of the functions are explained in more detail.

13.8 Ethernet Interface (only TNC 426, TNC 430)

Introduction

As an option, you can equip the TNC with an Ethernet card to connect the control as a **Client** in your network. The TNC transmits data through the Ethernet card in accordance with the Transmission Control Protocol/Internet Protocol (TCP/IP) family of protocols and with the aid of the Network File System (NFS). Since TCP/IP and NFS are implemented in UNIX systems, you can usually connect the TNC in the UNIX world without any additional software.

The PC world with Microsoft operating systems, however, also works with TCP/IP, but not with NFS. You will therefore need additional software to connect the TNC to a PC network. HEIDENHAIN recommends the following network software:

Operating System	Network Software
DOS, Windows 3.1, Windows 3.11, Windows NT	Maestro 6.0, from HUMMINGBIRD e-mail: support@hummingbird.com www: http:\\www.hummingbird.com
Windows 95	OnNet Server 2.0, from the FTP company e-mail: support@ftp.com www: http:\\www.ftp.com

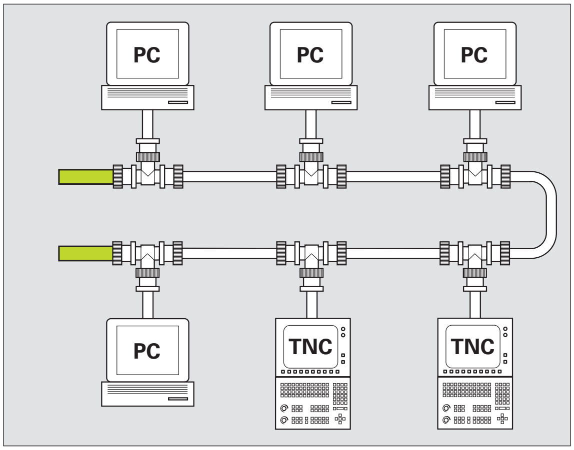
Installing an Ethernet card

Switch-off the TNC and the machine before you install an Ethernet card!

Read the installation instruction supplied with the Ethernet card!

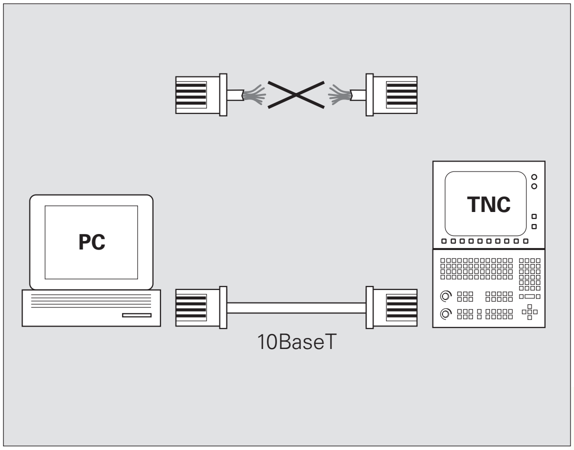
Connection Possibilities

You can connect the Ethernet card in your TNC to your network through a BNC connection (X26, coax cable 10Base2) or through the RJ45 connection (X25, 10BaseT). You can only use one of the two connections at one time. Both connections are metallically isolated from the control electronics.



BNC connection X26 (coaxial cable 10Base2, see figure at upper right)
The 10Base2 connection is also commonly known as Thin-Ethernet or CheaperNet. For the 10Base2 cable you need a BNC-T connector to connect the TNC to your network.

> The distance between two T-connectors must be at least 0.5 meters (1.7 ft).
>
> The number of T-connectors must not exceed 30.
>
> Open ends of the bus must be provided with terminal resistors of 50 ohms.
>
> The maximum cable segment length, i.e., the distance between two terminating resistors, is 185 m (600 ft). You can connect up to 5 cable segments with each other via signal amplifier (repeater).

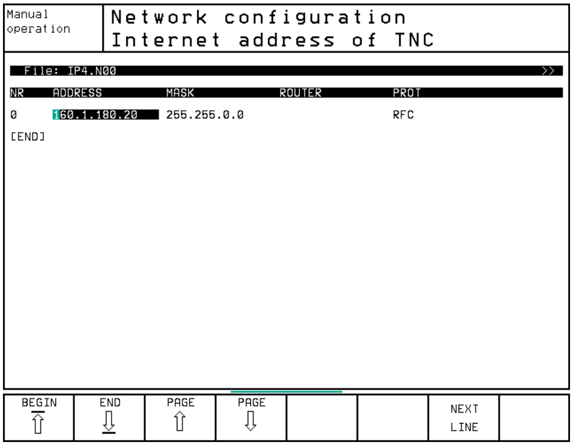


RJ45 connection X25 (10BaseT, see figure at center right)
For a 10BaseT connection you need a Twisted Pair cable to connect the TNC to your network.

> For unshielded cable, the maximum cable length between the TNC and a node is 100 meters (329 ft). For shielded cable, it is 400 meters (1300 ft).
>
> If you connect the TNC directly with a PC you must use a transposed cable.

Configuring the TNC

Make sure that the person configuring your TNC is a network specialist.



- In the Programming and Editing mode of operation, press the MOD key. Enter the code word NET123. The TNC will then display the main screen for network configuration.

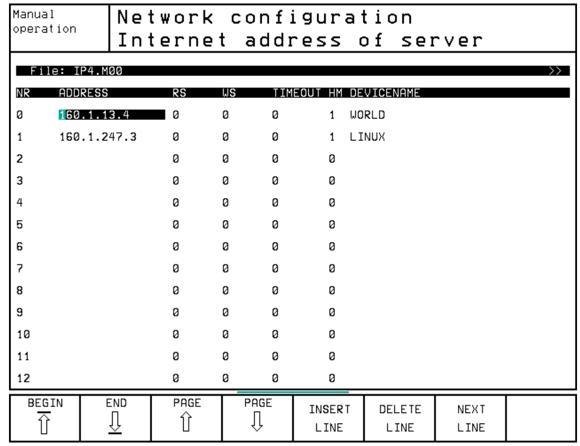
General network settings

- Press the DEFINE NET soft key to enter the general network settings (see figure at upper right) and enter the following information:

Setting	Meaning
ADDRESS	Address that your network manager must assign to the TNC. Input: four decimal numbers separated by points, e.g. **160.1.180.20**
MASK	The SUBNET MASK for expanding the number of available addresses within your network. Input: four decimal numbers separated by points. Ask your network manager for the number of your address, e.g. **255.255.0.0**
ROUTER	Internet address of your default router. Enter the Internet address only if your network consists of several parts. Input: four decimal numbers separated by points. Ask your network manager for the number of your subnet mask. e.g. 160.2.0.2.
PROT	Definition of the transmission protocol. **RFC**: Transmission protocol according to RFC 894 **IEEE**: Transmission protocol according to IEE 802.2/ 802.3
HW	Definition of the connection used **10BASET**: for use of 10BaseT **10BASE2**: for use of 10Base2
HOST	Name, under which the TNC identifies itself in the network. If you are using a host name, you must enter the „Fully Qualified Hostname" here. If you do not enter a name here, the TNC uses the so-called null authentication. The UID, GID, DCM and FCM settings specific to the device (see next page), are then ignored by the TNC.

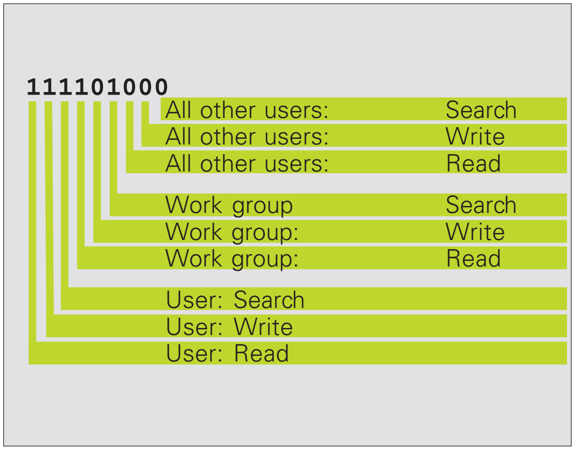
Network settings specific to the device

▶ Press the soft key DEFINE MOUNT to enter the network setting for a specific device (see figure at upper right). You can define any number of network settings, but you can manage only seven at one time.



Setting	Meaning
ADDRESS	Address of your server. Input: four decimal numbers separated by points. Ask your network manager for the number of your address. e.g. 160.1.13.4.
RS	Packet size in bytes for data reception. Input range: 512 to 4096. Input 0: The TNC uses the optimal packet size as reported by the server.
WS	Packet size in bytes for data transmission. Input range: 512 to 4096. Input 0: The TNC uses the optimal packet size as reported by the server.
TIMEOUT	Time in ms, after which the TNC repeats a Remote Procedure Call. Input range: 0 to 100 000. Standard input: 0, which corresponds to a TIMEOUT of 7 seconds. Use higher values only if the TNC must communicate with the server through several routers. Ask your network manager for the proper timeout setting.
HM	Definition of whether the TNC should repeat the Remote Procedure Call until the NFS server answers. **0**: Always repeat the Remote Procedure Call **1**: Do not repeat the Remote Procedure Call
DEVICENAME	Name that the TNC shows in the file manager for a connected device.
PATH	Directory of the NFS server that you wish to connect to the TNC. Be sure to differentiate between small and capital letters when entering the path.
UID	Definition of the user identification under which you access files in the network. Ask your network manager for the proper timeout setting.
GID	Definition of the group identification with you access files in the network. Ask your network manager for the proper timeout setting.

Setting	Meaning
DCM	Here you enter the **rights of access to files** on the NFS server (see figure at upper right). Enter a binary coded value. Example: 111101000 **0**: Access not permitted **1**: Access permitted
DCM	Here you enter the **rights of access to files** on the NFS server (see figure at upper right). Enter the value in binary coded form. Example: 111101000 **0**: Access not permitted **1**: Access permitted
AM	Definition of whether the TNC upon switch-on should automatically connect with the network. **0**: Do not connect automatically **1**: Connect automatically



Defining the network printer

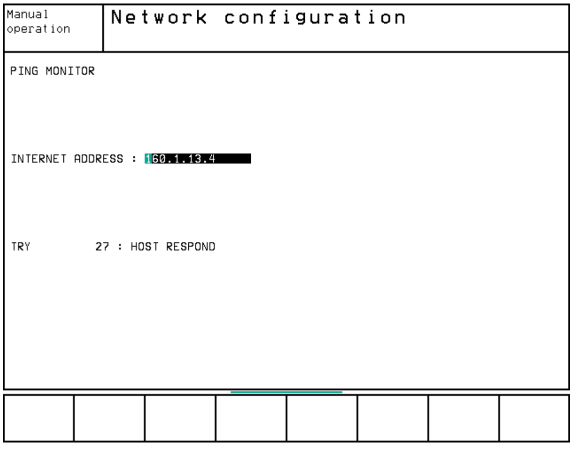
- Press the DEFINE PRINT soft key if you wish to print the files on the network printer directly from the TNC.

Setting	Meaning
ADDRESS	Address of your server. Input: four decimal numbers separated by points. Ask your network manager for the number of your address. e.g. 160.1.13.4.
DEVICE NAME	Name of the printer that the TNC shows when the PRINT soft key is pressed (see also „4.4 File Management with Additional Functions")
PRINTER NAME	Name of the printer in your network. Ask your network manager.

Checking the network connection

- Press the PING soft key.
- Enter the Internet address of the device with which you wish to check the connection, and confirm your entry with ENT. The TNC transmits data packets until you exit the test monitor by pressing the END key.

In the TRY line the TNC shows the number of data packets that were transmitted to the previously defined addressee. Behind the number of transmitted data packets the TNC shows the status:



Status display	Meaning
HOST RESPOND	Data packet was received again, connection is OK.
TIMEOUT	Data packet was not received, check the connection.
CAN NOT ROUTE	Data packet could not be transmitted. Check the Internet address of the server and of the router to the TNC.

Displaying the error log

- Press the SHOW ERROR soft key if you would like to see the error log. Here the TNC records all errors that have occurred in the network since the TNC was last switched on.

The listed error messages are divided into two categories:

Warnings are indicated with (W). Warnings occur when the TNC was able to establish the network connection, but had to correct settings in order to do so.

Error messages are indicated with (E). Error messages occur when the TNC was unable to establish a network connection.

Error message	Cause
LL: (W) CONNECTION xxxxx UNKNOWN USING DEFAULT 10BASET	The name you entered in DEFINE NET, HW was incorrect
LL: (E) PROTOCOL xxxxx UNKNOWN	The name you entered in DEFINE NET, PROT was incorrect
IP4: (E) INTERFACE NOT PRESENT	The TNC was unable to find an Ethernet card.
IP4: (E) INTERNETADDRESS NOT VALID	You used an invalid Internet address for the TNC.
IP4: (E) SUBNETMASK NOT VALID	The SUBNET MASK does not match the Internet address of the TNC.
IP4: (E) SUBNETMASK OR HOST ID NOT VALID	You used an invalid Internet address for the TNC, or you entered an incorrect SUBNET MASK, or you set all of the HostID bits to 0 (1)
IP4: (E) SUBNETMASK OR SUBNET ID NOT VALID	All bits of the SUBNET ID are 0 or 1
IP4: (E) DEFAULTROUTERADRESS NOT VALID	You used an invalid Internet address for the router.
IP4: (E) CAN NOT USE DEFAULTROUTER IP4: (E) I AM NOT A ROUTER	The default router does not have the same net ID or subnet ID as the TNC. You defined the TNC as a router.
MOUNT: <Device name> (E) DEVICENAME NOT VALID	The device name is either too long or it contains illegal characters.
MOUNT: <device name> (E) DEVICENAME ALREADY ASSIGNED	You have already defined a device with this name.
MOUNT: <Device name> (E) DEVICETABLE OVERFLOW	You have attempted to connect more than seven network drives to the TNC.
NFS2: <Device name> (W) READSIZE SMALLER THEN x SET TO x	The value that you entered for DEFINE MOUNT, RS is too small. The TNC sets RS to 512 bytes.
NFS2: <Device name> (W) READSIZE LARGER THEN x SET TO x	The value that you entered for DEFINE MOUNT, RS is too large The TNC sets RS to 4096 bytes.

Error message	Cause
NFS2: <Device name> (W) WRITESIZE SMALLER THEN x SET TO x	The value that you entered for DEFINE MOUNT, WS is too small. The TNC sets WS to 512 bytes.
NFS2: <Device name> (W) WRITESIZE LARGER THEN x SET TO x	The value that you entered for DEFINE MOUNT, WS is too large. The TNC sets WS to 4096 bytes.
NFS2: <Device name> (E) MOUNTPATH TOO LONG	The value that you entered for DEFINE MOUNT, PATH is too long.
NFS2: <Device name> (E) NOT ENOUGH MEMORY	At the moment there is too little main memory available to establish a network connection.
NFS2: <Device name> (E) HOSTNAME TOO LONG	The name you entered in DEFINE NET, HOST is too long.
NFS2: <Device name> (E) CAN NOT OPEN PORT	The TNC cannot open the port required to establish the network connection.
NFS2: <Device name> (E) ERROR FROM PORTMAPPER	The TNC has received implausible data from the portmapper.
NFS2: <Device name> (E) ERROR FROM MOUNTSERVER	The TNC has received implausible data from the mountserver.
NFS2: <Device name> (E) CANT GET ROOTDIRECTORY	The mount server does not permit a connection with the directory defined in DEFINE MOUNT, PATH.
NFS2: <Device name> (E) UID OR GID 0 NOT ALLOWED	You entered 0 for DEFINE MOUNT, UID or GID 0. The input value 0 is reserved for the system administrator.

13.9 Configuring PGM MGT (not TNC 410)

With this function you can determine the features of the file manager:

- Standard: Simple file management without directory display
- Expanded range: File management with additional functions and directory display

See also „section 4.3 Standard File Management" and „section 4.4 File Management with Additional Functions".

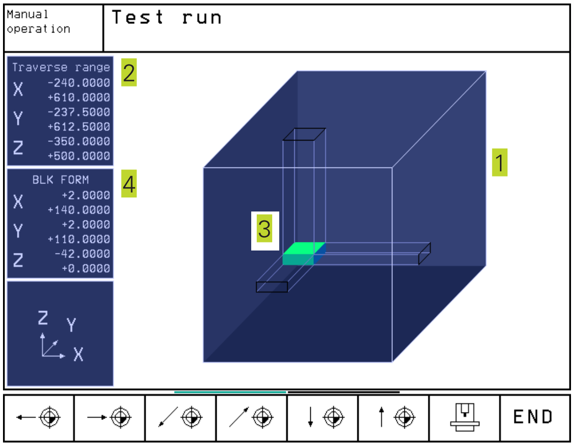
Changing the setting

- Select the file manager in the Programming and Editing mode of operation: press the PGM MGT key
- Select the MOD function: Press the MOD key
- Select the PGM MGT setting: using the arrow keys, move the highlight onto the PGM MGT setting and use the ENT key to switch between Standard and Enhanced

13.10 Machine-Specific User Parameters

The machine tool builder can assign functions to up to 16 user parameters. Your machine manual provides more detailed information.

13.11 Showing the Workpiece in the Working Space (not TNC 410)



This MOD function enables you to graphically check the position of the workpiece blank in the machine's working space and to activate work space monitoring in the Test Run mode of operation. This function is activated with the BLANK IN WORD SPACE soft key.

The TNC displays a cuboid 1 for the working space. Its dimensions are shown in the "Traverse range" (2) window. The TNC takes the dimensions for the working space from the machine parameters for the active traverse range. Since the traverse range is defined in the reference system of the machine, the datum of the cuboid is also the machine datum. You can see the position of the machine datum in the cuboid by pressing the soft key M91 in the 2nd soft-key row.

Another cuboid (3) represents the blank form. The TNC takes its dimensions (4) from the workpiece blank definition in the selected program. The workpiece cuboid defines the coordinate system for input. Its datum lies within the cuboid. You can see in the cuboid the position of the datum for input by pressing the corresponding soft key in the 2nd soft-key row.

For a test run it normally does not matter where the workpiece blank is located within the working space. However, if you test programs that contain movements with M91 or M92, you must graphically shift the workpiece blank to prevent contour damage. Use the soft keys shown in the table at right.

You can also activate the working-space monitor for the Test Run mode in order to test the program with the current datum and the active traverse ranges (see table at right, bottommost soft key).

Function	Soft key
Move workpiece blank to the left (graphically)	
Move workpiece blank to the right (graphically)	
Move workpiece blank forward (graphically)	
Move workpiece blank backward (graphically)	
Move workpiece blank upward (graphically)	
Move workpiece blank downward (graphically)	
Show workpiece blank referenced to the set datum	
Show the entire traversing range referenced to the displayed workpiece blank	
Show the machine datum in the working space	M91
(Show the position defined in the working space (e.g. tool change position) as defined by the machine tool builder	M92
Show the workpiece datum in the working space	
Enable (ON) or disable (OFF) work space monitoring	OFF / ON

13.12 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.

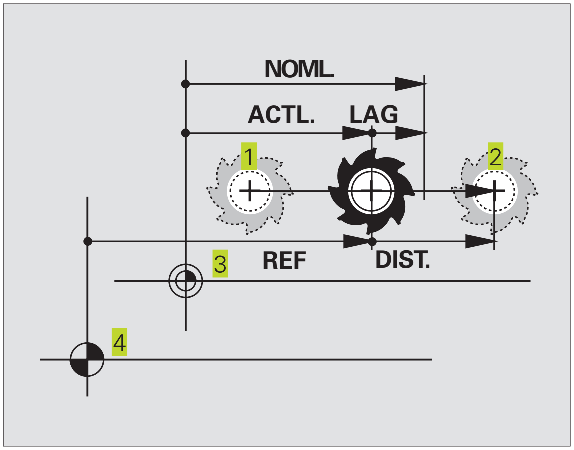
The figure at right shows the different tool positions:

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
Deflection of the measuring touch probe	DEFL.
Traverses that were carried out with handwheel superpositioning (M118) (only position display 2, not TNC 410)	M118

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.

13.13 Unit of Measurement

This MOD function determines whether the coordinates are displayed in millimeters (metric system) or inches.

- To select the metric system (e.g. X = 15.789 mm) set the Change mm/inches function to mm. The value is displayed 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.

13.14 Programming Language for MDI

The Program Input MOD function lets you switch between HEIDENHAIN conversational dialog or ISO format in the MDI mode:

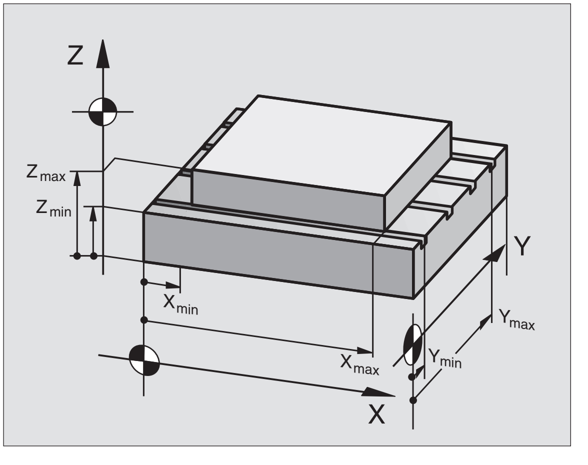
- To program in conversational dialog:
  Program input: HEIDENHAIN
- To program according to ISO:
  Program input: ISO

13.15 Selecting the Axes for Generating L Blocks (not TNC 410, only Conversational Dialog)

The axis selection input field enables you to define the current tool position coordinates that are transferred to an L block. To generate a separate L block, press the ACTUAL-POSITION-CAPTURE soft key. The axes are selected by bit-oriented definition similar to programming the machine parameters:

Axis selection	%11111	Transfer the X, Y, Z, IV and V axes
Axis selection	%01111	Transfer the X, Y, Z, IV axes
Axis selection	%00111	Transfer the X, Y and Z axes
Axis selection	%00011	Transfer the X and Y
Axis selection	%00001	Transfer the X axis

13.16 Axis Traverse Limits, Datum Display



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. If several traverse ranges are possible on your machine, you can set the limits for each range separately using the soft keys LIMIT SWITCH (1) to LIMIT SWITCH (3) (not TNC 410).

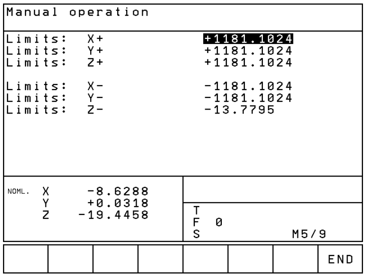
Working without additional traverse limits
To allow a machine axis to use its full range of traverse, enter the maximum traverse of the TNC (+/- 99999 mm) as the TRAVERSE RANGE.

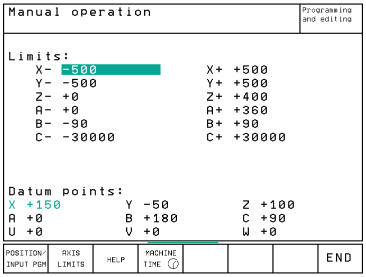
To find and enter the maximum traverse:

- Set the Position display mod function to 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.

- Enter the limits for axis traverse: Press the TRAVERSE RANGE soft key and enter the values that you wrote down as limits in the corresponding axes
- To exit the MOD function, press the END soft 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.



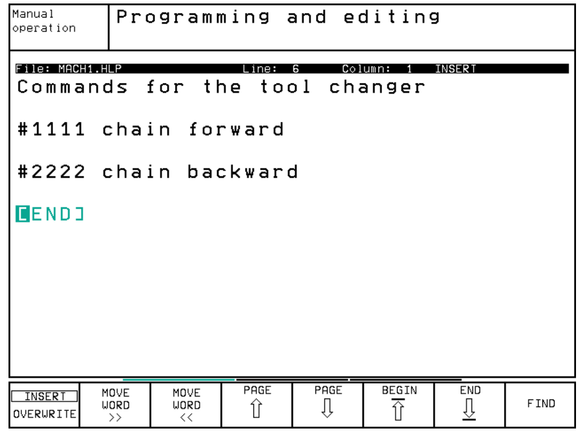


Datum display
The values shown at the lower left of the screen are the manually set datums referenced to the machine datum. They cannot be changed in the menu.

Traverse Range Limits for Test Run (only TNC 410)
It is possible to define a separate „traverse range“ for the Test Run and the programming graphics. Press the soft key TRAVERSE RANGE TEST (2nd soft-key row), after you have activated the MOD function.

In addition to the axis traverse limits, you can also define the position of the workpiece datum referenced to the machine datum.

13.17 The HELP Function



The HELP function can aid you in situations in which you need clear instructions before you can continue (for example, to retract the tool after an interruption of power). The miscellaneous functions may also be explained and executed in a help file.

The TNC 426, TNC 430 offers several help files, which you can select by means of the file management.

The HELP function is not available on every machine. The machine tool manual provides further information.

Selecting and executing a HELP function

▶ Select the MOD function: Press the MOD key

HELP

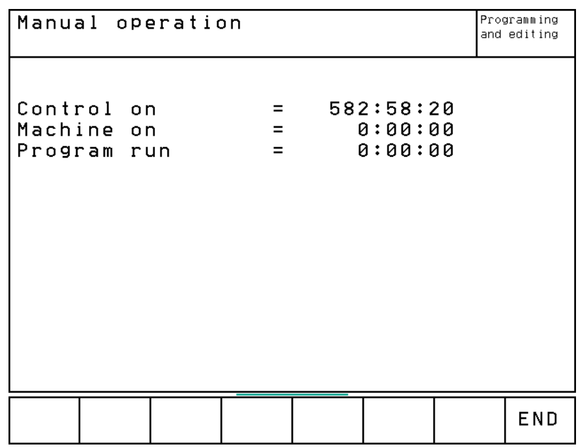
▶ To select the HELP function: Press the HELP soft key.

▶ On the TNC 426, TNC 430, call the file manager (PGM MGT key) and select a different help file, if necessary.

▶ Use the up and down arrow keys to select a line in the HELP file, which is marked with an #

▶ Use the NC start key to execute the selected HELP function

13.18 Operating Time (via Code Number for TNC 410)



The machine tool builder can provide further operating time displays. Refer to your machine tool manual.

The MACHINE TIME soft key enables you to show different operating time displays:

Operating time	Meaning
Control ON	Operating time of the control since its commissioning
Machine ON	Operating time of the machine tool since its commissioning
Program Run	Operating time for program-controlled operation since commissioning

14

Tables and Overviews

14.1 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 typical user parameters are:

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

Input possibilities for machine parameters

Machine parameters can be programmed as

- **Decimal numbers**
  Enter only the number
- **Pure binary numbers**
  Enter a percent sign (%) before the number
- **Hexadecimal numbers**
  Enter a dollar sign ($) before the number

Example:
Instead of the decimal number 27 you can also enter the binary number %11011 or the hexadecimal number $1B.

The individual machine parameters can be entered in the different number systems.

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 the 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.

External data 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**
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**	1 stop bit: **+128** 1 stop bit: **+192**
Integrating TNC interfaces EXT1 (5030.0) and EXT2 (5030.1) to external device	**MP5030.x** Standard transmission: **0** Interface for blockwise transfer: **1**

3-D touch probes and digitizing

Select touch probe
(only with option for digitizing with measuring touch probe; notTNC 410)

MP6200
Triggering touch probe: **0**
Measuring touch probe: **1**

Select signal transmission

MP6010
Touch probe with cable transmission: **0**
Touch probe with infrared transmission: **1**

Probing feed rate for triggering touch probes

MP6120
10 **to** 3000 **[mm/min]**

Maximum traverse to first probe point

MP6130
0.001 to **99 999.9999** [mm]

Safety clearance to probing point during automatic measurement

MP6140
0.001 to **99 999.9999** [mm]

Rapid traverse for triggering touch probes

MP6150
1 to **300 000** [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 3-D touch probe during calibration: 1 **to 88**

Multiple measurement for programmable probe functions (notTNC 410)

MP6170
1 to **3**

Confidence interval for multiple measurement (notTNC 410)

MP6171
0.001 to **0.999** [mm]

Infeed of the stylus when digitizing with the measuring touch probe (notTNC 410)

MP6310
0.1 to **2.0000** [mm] (recommended input value: 1 mm)

Measure center misalignment of the stylus when calibrating a measuring touch probe (notTNC 410)

MP6321
Measure center misalignment: **0**
Do not measure center misalignment: **1**

Assign touch probe axis to machine axis for a measuring touch probe (not TNC 410)

Ensure that the touch probe axes are correctly assigned to the machine axes. Wrong assignment could lead to a stylus break.

MP6322.0
Machine **X** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**

MP6322.1
Machine **Y** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**

MP6322.2
Machine **Z** axis parallel to touch probe axis X: **0**, Y: **1**, Z: **2**

Maximum stylus deflection of the measuring touch probe (not TNC 410)

MP6330
0.1 to **4.0000** [mm]

Feed rate for positioning measuring touch probes at MIN point and approaching the contour (not TNC 410)

MP6350
10 to **3 000** [mm/min]

Probe rate for measuring touch probe (not TNC 410)

MP6360
10 to **3 000** [mm/min]

Rapid traverse for measuring touch probes in the probe cycle (not TNC 410)

MP6361
10 to **3 000** [mm/min]

Feed rate reduction when the stylus of a measuring touch probe is deflected to the side (not TNC 410)

The TNC decreases the feed rate according to a preset characteristic curve. The minimum input value is 10% of the programmed digitizing feed rate.

MP6362
Feed rate reduction not active: **0**
Feed rate reduction active: **1**

Radial acceleration during digitizing for measuring touch probe (not TNC 410)

MP6370 enables you to limit the feed rate of the TNC for circular movements during digitizing. Circular movements are caused, for example, by sharp changes of direction.

As long as the programmed digitizing feed rate is less than the feed rate calculated with MP6370, the TNC will move at the programmed feed rate. Determine the appropriate value for your requirements by trial and error.

MP6370
0.001 to **5.000** [m/s^2] (recommended input value: 0.1)

Target window for digitizing contour lines with a measuring touch probe (not TNC 410)

When you are digitizing contour lines the individual contour lines do not end exactly in their starting points.

With machine parameter MP6390 you can define a square target window within which the end point must lie after the touch probe has orbited the model. Enter half the side length of the target window for the side length.

MP6390
0.1 to **4.0000** [mm]

Radius measurement with the TT 120 touch probe: Probing direction

MP6505.0 (traverse range 1) to 6505.2 (traverse range 3)
Positive probing direction in the angle reference axis (0° axis): **0**
Positive probing direction in the +90° axis: **1**
Negative probing direction in the angle reference axis (0° axis): **2**
Negative probing direction in the +90° axis: **3**

Probing feed rate for second measurement with TT 120, stylus shape, corrections in TOOL.T

MP6507
Calculate feed rate for second measurement with TT 120, with constant tolerance: **+0**
Calculate feed rate for second measurement with TT 120, with variable tolerance: **+1**
Constant feed rate for second measurement with TT 120: **+2**

Maximum permissible measuring error with TT 120 during measurement with rotating tool

Required for calculating the probing feed rate in connection with MP6570

MP6510
0.001 to **0.999** [mm] (recommended input value: 0.005 mm)

Feed rate for probing a stationary tool with the TT 120

MP6520
10 to **3000** [mm/min]

Radius measurement with the TT 120: Distance from lower edge of tool to upper edge of stylus

MP6530.0 (traverse range 1) to MP6530.2 (traverse range 3)
TNC 410: 1 traverse range

Clearance zone around the stylus of the TT 120 for pre-positioning

MP6540
0.001 to **99,999.999** [mm]

Rapid traverse for TT 120 in the probe cycle

MP6550
10 to **10 000** [mm/min]

M function for spindle orientation when measuring individual teeth

MP6560
0 to **88**

Measuring rotating tools: Permissible rotational speed at the circumference of the milling tool

Required for calculating rpm and probe feed rate

MP6570
1.000 to **120.000** [m/min]

Coordinates of the TT 120 stylus center relative to the machine datum

MP6580.0 (traverse range 1)
X axis

MP6580.1 (traverse range 1)
Y axis

MP6580.2 (traverse range 1)
Z-axis

MP6581.0 (traverse range 2) (not TNC 410)
X-axis

MP6581.1 (traverse range 2) (not TNC 410)
Y-axis

MP6581.2 (traverse range 2) (not TNC 410)
Z-axis

MP6582.0 (traverse range 3) (not TNC 410)
X-axis

MP6582.1 (traverse range 3) (not TNC 410)
Y-axis

MP6582.2 (traverse range 3) (not TNC 410)
Z-axis

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**

ISO programming: Set the block number increment

MP7220
0 to **150**

Inhibit particular file type

MP7224.0
Do not inhibit any file type: **+0**
Inhibit HEIDENHAIN programs: **+1**
Inhibit ISO programs: **+2**
Inhibit tool tables: **+4**
Inhibit datum tables: **+8**
Inhibit pallet tables: **+16**
(not TNC 410) Inhibit text files: **+32 (not TNC 410)**

Inhibit editing of particular file type (notTNC 410)

If a particular file type is inhibited, the TNC will erase all files of this type.

MP7224.1
Do not inhibit editor: **+0**
Inhibit editor for

- HEIDENHAIN programs: **+1**
- ISO programs: **+2**
- Tool tables: **+4**
- Datum tables: **+8**
- Pallet tables: **+16**
- Text files: **+32**

Configure pallet tables (notTNC 410)

MP7226.0
Pallet table inactive: **0**
Number of pallets per pallet table: **1** to **255**

Configure datum tables (notTNC 410)

MP7226.1
Datum table inactive: **0**
Number of datums per datum table: **1** to **255**

Program length as program control (notTNC 410)

MP7229.0
Blocks **100** to **9 999**

Program length up to which FK blocks are permitted (not TNC 410)

MP7229.1
Blocks **100** to **9 999**

Dialog language

MP7230 on TNC 410
German:**0**
English: **1**

MP7230 on TNC 426, TNC 430

English: **0**	Swedish:**7**
German: **1**	Danish:**8**
Czech: **2**	Finnish:**9**
French: **3**	Dutch:**10**
Italian: **4**	Polish: **11**
Spanish: **5**	Hungarian: **12**
Portuguese: **6**	

Set the internal clock of the TNC (not TNC 410)	**MP7235** Universal time (Greenwich Mean Time): **0** Central European Time (CET): **1** Central European Summertime: **2** Time difference to universal time: **-23** to **+23** [hours]
Configure tool tables	**MP7260** Inactive: **0** Number of tools generated by the TNC when a new tool table is opened: **1** to **254** If you require more than 254 tools, you can expand the tool table with the function APPEND N LINES (see also „5.2 Tool Data"; not TNC 410)
Configure pocket tables	**MP7261** Inactive: **0** Number of pockets per pocket table: **1** to **254**
Index tool numbers in order to be able to assign different compensation data to one tool number	**MP7262** Do not index: **0** Number of permissible indices: **1 to 9**
Soft key for pocket tables	**MP7263** Show the POCKET TABLE soft key in the tool table: **0** Do not show the POCKET TABLE soft key in the tool table: **1**

Configure tool table (To omit from the table: enter 0);
Column number in the tool table for

MP7266.0	Tool name – NAME: **0** to **28**; column width: 16 characters
MP7266.1	Tool length – L: **0** to **28**; column width: 11 characters
MP7266.2	Tool radius – R: **0** to **28**; column width: 11 characters
MP7266.3	Tool radius 2 – R2: **0** to **28**; column width: 11 characters (not TNC 410)
MP7266.4	Oversize length – DL: **0** to**28**; column width: 8 characters
MP7266.5	Oversize radius – DR: **0** to**28**; column width: 8 characters
MP7266.6	Oversize radius 2 – DR2: **0** to **28**; column width: 8 characters (not TNC 410)
MP7266.7	Tool locked – TL: **0** to**28**; column width: 2 characters
MP7266.8	Replacement tool – RT: **0** to**28**; column width: 3 characters
MP7266.9	Maximum tool life – TIME1: **0** to**28**; column width: 5 characters
MP7266.10	Maximum tool life for TOOL CALL – TIME2: **0** to **28**; column width: 5 characters
MP7266.11	Current tool life — CUR. TIME: **0** to **28**; column width: 8 characters
MP7266.12	Tool comment – DOC: **0** to **28**; column width: 16 characters
MP7266.13	Number of teeth – CUT.: **0** to **28**; column width: 4 characters
MP7266.14	Tolerance for wear detection in tool length – LTOL: **0** to **28**; column width: 6 characters
MP7266.15	Tolerance for wear detection in tool radius – RTOL: **0** to **28**; column width: 6 characters
MP7266.16	Cutting direction – DIRECT.: **0** to **28**; column width: 7 characters
MP7266.17	PLC status – PLC: **0** to **28**; column width: 9 characters
MP7266.18	Offset of the tool in the tool axis in addition to MP6530 – TT:L-OFFS: **0** to **28**; column width: 11 characters
MP7266.19	Offset of the tool between stylus center and tool center — TT:R-OFFS: **0** to **28**; column width: 11 characters
MP7266.20	Tolerance for break detection in tool length – LBREAK.: **0** to **28**; column width 6 characters
MP7266.21	Tolerance for break detection in tool radius – RBREAK.: **0** to **28**; column width 6 characters
MP7266.22	Tool length (Cycle 22) – LCUTS: **0** to **28**; column width: 11
MP7266.23	Maximum plunge angle (Cycle 22) – ANGLE.: **0** to **28**; column width: 7 characters
MP7266.24	Tool type – TYP: **0** to **28**; column width: 5 characters (only for conversational dialog, not TNC 410)
MP7266.25	Tool material – TMAT: **0** to **28**; column width: 16 characters (only for conversational programming, not TNC 410)
MP7266.26	Cutting data table – CDT: **0** to **28**; column width: 16 characters (only for conversational dialog, not TNC 410)
MP7266.17	PLC value – PLC-VAL: **0** to **28**; column width: 9 characters (not TNC 410)

Configure pocket tables; Column number in the tool table for (to omit from table enter 0):

MP7267.0
Tool number — T: **0** to **5**

MP7267.1
Special tool — ST: **0** to **5**

MP7267.2
Fixed pocket — F: **0** to **5**

MP7267.3
Pocket locked — L: **0** to **5**

MP7267.4
PLC status — PLC: **0** to **5**

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 defined via soft key F or 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: **0**

0.05 mm: **1**	0.001 mm: **4**
0.01 mm: **2**	0.0005 mm: **5** (not TNC 410)
0.005 mm: **3**	0.0001 mm: **6** (not TNC 410)

Display step for the Y axis

MP7290.1
For input values, see MP7290.0

Display step for the Z axis

MP7290.2
For input values, see MP7290.0

Display step for the IVth axis

MP7290.3
For input values, see MP7290.0

Display step for the Vth axis (not TNC 410)	
	MP7290.4 For input values, see MP7290.0
Display step for the 6th axis (not TNC 410)	
	MP7290.5 For input values, see MP7290.0
Display step for the 7th axis (not TNC 410)	
	MP7290.6 For input values, see MP7290.0
Display step for the 8th axis (not TNC 410)	
	MP7290.7 For input values, see MP7290.0
Display step for the 9th axis (not TNC 410)	
	MP7290.8 For input values, see MP7290.0
Disable datum setting (not TNC 410)	
	MP7295 Do not disable datum setting: **+0** Disable datum setting in the X axis: **+1** Disable datum setting in the Y axis: **+2** Disable datum setting in the Z axis: **+4** Disable datum setting in the IV axis: **+8** Disable datum setting in the V axis: **+16** Disable datum setting in the 6th axis: **+32** Disable datum setting in the 7th axis: **+64** Disable datum setting in the 8th axis: **+128** Disable datum setting in the 9th axis: **+256**
Disable datum setting with the orange axis keys	
	MP7296 Do not inhibit datum setting: **0** Disable datum setting with the orange axis keys: **1**
Reset status display, Q parameters and tool data	
	MP7300 Reset them all when a program is selected: **0** Reset them all when a program is selected and with M02, M30, END PGM: **1** Reset only status display and tool data when a program is selected: **2** Reset only status display and tool data when a program is selected and with M02, M30, END PGM: **3** Reset status display and Q parameters when a program is selected: **4** Reset status display and Q parameters when a program is selected and with M02, M30, END PGM: **5** Reset status display when a program is selected: **6** Reset status display when a program is selected and with M02, M30, END PGM: **7**

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**Display new BLK FORM in Cycle G53/G54 DATUM SHIFT referenced to the previous datum **+0**
Display new blank form in Cycle G53/G54 DATUM SHIFT referenced to the new datum: **+4** (not TNC 410)
Do not show cursor position during projection in three planes: **+0** (not TNC 410)
Show cursor position during projection in three planes: **+8** (not TNC 410)

Settings for the programming graphics (not TNC 426, TNC 430)

MP7311
Do not show penetration points as circle: **+0**
Show penetration points as circle: **+1**
Do not show meander paths in cycles: **+0**
Show meander paths in cycles: **+2**
Do not show compensated paths: **+0**
Show compensated paths: **+3**

Graphic simulation without programmed tool axis: Tool radius (not TNC 410)

MP7315
0 to **99 999.9999** [mm]

Graphic simulation without programmed tool axis: Penetration depth (not TNC 410)

MP7316
0 to **99 999.9999** [mm]

Graphic simulation without programmed tool axis: M function for start (not TNC 410)

MP7317.0
0 to **88** (0: Function inactive)

Graphic simulation without programmed tool axis: M function for end (not TNC 410)

MP7317.1
0 to **88** (0: Function inactive)

Set the screen saver (not TNC 410)

Enter the time after which the TNC should start the screen saver

MP7392
0 to **99** [min] (0: Function inactive)

Machining and program run

Cycle G85: Oriented spindle stop at beginning of cycle	**MP7160** Oriented spindle stop: **0** No oriented spindle stop: **1**
Effect of Cycle G72 SCALING FACTOR	**MP7410** SCALING FACTOR effective in 3 axes: **0** SCALING FACTOR effective in the working plane only: **1**
Tool data in programmable probe cycle G55 0	**MP7411** Overwrite current tool data by the calibrated data from the 3-D touch probe system: **0** Current tool data are retained: **1**
Transitional mode during contour milling (notTNC 426,TNC 430)	**MP7415.0** Insert rounding arc: **0** Insert third-degree polynomial (cubic spline, curve without sudden change in velocity): **1** Insert fifth-degree polynomial (curve without sudden change in acceleration): **2** Seventh-degree polynomial (curve without sudden change in acceleration rate of change): **3**
Settings for contour milling (notTNC 426,TNC 430)	**MP7415.1** Do not smoothen the contours: **+0** Smoothen the contours: **+1** Do not smoothen the velocity profile if a short straight-line segment lies between contour transitions: **+0** Smoothen the velocity profile if a short straight-line segment lies between contour transitions: **+2**

SL Cycles Group I

MP7420

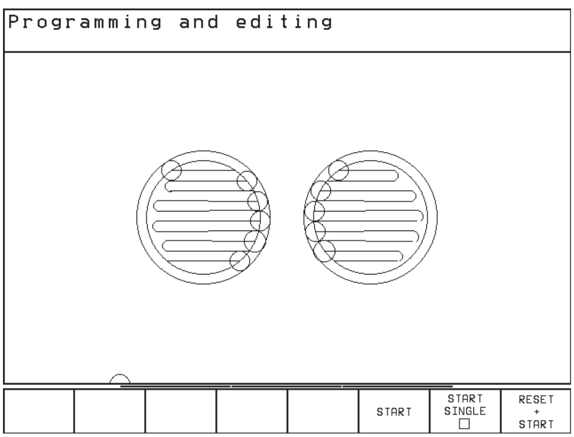
Mill channel around the contour — clockwise for islands and counterclockwise for pockets: **+0**
Mill channel around the contour — clockwise for pockets and counterclockwise for islands: **+1**
First mill the channel, then rough out the contour: **+0**
First rough out the contour, then mill the channel: **+2**
Combine compensated contours: **+0**
Combine uncompensated contours: **+4**
Complete one process for all infeeds before switching to the other process: **+0**
Mill channel and rough-out for each infeed depth before continuing to the next depth: **+8**

The following note applies to the Cycles G56, G57, G58, G59, G121, G122, G123 and G124:
At the end of the cycle, move the tool to the position that was last programmed before the cycle call: **+0**
At the end of the cycle, retract the tool in the tool axis only: **+16**

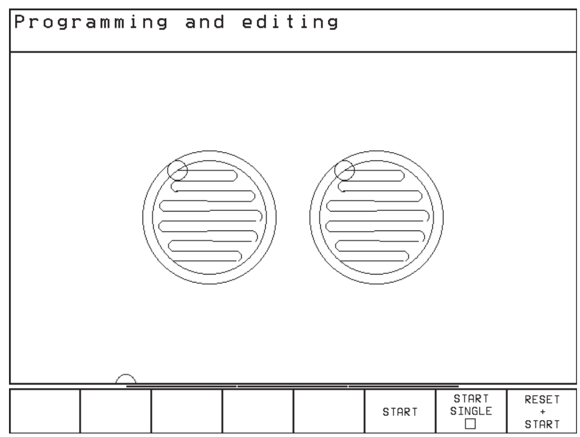
SL cycles, Group I, operating procedure (not TNC 426, TNC 430)

MP7420.1

Rough-out separate areas together, plunging with every pass: **+0**
Rough-out separate areas separately, plunging only once for each area **+1**
Bit 1 to bit 7: reserved



MP7420.1 = 0
(small circles = penetration)



MP7420.1 = 1

Cycle G75/G76 POCKET MILLING and Cycle G77/G78 CIRCULAR POCKET: overlap factor

MP7430
0.1 to **1.414**

Permissible deviation of circle radius between circle end point and circle starting point (not TNC 410)

MP7431
0.0001 to **0.016** [mm]

Behavior of M functions

> The k_V factors for position loop gain are set by the machine tool builder. Refer to your machine manual.

MP7440
Program stop with M06: **+0**
No program stop with M06: **+1**
No cycle call with M89: **+0**
Cycle call with M89: **+2**
Program stop with M functions: **+0**
No program stop with M functions: **+4**
k_v factors cannot be switched through M105 and M106: **+0** (not TNC 410)
k_v factors can be switched through M105 and M106: **+8** (not TNC 410)
Reduce the feed rate in the tool axis with M103 F.. , function inactive: **+0**
Reduce the feed rate in the tool axis with M103 F.. , function active: **+16**
Exact stop with rotary axes not active: **+0**
Exact stop for positioning with rotary axes active: **+32**

Running fixed cycles when M3 or M4 not active (not TNC 410)

MP7441
Error message when M3/M4 not active: **0**
Suppress error message when M3/M4 not active: **1**

Maximum permissible angle of directional change for constant contouring speed (effective for corners with R0 and for all inside corners); not TNC 426, 430.

This feature works both during operation with servo lag as well as with velocity feedforward.

MP7460
0.0000 to **179.9999** [°]

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

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

Datums from a datum table are referenced to the

MP7475
Workpiece datum: **0**
Machine datum: **1**

Running pallet tables (notTNC 410)

MP7683
Program run, single block: Run one line of the active NC program at every NC start: **+0**
Program run, single block: Run the entire NC program at every NC start: **+1**
Program run, full sequence: Run the entire NC program at every NC start: **+0**
Program run, full sequence: Run all NC programs up to the next pallet at every NC start: **+2**
Program run, full sequence: Run the entire NC program at every NC start: **+0**
Program run, full sequence: Run the entire pallet file at every NC start: **+4**

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** (not TNC 410)
HR 130 without additional keys: **2** (not TNC 410)
HR 330 with additional keys _ the handwheel keys for traverse direction and rapid traverse are evaluated by the PLC: **3** (not TNC 410)
HR 332 with twelve additional keys: **4**
(not TNC 410)
Multi-axis handwheel with additional keys: **5**
HR 410 with miscellaneous functions: **6**

Subdivision factor (notTNC 410)

MP7641
Interpolation factor is entered on the keyboard: **0**
Interpolation factor is set by the PLC: **1**

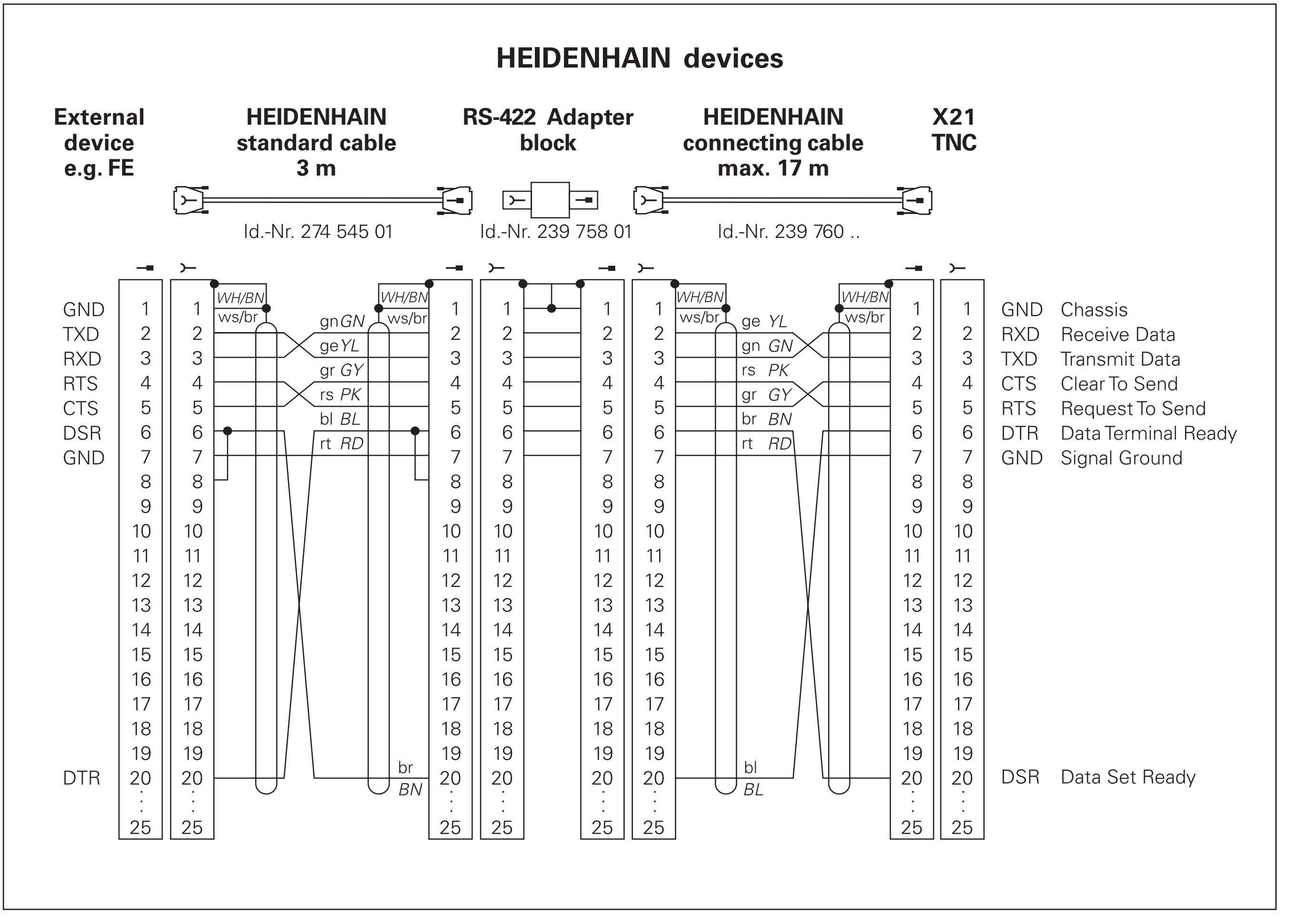
Machine functions that can be set for the handwheel by the machine tool builder (notTNC 410)

MP 7645.0	**0** to **255**
MP 7645.1	**0** to **255**
MP 7645.2	**0** to **255**
MP 7645.3	**0** to **255**
MP 7645.4	**0** to **255**
MP 7645.5	**0** to **255**
MP 7645.6	**0** to **255**
MP 7645.7	**0** to **255**

14.2 Pin Layout and Connecting Cable for the Data Interfaces

RS-232-C/V.24 Interface

HEIDENHAIN devices

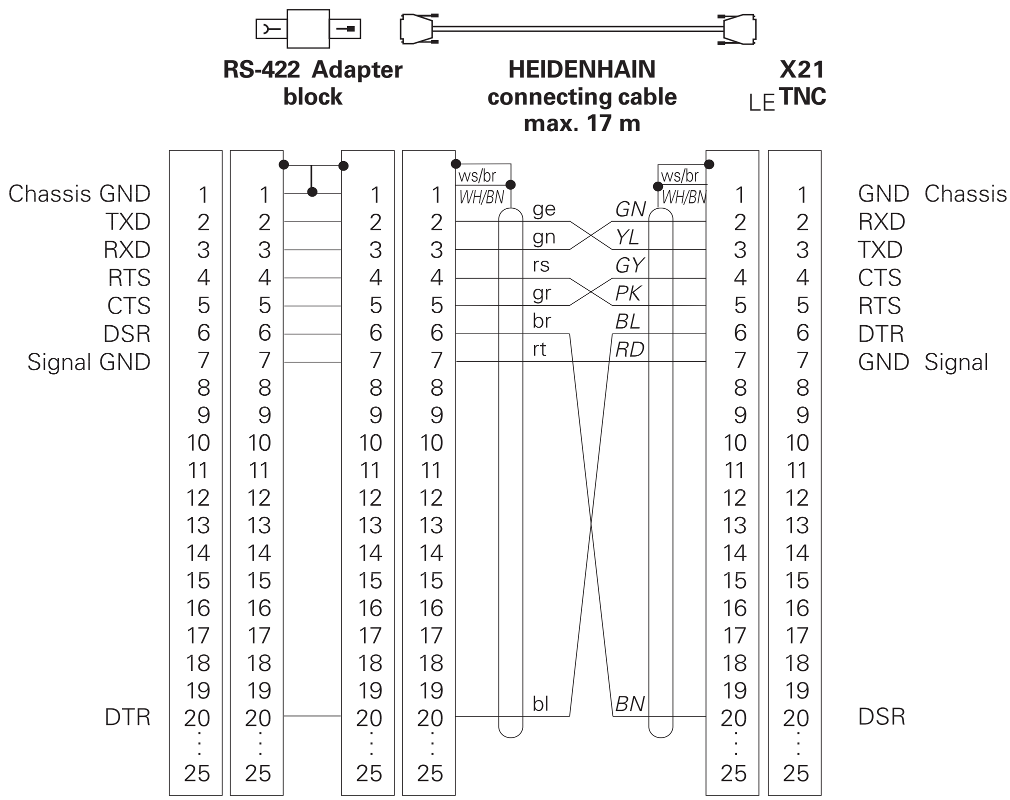


The connector pin layout on the adapter block differs from that on the TNC logic unit (X21).

Non-HEIDENHAIN devices

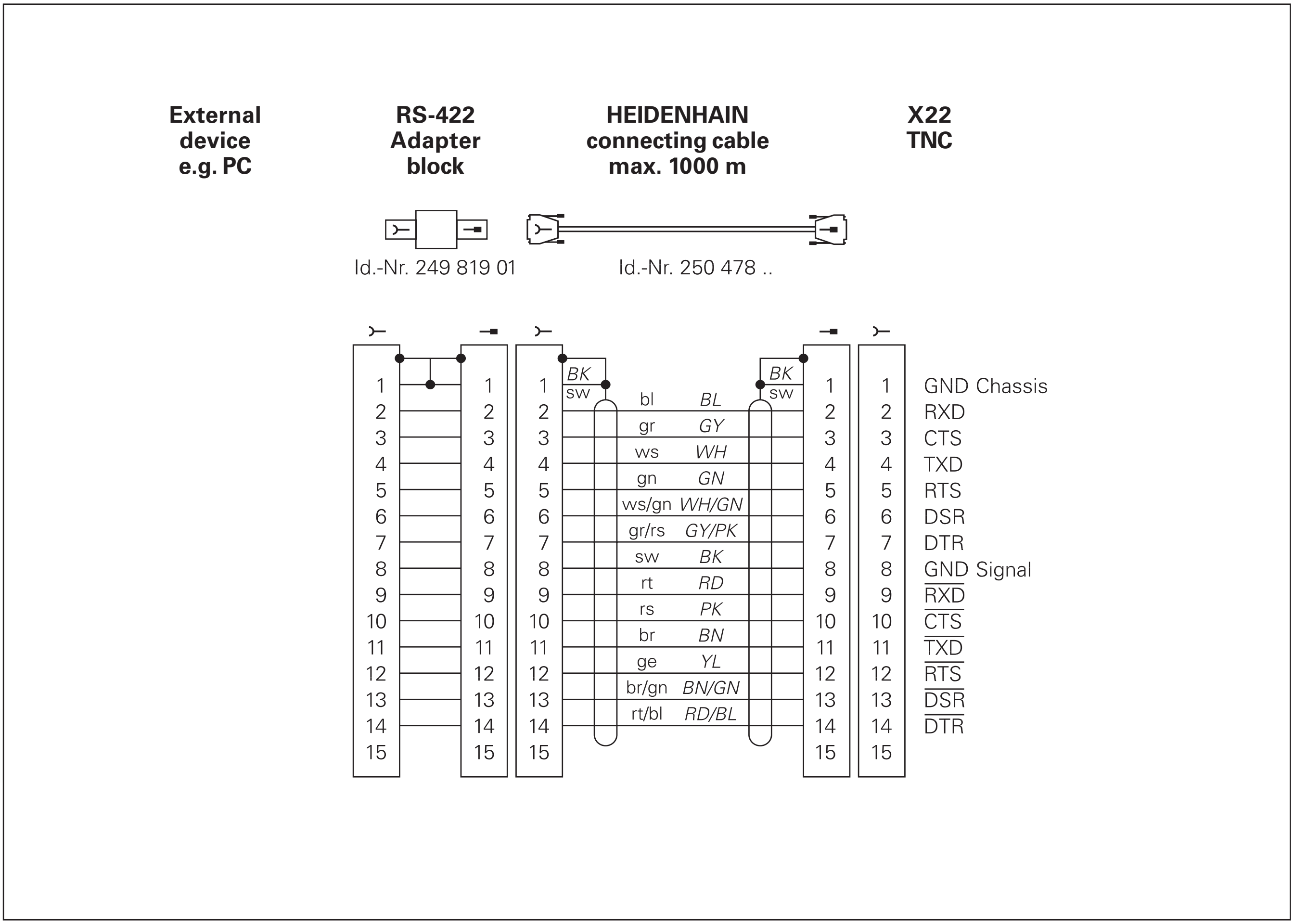
The connector pin layout of a non-HEIDENHAIN device may differ considerably from that on a HEIDENHAIN device.

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



RS-422/V.11 interface (not TNC 410)
Only non-HEIDENHAIN devices are connected to the RS-422 interface.

> The pin layouts on the TNC logic unit (X22) and on the adapter block are identical.



Ethernet interface RJ45 socket (option, not TNC 410)

Maximum cable length: Unshielded: 100 m
Shielded: 400 m

Pin	Signal	Description
1	TX+	Transmit Data
2	TX–	Transmit Data
3	REC+	Receive Data
4	Vacant –	
5	Vacant –	
6	REC–	Receive Data
7	Vacant –	
8	Vacant –	

Ethernet interface BNC socket (option, not TNC 410)

Maximum cable length: 180 m

Pin	Signal	Description
1	Data (RXI, TXO)	Inner conductor (core)
2	GND	Shielding

14.3 Technical Information

TNC features

Description	Contouring control for machines with up to 9 axes (TNC 410: 4 axes), with oriented spindle stop; TNC 410 CA, TNC 426 CB, TNC 430 CA with analog speed control, TNC 410 PA, TNC 426 PB, TNC 430 PB with digital speed control and integral current controller
Components	■ Logic unit ■ Keyboard unit ■ Visual display unit with soft keys
Data interfaces	■ RS-232-C / V.24 ■ RS-422 / V.11 (not TNC 410) ■ Ethernet interface (option, not TNC 410) Expanded data interface with LSV-2 protocol for remote Operating the TNC through the interface with the HEIDENHAIN software TNCremo (not TNC 410)
Simultaneous axis control for contour elements	
	■ Straight lines up to 5 axes (TNC 410: 3 axes) Export versions TNC 426 CF, TNC 426 PF, TNC 430 CE, TNC 430 PE: 4 axes ■ Circles up to 3 axes (with tilted working plane, TNC 410: 2 axes) ■ Helixes: 3 axes
Look Ahead	■ Defined rounding of discontinuous contour transitions (such as for 3-D surfaces); ■ Collision prevention with the SL cycle for open contours ■ Geometry precalculation of radius-compensated positions for feed rate adaptation with M120
Background programming	One part program can be edited while the TNC runs another program
Graphics	■ Interactive programming graphics ■ Test run graphics ■ Program run graphics (not TNC 410)
File types	■ HEIDENHAIN conversational programming ■ ISO programming ■ Tool tables ■ Cutting data tables (not TNC 410) ■ Datum tables ■ Point tables ■ Pallet files (not TNC 410) ■ Text files (not TNC 410) ■ Freely definable tables (not TNC 410) ■ System files

Program memory	■ Hard disk with 1.5 GB for NC programs (TNC 410: approx. 10,000 NC blocks with battery buffer backup) ■ Any number of files (TNC 410: up to 64 files)
Tool definitions	Up to 254 tools in program, any number of tools in tables (TNC 410: up to 254)
Programming support	■ Functions for approaching and departing the contour ■ Integrated pocket calculator (not TNC 410) ■ Structuring long programs (only conversational dialog, not TNC 410) ■ Comment blocks ■ Direct help on output error messages (context-sensitive, not TNC 410) ■ Help function for ISO programming (not TNC 426, TNC 430)

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 ■ B spline (not TNC 410)
Program jumps	■ Subprograms ■ Program section repeats ■ Program as Subprogram
Fixed cycles	■ Drilling cycles for drilling, pecking, reaming, boring, back 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 ■ Milling pockets and islands from a list of subcontour elements ■ Interpolation of cylinder surface (not TNC 410)

Coordinate transformations	■ Datum shift ■ Mirroring ■ Rotation ■ Scaling ■ Tilt the working plane (not TNC 410)
3-D touch probe applications	■ Touch probe functions for setting datums and for automatic workpiece measurement ■ Digitizing 3-D surfaces with the measuring touch probe (optional, only conversational dialog, not TNC 410) ■ Digitizing 3-D surfaces with the triggering touch probe (optional, only conversational dialog) ■ Automatic tool measurement with the TT 120 (only conversational dialog)
Mathematical functions	■ Basic arithmetic +, –, x and ¸ ■ Trigonometry sin, cos, tan, arcsin, arccos, arctan ■ Square root ($\sqrt{a}$) and root sum of squares ($\sqrt{a^2 + b^2}$) ■ Squaring (SQ) ■ Powers (^) ■ Constant PI (3.14) ■ Logarithms ■ Exponential functions ■ Negation (NEG) ■ Forming an integer (INT) ■ Forming an absolute number (ABS) ■ Truncating values before the decimal point (FRAC) ■ Logical comparisons (greater than, less than, equal to, not equal to)

TNC Specifications

Block processing time	4 ms/block, TNC 410: 6 ms/block, 20 ms/block for blockwise execution via data interface
Control loop cycle time	■ TNC 410 path interpolation: 6 ms ■ TNC 426 CB, TNC 430 CA: Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (contour) ■ TNC 426 PB, TNC 430 PB: Contouring interpolation: 3 ms Fine interpolation: 0.6 ms (speed)
Data transfer rate	Maximum 115,200 baud via V.24/V.11 Maximum 1 Mbaud via Ethernet interface (optional, not TNC 410)
Ambient temperature	■ Operation: 0° C to +45° C (32° to 113° F) ■ Storage: –30° C to +70° C (–22° to 158° F)
Traverse range	Maximum 100 000 mm (2540 inches) TNC 410: Maximum 30 000 mm (1181 inches)
Traversing speed	Maximum 300 m/min (11 811 ipm) TNC 410: Maximum 100 m/min
Spindle speed	Maximum 99 999 rpm
Input range	■ Minimum 0.1µm (0.00001 inches) or 0.0001° (TNC 410: 1 µm) ■ Maximum 99 999.999 mm (3937 inches) or 99 999.999° TNC 410: Maximum 30 000 mm (1181 inches) or 30 000.000°

14.4 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 buffer batteries are located next to the power supply unit in the logic unit (round, black case). 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“

14.5 Addresses (ISO)

G functions

Group	G	Function	Blockwise function	See page
Positioning	00	Straight-line interpolation, Cartesian coordinates, rapid traverse		102
	01	Straight-line interpolation, Cartesian coordinates		102
	02	Circular interpolation, Cartesian coordinates, clockwise	■ (with R)	103
	03	Straight-line interpolation, Cartesian coordinates, counterclockwise	■ (with R)	103
	05	Circular interpolation, Cartesian coordinates, without indication of direction		103
	06	Circular interpolation, Cartesian coordinates, tangential contour approach		106
	07	Paraxial positioning block	■	
	10	Straight-line interpolation, polar coordinates, rapid traverse		112
	11	Straight-line interpolation, polar coordinates		112
	12	Circular interpolation, polar coordinates, clockwise		112
	13	Circular interpolation, polar coordinates, counterclockwise		112
	15	Circular interpolation, polar coordinates, without indication of direction		112
	16	Circular interpolation, polar coordinates, tangential contour approach		113
Drilling cycles	83	Pecking		146
	84	Tapping with a floating tap holder		158
	85	Rigid tapping		161
	86	Thread cutting (not TNC 410)		164
	200	Drilling		147
	201	Reaming		148
	202	Boring		149
	203	Universal drilling		150
	204	Back boring		152

Group	G	Function	Blockwise function	See page
Drilling cycles	205	Universal pecking (only NC software 280 474-xx)		153
	206	Tapping with floating tap holder (only NC software 280 474-xx)		158
	207	Rigid tapping (only NC software 280 474-xx)		161
	208	Bore milling (only NC software 280 474-xx)		155
Cycles for milling	74	Slot milling		178
Pockets, studs and slots	75	Rectangular pocket milling in clockwise direction		169
	76	Circular path in counterclockwise direction		169
	77	Circular pocket milling in clockwise direction		173
	78	Circular pocket milling in counterclockwise direction		173
	210	Slot milling with reciprocating plunge		179
	211	Round slot with reciprocating plunge		181
	212	Rectangular pocket finishing		170
	213	Rectangular stud finishing		172
	214	Circular pocket finishing		175
	215	Circular stud finishing		176
Cycles for creating	220	Circular pattern		185
point patterns	221	Linear pattern		186
Cycles for creating	37	Definition of pocket contour		190/197
complex contours	56	Pilot drilling of the contour pocket (in connection with G37) SLI		181
	57	Rough-out of the contour pocket (in connection with G37) SLI		192
	58	Contour milling in clockwise direction (in connection with G37) SLI		194
	59	Contour milling in counterclockwise direction (in connection with G37) SLI		194
	120	Contour data (not TNC 410)		199
	121	Pilot drilling (in connection with G37) SLII (not TNC 410)		200
	122	Rough-out (in connection with G37) SLII (not TNC 410)		201
	123	Floor finishing (in connection with G37) SLII (not TNC 410)		202
	124	Side finishing (in connection with G37) SLII (not TNC 410)		203
	125	Contour train (in connection with G37, not TNC 410)		204
	127	Cylinder surface (in connection with G37, not TNC 410)		206
	128	Cylindrical surface slot (in connection with G37, only NC software 280 474-xx)		208
Cycles for multipass milling	60	Running point tables (not TNC 410)		214
	230	Multipass milling of smooth surfaces		216
	231	Multipass milling of tilted surfaces		218
Coordinate transformation	28	Mirror image		226
cycles	53	Datum shift in a datum table		223
	54	Datum shift in program		222
	72	Scaling factor		228
	73	Rotation of the coordinate system		227
	80	Machining plane (not TNC 410)		229
Special Cycles	04	Dwell time	■	236
	36	Oriented spindle stop		237
	39	Cycle for program call, program call via G79	■	236
	62	Tolerance deviation for fast contour milling (not TNC 410)		238

Group	G	Function	Blockwise function	See page
	79	Call the cycle	■	141
Selection of the machining plane	17	Plane selection XY, tool axis Z		96
	18	Plane selection ZX, tool axis Y		96
	19	Plane selection YZ, tool axis X		96
	20	Tool axis IV		96
	24	Chamfer with length R		101
	25	Corner rounding with R		106
	26	Tangential approach of a contour with R		99
	27	Tangential departure of a contour with R		99
	29	Transfer the last nominal position value as a pole		110
Define the workpiece blank	30	Define workpiece blank for graphics, min. point		61
	31	Define workpiece blank for graphics, max. point		61
	38	Program run STOP		284
Path compensation	40	No tool compensation (R0)		90
	41	Tool radius compensation, to the left of the contour (RL)		90
	42	Tool radius compensation, to the right of the contour (RR)		90
	43	Paraxial compensation, lengthening (R+)		90
	44	Paraxial compensation, shortening (R–)		90
	51	Next tool number (in a central tool memory)	■	87
	55	Touch probe function		308
Unit of measure	70	Unit of measure: inches (set at start of program)		60
	71	Unit of measure: millimeters (set at start of program)		60
Dimensions	90	Absolute dimensions		35
	91	Incremental dimensions		35
	98	Setting a label number	■	220
	99	Define the tool	■	80

Designation	Function
%	Program start or program call
#	Datum number with Cycle G53
A	Rotation about X-axis
B	Rotation about Y-axis
C	Rotation about Z-axis
D	Definition of parameters (program parameters Q)
DL	Length wear compensation with tool call
DR	Radius wear compensation with tool call
E	Tolerance for M112 and M124
F	Feed rate
F	Dwell time G04
F	Scaling factor with G72
F	Factor for feed-rate reduction with M103
G	Preparatory function
H	Polar coordinates angle in incremental value/absolute value
H	Angle of rotation with G73
H	Tolerance angle for M112

Designation	Function
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Setting a label number with G98
L	Jump to a label number
L	Tool length with G99
LA	Number of blocks for block scan with M120
M	Miscellaneous functions
N	Block number
P	Cycle parameters in machining cycles
P	Parameters in parameter definitions
Q	Program parameters/Cycle parameters Q
R	Polar coordinate radius
R	Circular radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Chamfer section with G24
R	Tool radius with G99
S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
U	Linear movement parallel to X-axis
V	Linear movement parallel to Y-axis
W	Linear movement parallel to Z-axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

Parameter definition	Function	See page
D00	Assign	254
D01	Addition	254
D02	Subtraction	254
D03	Multiplication	254
D04	Division	254
D05	Root	254
D06	Sine	256
D07	Cosine	256
D08	Root sum of squares ($c = \sqrt{a^2 + b^2}$)	256
D09	If equal, go to	257
D10	If not equal, go to	257
D11	If greater than, go to	257
D12	If less than, go to	257
D13	Angle (angle from $c \cdot \sin \alpha$ and $c \cdot \cos \alpha$)	256
D14	Error number	259
D15	Print	259
D19	Assignment PLC markers	259

M	Effect of M function	Effective at block - start	end	page
M00	Stop program run/spindle STOP/coolant OFF		■	**121**
M01	Optional program STOP		■	**293**
M02	Stop program/Spindle STOP/Coolant OFF/Clear status display (depending on machine parameter)/Go to block 1		■	**121**
M03	Spindle ON clockwise	■		
M04	Spindle ON counterclockwise	■		
M05	Spindle STOP		■	**121**
M06	Tool change/Stop program run (depending on machine parameter)/Spindle STOP		■	**121**
M08	Coolant ON	■		
M09	Coolant OFF		■	**121**
M13	Spindle ON clockwise/coolant ON	■		
M14	Spindle ON counterclockwise/Coolant ON	■		**121**
M30	Same function as M02		■	**121**
M89	Vacant miscellaneous function **or**	■		
	Cycle call, modally effective (depending on machine parameter)		■	**143**
M90	Only in lag mode: Constant contouring speed at corners		■	**124**
M91	Within the positioning block: Coordinates are referenced to machine datum	■		**121**
M92	Within the positioning block: Coordinates are referenced to position defined by machine tool builder, such as tool change position	■		**121**
M94	Reduce display of rotary axis to value under 360°	■		**135**
M97	Machine small contour steps		■	**129**
M98	Machine open contours completely		■	**130**
M99	Blockwise cycle call		■	**143**

M	Effect of M function	Effective at block - start	end	page
M101	Automatic tool change with replacement tool if maximum tool life has expired	■		
M102	Reset M101		■	**89**
M103	Reduce feed rate during plunging to factor F (percentage)	■		**131**
M104	Reactivate datum last set in the manual mode of operation	■		**123**
M105	Machining with second kv factor	■		
M106	Machining with first kv factor	■		**350**
M107	Suppress error message for replacement tools	■		
M108	Reset M107		■	**89**
M109	Constant contouring speed at tool cutting edge (increase and decrease feed rate)	■		
M110	Constant contouring speed at tool cutting edge (feed rate decrease only)	■		
M111	Reset M109/M110		■	**132**
M112	Entering contour transitions between contour elements; Enter the tolerance of contour deviation via T.	■		
M113	Cancel M112			**125**
M114	Automatic compensation of machine geometry when working with tilted axes:	■		
M115	Reset M114		■	**136**
M116	Feed rate for angular axes in mm/min	■		
M117	Reset M116		■	**134**
M118	Superimpose handwheel positioning during program run	■		**133**
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)	■		**132**
M124	Contour filter	■		**127**
M126	Shorter-path traverse of rotary axes	■		
M127	Reset M126		■	**134**
M128	Maintain the position of the tool tip when positioning with tilted axes (TCPM)	■		
M129	Reset M128		■	**137**
M130	Moving to position in an untilted coordinate system with a tilted working plane	■		**123**
M134	Exact stop at nontangential contour transitions when positioning with rotary axes	■		
M135	Reset M134		■	**139**
M136	Feed rate F in micrometers per spindle revolution	■		
M137	Reset M136		■	**131**
M138	Select tilting axes	■		**139**
M200	Laser cutting: Output programmed voltage directly	■		
M201	Laser cutting: Output voltage as a function of distance	■		
M202	Laser cutting: Output voltage as a function of speed	■		
M203	Laser cutting: Output voltage as a function of time (ramp)	■		
M204	Laser cutting: Output voltage as a function of time (pulse)	■		**140**

Contour cycles

Sequence of program steps for machining with several tools	
List of subcontour programs	G37 P01 ...
Define contour data	G120 Q1 ...
Drill define/call Contour cycle: pilot drilling Cycle call	G121 Q10 ...
Roughing mill define/call Contour cycle: rough-out Cycle call	G122 Q10 ...
Finishing mill define/call Contour cycle: floor finishing Cycle call	G123 Q11 ...
Finishing mill define/call Contour cycle: side finishing Cycle call	G124 Q11 ...
End of main program, return	**M02**
Contour subprograms	G98 ... G98 L0

Radius compensation of the contour subprograms

Contour	Programming sequence of the contour elements	Radius compens.
Inside (pocket)	Clockwise (CW) Counterclockwise (CCW)	G42 (RR) G41 (RL)
Outside (island)	Clockwise (CW) Counterclockwise (CCW)	G41 (RL) G42 (RR)

Coordinate transformations

Coordinate transformation	Activate	Cancel
Datum shift	G54 X+20Y+30 Z+10	G54 X+0Y+0 Z+0
Mirror image	G28 X	G28
Rotation	G73 H+45	G73 H+0
Scaling factor	G72 F0,8	G72 F1
Machining plane	G 80 A+10 B+10 C15	G80

Q-parameter definitions

D	Function
00	Assign
01	Addition
02	Subtraction
03	Multiplication
04	Division
05	Root
06	Sine
07	Cosine

D	Function
08	Root sum of squares $c = \sqrt{a^2+b^2}$
09	If equal, go to label number
10	If not equal, go to label number
11	If greater than, go to label number
12	If less than, go to label number
13	Angle from c • sin a and c • cos a)
14	Error number
15	Print
19	Assignment PLC

ISO function overview

TNC 410, TNC 426, TNC 430

M functions

M00 M01 M02	Stop program run/Spindle stop/Coolant off Optional program run interruption (not TNC 426, TNC 430) Stop program run/Spindle stop/Coolant off Delete status display (depending on machine parameter) Go to block 1
M03/M04 M05	Spindle on clockwise / counterclockwise Spindle stop
M06	Tool change, if needed, spindle stop/program run stop
M08/M09	Coolant on / coolant off
M13 M14	Spindle ON clockwise/Coolant ON Spindle ON counterclockwise/Coolant ON
M30	Same as M02
M89	Free miscellaneous function or cycle call, modal
M99	Cycle call, non-modal
M90	Constant contouring speed at inside corners and uncompensated corners
M91 M92	Coordinates in positioning block are referenced to the machine datum Coordinates in positioning block are referenced to a position defined by the machine builder
M94	Reduce display of rotary axis to value under 360°
M97 M98	Path compensation on outside corners: point of intersection instead of transition arc End of path compensation, non-modal
M101 M102	Automatic tool change with sister tool if maximum tool life has expired. Reset M101
M103	Reduce plunging rate to factor F (percent)
M104	Reactivate datum last set in the manual mode of operation (only NC 280 474-xx)
M105 M106	Machining with second k_v factor (not TNC 410) Machining with first k_v factor (not TNC 410)
M107 M108	Suppress error message for replacement tools with oversize (in blockwise transfer, not TNC 410) Reset M107
M109 M110 M111	Constant contouring speed at the tool cutting edge on inside and outside corners Constant contouring speed at the tool cutting edge on inside corners Feed rate refers to the tool path center (standard setting)
M112 M113	Enter contour transition between two contour elements; Enter contour deviation tolerance via T Reset M112 (not TNC 426, TNC 430)
M114 M115	Automatic compensation of machine geometry when working with tilted axes (not TNC 410) Reset M114 (not TNC 410)
M116 M117	Feed rate for rotary axes in mm/min (not TNC 410) Reset M116
M118	Superimpose handwheel positioning during program run (not TNC 410)
M120	Pre-calculate radius-compensated contour (LOOK AHEAD)
M124	Contour filter (not TNC 426, TNC 430)
M126 M127	Shorter-path traverse of rotary axes Reset M126
M128 M129	Maintain the position of the tool tip when positioning with tilted axes (not TNC 410) Reset M128 (not TNC 410)
M130	Moving to position in an untilted coordinate system with a tilted working plane (not TNC 410)
M134 M135	Exact stop at nontangential contour transitions when positioning with rotary axes (not TNC 410) Reset M134 (not TNC 410)
M136 M137	Feed rate F in micrometers per spindle revolution Reset M136
M138	Select tilting axes
M200...M204	Functions for laser cutting machines (not TNC 410)

G functions and addresses

G functions

Tool movements

	G00	Straight-line interpolation, Cartesian coordinates, rapid traverse
	G01	Straight-line interpolation, Cartesian coordinates
	G02	Circular interpolation, Cartesian coordinates, clockwise
	G03	Circular interpolation, Cartesian coordinates, counterclockwise
	G05	Circular interpolation, Cartesian coordinates, without indication of direction
	G06	Circular interpolation, Cartesian coordinates, tangential contour approach
*	G07	Paraxial positioning block
	G10	Straight-line interpolation, polar coordinates, rapid traverse
	G11	Straight-line interpolation, polar coordinates
	G12	Circular interpolation, polar coordinates, clockwise
	G13	Circular interpolation, polar coordinates, counterclockwise
	G15	Circular interpolation, polar coordinates, without indication of direction
	G16	Circular interpolation, polar coordinates, tangential contour approach

Chamfer/Rounding/Approach contour/Depart contour

*	G24	Chamfer with length R
*	G25	Corner rounding with radius R
*	G26	Tangential contour approach with radius R
*	G27	Tangential contour departure with radius R

Define the tool

*	G99	With tool number T, length L, radius R

Tool radius compensation

	G40	No tool radius compensation
	G41	Tool radius compensation, left of the contour
	G42	Tool radius compensation, right of the contour
	G43	Paraxial compensation for G07, lengthening
	G44	Paraxial compensation for G07, shortening

Blank form definition for graphics

	G30	(G17/G18/G19) min. point
	G31	(G90/G91) max. point

Drilling cycles

	G83	Pecking
	G84	Tapping with a floating tap holder
	G85	Rigid tapping
	G86	Thread cutting (not TNC 410)
	G200	Drilling
	G201	Reaming
	G202	Boring
	G203	Universal drilling
	G204	Back boring
	G205	Universal peck drilling (only NC software 280 474-xx)
	G206	Tapping with floating tap holder (only NC software 280 474-xx)
	G207	Rigid tapping (only NC software 280 474-xx)
	G208	Bore milling (only NC software 280 474-xx)

Cycles for milling pockets, studs and slots

	G74	Slot milling
	G75	Rectangular pocket milling in clockwise direction
	G76	Rectangular pocket milling in counterclockwise direction
	G77	Circular pocket milling in clockwise direction
	G78	Circular pocket milling in counterclockwise direction
	G210	Slot milling with reciprocating plunge
	G211	Round slot with reciprocating plunge
	G212	Rectangular pocket finishing
	G213	Rectangular stud finishing
	G214	Circular pocket finishing
	G215	Circular stud finishing

Cycles for creating point patterns

	G220	Circular pattern
	G221	Linear pattern

SL cycles, group 1

	G37	Contour geometry, list of subcontour program numbers
	G56	Pilot drilling
	G57	Rough-out
	G58	Contour milling in clockwise direction (finishing)
	G59	Contour milling, counterclockwise (finishing)

SL cycles, group 2 (not TNC 410)

	G37	Contour geometry, list of subcontour program numbers
	G120	Contour data (applies to G121 to G124)
	G121	Pilot drilling
	G122	Rough-out
	G123	Floor finishing
	G124	Side finishing
	G125	Contour train (machining open contour)
	G127	Cylinder surface

G functions

Coordinate transformations

	G53	Datum shift in datum table
	G54	Datum shift in program
	G28	Mirror image
	G73	Rotation of the coordinate system
	G72	Scaling factor (reduce or enlarge contour)
	G80	Tilting the Working Plane

Cycles for multipass milling

	G60	Running point tables (not TNC 410)
	G230	Multipass milling of smooth surfaces
	G231	Multipass milling of tilted surfaces

Special Cycles

*	G04	Dwell time with F seconds
	G36	Oriented spindle stop
*	G39	Program call
	G62	Tolerance deviation for fast contour milling (not TNC 410)

Define machining plane

	G17	Working plane: X/Y; tool axis: Z
	G18	Working plane: Z/X; tool axis: Y
	G19	Working plane: Y/Z; tool axis: X
	G20	Tool axis IV

Dimensions

	G90	Absolute dimensions
	G91	Incremental dimensions

Unit of measure

	G70	Inches (set at start of program)
	G71	Millimeters (set at start of program)

Other G functions

	G29	Transfer the last nominal position value as a pole (circle center)
	G38	Program run STOP
*	G51	Next tool number (with central tool file)
	G55	Programmable probing function
*	G79	Call the cycle
*	G98	Set label number

*) Non-modal function

Addresses

Add.	Function
%	Start of program
%	Program call
#	Datum number with G53
A	Rotation about X-axis
B	Rotation about Y-axis
C	Rotation about Z-axis
D	Q-parameter definitions
DL	Wear compens. of length with T
DR	Wear compens. of radius with T
E	Tolerance with M112 and M124
F	Feed rate
F	Dwell time with G04
F	Scaling factor with G72
F	Reduction factor F with M103
G	G functions
H	Polar coordinate angle
H	Angle of rotation with G73
H	Max. permissible angle with M112
I	X coordinate of the circle center/pole
J	Y coordinate of the circle center/pole
K	Z coordinate of the circle center/pole
L	Setting a label number with G98
L	Jump to a label number
L	Tool length with G99

Add.	Function
M	M functions
N	Block number
P	Cycle parameters in machining cycles
P	Value or Q parameter in Q-parameter definition
Q	Q parameter
R	Polar coordinate radius
R	Circular radius with G02/G03/G05
R	Rounding radius with G25/G26/G27
R	Tool radius with G99
S	Spindle speed
S	Oriented spindle stop with G36
T	Tool definition with G99
T	Tool call
T	Next tool with G51
U	Axis parallel to X axis
V	Axis parallel to Y axis
W	Axis parallel to Z axis
X	X axis
Y	Y axis
Z	Z axis
*	End of block

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

318 179-24 · 4/99 · pdf · Printed in Germany · Subject to change without notice